Operator's Manual

WavePro® 7 Zi
SDA 7 Zi
DDA 7 Zi

LeCroy

WavePro 7Zi

Oscilloscope

Operator's Manual

August, 2008

LeCroy Corporation
700 Chestnut Ridge Road
Chestnut Ridge, NY, 10977-6499
Tel: (845) 578-6020, Fax: (845) 578 5985

Warranty

NOTE: THE WARRANTY BELOW REPLACES ALL OTHER WARRANTIES, EXPRESSED OR IMPLIED, INCLUDING BUT NOT LIMITED TO ANY IMPLIED WARRANTY OF MERCHANTABILITY, FITNESS, OR ADEQUACY FOR ANY PARTICULAR PURPOSE OR USE. LECROY SHALL NOT BE LIABLE FOR ANY SPECIAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES, WHETHER IN CONTRACT OR OTHERWISE. THE CUSTOMER IS RESPONSIBLE FOR THE TRANSPORTATION AND INSURANCE CHARGES FOR THE RETURN OF PRODUCTS TO THE SERVICE FACILITY. LECROY WILL RETURN ALL PRODUCTS UNDER WARRANTY WITH TRANSPORT PREPAID.

The oscilloscope is warranted for normal use and operation, within specifications, for a period of three years from shipment. LeCroy will either repair or, at our option, replace any product returned to one of our authorized service centers within this period. However, in order to do this we must first examine the product and find that it is defective due to workmanship or materials and not due to misuse, neglect, accident, or abnormal conditions or operation.

LeCroy shall not be responsible for any defect, damage, or failure caused by any of the following: a) attempted repairs or installations by personnel other than LeCroy representatives or b) improper connection to incompatible equipment, or c) for any damage or malfunction caused by the use of non-LeCroy supplies. Furthermore, LeCroy shall not be obligated to service a product that has been modified or integrated where the modification or integration increases the task duration or difficulty of servicing the oscilloscope. Spare and replacement parts, and repairs, all have a 90-day warranty.

The oscilloscope's firmware has been thoroughly tested and is presumed to be functional. Nevertheless, it is supplied without warranty of any kind covering detailed performance. Products not made by LeCroy are covered solely by the warranty of the original equipment manufacturer.

Internet: www.lecroy.com

Manufactured under an ISO 9000 Registered Quality Management System. Visit www.lecroy.com to view the certificate.		This electronic product is subject to disposal and recycling regulations that vary by country and region. Many countries prohibit the disposal of waste electronic equipment in standard waste receptacles. For more information about proper disposal and recycling of your LeCroy product, please visit www.lecroy.com/recycle.

916494 RevA

TABLE OF CONTENTS

Welcome

Thank you for purchasing a LeCroy product. We're certain you'll be pleased with the detailed features so unique to our instruments.

This WavePro 700Zi Operator's Manual provides information in the following manner:

WavePro 700Zi Features

- First, we cover functions like **Cable De-Embedding**, **Sequence Sampling Mode**, **Spectrum Analyzer**, and **Trigger Scan**.
- Then, we acquaint you with your new instrument in a section named **Getting to Know WavePro**.

  This section contains two sub-sections showing the **Hardware** (explaining the physical features of your new instrument) and **Basic Controls** (demonstrating the relationship between some front panel and screen layout controls) of the instrument.

Comprehensive Core Functions (1000Base-T through Vertical)

The second section covers all the core functions of your instrument.

Compatible Options and Accessories

The next sections cover related **Options** and **Accessories** available for your product.

Reference

We've set aside this Reference section to contain contact information for various LeCroy offices. Here, we cover items like the **Specification**, **Technical Support** contact information, and **Safety Requirements**.

Support

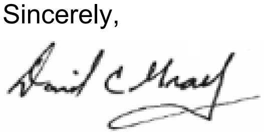
When your product is delivered, verify that all items on the packing list or invoice copy have been shipped to you. Contact your nearest LeCroy customer service center or national distributor if anything is missing or damaged. If there is something missing or damaged, and you do not contact us immediately, we cannot be responsible for replacement. If you have any problems with your product, please refer to the Technical Support contacts located in the Reference section. It also contains the product Specification and Safety Requirements.

The Online help (located on the Help menu on your instrument) contains most of the more detailed information found in this manual.

Thank You

We truly hope these materials provide increased comprehension when using LeCroy's fine products.

Sincerely,



David C. Graef

LeCroy Corporation
Vice President and Chief Technology Officer

Cable De-Embedding

When making measurements on serial data signals, losses in the cables used in the test setup can reduce the accuracy of your signal (for example, signal amplitude and risetime), as well as introduce Inter-Symbol Interference. These cable effects can dramatically alter your serial data measurements and potentially create mask test violations. The Cable De-Embedding option allows you to quickly specify the characteristics of the cables (typically found on the cable's data sheet) in your test setup and analyze your signal with the effects of the cables removed.

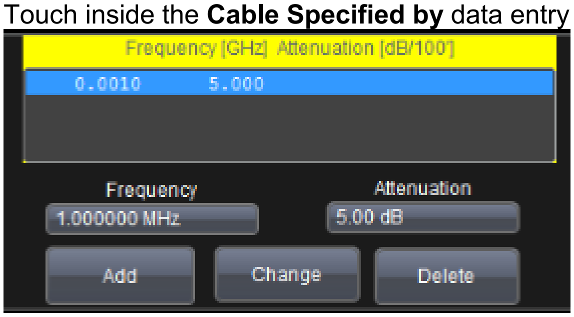
Setting Up Cable De-Embedding

Each of the four Channel menus has its own Cable De-Embedding dialog so that you can individually describe the cable that is being used on each channel. You can specify your cable characteristics by entering either the attenuation table for the cable or two attenuation constants of the loss model for the cable (provided by the cable manufacturer). Then, you must specify the delay and physical length of the cable that you are using in your setup, and you are ready to view your corrected signal. When you remove the effects of the cable, the compensated signal is a more accurate representation of what was actually transmitted.

1. Touch **Vertical** → **Channel***x* **Setup** from the menu bar.
2. On the **Channel***x* **Setup** dialog, touch to place a check mark in the **Cable De-Embedding** checkbox. A **Cable De-Embedding** tab will be displayed.
3. Touch the **Cable De-Embedding** tab.
4. Touch inside the **Cable Specified by** data entry field and select **Attenuation Constants** if you want to specify your cable characteristics by entering the two attenuation constants of the loss model for the cable (provided by the cable manufacturer).

   OR

   Touch inside the **Cable Specified by** data entry field and select Attenuation Table

   if you want to specify your cable characteristics by entering the attenuation table for the cable (provided by the cable manufacturer). Enter an **Attenuation** value and a **Frequency** and click **Add** to add them to the Attenuation Table. If you want to edit a row in the table, highlight the row, change the **Attenuation** and **Frequency** and click **Change**. If you want to delete a row, highlight the row and click **Delete**.
5. Touch inside the **Delay Specified by** data entry field to specify the delay. The cable delay can be Propagation **Velocity**, **Nominal Delay**, or a **Dielectric Constant**.
6. Touch inside the **Cable Length** data entry field and enter the physical length of the cable in inches.
7. Click **Apply** to initiate the cable de-embedding for the signal on this channel so that the effects of the cable are removed.

Saving Cable Configurations

You should save cable configurations so that you can easily load them at another time. The current cable configuration will not be preserved after exiting the application unless you save it.

1. On the **Cable De-Embedding** dialog, touch inside the **File Name [DefaultCable]** data entry field and enter a file name using the pop-up keyboard.

   OR

   Touch the **Browse** button and select a location and file name.
2. Touch the **Save** button.

Note: You can load previously saved cable configurations by touching the **Browse** button, locating the file and then clicking **Load**.

Sequence Sampling Mode – Working with Segments

Using Sequence Mode, you can store up to 15,000 triggered events as "segments" into the oscilloscope's memory. This is ideal when capturing many fast pulses in quick succession or when capturing few events separated by long time periods. The instrument can capture in fine detail complicated sequences of events over large time intervals, while ignoring the uninteresting periods between the events. You can also make time measurements between events on selected segments using the full precision of the acquisition timebase.

Sequence mode offers a number of unique capabilities:

- You can acquire up to four channels simultaneously.
- You can minimize dead time between trigger events for consecutive segments.
- You can view time stamps for acquisitions.
- You can zoom segments or used them as input to math functions.
- You can combine sequence mode with an advanced trigger to isolate a rare event, capture all instances over hours or days, and view/analyze each afterwards.
- You can use Sequence mode in remote operation to take full advantage of the instrument's high data-transfer capability.

In Sequence mode, the complete waveform consists of a number of fixed-size segments acquired in single-shot mode (see the instrument specifications for the limits). The oscilloscope uses the sequence timebase setting to determine the capture duration of each segment: 10 x time/div. Along with this setting, the oscilloscope uses the desired number of segments, maximum segment length, and total available memory to determine the actual number of samples or segments, and time or points.

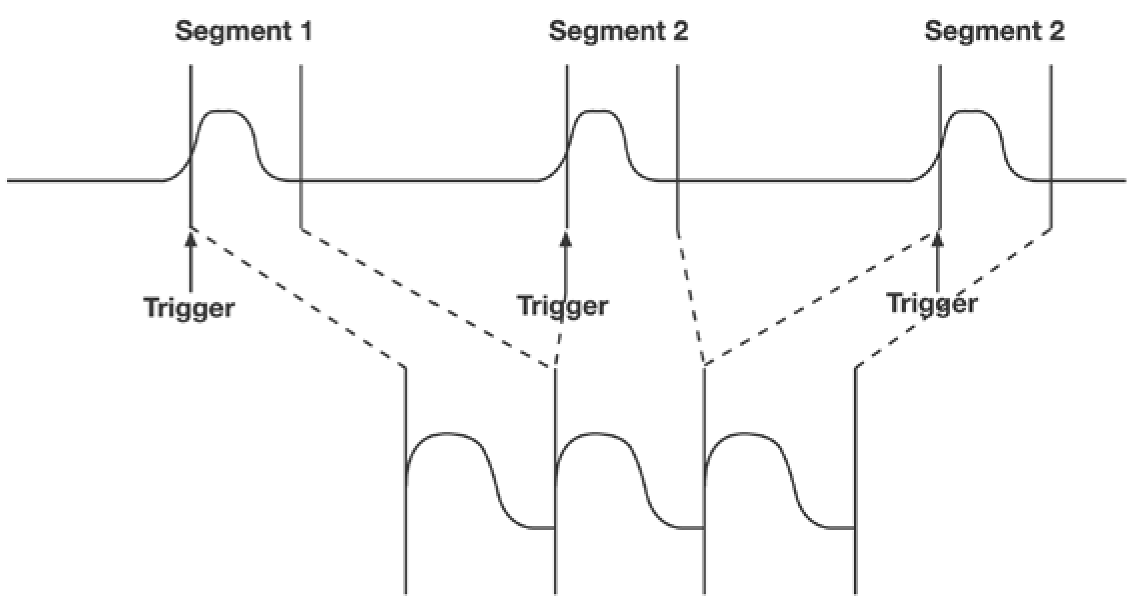
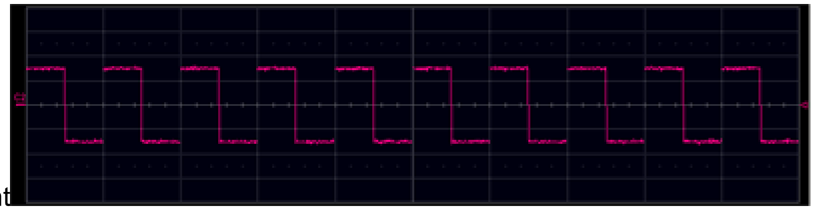
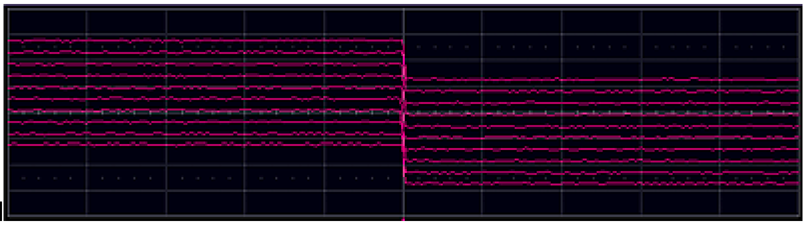
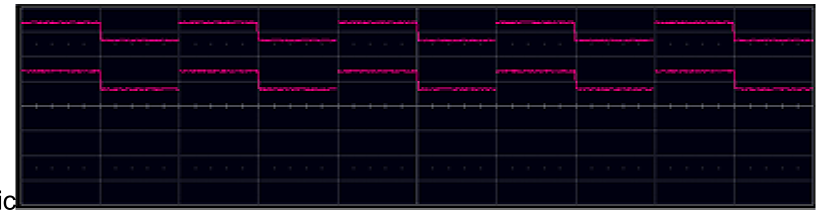
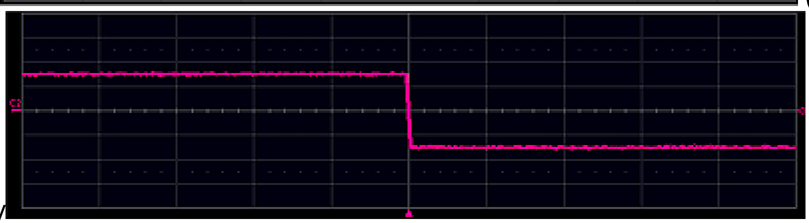
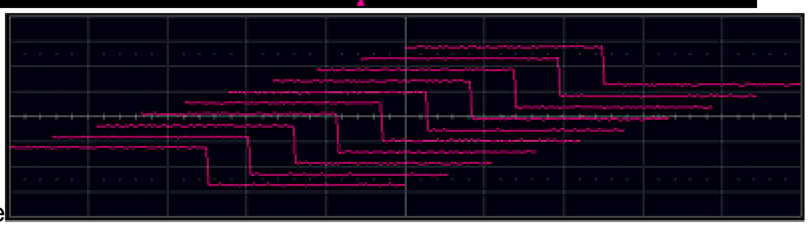


Figure 3-1. How the instrument captures segments

Sequence Display Modes

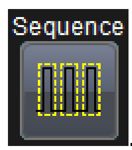
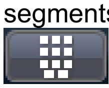
The instrument gives you a choice of five ways to display your segments:

- Adjacent 
- Waterfall  (cascaded)
- Mosaic  (tiled)
- Overlay 
- Perspective 

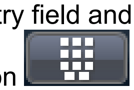
Setting Up Sequence Mode

When setting up Sequence Mode, you define the number of fixed-size segments acquired in single-shot mode (see the instrument specifications for the limits). The oscilloscope uses the sequence timebase setting to determine the capture duration of each segment. Along with this setting, the oscilloscope uses the number of segments, maximum segment length, and total available memory to determine the actual number of samples or segments, and time or points.

Figure 3-2. Setting up Sequence Mode (Adjacent)

1. Touch **Timebase** → **Horizontal Setup** on the menu bar.
2. Under **Sampling Mode**, touch the **Sequence** mode button .
3. Click the **Sequence** tab.
4. Under **Acquisition Settings**, touch inside the **Num Segments** data entry field and enter the number of segments you want to display, using the slider bar at the bottom of the window. Click the keypad button  on the slider bar to enter a value using the pop-up numeric keypad.

> **Note:** The number of segments you choose to display (80 maximum) can be less than the total number of segments in the waveform. For example, in the pop-up images above, the number of display segments is 10, but the total number of segments entered in the timebase dialog's **Num Segments** field is 100.

5. Touch the **Enable Timeout** checkbox.
6. Touch inside the **Timeout** data entry field and enter a timeout value, using the slider bar at the bottom of the window. Click the keypad button  on the slider bar to enter a value using the pop-up numeric keypad.

> **Note:** Use the sequence mode timeout to automatically interrupt the sequence acquisition if the timeout value is exceeded without a valid trigger. The timeout period accounts for instances when a **Num Segments** miscount occurs for some reason and the scope waits indefinitely for an unforthcoming segment. During that time, no oscilloscope functions are accessible. By means of a timeout value, however, the acquisition will be completed, the waveform displayed, and control of the oscilloscope returned to the user after the timeout has elapsed.

5. Under **Display Settings**, touch inside the **Display mode** field, and select a sequence mode display from the pop-up menu.
6. Touch the SINGLE trigger front panel button.

Note: Once a single acquisition has started, you can interrupt the acquisition at any time by pressing the **SINGLE** front panel button a second time or by pressing the STOP front panel button. In this case, the segments already acquired will be retained in memory.

Zooming Segments in Sequence Mode

You can zoom individual segments easily using the QUICKZOOM front panel button. When you zoom, the zoom traces default to Segment 1. Channel descriptors indicate the total number of segments acquired. Zoom descriptors indicate [Seg #] and #Segments in the Zoom. You can scroll through the segments using ZOOM front panel position knob.

Figure 3-3. Sample Zoom of Segments in Sequence Mode

1. Touch the front panel **Quickzoom** button .
2. Turn the ZOOM front panel position knob to scroll through the segments.
3. To vary the degree of zoom, touch the newly created **Zx** trace label. The setup dialog for the zoom (Z1 to Z4) opens. It shows the current horizontal and vertical zoom factors.
4. If you want to increase or decrease your horizontal or vertical zoom in small increments, touch the **Var.** checkbox to enable variable zooming. Now with each touch of the zoom control buttons 

, the degree of magnification will change by a small increment.

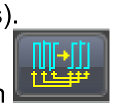
OR

If you want to zoom in or out in large standard increments with each touch of the zoom control buttons, leave the **Var.** checkbox unchecked.

OR

If you want to set exact horizontal or vertical zoom factors, touch inside the Horizontal **Scale/div** data entry field and enter a time-per-div value, using the pop-up numeric keypad. Then touch inside the Vertical **Scale/div** field and enter a voltage value.

Displaying an Individual Segment

1. Touch **Math** → **Math Setup** on the menu bar.
2. Touch a function tab (**F1** to Fx - The number of available math traces depends on the software options loaded on the oscilloscope. Refer to the specifications for details).
3. Touch inside the **Operator1** field and select the **Segment** button  from the pop-up menu.
4. In the dialog on the right, touch the **Select** tab.
5. Touch inside the **First Selected** data entry field and use the slider bat at the bottom of the window to select the first segment you want to display. Click the keypad button  on the slider bar to enter a value using the pop-up numeric keypad.

> **Note:** In Persistence mode, the segments are automatically overlaid one on top of the other in the display. In non-Persistence mode, they appear separately on the grid.

Viewing Time Stamps

You can view time stamps for each segment.

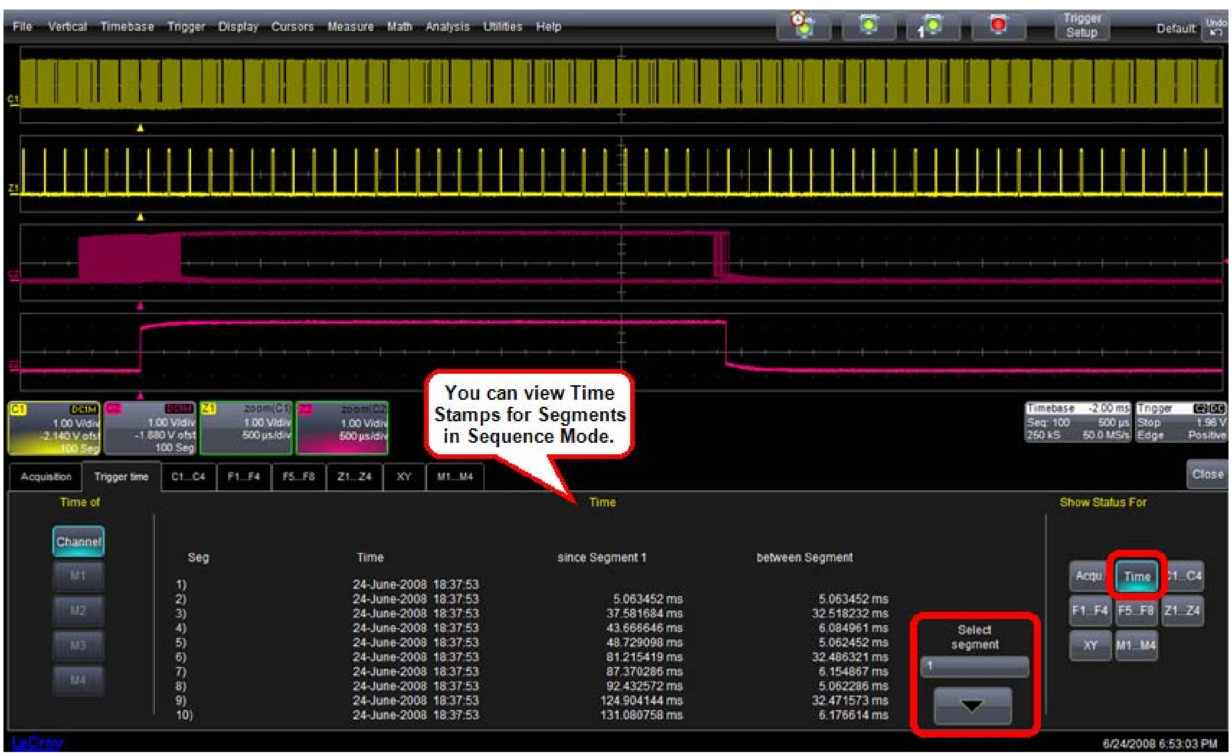


Figure 3-4. View Segment Time Stamps

1. Touch **Timebase** → **Acquisition Status** on the menu bar.

   OR

   Touch **Vertical** → **Channel Status** on the menu bar.
2. Touch the **Trigger Time** tab.
3. Under **Show Status For**, touch the **Time** button.

4. Touch inside the **Select Segment** field and enter a segment number, using the pop-up keypad or touch the arrow buttons to scroll through segment times.

Spectrum Analyzer

The Spectrum Analyzer and Advance FFT option in the WavePro 700Zi series oscilloscopes will help you use the Fast Fourier Transform (FFT) in your measurements. If you are familiar with RF spectrum analyzers, you can start using the FFT with little or no concern about the details of setting up an FFT. The Spectrum Analyzer controls are the same as you would find on an RF spectrum analyzer. You can set the span, center frequency and resolution bandwidth and the oscilloscope automatically configures the acquisition and FFT controls to obtain the desired spectrum view.

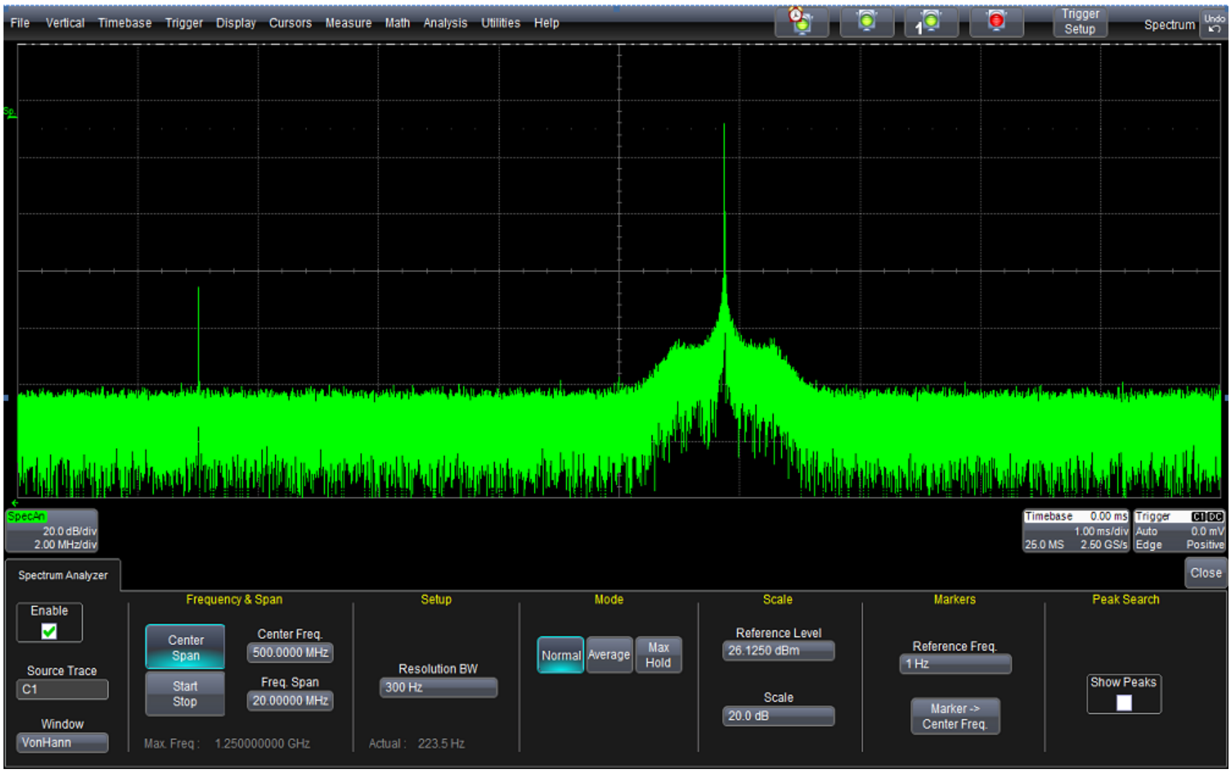
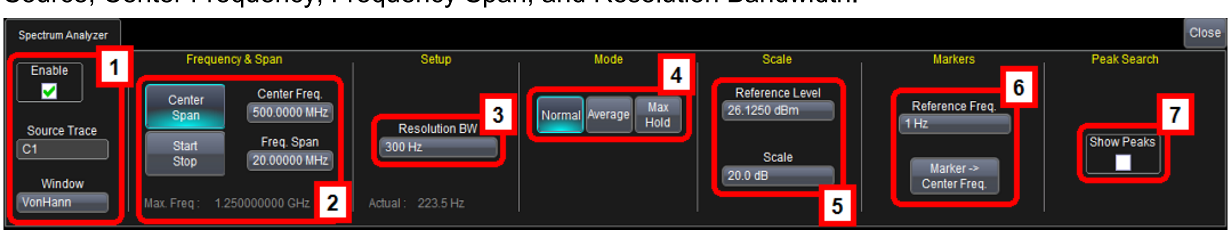


Figure 3-5. Sample dialog and display for the Spectrum Analyzer

Running the Spectrum Analyzer

You can run the Spectrum Analyzer by touching **Analysis** → **Spectrum Analyzer** from the menu bar. When you run the Spectrum Analyzer, you set up controls in the Spectrum Analyzer dialog from left to right, including a Source, Center Frequency, Frequency Span, and Resolution Bandwidth.



Number	Description
1	Enable the **Spectrum Analyzer** and choose the **Source Trace**.
2	**Center Span** is similar to adjusting the position of the FFT zoom trace. You enter the **Center Frequency** and the **Frequency Span**. **Frequency Span** is similar to adjusting the zoom scale of the FFT. This does not change the sample rate or memory. The Spectrum Analyzer reports the maximum frequency that can be observed. ***OR***

Number	Description
	Start Stop provides another way to adjust the position of the FFT zoom trace. You can specify the Starting and Stopping Frequencies
3	**Resolution Bandwidth** is equivalent to changing the Timebase setting to increase or decrease memory in FFT mode. Reducing the Resolution Bandwidth equals more memory. The Spectrum Analyzer reports back adapted values for the resolution bandwidth if the value entered is not achievable.
4	There are three modes: **Normal**, **Average**, and **Max Hold**. In Averaging mode, you can enter the number of spectra to be averaged. Averaging is effective in reducing the noise of the signal to see more of the harmonic or carrier detail. Max (Peak) Hold mode is useful for swept frequency measurements where it shows the history of peak values across the frequency axis. Max Hold shows the maximum level the signal reaches. it is also useful for finding infrequent spurs.
5	**Reference Level** sets the amplitude of the top of the screen. **Scale** is the same as adjusting the Vertical Gain knob in FFT mode.
6	Using **Markers**, you can set the reference frequency and display a marker for the center frequency.
7	**Show Peaks** lets you label and tabulate peaks. When Show Peaks is checked, the significant peaks are marked with a frequency and gain stamp and a table listing the peaks, ordered by amplitude from highest to lowest is displayed. This is very useful in identifying harmonics or peaks that may be hidden due to poor resolution.

TriggerScan

TriggerScan is a debugging tool that helps you quickly find rare waveform glitches and anomalies. With TriggerScan, you can build a list of trigger setups to look for rare events and automatically sequence through each one. TriggerScan can use any type of trigger setup available including edge, width, and qualify as well as Smart Triggers (such as, glitch and runt triggers). TriggerScan automates two key processes in triggering rare events:

1. Trains the system by looking at normal acquired waveforms. During the training, the oscilloscope analyzes the waveforms to determine what waveforms normally look like. Using this information, it generates a list of smart trigger setups to trigger on abnormal situations.
2. Loads the smart trigger setups from the Trainer and cycles through these. As triggers occur, they are overlaid on the screen. All acquisition settings are preserved and you can use all the functions of the oscilloscope to find the root cause of these anomalies including, WaveScan, Histograms, and advanced analysis.

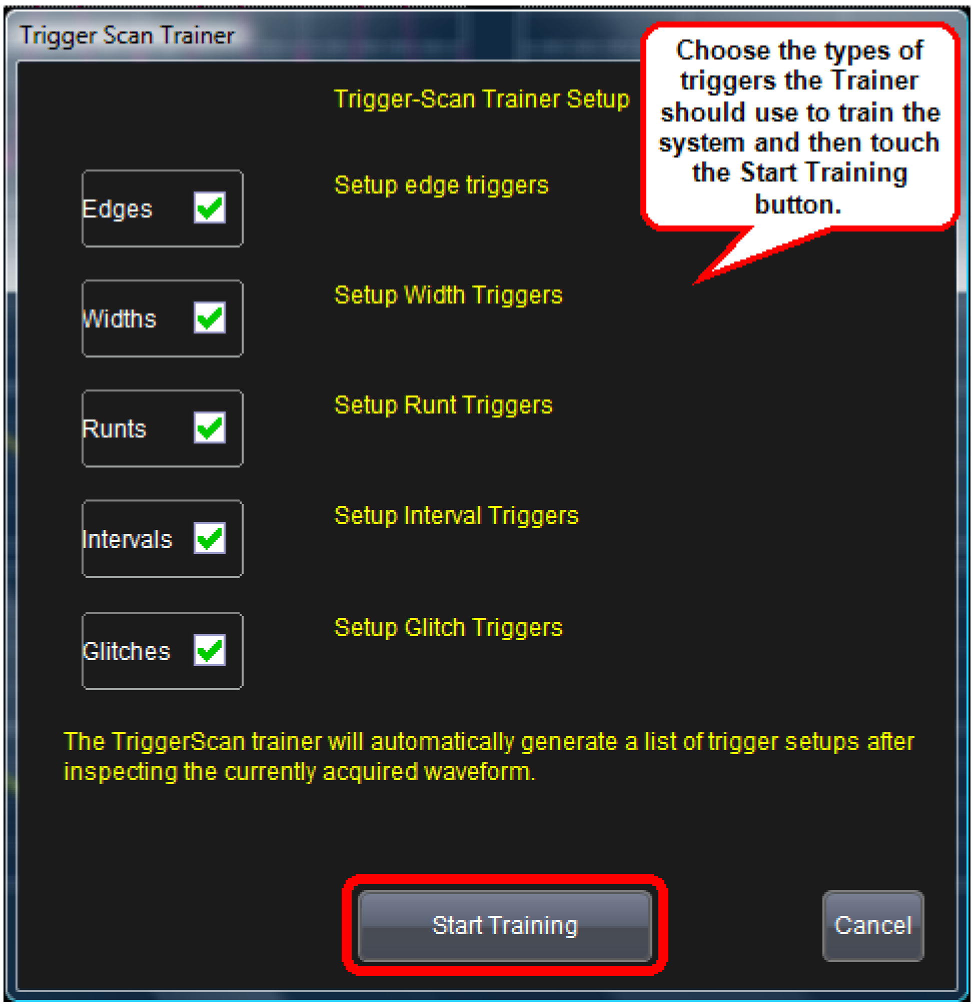
Training TriggerScan

The TriggerScan Trainer inspects a currently acquired waveform and automatically builds a list of common trigger setups used to find rare events.

PLEASE NOTE THE FOLLOWING:

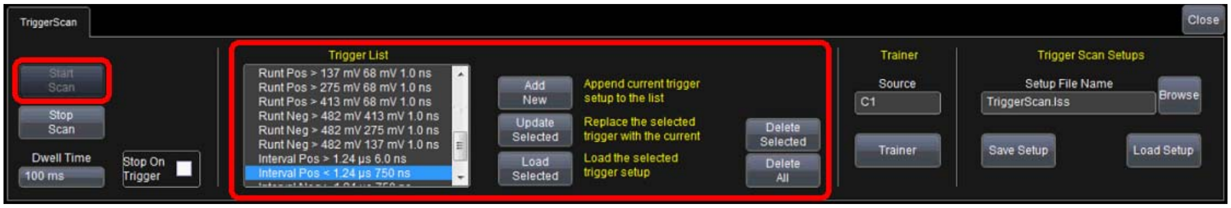
- You must acquire and display at least 3 cycles of a signal before running the Trainer.
- You should run the Trainer if you want to change the trigger types or if you change the channel or signal.

1. Touch **Trigger** → **Trigger Setup** from the menu bar.
2. On the **Trigger** dialog, click the **TriggerScan** tab.
3. Touch inside the **Source** data entry field and select a channel as the source for the training.
4. Touch the **Trainer** button.
5. On the **TriggerScan Trainer Setup** window, choose the types of triggers the Trainer should use to train the system and then touch the **Start Training** button. The training begins. When it is complete, a list of smart trigger setups is displayed in the Trigger List.



Starting TriggerScan

After you have run the Trainer, the Trigger List displays a list of smart trigger setups. You can add or remove trigger setups. You can also update the selected smart trigger setup. Once you have made any changes to the Trigger List, you are ready to start scanning.



1. Touch **Trigger** → **Trigger Setup** from the menu bar.
2. On the **Trigger** dialog, click the **TriggerScan** tab.
3. If you want to add a new trigger setup, touch the **Trigger** tab and set the new trigger. Then, touch the **Add New** button to add the new trigger to the Trigger List.
4. If you want to delete a trigger setup, highlight the setup in the Trigger List and touch the **Delete Selected** button.

Note: If you want to delete all trigger setups in the Trigger List, touch the **Delete All** button.

5. If you want to replace the selected trigger setup with the current trigger setup, highlight the setup in the Trigger List and touch the **Update Selected**button.
6. Once you have made any changes to the Trigger List, touch Start Scan. The oscilloscope automatically sequences through all the trigger setups.

PLEASE NOTE THE FOLLOWING:

- You can tune the dwell time that the scope will wait before loading the next trigger setup using the **Dwell Time** data entry field.
- If you have Persistence display mode enabled, all trigger events are recorded on the display. Refer to the **Persistence Setup** topic for instructions on enabling Persistence display mode.
- If you want TriggerScan to stop when the scope triggers next, check the **Stop On Trigger** checkbox. You can use this to isolate specific trigger setups.

Saving TriggerScan Setups

You should save TriggerScan setups once you have made any modifications to the Trigger List. The current Trigger List will not be preserved after exiting the application unless you save it.

1. On the **TriggerScan** dialog, touch inside the **Setup File Name** data entry field and enter a file name using the pop-up keyboard.

   OR

   Touch the **Browse** button and select a location and file name.
2. Touch the **Save Setup** button.

Note: You can load previously saved TriggerScan setups by touching the **Browse** button, locating the file and then clicking **Load Setup**.

Front Panel

Numbered labels on this graphic correspond with descriptions in the table.

Number and Description	Number and Description	Number and Description
1. Power Button 2. Channel Row LED Indicator 3. Channel Inputs 4. Auxiliary Input and Output	5. Volume Control and Mute Button 6. Ground Connector 7. Speaker 8. Fast Edge, Recovered Clock, and Data Outputs	9. USB Ports 10. Detachable Front Panel-Remote Control 11. Release Switch for Front Panel-Remote Control

Detaching and Attaching the Front Panel

Detach the front panel from the oscilloscope by sliding the detachment lever to the left and pulling at the right.

Attach the front panel by inserting the lower part first, sliding the detachment lever to the left, and then pushing the top in place.

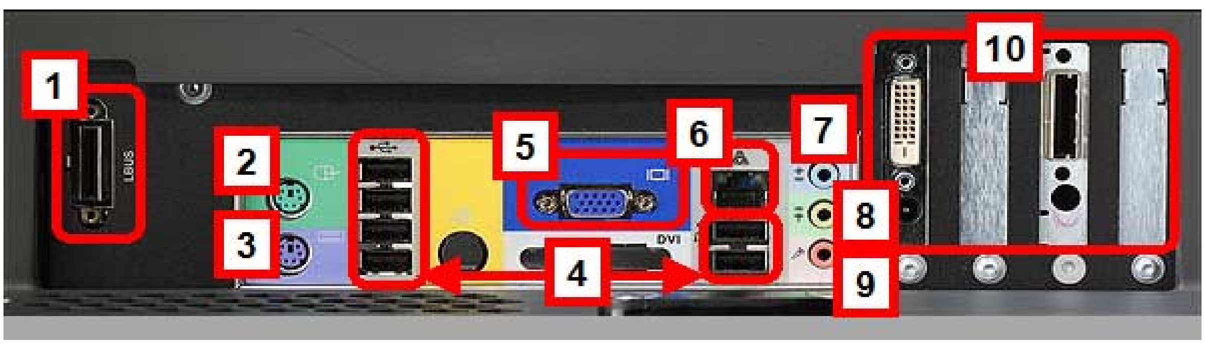
Front Panel as a Remote Control

While detached, the front panel can then act as a remote control. Just plug-and-play connect it to the oscilloscope using a **USB - A** to **USB - Mini B** cable.

Side Panel

The WavePro Zi side panel is located on the right side (facing the front of the instrument) as follows:

Numbered labels on this graphic correspond with descriptions in the table.

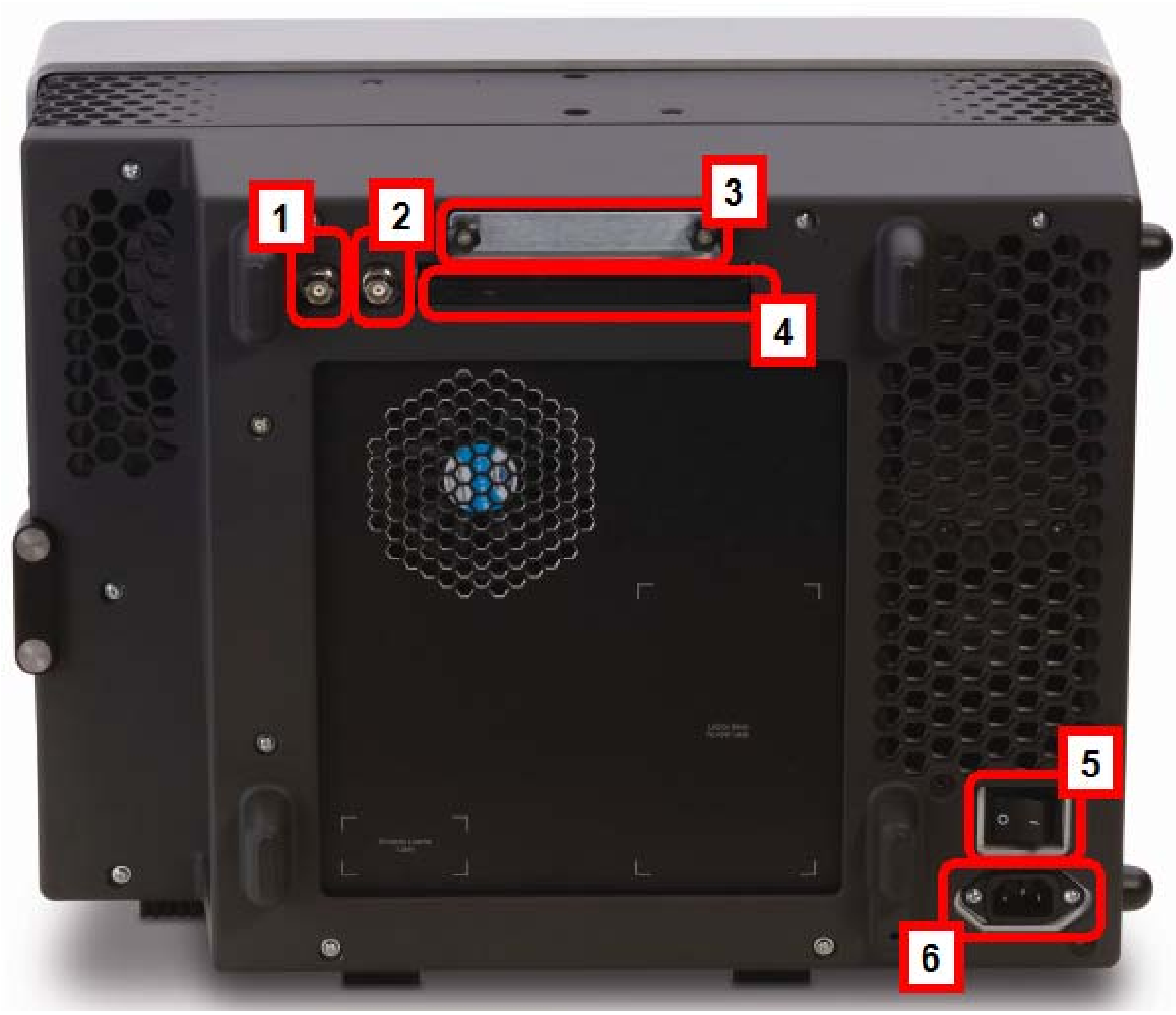


Number and Description	Number and Description	Number and Description
1. LBUS (LeCroy Bus) 2. Mouse 3. Keyboard 4. USB Ports	5. External VGA Monitor 6. Ethernet Port 7. Line In	8. Speakers 9. Microphone 10. PCI Expansion Slots for DVI (for standard display hardware), LSIB, and other options (LeCroy External Display WPZi-EXTDISP-15 option).

Note: Consult your system administrator when connecting to an internal LAN.

Back Panel

Numbered labels on this graphic correspond with descriptions in the table.



Number and Description	Number and Description	Number and Description
1. External Clock Input (Grounded EMI Shield required when port is not in use) 2. External Clock Output	3. Removable Hard Drive 4. DVD-CD + R Drive	5. Power Switch 6. DC Power Plug

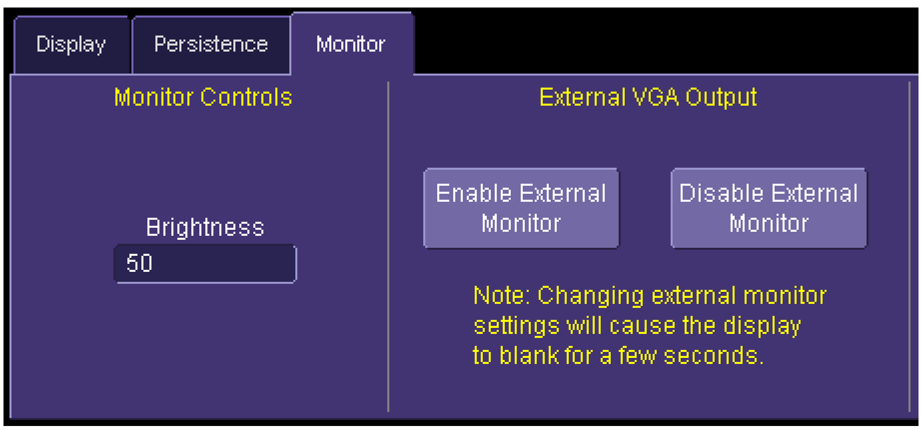
External Monitor

Setting up the External Monitor involves a connection and a few touch screen selections. It's a hot swap connection, so there's no need to restart the instrument once you've connected.

1. Plug your external monitor (WPZi-EXTDISP15 option) display into the **DVI** and **DC 12** connections on the PCI slot (located on the side of the oscilloscope). Connect your **USB** plug (where applicable) to an available port (also on the side of the instrument).

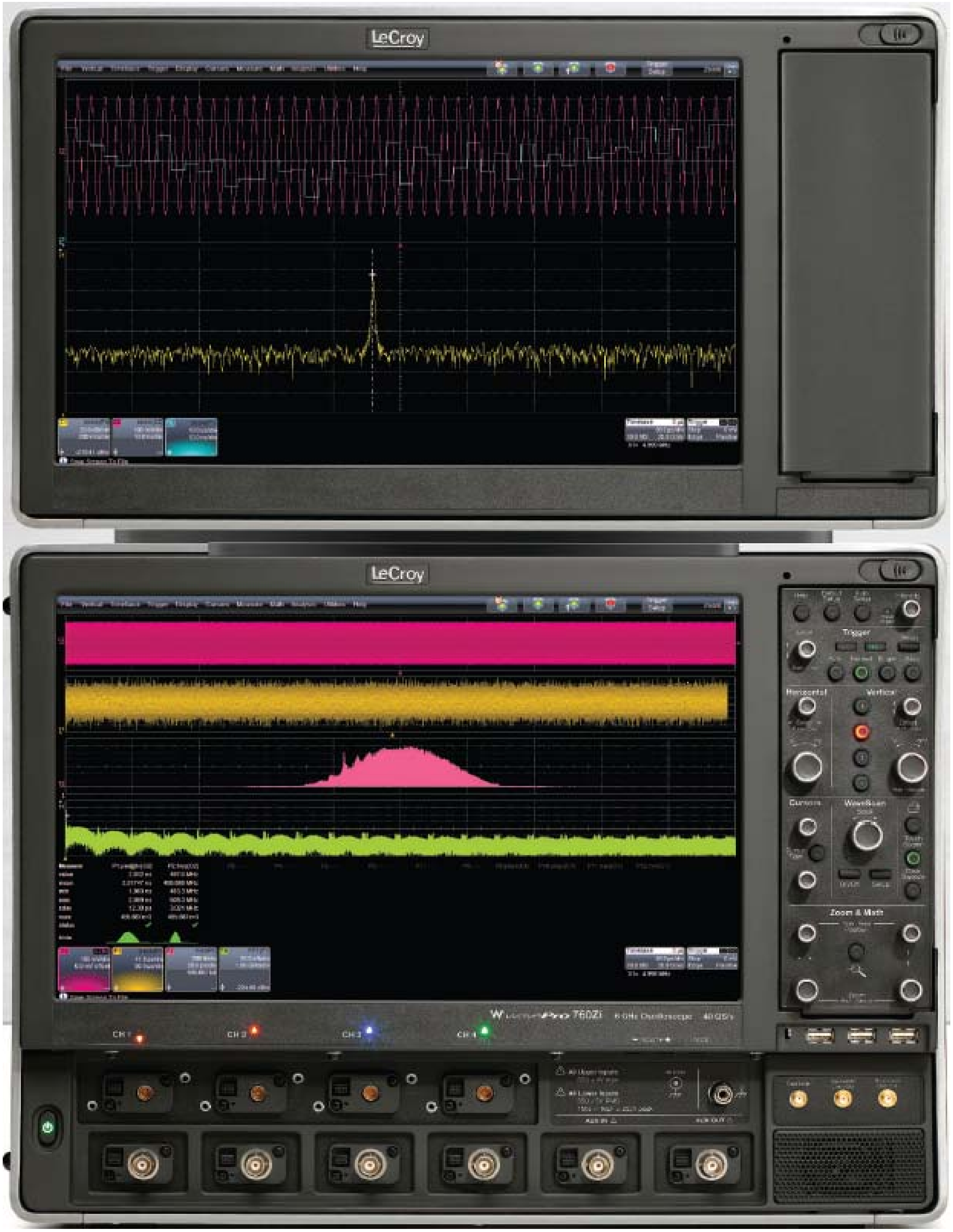
Note: To connect a standard VGA monitor, connect to the VGA Port.

2. Turn on the oscilloscope, let the instrument boot and then touch **Display** → **Display Setup...** from the menu bar.
3. Touch the **Monitor** tab and then the **Enable External Monitor** button.



4. Touch inside the **Brightness** field and adjust brightness as necessary.

The following picture shows a WavePro 700Zi with the LeCroy external monitor attached (optional WPZi-EXTDISP15).



Dual Channel Acquisition

Combining Channels

Channels can be combined to increase sample rate, memory, or both in order to capture and view a signal in all its detail. When you combine channels, uncombined channels like EXT BNC remain available for triggering, even though they are not displayed.

In 2-channel operation, channels 2 and 3 are made active. Channels 1 ***or*** 2, and 3 ***or*** 4 can be used in Auto operation for 20 GS/s. When channels are paired in the following combinations, the maximum sampling rate is doubled and the record length is greatly increased:

Channel Combinations Receiving 40 GS/s
Channels 1 and 3
Channels 1 and 4
Channels 2 and 3
Channels 2 and 4

Note: These sample rates apply to all WavePro Zi oscilloscopes (except the 715Zi without the WPZi 1GHz 4x20 GS option).

In short, sampling can be maximized to 40GS/s using two channel combinations except for channels 1 ***and*** 2, or channels 3 ***and*** 4 (which always yield 20 GS/s. Use best practice by choosing channel 1 or 2 for your first input, and channel 3 or 4 for the second.

Refer to Acquisition Modes in the specifications for maximum sample rates.

Combining Channels Procedure

Set up channel combinations on the interface as follows:

1. Touch **Timebase** → **Horizontal Setup** on the menu bar and the **Horizontal Setup** dialog is shown.
2. Under **Active Channels**, touch **4**, **2** or **Auto**. The maximum sample rate is shown alongside each button.

Hardware and Software Controls

The following Basic Control topics cover the general usage of the **hardware** buttons located on the oscilloscope's front panel and the screen control interface elements of the **software**.

Front Panel Controls

Note: Some front panel controls correspond with screen layout controls in specific ways. For example, the **Print** front panel general control button corresponds with the **Hardcopy** function at **Utilities** → **Utilities Setup** → **Hardcopy**.

The detachable WavePro Zi front panel.

Front Panel Groupings

The front panel is divided into sections based on functions. The following sections explain them in a bit more detail.

Miscellaneous Setup Controls

- **Help** - Press to open the LeCroy Online Assistant where you can click to open the oscilloscope online help table of contents, index, or search for a topic using a keyword. If the second monitor is installed, the online help opens on the second monitor.
- **Default Setup** - Press to reset the oscilloscope's settings to the default configuration. Corresponds with screen menu selection: **File** → **Recall Setup** → **Recall Default Setup**. For a list of default settings, see **Save/Recall** → **Saving and Recalling Scope Settings**.
- **Auto Setup** - Press once and the **Auto Setup** flyout menu opens. Press the **Auto Setup** button on the flyout menu to perform a full auto setup. Press a **Channel Find Scale** button on the flyout menu to perform a quick auto setup for that channel only. Press the AUTO SETUP front panel button twice to perform the last selection from the **Auto Setup** flyout menu (the default is to perform a full auto setup).
- **WaveStream** - Indicates when the acquisition mode is set to WaveStream.
- **Intensity** - Press to toggle between the most recently selected Acquisition/Sampling Mode and WaveStream mode. When you turn the knob, if WaveStream is ON, the WaveStream display intensity changes. When you turn the knob, if WaveStream is OFF, changes the Intensity setting. Corresponds with the screen menu selection: **Display** → **Display Setup**.

Trigger Front Panel Controls

- **Level** - Press to toggle between 50% trigger level and the previous level setting. Turn to change the trigger threshold level. This level is indicated on the Trigger label.
- **Trigger** - Indicates the trigger status as **READY** and **TRIG'D**. READY is lit when the trigger is armed. TRIG'D is lit momentarily when a trigger occurs. A fast trigger rate will cause the light to stay lit continuously.
- **Setup** - Press once to open the **Trigger Setup** dialog. Corresponds with screen menu selection: **Trigger** → **Trigger Setup**. Press the Trigger SETUP front panel button again to close the **Trigger Setup** dialog.
- **Auto** - Press to turn on **Auto Trigger** mode, which triggers the oscilloscope after a time-out, even if the trigger conditions are not met.
- **Normal** - Press to turn on **Normal Trigger** mode, which triggers the oscilloscope each time a signal is present that meets the conditions set for the type of trigger selected.
- **Single** - Press to turn on **Single Trigger** mode, which arms the oscilloscope to trigger once (single-shot acquisition) when the input signal meets the trigger conditions set for the type of trigger selected. If the scope is already armed, it will force a trigger.
- **Stop** - Press to prevent the scope from triggering on a signal. If you boot up the instrument with the trigger in **Stop** mode, the message "no trace available" will be displayed. Press the Trigger AUTO front panel button to display your trace.

Horizontal Front Panel Controls

Note: Horizontal front panel controls correspond with screen menu selection: **Timebase** → **Horizontal Setup**.

- **Delay** - Press to toggle between a zero horizontal delay value and the previous horizontal delay value. Turn to change the horizontal delay value.
- **Time** - Turn to set the time/division of the oscilloscope timebase (acquisition system). The time scale is adjusted from the left edge of the display.

Vertical Front Panel Controls

Note: Vertical front panel controls correspond with screen menu selection: **Vertical** → **Channel Setup**.

- **Channels - Press a Channel 1-4 front panel button to turn the channel on or off. When a Channel 1-4 front panel button is lit, the Vertical Offset and Volts/Div knobs are "active" for that channel only.**
- **Offset** - Press to toggle between a zero vertical offset value and the previous vertical offset value for the selected channel. Turn to change the vertical offset value for the selected channel.
- **Gain** - Press to toggle between fixed and variable gain adjustment. Turn to change the gain value.

Cursors Front Panel Controls

Note: Cursors front panel controls correspond with screen menu selection: **Cursors** → **Cursors Setup**.

- **Absolute Cursor** - Press to set the cursor position to the default 25% position, either horizontal or vertical (depending on which cursor type you are using). Turn to adjust the position of the cursor (absolute cursor or one of the two relative cursors).
- **Cursor Type** - Press one to turn cursors ON to the last cursor type selected. Each subsequent press toggles through the cursor types (**Off**, **Abs Horizontal**, **Rel Horizontal**, **Abs Vertical**, or **Rel Vertical**).
- **Relative Cursor** - Press to set the relative cursor to the default 75% position, either horizontal or vertical (depending on which cursor type you are using)Turn to adjust relative cursors. It does not work with absolute cursor types.

WaveScan Front Panel Controls

Note: WaveScan front panel controls correspond with screen menu selection: **Analysis** → **WaveScan**.

- **Scroll** - If WaveScan is On, turn to scroll through the table of WaveScan-filtered events. If WaveScan is Off, turn to adjust the selected data entry field.
- **On/Off** - Press to toggle between WaveScan On and Off. This is the same as checking **Enable** in the **WaveScan** dialog. When you turn WaveScan On, it turns on the WaveScan to the last enabled mode on the last selected Channel with the last selected Filter method. It also turns on the Z1 trace with its source as the WaveScan Channel and changes whatever display grid was selected to a dual grid with channels in the top grid and the WaveScan Z1 trace in the bottom grid.
- **Setup** - Press once to open the **WaveScan** dialog. Corresponds with screen menu selection: **Analysis** → **WaveScan**. Press the WaveScan SETUP front panel button again to close the **WaveScan** dialog.

General Controls Front Panel Controls

- **Print** - Press once and the **Print** flyout menu opens.
  Press the **Print Now** button on the flyout menu to print the current screen image using the Print settings on the **Utilities** → **Utilities Setup** → **Hardcopy** dialog.
  Press the **Print Setup** button on the flyout menu to open the **Utilities** → **Utilities Setup** →**Hardcopy**.
  Press the **Save Screen to file** button on the flyout menu to save the screen image as a file to storage media such as a USB drive or hard drive using the File settings on the **Utilities** → **Utilities Setup** → **Hardcopy** dialog (incrementing the filename by one each time you touch **Save Screen to File**).
  Press the **Save Screen to Clipboard** button on the flyout menu to save the current screen image to the clipboard.
  Press the **Send Screen to Email** button to send the current screen image using the email settings on the **Utilities** → **Utilities Setup** → **Hardcopy** dialog.
  Press the **Create Notebook Entry** button on the flyout menu to create a new lab notebook entry using the **Enter Notebook Info** window.
  Press the PRINT front panel button twice to perform the last selection from the **Print** flyout menu (the default is to print the screen image).
- **Touch Screen** - Press to toggle the touch screen on and off.
- **Clear Sweeps** - Press to clear data from multiple sweeps (acquisitions) including: persistence trace displays, averaged traces, parameter statistics, and Histicons. During waveform readout, cancels readout. This is the same as pressing **Clear Sweeps** on the **Measure** → **Measure Setup** or **Math** → **Math Setup dialogs**.

Zoom and Math Front Panel Controls

Note: Zoom and Math front panel controls correspond with screen menu selection: **Math** → **Zoom Setup**.

- **Horizontal Position** - Press to reset the horizontal zoom position to zero. Turn to change the horizontal position of the selected math or zoom trace.
- **Horizontal Ratio** - Press to toggle between fixed and variable horizontal zoom ratio adjustment. Turn to change the horizontal zoom ratio of the selected math trace.

- **Quick Zoom** - Press to automatically display magnified views of up to four signal inputs on multiple grids. With four input signals, the signals are displayed along with four zoom traces, each on its own grid. Pressing this button also turns off all other traces.
- **Vertical Position** - Press to reset the vertical zoom position to zero. Turn to change the vertical position of the selected math or zoom trace.
- **Vertical Ratio** - Press to toggle between fixed and variable vertical zoom ratio adjustment. Turn to change the vertical zoom ratio of the selected math trace.

Screen Layout, Groupings, and Controls

The instrument's screen is divided into three main sections:

- **Menu Bar**
- **Signal Display Grid**
- **Dialog(s)**

Note: Some front panel controls correspond with screen layout controls in specific ways. For example, the **Print** front panel general control button corresponds with the **Hardcopy** function at **Utilities** → **Utilities Setup** → **Hardcopy**.

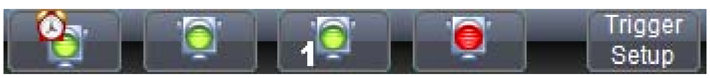
Menu Bar

The top of the screen contains a menu bar of commonly used functions. Whenever you touch one of these buttons and make a selection from its drop-down menu, the dialog area at the bottom of the screen displays the corresponding dialog.

Specific Menu Bar functions are referenced using arrow-separated path descriptions. For example, the **Save Setup** function is referenced as **File** → **Save Setup**.

The Quick Access Toolbar

The Quick Access toolbar is located on the right side of the menu bar. You can use these toolbar buttons to quickly access trigger functions.



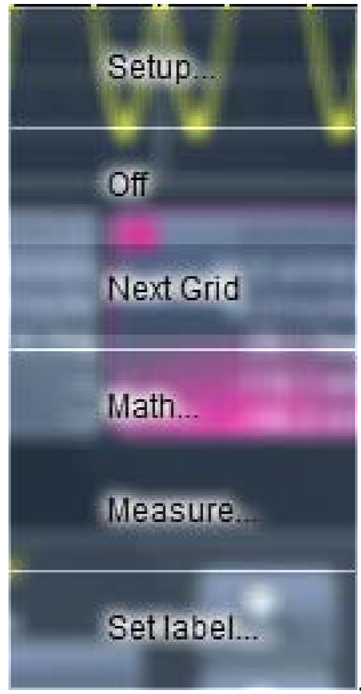
- **Auto** - Press to turn on **Auto Trigger** mode, which triggers the oscilloscope after a time-out, even if the trigger conditions are not met.
- **Normal** - Press to turn on **Normal Trigger** mode, which triggers the oscilloscope each time a signal is present that meets the conditions set for the type of trigger selected.
- **Single** - Press to turn on **Single Trigger** mode for the selected channel, which arms the oscilloscope to trigger once (single-shot acquisition) when the input signal meets the trigger conditions set for the type of trigger selected. If the scope is already armed, it will force a trigger.
- **Stop** - Press to prevent the scope from triggering on a signal. If you boot up the instrument with the trigger in **Stop** mode, the message "no trace available" will be displayed.
- **Trigger Setup** - Press to open the **Trigger Setup** dialog. Corresponds with screen menu selection: **Trigger** → **Trigger Setup**.

The Signal Display Grid

You can set up the signal display area by touching **Display** → **Display Setup** from the menu bar. The **Display** dialog offers a choice of grid combinations and can also set the grid intensity.

The following Display Grid features also provide assistance when using the oscilloscope:

- **Descriptor Labels** - For more information, go to the **Trace Descriptors** topic.

- **Pop-up Menu** - You can click on a waveform to open a pop-up menu 

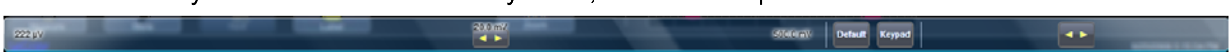
. From this pop-up menu, you can perform the following functions:
    - Open the Setup dialog for the trace
    - Turn the trace descriptor label off
    - Open the Math dialog for the trace
    - Open the Measure dialog for the trace
    - Annotate the selected trace

Dialog Area

The lower portion is where information is shown, selections are made, and data is input. Typically they are organized into tabular displays, subtabs, or pop-up dialogs. The dialog area is controlled by touch screen buttons and front panel buttons.

The following Dialog Area controls also provide assistance when entering data:

- **Slider Bar** - When you click in some data entry fields, a slider bar opens at the bottom of the screen.

  You can use the slider bar on the left to make fine adjustments to the value. You can use the value slider on the right to make coarse adjustments to the value. You can click on the value slider to set the field to a specific value. Click the **Default** button on the slider bar to set the field to the default value.

  Click the keypad button  on the slider bar to set the value using the pop-up numeric keypad.
- **Flyout Menu** - The Print and Auto Setup front panel buttons open flyout menus. For more information on these menus, go to the **Front Panel Groupings** topic.

Universal ProBus/ProLink Interface

LeCroy's ProBus probe system provides a complete measurement solution from probe tip to oscilloscope display. ProBus allows you to control gain and offset directly from your front panel. It is particularly useful for voltage, differential, and current active probes. It uploads gain and offset correction factors from the ProBus EPROM's and automatically compensates to achieve fully calibrated measurements.

This intelligent interconnection between your instrument and a wide range of accessories offers important advantages over standard BNC and probe ring connections. ProBus ensures correct input coupling by auto-sensing the probe type, thereby eliminating the guesswork and errors that occur when attenuation or amplification factors are set manually.

The LeCroy WavePro 700Zi series oscilloscopes with bandwidths of 4 to 6 GHz include universal ProBus/ProLink probe interfaces. As shown in the following figure, each channel has a high bandwidth ProLink connector and a 50 Ω/1 MΩ ProBus connector.

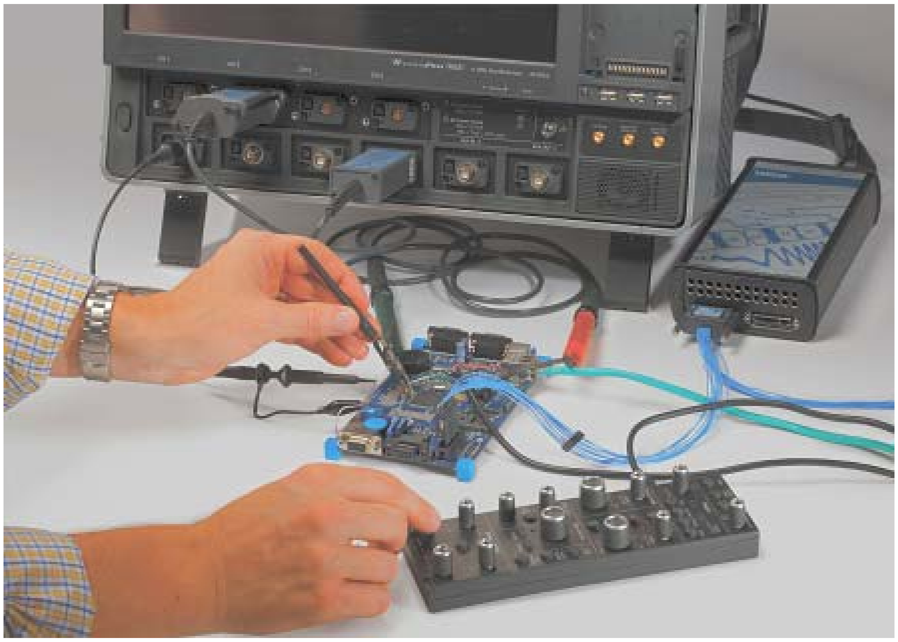


Figure 3-6. A WavePro760 Zi showing the four sets of ProBus/ProLink probe interfaces. This oscilloscope is compatible with all LeCroy probes.

This unique interface provides the user the greatest flexibility in probe selection in any oscilloscope. The ProLink inputs provide a high integrity, high bandwidth interchangeable interface to SMA cables, LeCroy ProLink probes, and accessories. ProLink is used for higher bandwidth probes.

The ProBus interface offers both 50 Ω/1 MΩ input impedance and provides probe power and control for a wide range of probes such as high impedance passive probes, high impedance active probes, current probes, high voltage probes, differential probes. ProBus also includes sense rings for detecting passive probes. All scopes with bandwidths over 4 GHz, except the WavePro 740Zi and760Zi, use 50 Ω inputs and provide 1 MΩ impedance by means of external adapters.

The following figure shows a typical channel setup. The input selection is on the left hand side of the dialog box. In this case the A input, the ProLink interface is selected. Each input selection has its own independent settings for Probe attenuation, bandwidth and coupling.

Figure 3-7. The channel screen layout showing Input A's ProLink interface controls setup

> **Note**: Input A (Upper) and B (Lower) refer to the two rows of input channels specifically available on WavePro 740 and 760Zi. WavePro 715, 725, and 735Zi all have only one row of input channels on the front panel, and one corresponding selected input on the screen layout.

The setup for the ProBus interface is shown in the following figure. This interface also allows the selection of input impedance, 50 Ω/1 MΩ.

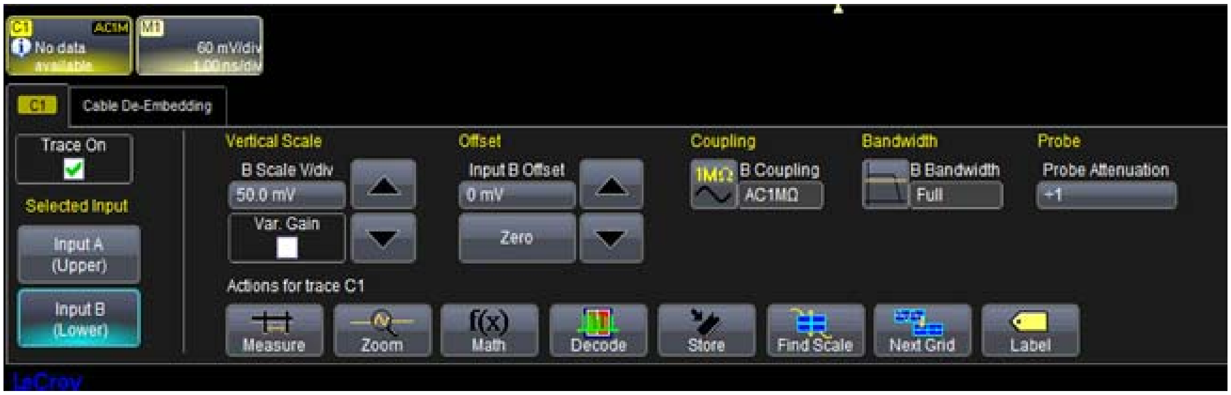


Figure 3-8. The channel screen layout showing Input B's ProBus interface controls setup

Another universal probe interface advantage is the ability to switch between two probe setups. This function is remotely programmable allowing the interface to operate like a switch.

The characteristics of the universal ProBus/ProLink Interface are shown in the following table. Each of the interfaces provide power to the probe. An I^2C interface allows communication with the probe which permits probe recognition and interaction. This adds to the flexibility of the system as the scope can sense and control each probe.

Probe Interface	ProBus (Bandwidth)	ProLink (Bandwidth)
50 Ω	3.5 GHz	4-6 GHz
1 MΩ	500 MHz	NA

Table 3-1.Probe interface characteristics

The WavePro 740Zi and 760 Zi oscilloscopes offer the universal ProBus/ProLink probe interface which allows dual inputs in the DC to 3.5 GHz range and a dedicated high bandwidth interface in the range of 4 through 6 GHz.

The ability to select four out of eight inputs provides unheard of connection flexibility and eliminates the need for costly, easy to lose adapters.

ProLink Interface

For some instruments, LeCroy's ProLink Adapters (LPA) give you the ability to connect your signal in one of three ways:

- BMA connector (only on 4 and 6 GHz models)
- SMA using the BMA-to-SMA adapter (4 are sent standard on 4 and 6 GHz models)
- BNC using BMA-to-BNC adapter (optional)

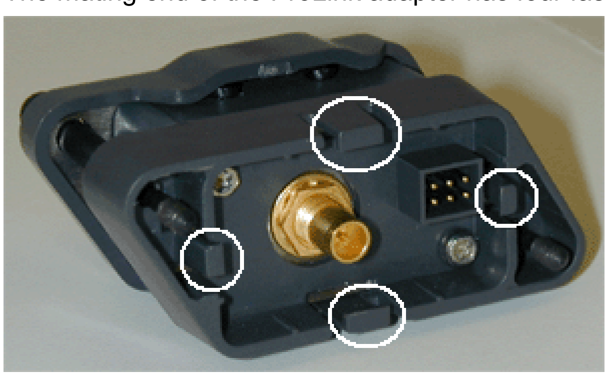
Figure 3-9. BMA-to-SMA Adapter (1) and BMA-to-BNC Adapter (2)

Figure 3-10. BMA Female Connector (1), ProLink BMA-to-SMA Adapter (Installed, 2), and ProLink BMA-to-BNC Adapter (Installed, 3)

> **Note**: When connecting an active probe to the instrument, be sure to use a ProLink BMA-to-BNC adapter (item 3 in the figure). Do not plug the probe directly into the front panel connector (item 1) without an adapter.

Connecting the Adapters

The mating end of the ProLink adapter has four fastening clips as follows:

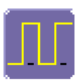
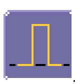
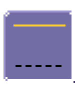
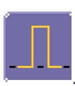


When installing an adapter on the instrument's connector panel, align the male 6-pin connector with the female connector and push the adapter straight in. There will be some resistance and you'll hear clicks as the four clips slide into place. Then tighten the captive screws.

When removing an adapter, loosen the two captive screws. Push down on the adapter to unseat the clips. This will require some force and will be initially noisy, but no damage will result to the connector, the floating female BMA connector, or the pins, which can be 15 degrees off axis when being mated or unmated.

Auxiliary Output Signals

In addition to a calibration signal, the following signals can be output through the AUX OUTPUT connector:

	**Square Wave**
	**Trigger Out** - can be used to trigger another scope
	**DC level** - a reference level (not available on all instruments)
	**Trigger Enabled** - can be used as a gating function to trigger another instrument when the scope is ready
	**Pass/Fail** - allows you to set a pulse duration from 1 ms to 500 ms; generates a pulse when pass/fail testing is active and conditions are met.
	**Aux Output Off** - turns off the auxiliary output signal

Auxiliary Output Setup

1. Touch **Utilities** → **Utilities Setup...** from the menu bar.
2. Touch the **Aux Output** tab.
3. If you want a 1 kHz, 1 V square wave, touch the corresponding button.
4. If you want a specialized output, touch one of the buttons under **Use Auxiliary Output For**.
5. Touch inside the **Amplitude** data entry field and enter a value, using the pop-up numeric keypad. If you want a TTL level signal, touch the **TTL Level** checkbox. The **Amplitude** field will accordingly become unavailable.
6. If you selected Square Wave, touch inside the **Frequency** data entry field and enter a value, using the pop-up keypad. You can set a value from 5.0 Hz to 5 MHz.
7. If you selected Pass/Fail, touch inside the **Pulse Duration** field and enter a value from 1 ms to 500 ms, using the pop-up numeric keypad.

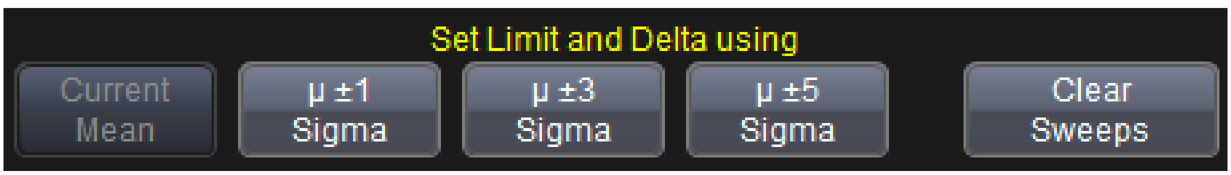
Pass / Fail Testing

Comparing Parameters

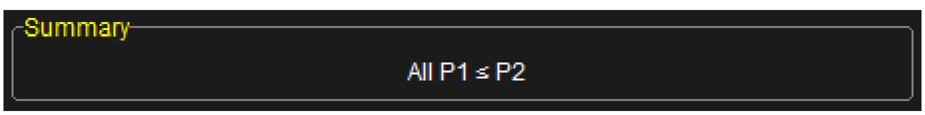
Each Pass/Fail input (**Qx**) can compare a different parameter result to a user-defined limit (or statistical range) under a different condition.

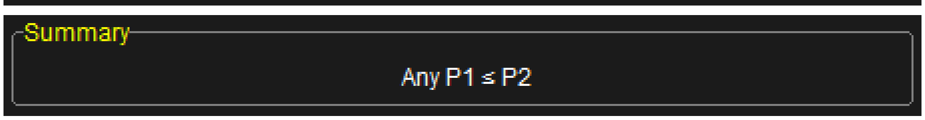
The conditions are represented by these comparison operators:

At the touch of a button, test results can also be compared to these standard statistical limits:



In Dual Parameter Compare mode, your X-Stream oscilloscope can take parameter results measured on two different waveforms and compare them to each other. If desired, set your test to be true if **Any** or **All** waveforms fit the criteria set by the comparison condition. Setups are conveniently shown in the Summary box of the **Qx** dialog as follows:





Mask Tests

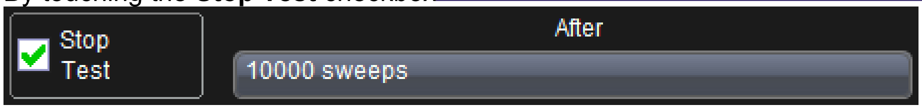
Mask testing can be done using an existing mask, or by using a mask created from your actual waveform, with vertical and horizontal tolerances that you define. Existing masks can be loaded from a floppy disk or from a network.

You can set your mask test to be True for waveforms All In, All Out, Any In, or Any Out. For example, if you select All In, the test will be False if even a single waveform falls outside the mask.

Masks that you create from your waveform can be confined to just a portion of the trace by use of a measure gate. Refer to the **Measure Gate** topic for more information on how this feature works.

Actions

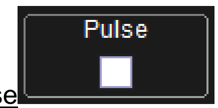
By touching the **Stop Test** checkbox 



in the **Actions** dialog, you can set up the test to end after a predetermined number of sweeps that you decide.

You can also decide the actions to occur upon your waveforms' passing or failing, by selecting one or all of the following:

- stop
- audible alarm
- print image of display
- emit pulse (if available)
- save waveform

The selection Pulse 

causes a pulse to be output through the Aux Out connector at the front of the oscilloscope. This pulse can be used to trigger another oscilloscope. You can set the amplitude and width of the pulse as described in Auxiliary Output Signals.

Depending on your scope model, you can configure up to 8 pass/fail conditions. The boolean conditions to determine if your waveform passes are as follows:

All True	All False
Any True	Any False
All Q1 to Q4 **Or** All Q5 to Q8	Any Q1 to Q4 **And** Any Q5 to Q8

Pass/Fail Testing Setup

Initial Setup

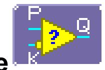
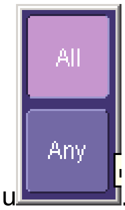
1. Touch **Analysis** → **Pass/Fail Setup...** from the menu bar.
2. Touch the **Actions** tab.
3. Touch the **Enable Actions** checkbox. This causes selected actions to occur on your waveform's passing or failing a test.
4. Touch the **Summary View** checkbox to enable a line of text 

showing a concise status of the last waveform and keeping a running count of how many sweeps have passed.
5. Touch inside the **Pass If** field, and select a boolean condition from the choices shown.
6. If you want to set up the test to end after a finite number of sweeps, touch the **Stop Test** checkbox. Then touch inside the **After** data entry field and enter a value, using the pop-up numeric keypad.
7. Use either the **Pass** 

or **Fail** 

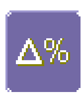
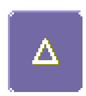
button on the **If** section of the dialog to set actions taking place when your waveform passes or fails the test.
8. The **Then** section of the dialog provides **stop test**, **sound alarm**, **print result**, **emit pulse**, or **save** (the waveform) actions. If you want to have the results printed and your scope is not equipped with a printer, be sure that the it is connected to a local or network printer. Refer to the **Printing** topics for more information.
9. If you want to save your waveform automatically, touch the **Save Setup** checkbox. This will take you out of the current dialog and will open the **Save Waveform** dialog. Refer to the **Save Recall →Saving and Recalling Waveforms** topic for more information.
10. Test Pass/Fail conditions by touching the **Force Actions Once** button. Press the **Clear All** button to quickly uncheck all checkboxes and make new selections.

Comparing a Single Parameter

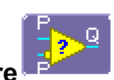
1. Touch **Analysis** → **Pass/Fail Setup...** from the menu bar.
2. Touch one of the **Qx** tabs and a setup dialog for the position is shown.
3. Touch inside the **Source1** field and select a source from the pop-up menu.
4. Touch inside the **Condition** field in the main dialog and select **ParamCompare** .
5. Touch inside the **Compare Values** field and select **All** or **Any** from the pop-up menu 

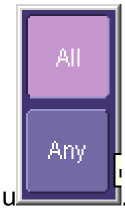
. By selecting **All**, the test will be true only if every waveform falls within the limit that you will set. By selecting **Any**, the test will be true if just one waveform falls within the limit.
6. Touch inside the **Condition** field on the **ParamCompare** part of the dialog and select a math operator from the pop-up menu 

.
7. Touch inside the **Limit** field and enter a value using the pop-up numeric keypad. The value entered acquires the dimensions corresponding to the parameters being tested. For example, if you are testing a time parameter, the unit is seconds.

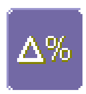
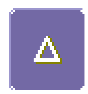
If you chose either **WithinDeltaPct**  or **WithinDeltaAbs**  from the **Condition** menu, you also have the choice of setting the limit by means of the statistical buttons at the bottom of the **ParamCompare** dialog:

Comparing Dual Parameters

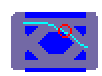
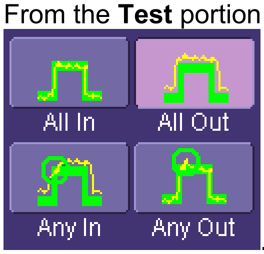
1. Touch **Analysis** → **Pass/Fail Setup...** from the menu bar.
2. Touch one of the **Qx** tabs and a setup dialog for the position is shown.
3. Touch inside the **Condition** field in the main dialog and select **DualParamCompare** .
4. Touch inside the **Source1** and **Source2** fields and select a source from the pop-up menu.
5. Touch inside the **ParamCompare** mini-dialog field and select a source from the pop-up menu.

6. Touch inside the **Compare Values** field and select **All** or **Any** from the pop-up menu 

. By selecting **All**, the test will be true only if every waveform falls within the limit that you will set. By selecting **Any**, the test will be true if just one waveform falls within the limit.
7. Touch inside the **Condition** field in the **ParamCompare** mini-dialog and select a math operator from the pop-up menu 

.
8. Touch inside the **Limit** field and enter a value, using the pop-up numeric keypad. This value takes the dimension of the parameter that you are testing. For example, if you are testing a time parameter, the unit is seconds.
9. If you chose either **WithinDeltaPct**  or **WithinDeltaAbs**  from the **Condition** menu, touch inside the **Delta** field and enter a value.

Mask Testing

1. Touch **Analysis** → **Pass/Fail Setup...** from the menu bar.
2. Touch one of the **Qx** tabs and a setup dialog for the position is shown.
3. Touch inside the **Source1** field and select a source from the pop-up menu.
4. Touch inside the **Condition** field in the main dialog and select **Mask Test**  .
5. From the **Test** portion of the dialog, make a selection in the **Test is True when** group of buttons 

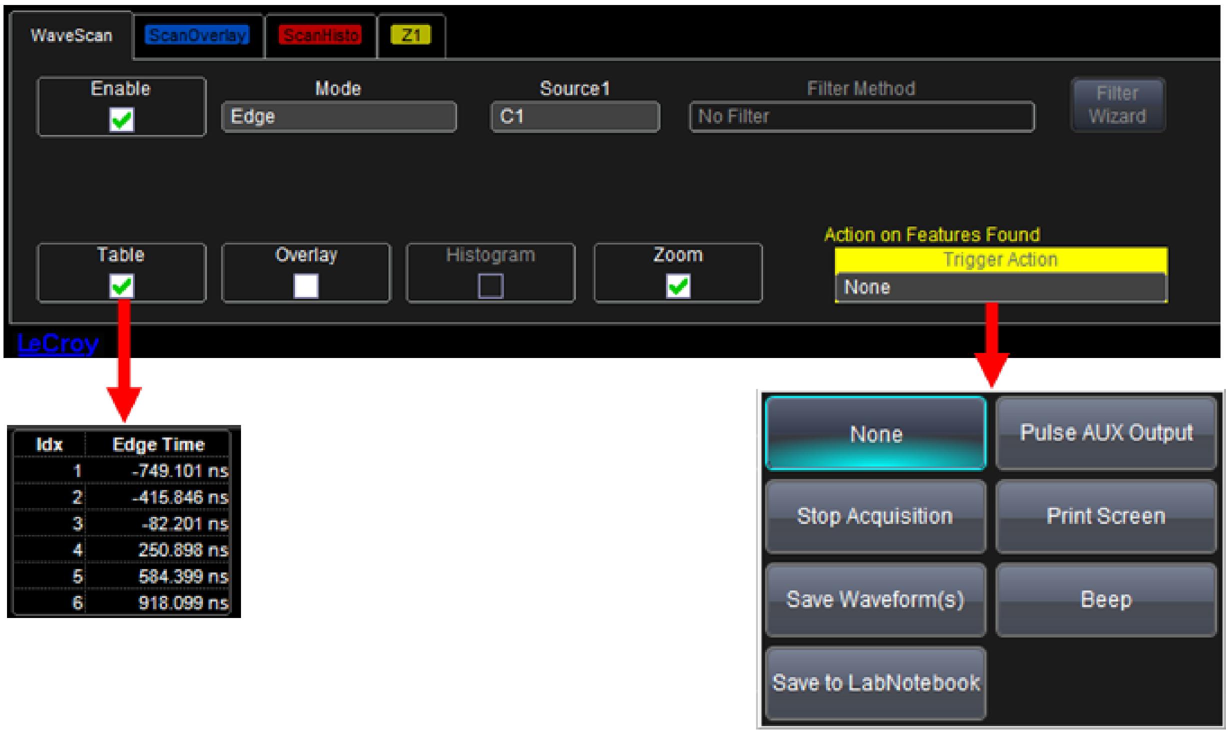
. This selection means, for example, that if you select **All In** the test is False when a single waveform falls outside the mask.
6. From the **Show Markers** section of the dialog, choose whether or not to have mask violations displayed.
7. If you are loading a pre-existing mask, touch the **Load Mask** tab, and then the **File** button. You can then enter the file name or browse to its location.
8. If you want to make a mask from your waveform, touch the **Make Mask** tab.
9. Touch inside the **Ver Delta** and **Hor Delta** fields and enter boundary values, using the pop-up numeric keypad.
10. Touch the **Browse** button to create a file name and location for the mask if you want to save it.
11. Touch the **Gate** tab, then enter values in the **Start** and **Stop** fields to constrain the mask to a portion of the waveform. Or, you can simply touch and drag the Gate posts, which initially are placed at the extreme left and right ends of the grid.

Introduction to WaveScan

WaveScan enables you to search for unusual events in a single capture, or to scan for an event in many acquisitions over a long period of time. You can select from more than 20 search modes (frequency, rise time, runt, duty cycle, etc.), apply a search condition (slope, level, threshold, hysteresis), and begin scanning in a post-acquisition environment. Since the scanning modes are not simply copies of the hardware triggers, but "software triggers," the capability is much greater.

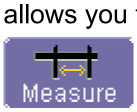
For instance, there is no “frequency” trigger in any oscilloscope, yet WaveScan allows frequency to be quickly scanned for. You can accumulate a data set of unusual events that are separated by hours or days, enabling faster debugging. The events are time stamped and indexed in a table from which you can select them for viewing individually.

You can also set actions to occur automatically when unusual events are found: stop the acquisition, emit an audible beep, output a pulse, print the screen, save the waveform, or create a LabNotebook™ entry.



Signal Views

WaveScan provides distinct views of your signal:

- Source view highlights all occurrences of edges that meet your criteria.
- Scan Overlay (not available in WaveSurfer Xs scopes) places all captured edges one on top of the other in a separate grid. You can apply monochromatic persistence in this view.
- Scan Histogram (not available in WaveSurfer Xs scopes) provides a statistical view of edges that meet your search criteria.
- Zoom view allows you to expand a waveform feature vertically and horizontally; to apply further processing  ; to store  it, or to apply a descriptive label  to the feature.

> **Note:** The number of grids displayed varies from one to three grids depending on which views are enabled. WaveScan handles this function automatically, and there is no option to move traces from one grid to another, as would be the case under normal (non-WaveScan) operation.

Search Modes

Search modes are used to locate anomalies during acquisition.

- Edge - for detecting the occurrence of edges; selectable slope and level
- Non-monotonic - for detecting threshold re-crosses; selectable slope, hysteresis, and level
- Runt - for detecting pulses that fail to cross a threshold; selectable polarity and thresholds
- Measurement - for filtering and performing parameter measurements

Parameter Measurements

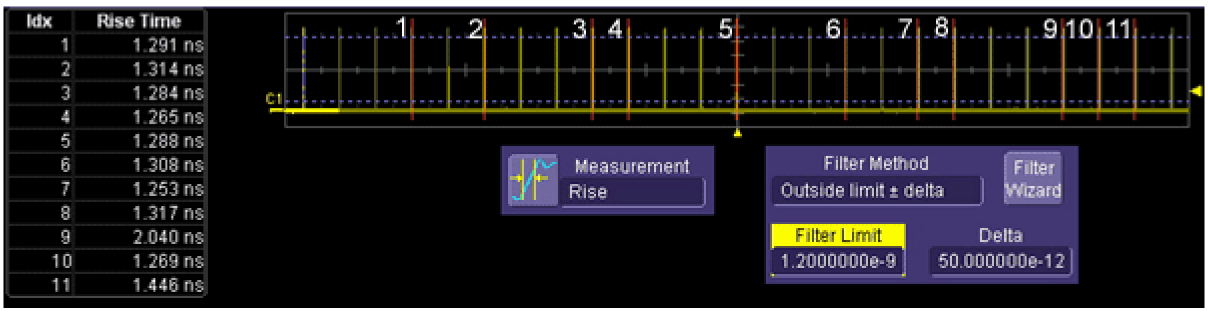
Besides parameter measurements made during acquisition, post-acquisition measurements can also be made. The number of parameters available depends on the options loaded on your instrument. Measurements are made only on the events defined by your filter (search criteria). A Filter Wizard is provided to quickly establish statistical criteria such as ±1, 3, or 5 sigma.

Sampling Mode

Whenever WaveScan is enabled, the instrument reverts to Real-time sampling mode.

Source View

The top trace on the screen is the source (channel, math, memory) trace. This trace shows all captured edges and highlights those that fit your search criteria. For example, in this figure we are searching for slow rising edges that fall outside a time window:



In this acquisition, WaveScan has located eleven edges that fit the search criteria (filter) of greater than 1.2 ns with a delta of 50 ps. Each of the found edges is highlighted with a red rectangle, and indexed to a table entry at left. Rise time values are also included in the table.

Level Markers

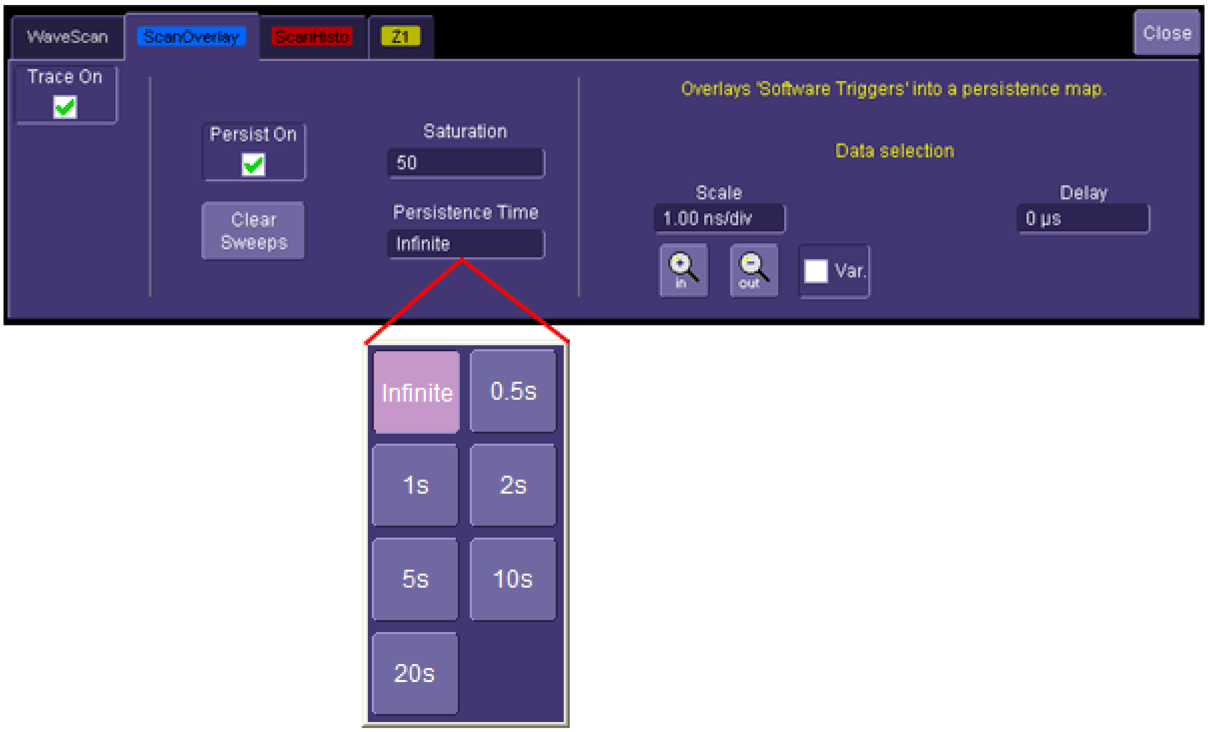
Markers are provided on the source trace to show measurement criteria such as level and thresholds. In the example above, the level markers indicate 10% and 90% for the standard Rise Time parameter measurement.

Level markers are displayed only while the scope is in WaveScan mode. Once the WaveScan **Enable** checkbox is unchecked, the level markers disappear.

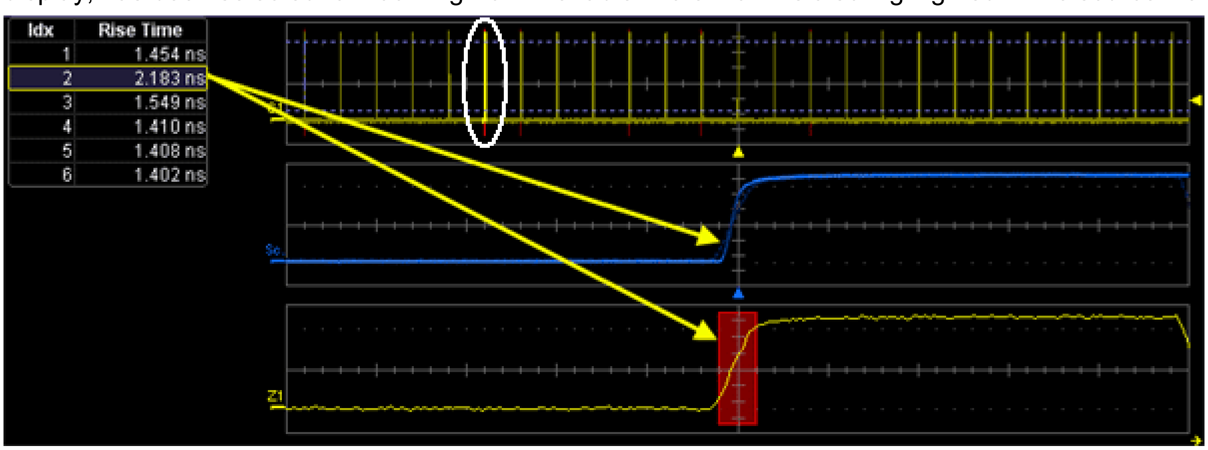
Scan Overlay

This display mode shows all edges in an acquisition overlaid one on top of the other. By default, monochromatic persistence is turned on for the scan overlays, but you have the option to disable it. Saturation and persistence time controls are also available.

> **Note:** The **Saturation** and **Persistence Time** set in the **ScanOverlay** dialog control the settings in the **Display → Persistence** setup dialog, and vice versa.



In the following acquisition, we have scanned for rise times greater than 1.4 ns. WaveScan has located six edges that meet this criteria, one of which is greater than 2 ns. This slowest edge, which appears in the persistence display, has been selected for zooming from the table. Note that it is also highlighted in the source view (top grid).

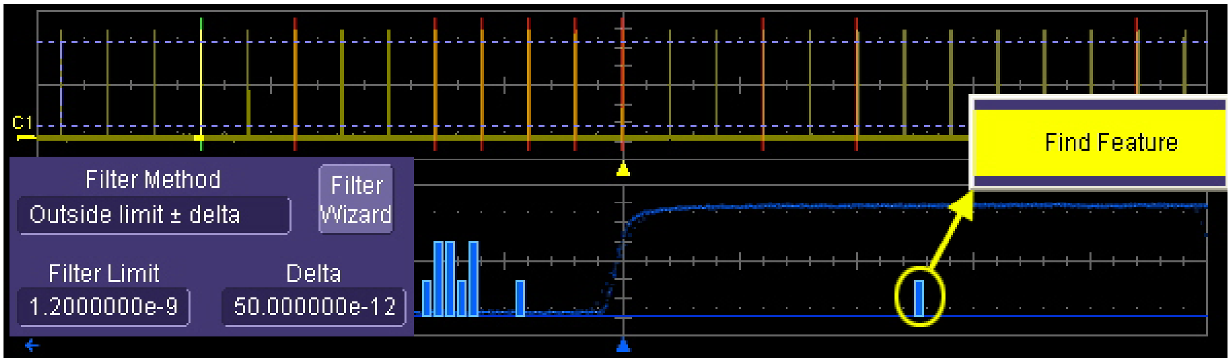


The horizontal scale of the overlaid traces can be expanded independently of the zoom trace. However, the previous example shows them in the same scale as the zoom trace (5 ns/div).

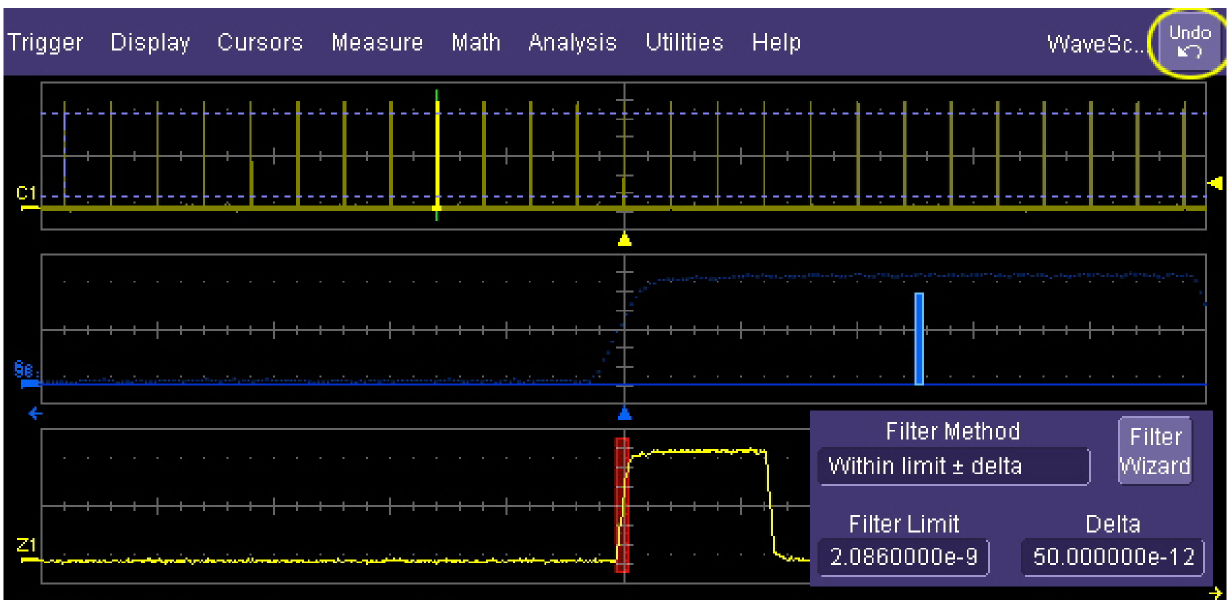
Scan Histogram

By enabling ScanHistogram, a histogram corresponding to your search criteria is superimposed on the overlay trace. In the following example, the Rise 10-90% parameter measurement has been applied, but only edges slower than 1.2 ns with a delta of 50 ps are accumulated in the histogram.

Another feature of WaveScan is that you can select a single bin of the histogram for analysis by touching or clicking it. A confirmation pop-up button then appears:



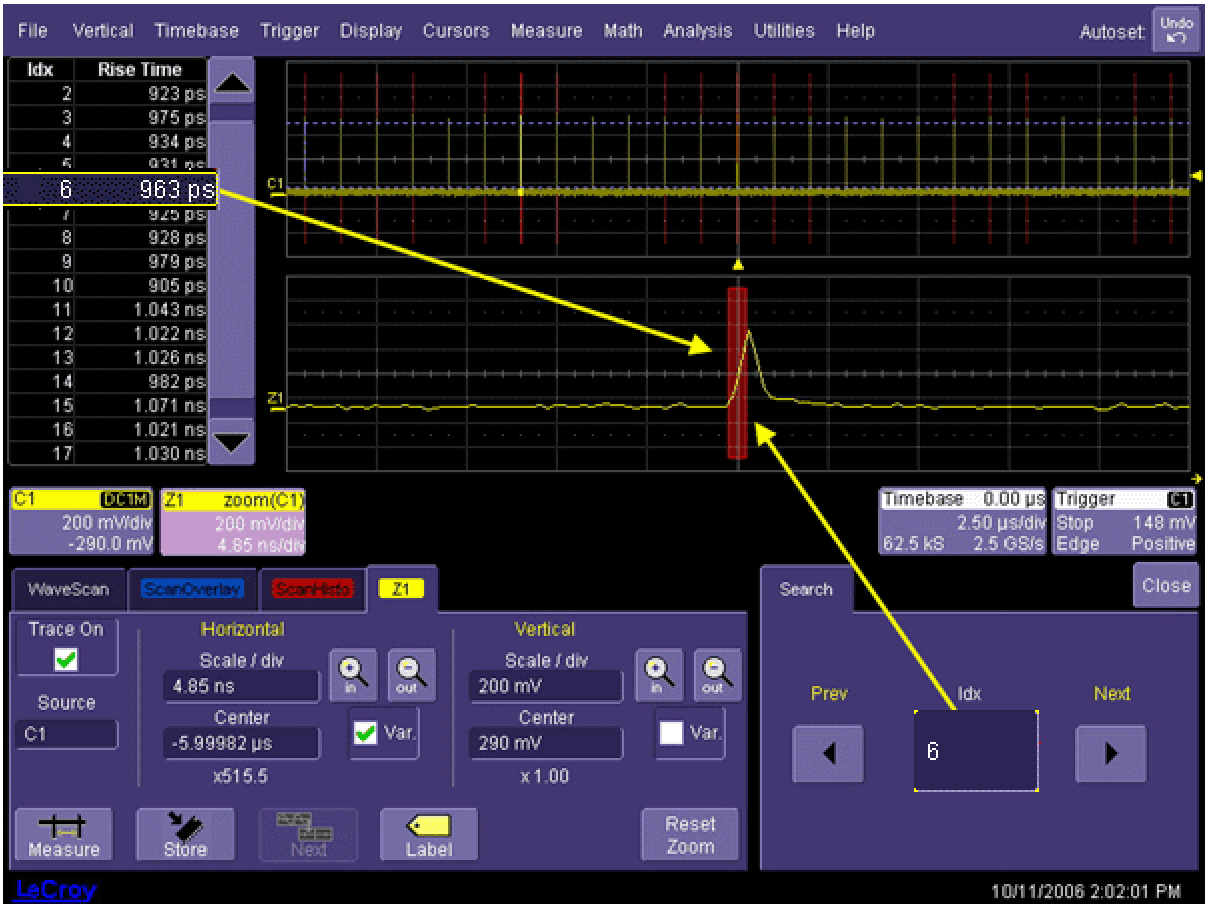
After **Find Feature** is confirmed, only the single bin of the histogram remains and information about the contents of the bin is displayed in the Filter Method area:



You can recall the original filter conditions by touching the **Undo** button at the top-right corner of the screen.

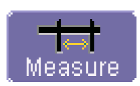
Zoom View

An individual edge can be zoomed by selecting it from the table of found events at the left of the screen. You can also scroll through the table using the **Prev/Next** scroll buttons in the **Search** dialog, or select an event by touching inside the **Idx** field and entering an index number, using the pop-up keypad.



Front panel **ZOOM** controls can be used to vary the magnification and position of the zoomed trace. Or, you can use the on-screen **In/Out** buttons 

.

In Zoom View, you can also apply further processing 

, store 

, or apply a descriptive label 

to the zoom trace.

Edge Mode

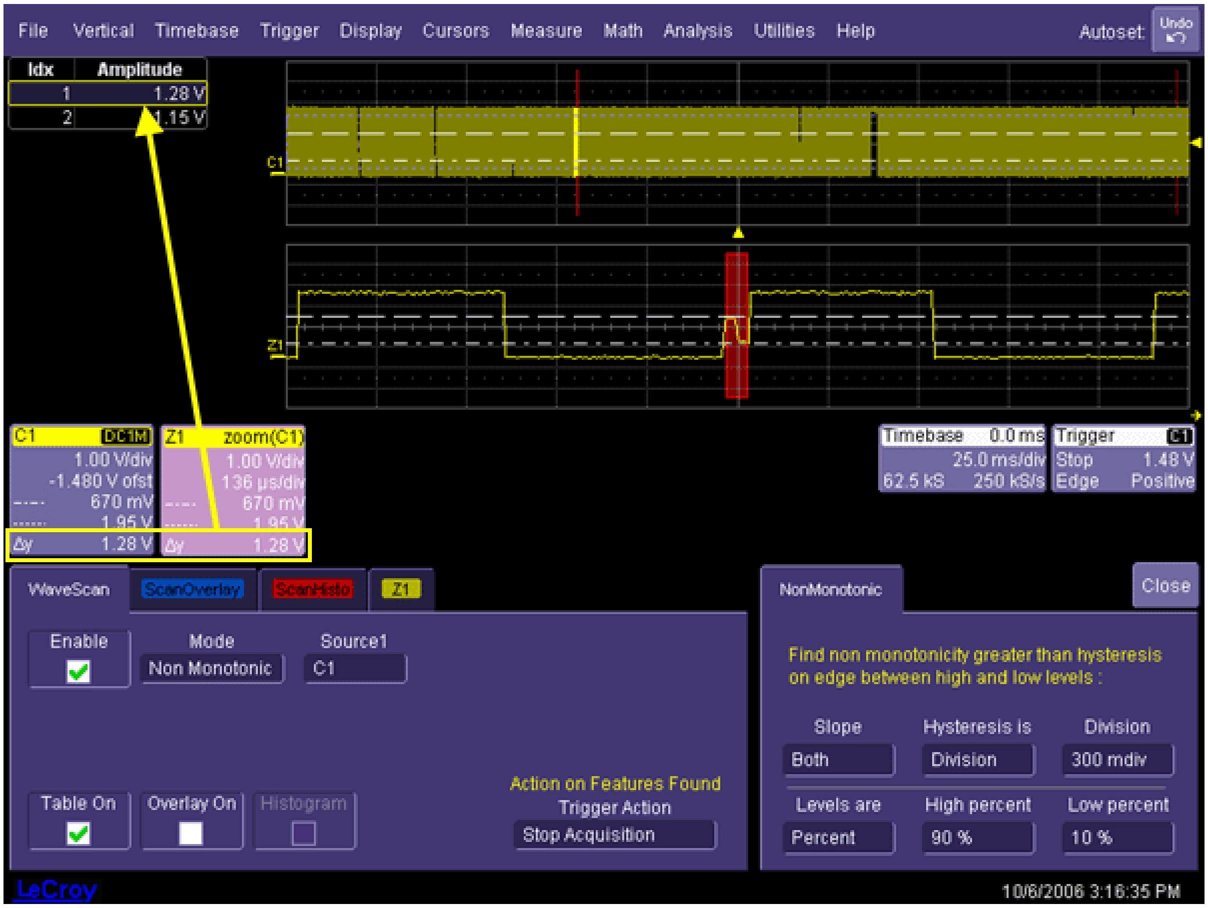
This search mode locates all edges in an acquisition and presents them time-stamped in a table. You can select positive, negative, or both edges. When the acquisition is stopped, scan filters contained in Measurements mode can be applied to the edges to find specific characteristics.

A level marker, corresponding to 50% in this case, is displayed in the source trace at top.

As with other search modes, a trigger action can be set for Edge mode also.

Non-monotonic Mode

The Non-monotonic search mode looks for edges that cross a threshold more than once between high and low levels. All events that meet the criteria of slope, hysteresis, and level are presented in a table and highlighted in the source trace. The value displayed in the table is the difference of the max. and min. of the non-monotonicity. This can be confirmed with cursors:

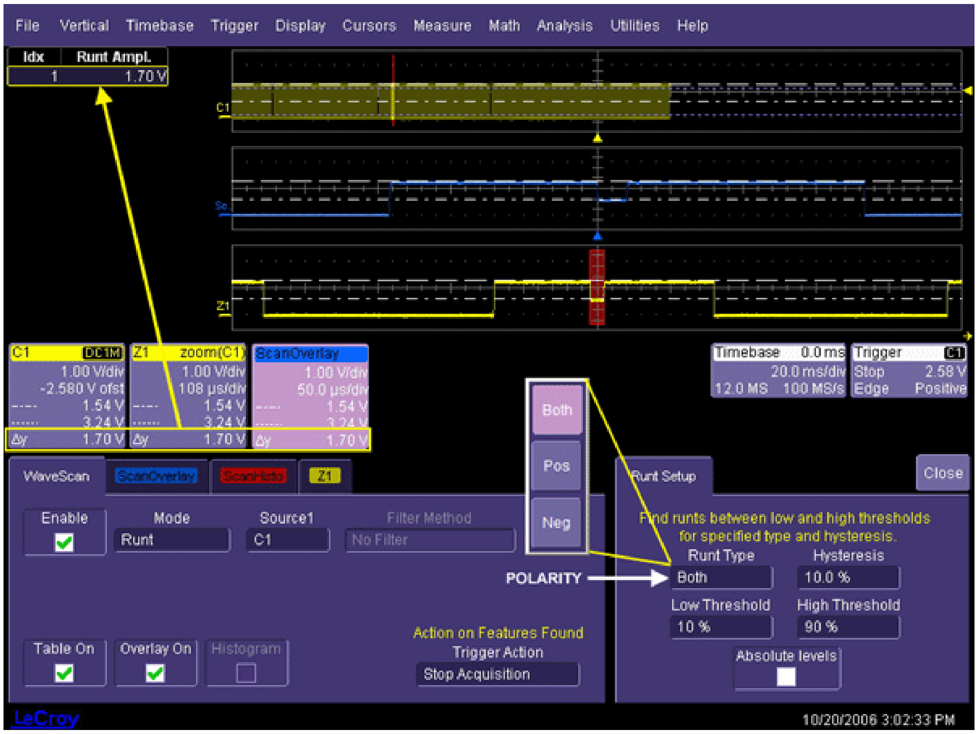


The hysteresis value is used to eliminate noise. A non-monotonicity is detected only when its amplitude is greater than the hysteresis. Therefore, when setting a hysteresis level, set a value that is greater than the amplitude of the noise.

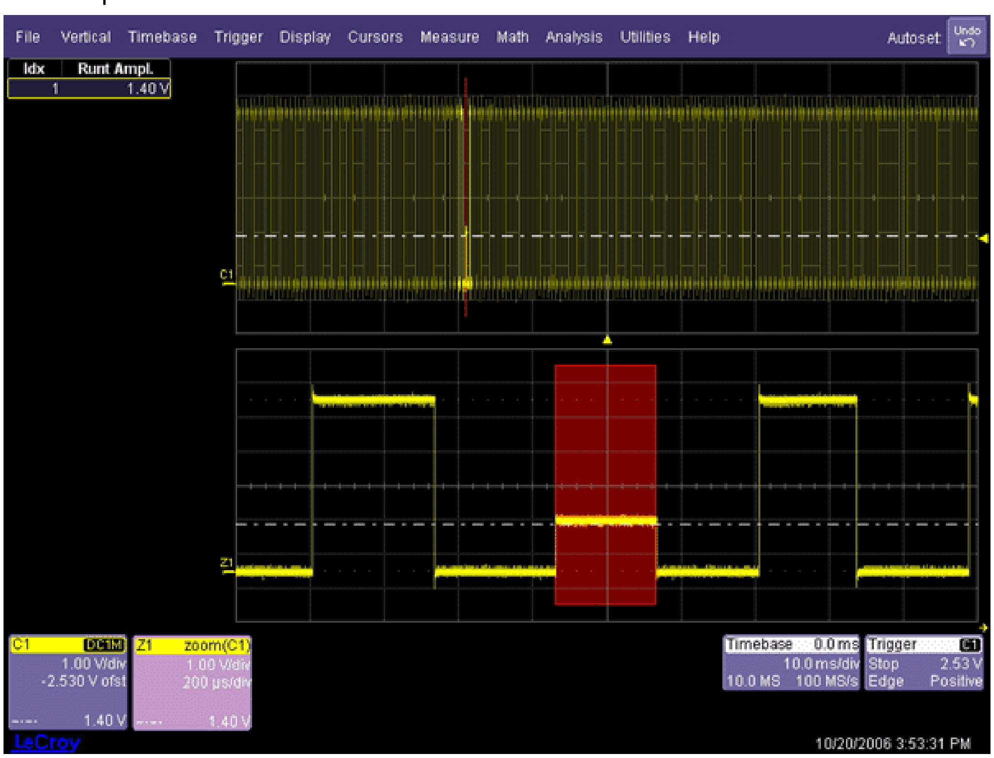
Runt Mode

This search mode looks for pulses that fail to transit a given region. You can search for positive-going or negative-going runts, or both. An adjustable hysteresis band is provided to eliminate noise.

In the case of negative-going runt pulses, the value displayed in the table is the difference (delta) of the high level of the signal and the runt amplitude, i.e., where the runt bottoms out. This can be confirmed by placing cursors on the runt pulse and reading the delta Y value in the trace labels:

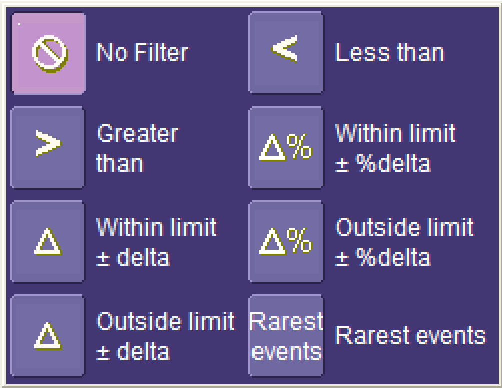


In the case of positive-going runt pulses, the value displayed in the table is the absolute value of the amplitude of the runt pulse:



Measurement Mode

This search and scan mode lets you apply filters to your acquisitions to highlight only the features of interest. Post-acquisition, you can apply other filters to the acquisition, or make different parameter measurements on it.



For example, in this acquisition a **Rise** time (10-90%) parameter measurement is applied to fast edges during acquisition. We are interested only in edges with a rise time slower than 1 ns. Therefore, the **Greater Than** filter method is selected, with a value of 1 ns; four edges pass during acquisition:

Scan Filters

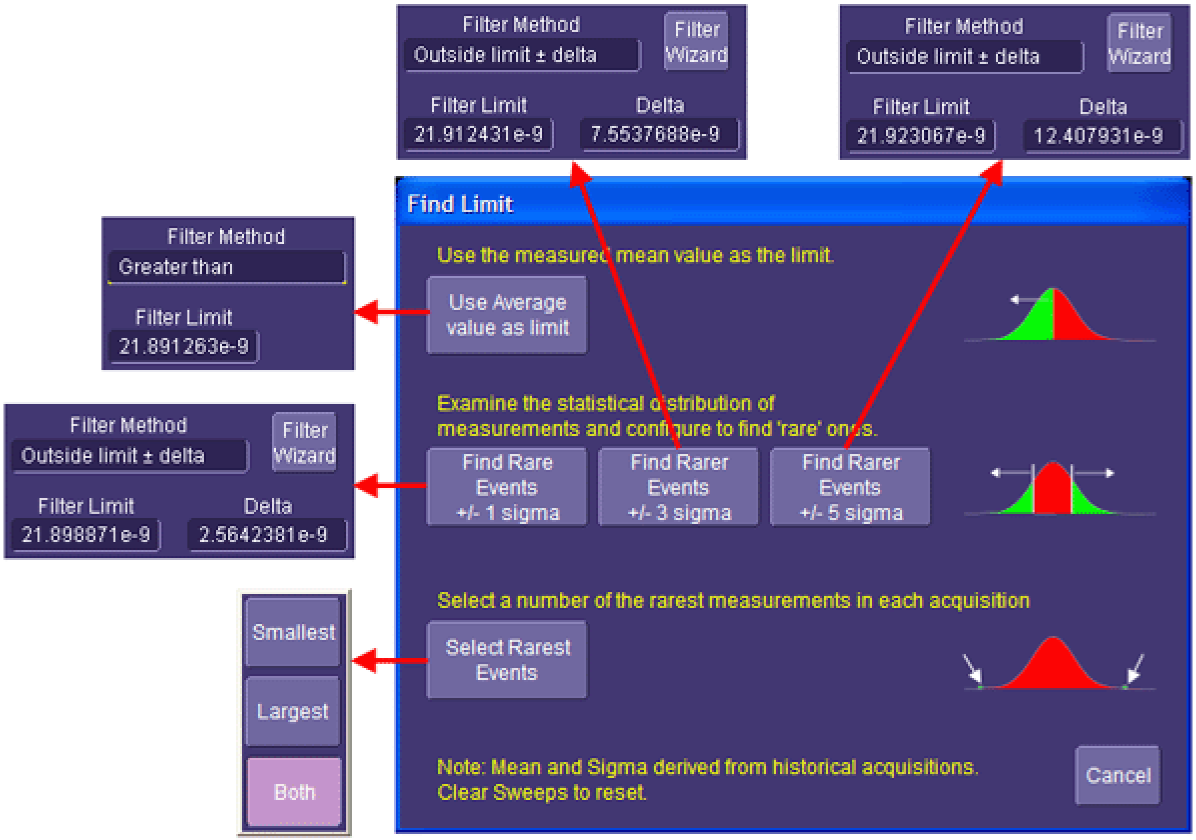
But, now we want to look at fall times greater than 3 ns; fifteen falling edges from the same acquisition pass this new filter:

You can also decide how anomalous the features located should be by using the **Filter Wizard** to select the average value as the limit, to search for rarest events, or to apply statistical criteria.

Filter Wizard

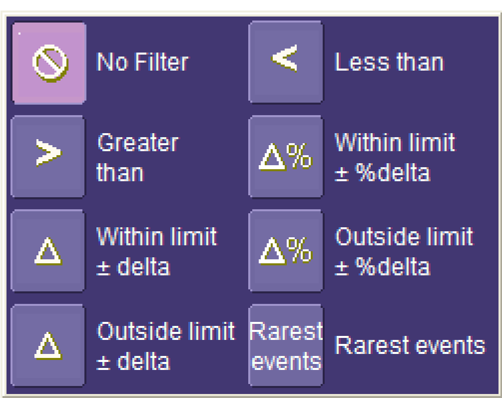
You can decide how anomalous the features searched or scanned for should be by using the average value as the limit, by searching for rarest events, or by applying statistical criteria: ±1, 3, or 5 sigma.

In the following example, there were several slow edges in a repetitive waveform. For each filtering method selectable from the wizard, the rise time used as the filter limit and delta are calculated automatically.



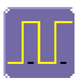
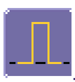
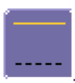
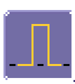
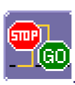
Filter Methods

While the Filter Wizard provides a quick way to apply filtering criteria automatically, the **Filter Method** menu and related fields let you manually set up filtering criteria. And unlike the Filter Wizard selections, which are restricted to absolute deltas, the **Filter Method** menu also offers deltas as percentages:



Auxiliary Output Signals

In addition to a calibration signal, the following signals can be output through the AUX OUTPUT connector:

	**Square Wave**
	**Trigger Out** - can be used to trigger another scope
	**DC level** - a reference level (not available on all instruments)
	**Trigger Enabled** - can be used as a gating function to trigger another instrument when the scope is ready
	**Pass/Fail** - allows you to set a pulse duration from 1 ms to 500 ms; generates a pulse when pass/fail testing is active and conditions are met.
	**Aux Output Off** - turns off the auxiliary output signal

Auxiliary Output Setup

1. Touch **Utilities** → **Utilities Setup...** from the menu bar.
2. Touch the **Aux Output** tab.
3. If you want a 1 kHz, 1 V square wave, touch the corresponding button.
4. If you want a specialized output, touch one of the buttons under **Use Auxiliary Output For**.
5. Touch inside the **Amplitude** data entry field and enter a value, using the pop-up numeric keypad. If you want a TTL level signal, touch the **TTL Level** checkbox. The **Amplitude** field will accordingly become unavailable.
6. If you selected Square Wave, touch inside the **Frequency** data entry field and enter a value, using the pop-up keypad. You can set a value from 5.0 Hz to 5 MHz.
7. If you selected Pass/Fail, touch inside the **Pulse Duration** field and enter a value from 1 ms to 500 ms, using the pop-up numeric keypad.

Customization Overview

The instrument provides powerful capability to add your own parameters, functions, display algorithms, or other routines to the scope user interface without having to leave the instrument application environment. You can customize the instrument to your needs by using the power of programs such as Excel™, Mathcad™, and MATLAB™, or by scripting in VBS. Whichever method you use, the results appear on the instrument's display together with the signals that you started with. This ability offers tremendous advantages in solving unique problems for a large range of applications, with comparatively little effort from you.

Caution

Accessing the scope's automation interface from within an XDEV custom processor (VBScript, MATLAB, Excel, etc.) is NOT recommended.
Cases where the scope's behavior cannot be guaranteed, or worse, cases which can cause the scope's software to crash include the following:

1. Changing "Upstream" Controls
Upstream controls are considered to be any control that, if changed, could provoke an infinite loop. An example would be a VBScript processor, in F1, which uses C1 as a source. If this processor changes the offset or vertical scale of C1 as a result of examining its input (C1) data, an infinite loop could occur. This could eventually cause the scope software to crash. This is not limited only to changing upstream channel controls, but includes any upstream processing also.

2. Accessing Other Results
Access to results (waveform, measurement, etc.), other than the scripting processor's own inputs, may cause incorrect measurements. The reason for this is simple: the scope contains a complex algorithm to determine in which order results are computed. This algorithm ensures that all inputs required by a processor are computed before the processor itself. If a VBScript processor decides to access, via automation, results other than those supplied to its inputs, the scope's dependency algorithm cannot be used. Therefore, the results accessed may not be coherent (they may be from a previous acquisition, or worse, could cause an infinite computation loop).

3. Reconfiguring Math or Measurements
Reconfiguring (adding or removing) Math and/or Measurements from within a custom processing function is not recommended, especially when the reconfiguration would cause the custom processor to remove itself.

Instrument customization provides these important capabilities:

- You can export data to programs, without leaving the instrument environment.
- You can get results back from those programs, and display them on the instrument, without leaving the instrument application environment.
- Once the result is returned, you can perform additional scope operations, such as measuring with cursors, applying parameters, or performing additional functions on the waveform, in exactly the same way as for a normal waveform.
- You can program the scope yourself.

The instrument does not just provide connectivity with data downloads to other programs. It provides true customizable interaction with these other programs, and allows you to truly customize the scope to do the exact job you want it to do. The advantages to this are many:

- You can use the standard processing power of the instrument to do most of your calculations
- You only need to write the function, parameter, display algorithm, etc. that specifically applies to your need and that the instrument doesn't contain.
- You can view the final result on the instrument display, and use all of the instrument's tools to understand the result.
- You can do additional processing on the result by applying either standard instrument parameters, functions, etc. to the returned result, or even more powerfully, adding chained customized functions. For example, you can do an Excel calculation on a result with a MATLAB function applied to it.

Solutions

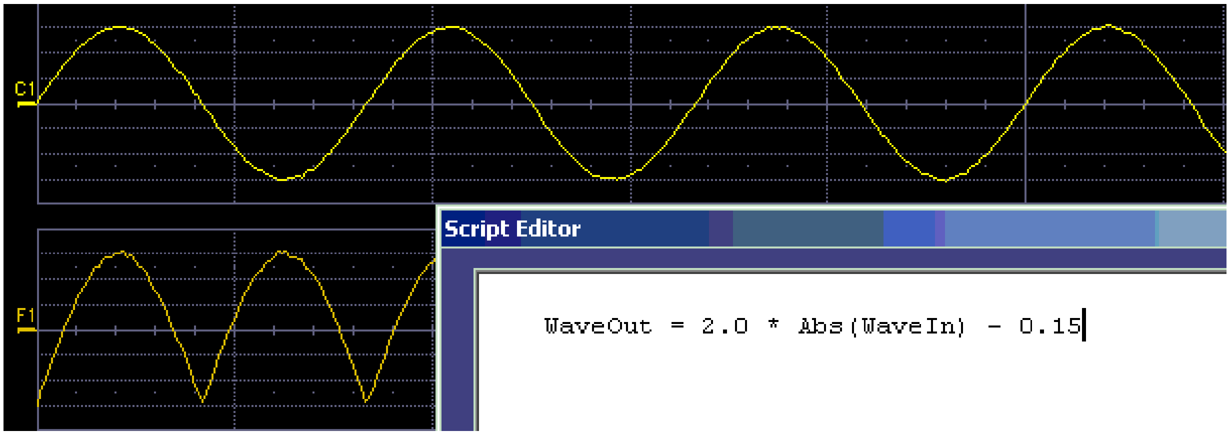
Engineers do not buy equipment; they buy solutions. But what solutions can be reached from a set of instrument waveform data? In principle, anything that can be logically derived from those data, given the limitations of signal-to-noise ratio and processing time. Here are some examples of what can be done with a customized instrument:

- Changing the units of a grid to joules, newtons, amps, etc.
- Creating a new waveform by manipulating the data of one or two input waveforms
- Creating a new waveform without using any of the input data
- Creating a new parameter by manipulating the data of one or two input waveforms
- Changing a vertical scale or a horizontal scale from linear to non-linear

You don't have to use all the data from the input waveforms: you can select data from one or more segments, which need not be aligned in the two-input waveforms.

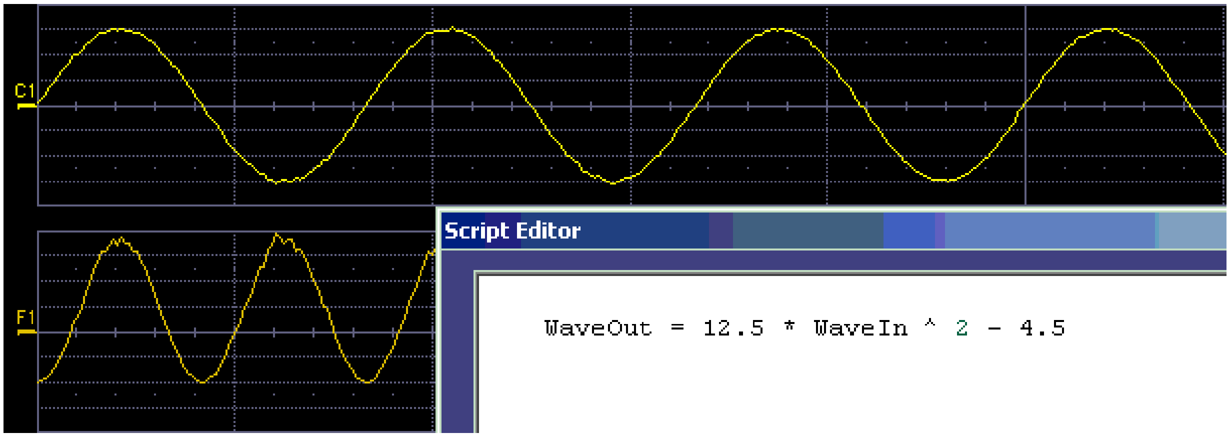
Examples

Example 1: Simple math functions using VBScript



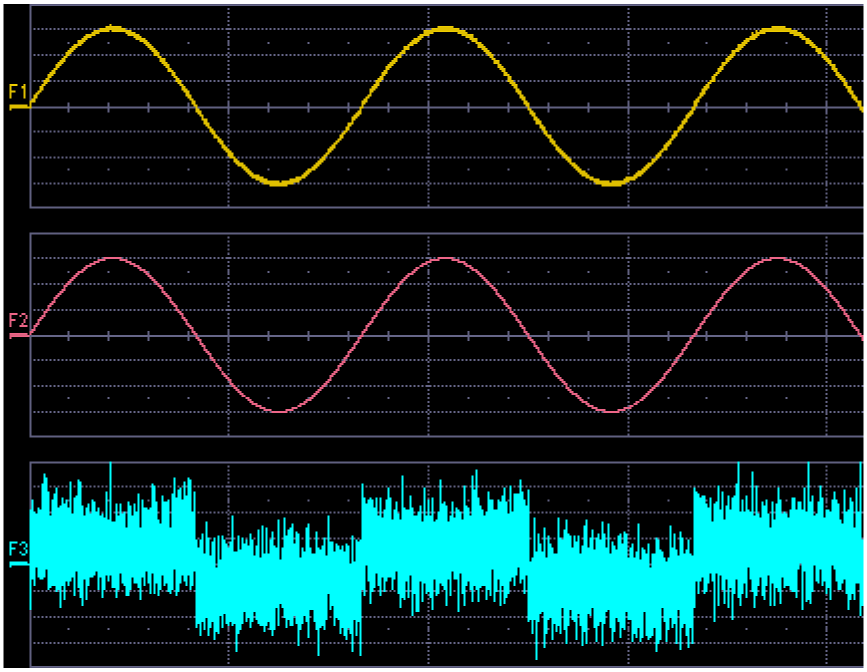
WaveOut is the waveform being returned to the instrument (F1 in this case). WaveIn is the input waveform (C1 in this case) You can see that the F1 result is displayed on the scope, and can be processed further.

Example 2: Another simple math functions using VBScript



Example 3 below doesn't use the input data at all. The middle waveform (F2) is a "golden waveform", in this case a perfect sine (subject to 16-bit resolution), that was created using a VBScript. The lower trace (F3) is a subtraction of the acquired waveform (upper trace) and the golden waveform. The subtraction (of course) contains all the noise, but it also shows the presence of a very small square wave signal.

Example 3



Here is the VBScript that produced the "golden sine" (F2 above):

```
Frequency = 3000000.0    ' Frequency of real data
SampleTime = InResult.HorizontalPerStep
Omega = 2.0 * 3.1416 * Frequency * SampleTime
Amplitude = 0.15         ' Amplitude of real data
    For K = 0 To LastPoint
    newDataArray(K) = Amplitude * Sin(Omega * K)
    Next
OutResult.DataArray(True) = newDataArray ' Data in volts
```

OutResult.DataArray is the waveform returned to the scope and displayed on the scope as the F2 waveform.

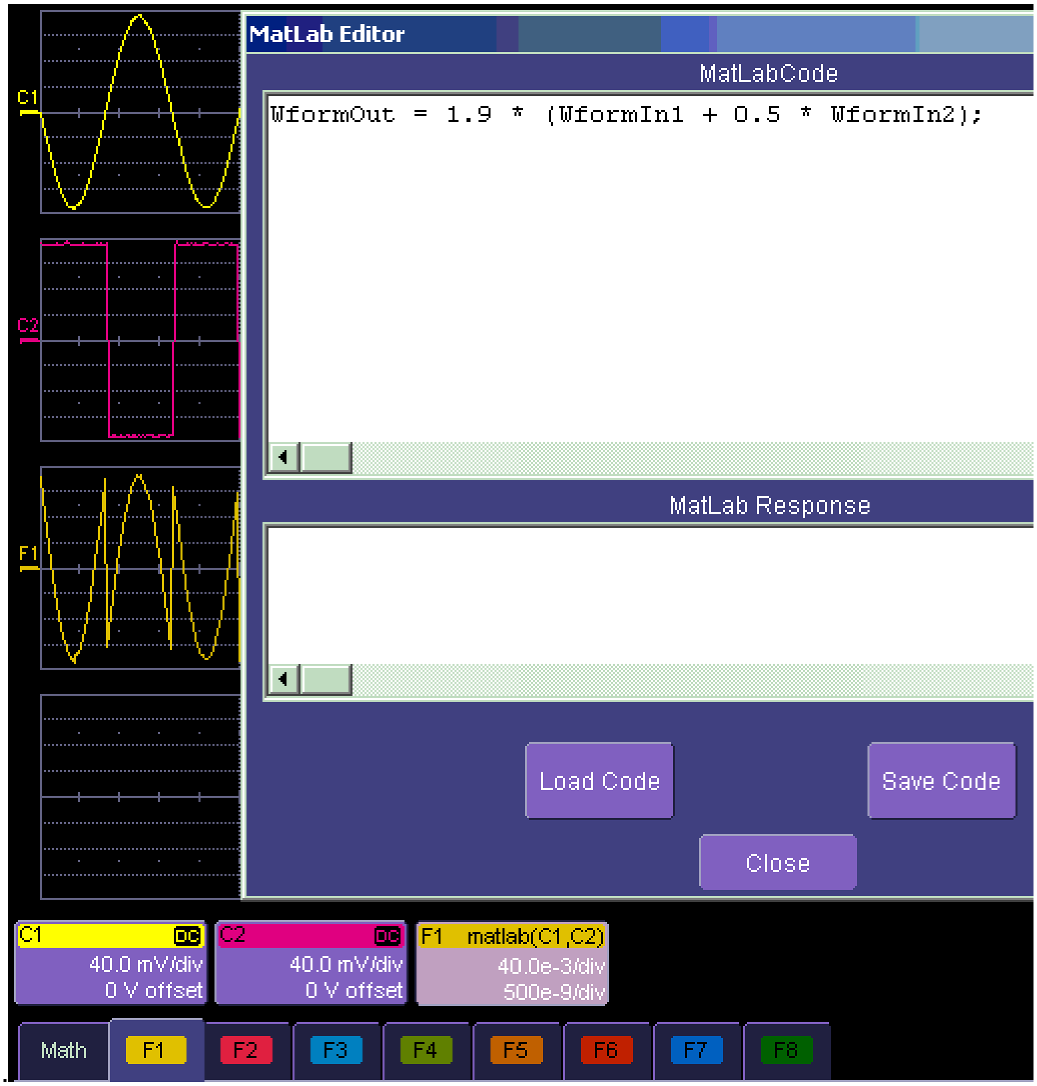
Example 4

Touch here.

Example 4 is a measurement of DVI (Digital Video Interface) Data-Clock skew jitter measurement, using a VBScript to emulate the PLL.

In this example, a customer was not able to probe the desired clock signal. The only probing point available was the output differential clock signal (C2). However, that clock was a factor of 10 slower than the clock embedded in the data signal (C3). By using a VBScript to create a clock waveform of the appropriate frequency (waveform F1), the customer was able to display and measure data-clock skew using a LeCroy instrument function and parameter.

Example 5



Touch here.

Summary

The examples above illustrate only the capability to use VBScript and MATLAB. The instrument with the LeCroy XMAP software option allows you to use Excel, MATLAB, and VBScript in this manner. Of course, you will need to load Excel, Mathcad, or MATLAB in the scope (VBScript does not require any additional software) to take advantage of the capability. You can think of these functions as "subroutines" of the instrument's main software, which take in waveform data and other variables like vertical scale and offset, and horizontal scale and offset. These functions then return a waveform or a parameter as required. In addition, you can view the calculated data directly in Excel, MATLAB, if you desire.

What is Excel?

Excel is a program within Microsoft Office. With it you can place data in the cells of a spreadsheet, calculate other values from them, prepare charts of many kinds, use mathematical and statistical functions, and communicate with other programs in Office. From the instrument you can send data to Excel (where processing can take place) and return the results to the instrument.

What is MATLAB?

MATLAB is a software package from MathWorks that provides an environment for work in computation and mathematics. An interactive language and graphics are provided.

What is VBS?

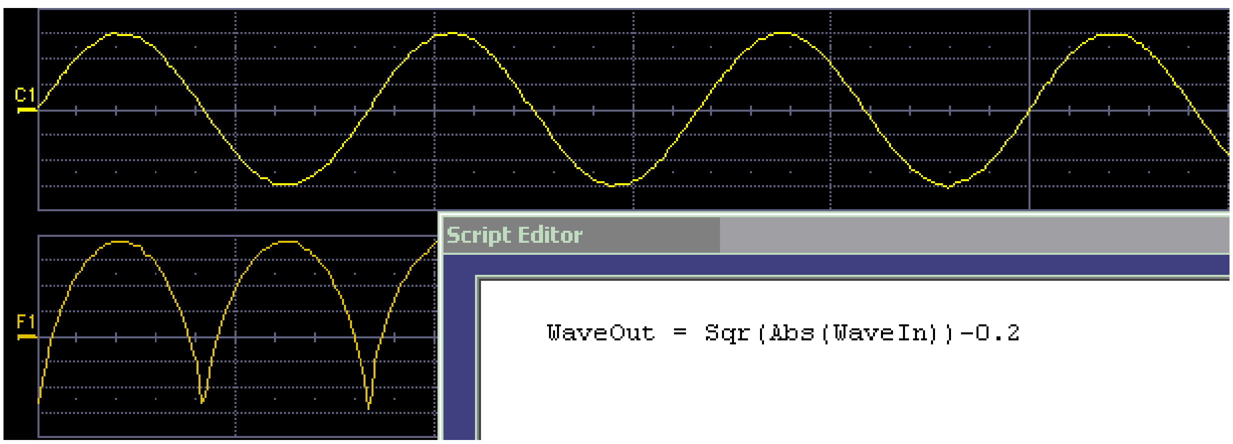
VBS is a programming language, but you don't write it in a special environment such as C++ or Visual Basic; you write it within your own application. In the instrument, a few clicks or button pushes will get you into an editing panel where you can write what you want. You cannot crash the scope, or in any other way interfere with its workings, because the system is completely protected.

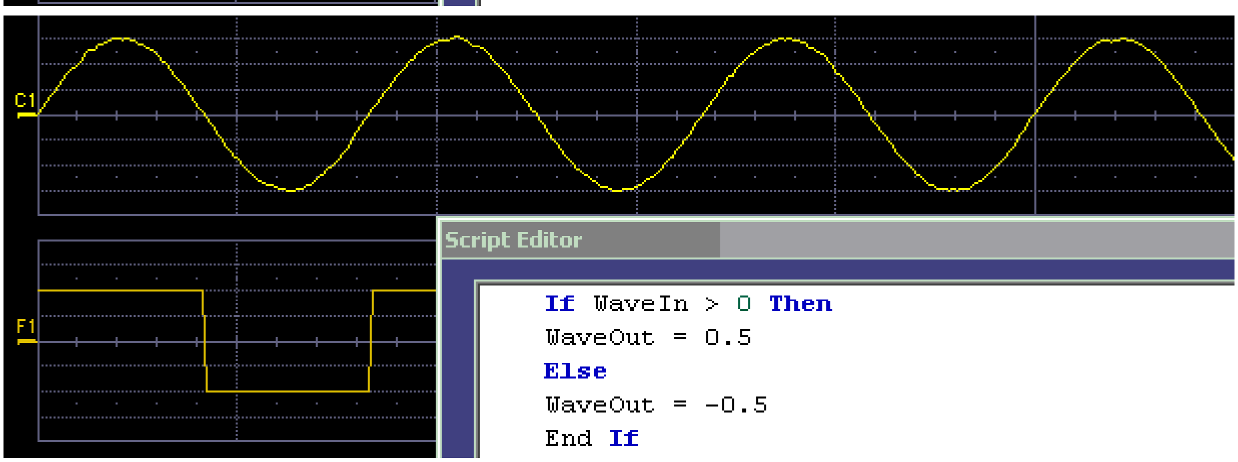
A product of Microsoft and a subset of Visual Basic, VBS can be learned very quickly if you have some experience in any programming language. The VBS processing function can collect a number of useful variables from the scope, including waveform data and useful variables such as volts per division and time per division. The output from a script can be a waveform or a parameter, and you can choose your own values for variables such as volts per division.

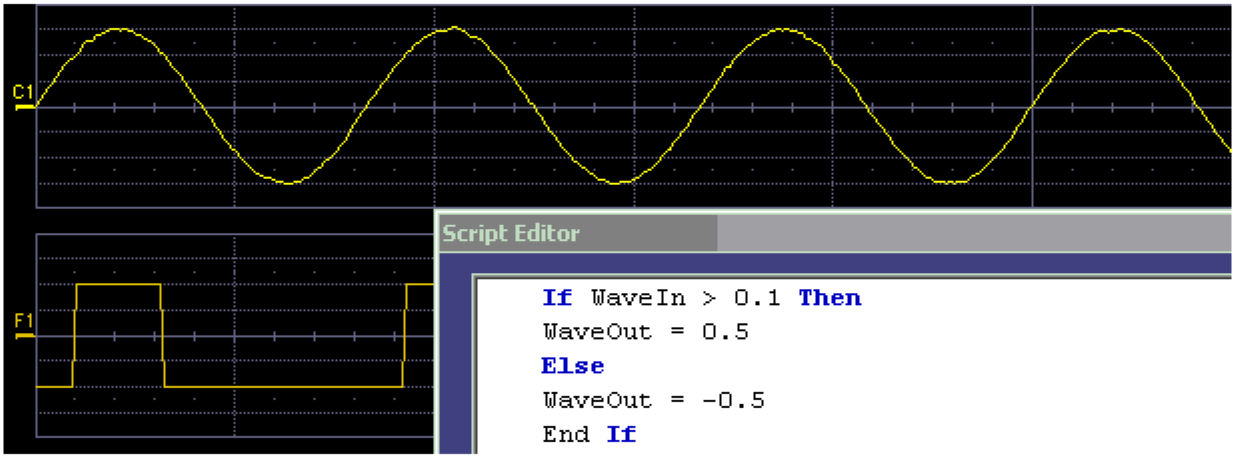
The idea of a VBS function is that you start with an input waveform, operate on some or all of the values with a script, and show the result on a scope grid, like any other waveform.

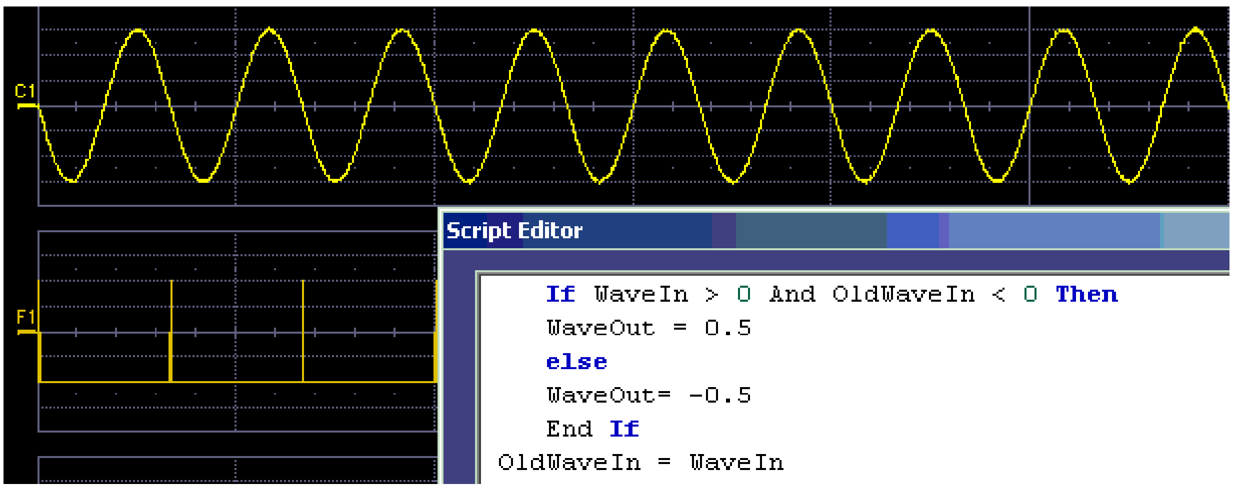
VBScript customization is built into the instrument, so no additional programs need to be loaded to take advantage of this capability.

The following diagrams were made by changing a small part, in some cases just one line, of a standard VBScript. VBS is a well-known standard language, with excellent support documentation, and it is easy to use in several different environments.









These examples are purely illustrative, but you can easily imagine that with a VBScript you can add value to the scope in a very short time. This gives you an instrument that does exactly what you want, time after time, by using your stored setups and scripts.

What Can You Do with a Customized Scope?

If you require a result that can be derived logically from the input waveform, you can do it. Many calculations can be done with remarkably small scripts, but if you have no time for scripting, you can use one of the proprietary packages, such as Excel, MATLAB, which offer immense processing power.

Scaling and Display

Scripting and programming allow a large variety of opportunities. You may, for example, be using transducers. If so, you can change the units of your waveforms, and write N (newtons), J (joules) and so on, and you can introduce scaling factors. If the transducers are non-linear, you can correct for that, too. You can also transform horizontal scales and vertical scales by manipulating the data. Logarithmic scales in amplitude and frequency are often required. Squaring and taking square roots are needed in certain applications. Here is a picture showing some graphs related to white noise, showing ways of detecting small deviations from the true distribution. The lower two graphs were generated and placed in one trace using a VBScript.

In the next example, four graphs are placed in one trace.

Golden Waveforms

This is a rich field for VBS. An example was given earlier. The only limits to the shapes that can be generated are the vertical resolution and the number of samples.

A practical example – DVI Data-Clock skew

The next example is a measurement of DVI Data-Clock skew jitter measurement, using a VBScript to emulate the PLL. A solution to a practical measurement problem was shown earlier.

These are just a few of the many solutions that can be created.

Calling Excel Directly from the Oscilloscope

Excel can be directly called from the instrument in two ways:

Calling Excel:	Description:	Result:
Using a function	F1 through Fx[The number of math traces available depends on the software options loaded on your scope. See Specifications.]	Excel returns a waveform
Using a parameter	P1 through Px[The number of parameters available depends on the software options loaded on your scope. See Specifications.]	Excel returns a parameter

In both cases, one call to Excel can use two separate waveforms as input.

PLEASE NOTE THE FOLLOWING:

- Excel has a calculation algorithm of 64,000 points (32,000 if you have created a chart in Excel). Therefore, make sure that your acquisition has less than this number of points if you are going to use an Excel calculation.
- To use this capability, you must have the LeCroy XMAP software option and Excel loaded in your instrument. Select **Minimize** from the instrument's **File** menu to access the Excel program directly.

How to Select a Math Function Call

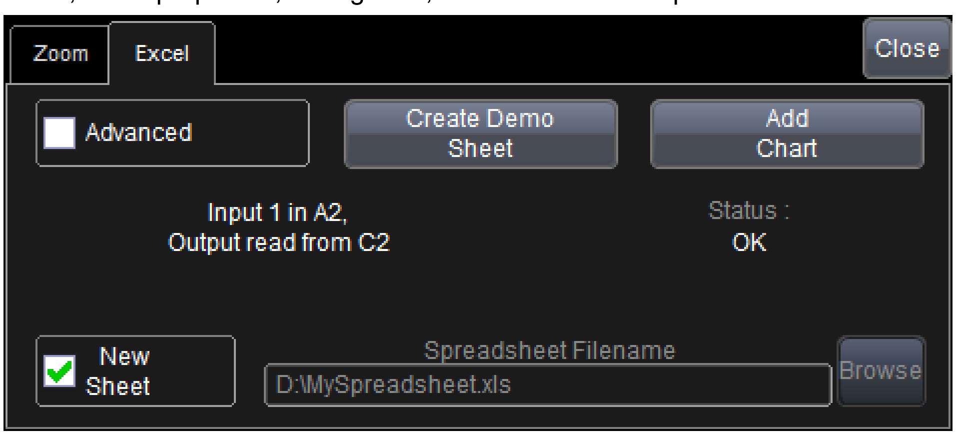
The Excel math function is selected from the **Math Operator** menu, where it appears in the **Custom** group.

How to Select a Parameter Function Call

The Excel Parameter function is selected from the **Select Measurement** menu, where it appears in the **Custom** group.

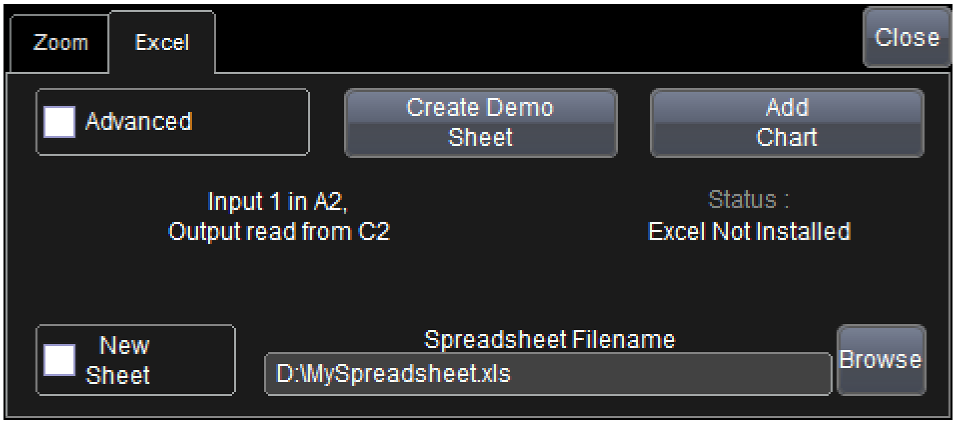
Excel Control Dialog

Once you have invoked an Excel call, you will see a dialog at the right of the screen, allowing you to control the zoom, Excel properties, linking cells, and scale of the output trace from Excel:



Entering a File Name

If you uncheck the **New Sheet** checkbox, you can enter the file name of an existing file.

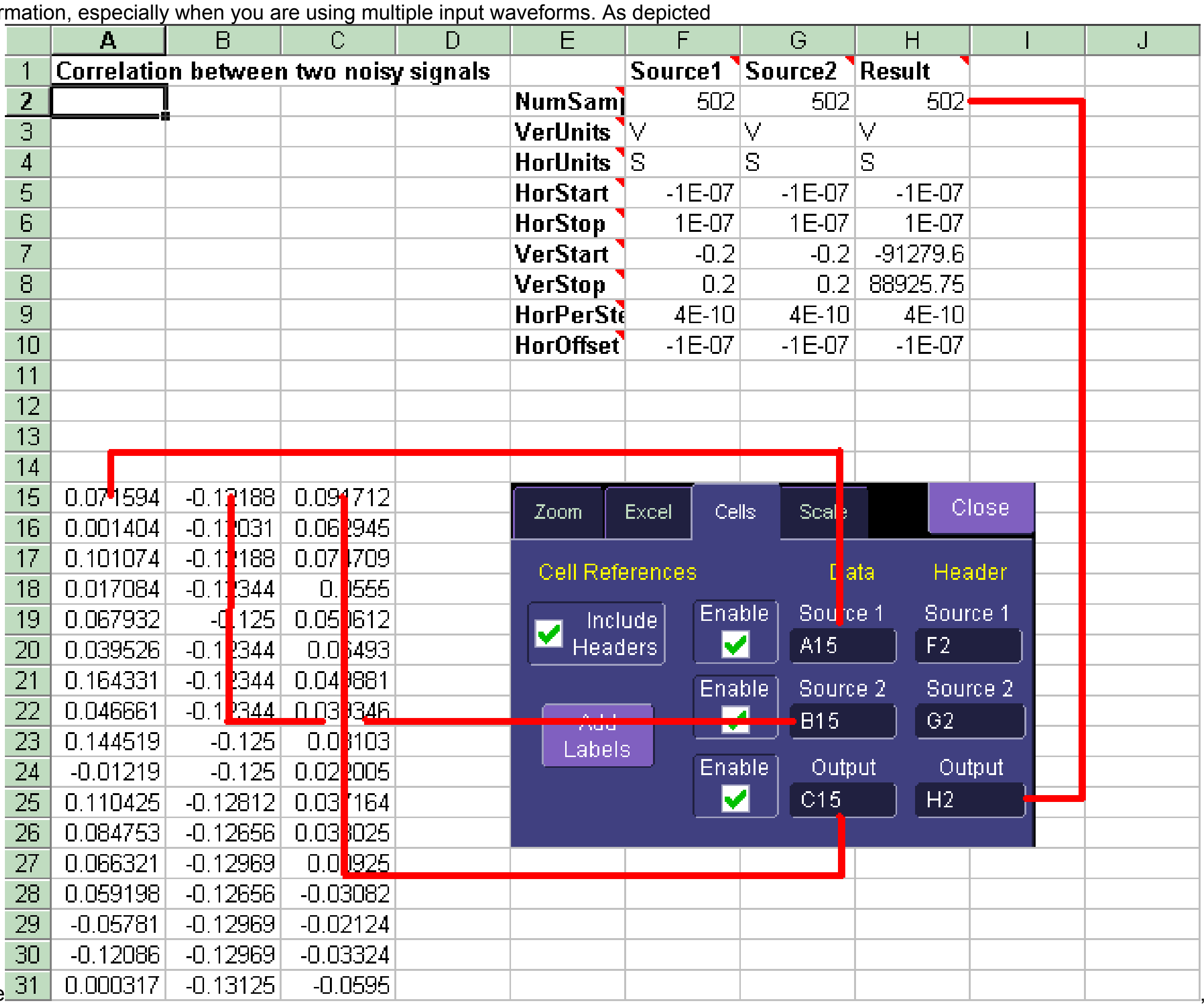


Create Demo Sheet Calls up a default Excel spreadsheet.

Add Chart Adds charts of your waveforms to Excel. You can go into Excel and create as many charts as you want.

Organizing Excel Sheets

The **Cells** tab allows you to organize your Excel chart. When placing the components in the sheet, be careful to avoid over-writing needed information, especially when you are using multiple input waveforms. As depicted here



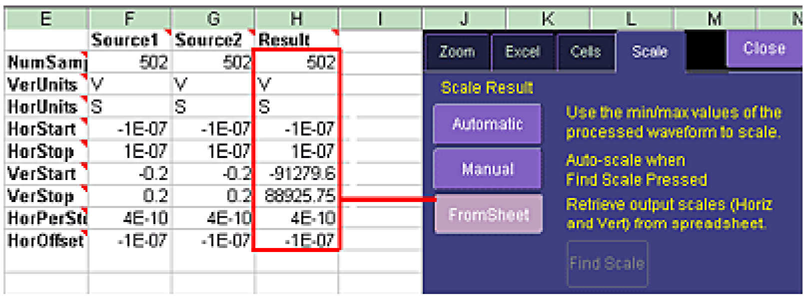
, the instrument panel has been pasted over the Excel sheet.

There are three arrays of data for the three waveforms: up to two inputs and one output. There are corresponding small arrays of information about each trace.

Setting the Vertical Scale

The vertical scale of the output waveform from Excel may be set in three ways:

Set Vertical Scale:	Description:
Automatic	For each acquisition, the instrument fits the waveform into the grid.
Manual	For one acquisition, click Find Scale; the instrument fits the current waveform into the grid. All subsequent acquisitions will use this scale until you make a change.
From Sheet	The scale is taken from the specified cells in the Excel sheet, H2 through H10 in the example above, where cell H2 was specified as the top of the data set, as depicted below.



Trace Descriptors

The next figure explains the meanings of the descriptors for each trace.

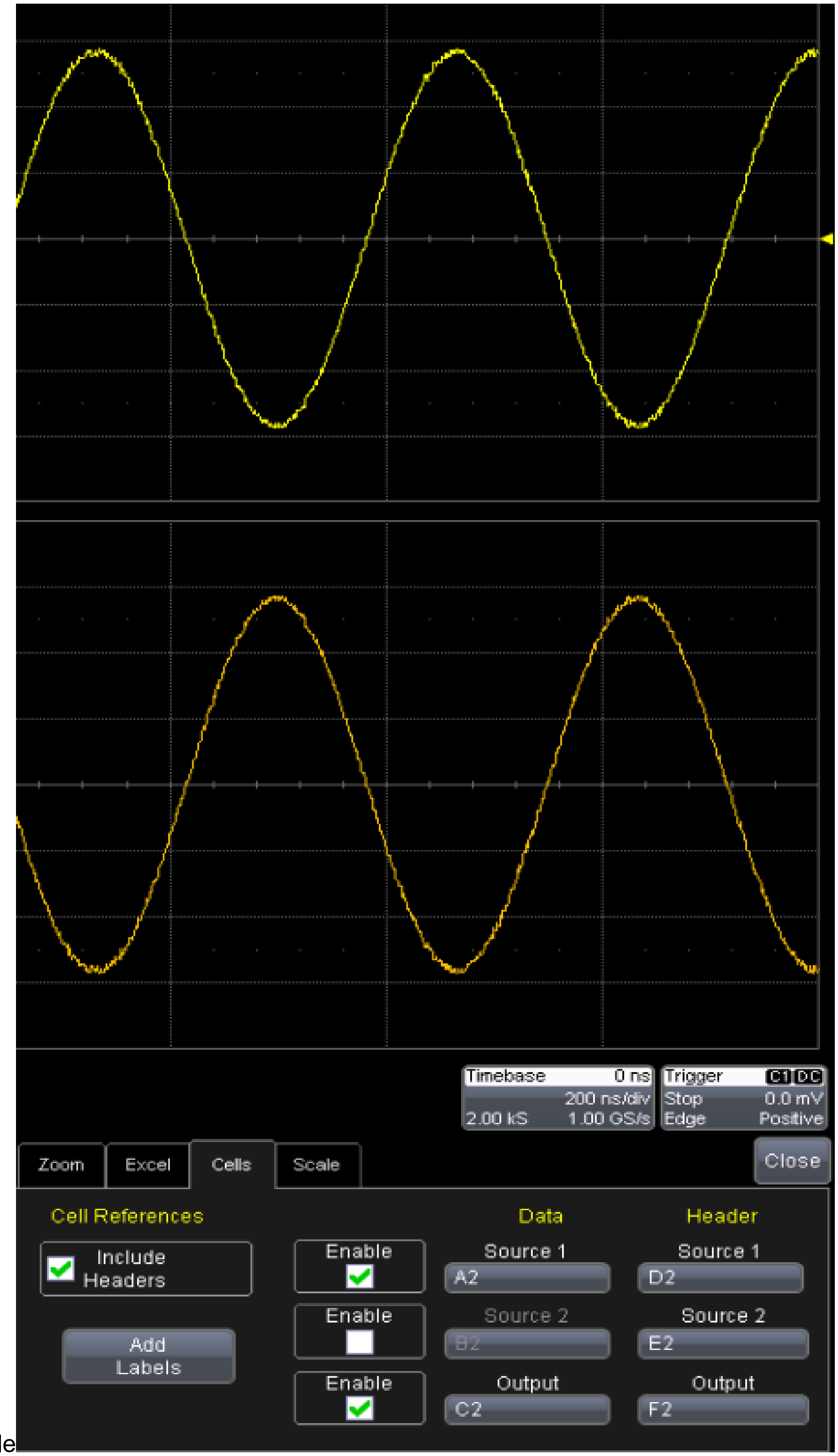
C	D	E	F	G	H
ı two noisy signals			**Source1**	**Source2**	**Result**
Samples in trace		**NumSam**	502	502	502
Vertical units		**VerUnits**	V	V	V
Horizontal units		**HorUnits**	S	S	S
Left edge of grid		**HorStart**	-5E-08	-5E-08	-5E-08
Right edge of grid		**HorStop**	1.5E-07	1.5E-07	1.5E-07
Bottom of grid		**VerStart**	-0.2	-0.2	-0.32585
Top of grid		**VerStop**	0.2	0.2	0.266995
Time per sample		**HorPerSte**	4E-10	4E-10	4E-10
Trigger position		**HorOffset**	-5E-08	-5E-08	-5E-08

Multiple Inputs and Outputs

If you invoke two or more instrument parameter functions or waveform functions that call Excel, you will find that they all refer to the same spreadsheet by default. Thus, your spreadsheet can use the data from several waveforms, and you can derive many different combinations of output parameters and waveforms, including some of each, from your spreadsheet. You only have to be careful about the positioning of your cell ranges within the sheet so that no conflicts occur.

Because filling cells in the spreadsheet is a relatively slow process, all unwanted sources (inputs) should be left disabled (unchecked). For example, if you want one waveform and two parameters derived from the data of three waveforms, you can have one function with both sources enabled, one with one source enabled, and one with no sources enabled. The alternative is to use one input in each function.

Simple Excel Example 1



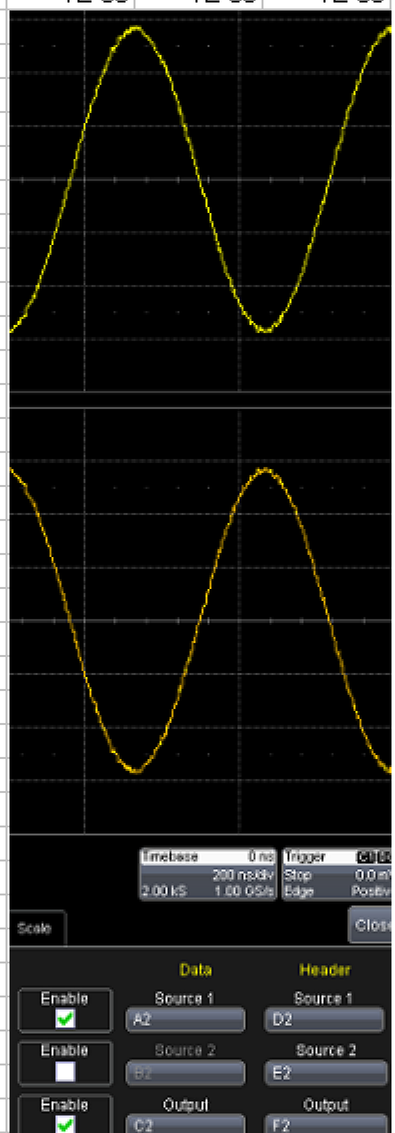
In this example we use Excel to invert or negate a waveform. The first figure shows a part of the screen. The upper trace is the original signal. The lower is the result from Excel.

The dialog is the one that controls the location of the data in the Excel worksheet.

The input data are in columns A and B (though, only the first is used) and the output is in column C. All have been set to start at row 2, allowing space for a title in row 1.

Columns D, E and F contain the headers for the three waveforms. These are the set of numbers that provide the description of the scope settings, such as vertical scale and offset, and number of samples.

	A	B	C	D	E	F
1						
2	0.150391		-0.15039	2002	2002	2002
3	0.150391		-0.15039	V	V	V
4	0.146484		-0.14648	S	S	S
5	0.148438		-0.14844	-1E-06	-1E-06	-1E-06
6	0.150391		-0.15039	0.000001	0.000001	0.000001
7	0.150391		-0.15039	-0.212	-0.21328	-0.21328
8	0.150391		-0.15039	0.212	0.213281	0.213281
9	0.148438		-0.14844	1E-09	1E-09	1E-09
10	0.148438		-0.14844	-1E-06	-1E-06	-1E-06
11	0.146484		-0.14648			
12	0.144531		-0.14453			
13	0.146484		-0.14648			
14	0.144531		-0.14453			
15	0.142578		-0.14258			
16	0.142578		-0.14258			
17	0.138672		-0.13867			
18	0.144531		-0.14453			
19	0.142578		-0.14258			
20	0.136719		-0.13672			
21	0.134766		-0.13477			
22	0.140625		-0.14063			
23	0.132813		-0.13281			
24	0.132813		-0.13281			
25	0.136719		-0.13672			
26	0.132813		-0.13281			
27	0.130859		-0.13086			
28	0.126953		-0.12695			
29	0.130859		-0.13086			
30	0.128906		-0.12891			
31	0.128906		-0.12891			
32	0.121094		-0.12109			
33	0.121094		-0.12109			
34	0.123047		-0.12305			
35	0.117188		-0.11719			
36	0.119141		-0.11914			
37	0.117188		-0.11719			
38	0.117188		-0.11719			
39	0.111328		-0.11133			
40	0.111328		-0.11133			
41	0.109375		-0.10938			
42	0.107422		-0.10742			
43	0.103516		-0.10352			



In this figure, the panel has been pasted onto the Excel sheet for comparison:

To get the output values in column C, we set C2 = - A2 and copy this formula down the column. This is the only action needed in Excel, and can be seen in the next figure:

C2 = =-A2

	Name Box	B	C	D	E	F
1						
2	-0.00781		0.007813	502	502	502
3	-0.03281		0.032813	V	V	V
4	-0.06094		0.060938	S	S	S
5	-0.08438		0.084375	-2.5E-06	-2.5E-06	-2.5E-06
6	-0.10781		0.107813	2.5E-06	2.5E-06	2.5E-06
7	-0.125		0.125	-0.2	-0.2	-0.2125
8	-0.13906		0.139063	0.2	0.2	0.214063
9	-0.14688		0.146875	1E-08	1E-08	1E-08
10	-0.14688		0.146875	-2.5E-06	-2.5E-06	-2.5E-06
11	-0.14844		0.148438			
12	-0.13906		0.139063			
13	-0.12812		0.128125			
14	-0.11094		0.110937			
15	-0.09375		0.09375			
16	-0.06563		0.065625			
17	-0.04219		0.042188			
18	-0.01406		0.014063			
19	0.015625		-0.01563			
20	0.04375		-0.04375			

Simple Excel Example 2

In this

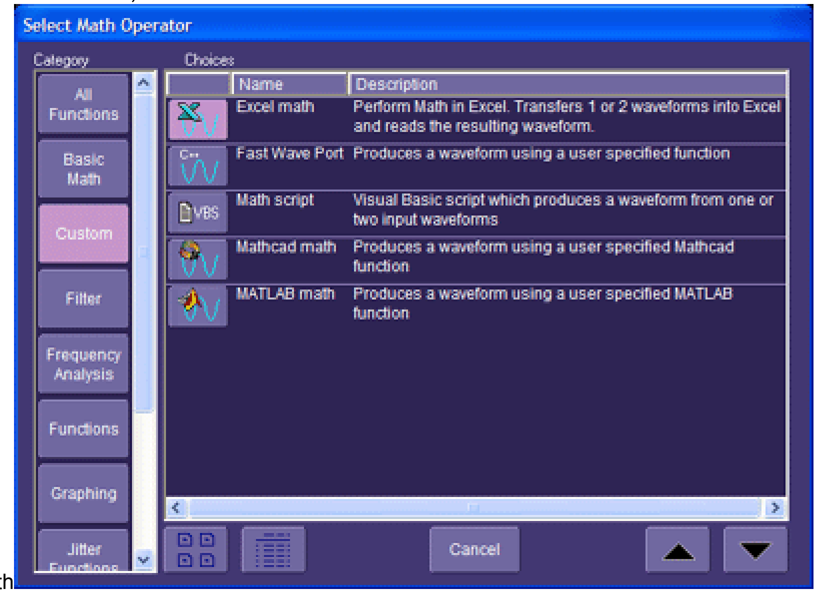
example we use Excel to invert or negate a waveform. The first figure shows a part of the instrument screen. The upper trace (C1) is the original signal. The lower trace (F1) is the result calculated in Excel and displayed on the screen.

The input data is in columns A and B (though by default, only a single input/column is used), and the output is in column C. All have been set to start at row 2 (which allows for a header in row 1).

To create this waveform, you would simply do the following:

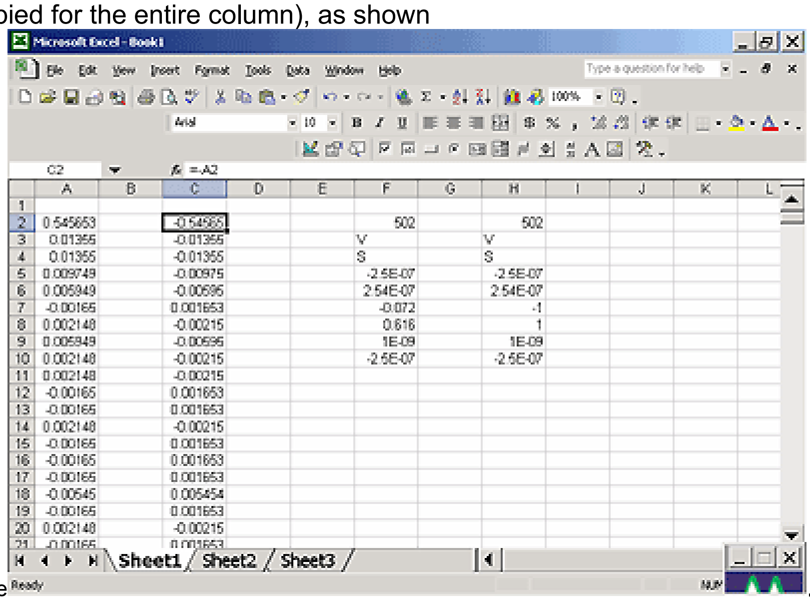
1. Ensure that your acquisition has no more than 64 kpts (the Excel calculation limit)

2. Choose a function, and select



ExcelMath as **Operator1** for the function. Excel will open automatically in the background.

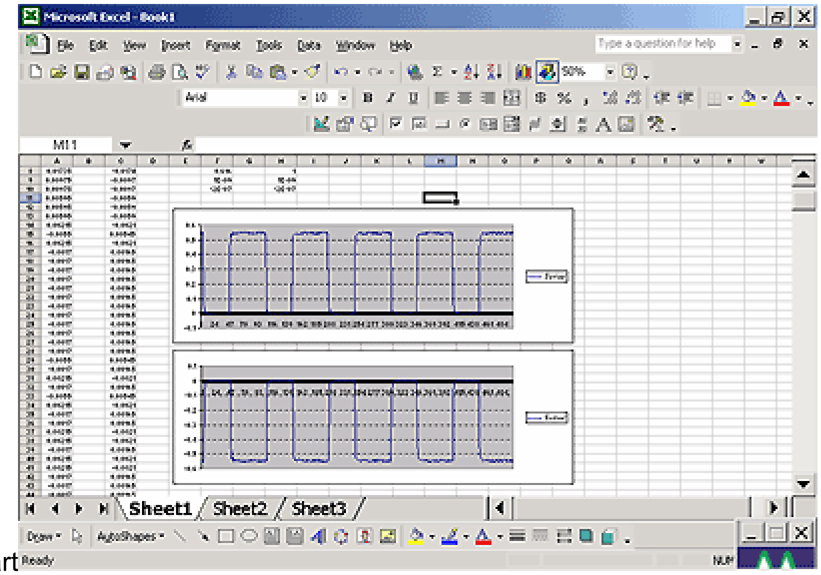
3. Choose File, Minimize from the menu bar to minimize the instrument display and open the Excel program.
4. Create your formula for each data point in column A (in this case, our formula for cell C2 is **–A2**, copied for the entire column), as shown



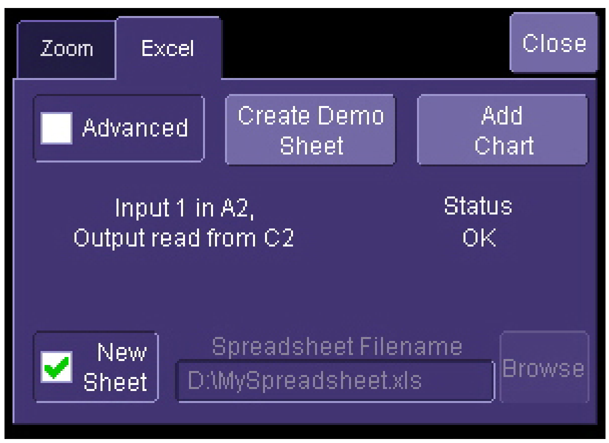
here.

5. Retrigger the scope (if it is not currently triggering)
6. Return to the program

Note that the only action that was needed in Excel was to create the formula in column C for each data point in column A. The instrument automatically opens Excel, puts the waveform data in the correct columns, and returns the calculated data back to the display as the chosen F trace. This Excel-calculated trace can have further measurements or math calculations performed on it, if desired.

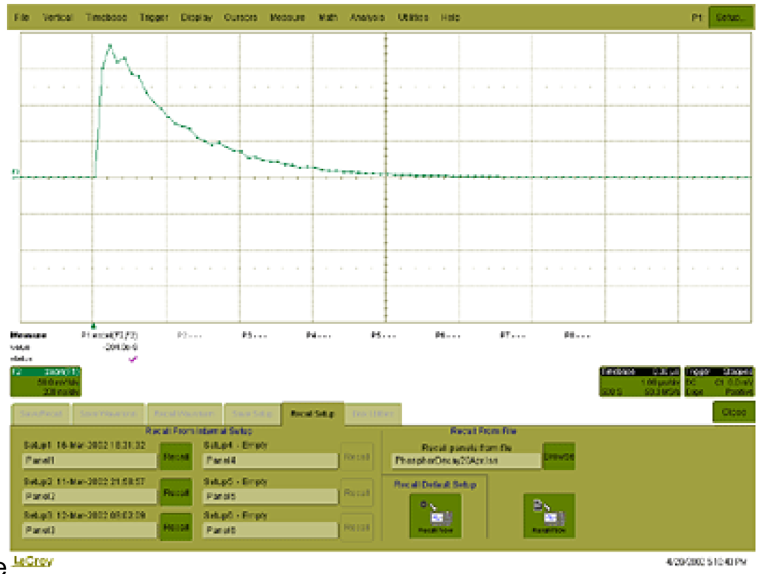


You can also create a chart of the data in Excel automatically and view the data there. Simply press the **Add Chart** button in the instrument's Excel dialog and a chart of the input (top chart) and Excel calculated output (bottom chart) will be automatically created in the spreadsheet. The chart will be updated automatically as the scope is triggered.



Exponential Decay Time Constant

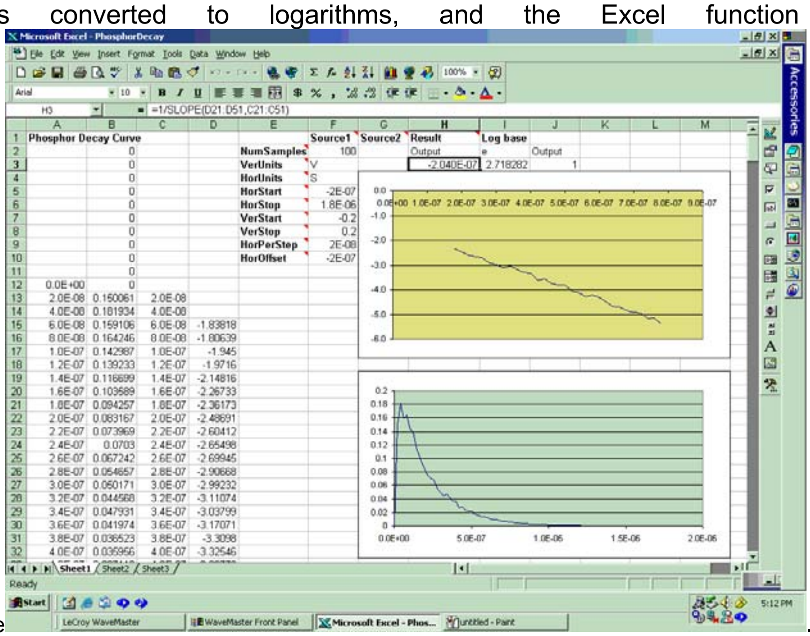
This example calculates the time constant of an exponentially falling pulse, such as the light output of a phosphor.



The first figure shows a typical pulse, including pseudo-random noise, generated by a VBScript.

The pulse was generated by a formula of the form $e^{(1 - t/TC1)} * e^{-t/TC2}$, where TC1 and TC2 are time constants, The requirement is to measure the time constant TC2, using the portion of the trace where TC1 has negligible effect. This was done using Function F1, which is not a part of the measurement process.

For the actual measurement, Parameter P1 was set up as an Excel call. In Excel, the selected portion of the trace was converted to logarithms, and the Excel function SLOPE was used, as shown here



.

Here we see the input data in column B (with a time scale in A) created using the contents of cell F9, Horizontal Per Step. The logarithmic data are in column D, with the time scale repeated in C. The output appears in cell H3, using the formula **=1/SLOPE(D21:D51,C21:C51)**.

Required files:

Setup: PhosphorDecay20Apr.lss

F1 Generator: PhosphorPulseGen.txt

P1 Excel: PhosphorDecay.xls

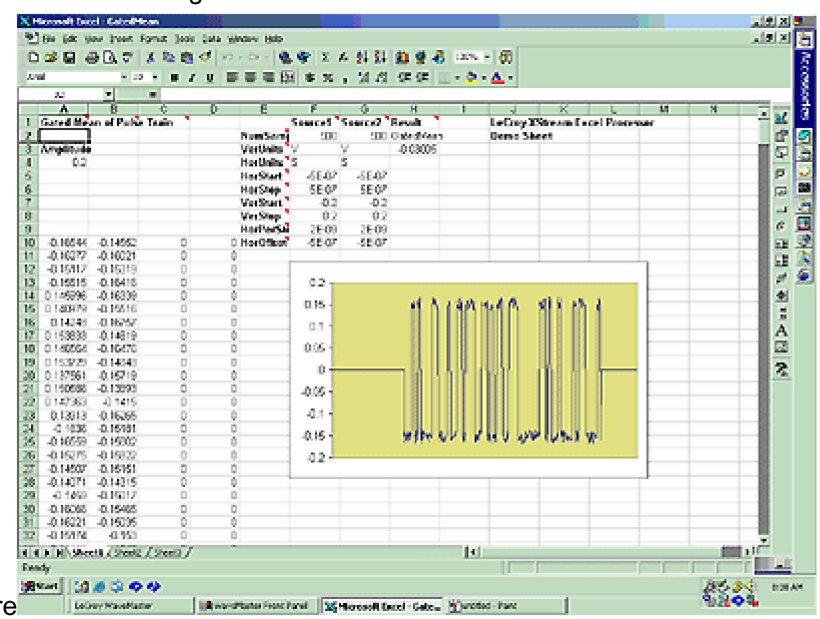
Gated Parameter Using Excel

This example calculates a parameter of a waveform, in a region of interest defined by the leading edges of two pulses in a separate waveform.

This figure shows the instrument screen:

The traces were made using VBS scripts in functions F1 and F2, based on pseudo-random numbers to provide noise and varying pulse widths. Randomize Timer:Randomize Timer was used in both scripts to ensure that successive acquisitions produced different data. Script F1 generates pulses with widths that are multiples of a set clock period. F2 generates one pulse in the first half of the time window, and one pulse in the second half. Both pulses are constrained to coincide with the clock pulses of F1. F1 and F2 are used here only as simulations and are not part of the measurement process, which only uses P1.

The call to Excel is made through Parameter P1.



The next figure shows a part of the Excel workbook.

Here we see the gated waveform that has been created in Excel. The Mean parameter during the region of interest (ROI) is placed in cell H3.

How Does this Work?

The amplitude of the signal is about 0.3 volts, and the screen height is 0.4 volts, as derived from cells F7 and Fx. A threshold value for amplitude was calculated by placing 0.5 * (Fy – Fx) in cell A4.

Remember that in the instrument the sources were defined to be A10 and B10. This means that the first point on the waveform will be read into A10, and, since the waveform has 500 points, the last point will be read into A510. The same holds true for F2 and column B, since F2 is assigned as Source2, and data is defined to write into column B starting with cell B10.

To create the gating function in column C, the cell C10 was given the following formula:

IF ((B10 – B9) > A4, 1 – C9, C9). This was copied down the column. Column D, the output column, is simply A * C.

The output was defined as cell H3.

The required mean in cell H3 is given by SUM (D10 : D509) / SUM (C10 : C509), for a 500 point waveform.

Requires files:

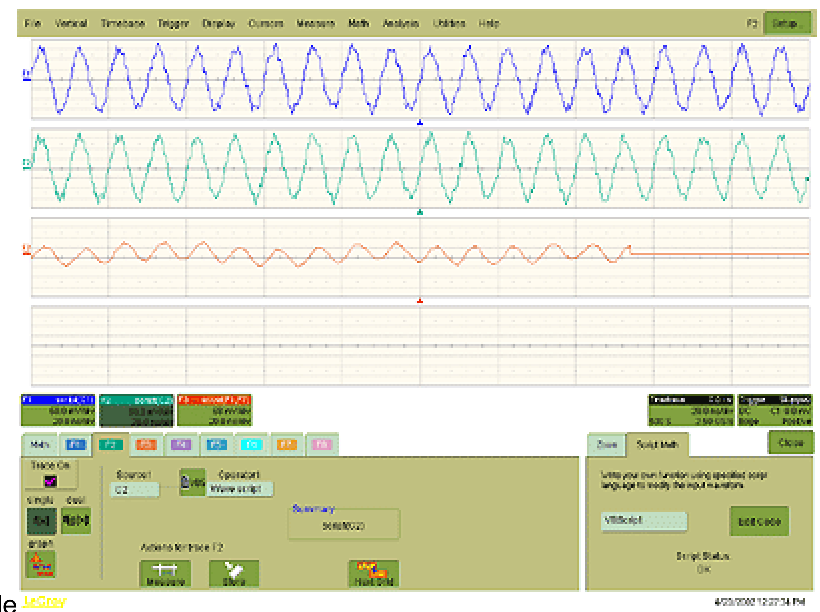
Setup: GatedParameterExcel.lss

Function F1: RandomPulses22Apr.txt

Function F2: RandomGate22Apr.txt

Parameter P1: GatedMean.xls

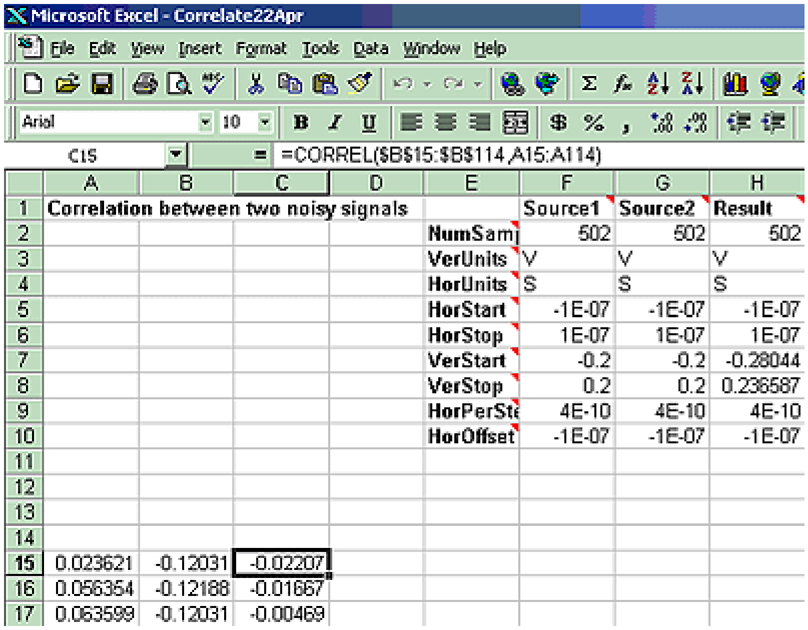
Correlation Excel Waveform Function



This example uses an Excel waveform function to examine the cross-correlation between two signals, which are both noisy sinusoidal segments. The correlation trace is, of necessity, shorter than the input traces.

The noise was generated using pseudo-random numbers. Randomize Timer was included in the VBScript to ensure that the two traces differed, and that subsequent acquisitions differed. Functions F1 and F2 are included only to simulate signals, and are not part of the measurement process, which is performed by F3.

This example used the CORREL (Array1, Array2) function of Excel, as depicted below:



Required files:

Setup: CorrelateExcel22Apr.lss

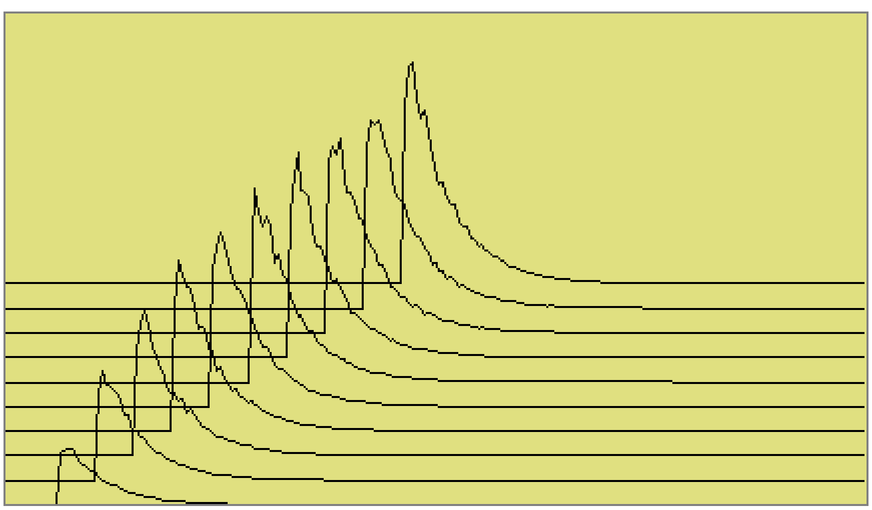
Function F1: NoisySine22Apr.txt

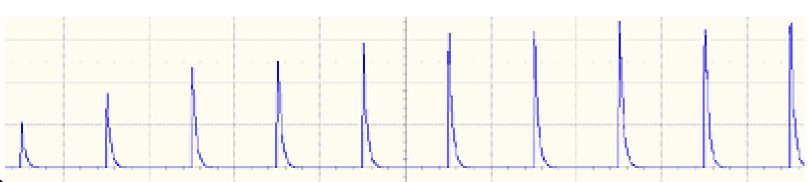
Function F2: NoisySine22Apr.txt

Function F3: Correlate22Apr.xls

Multiple Traces on One Grid

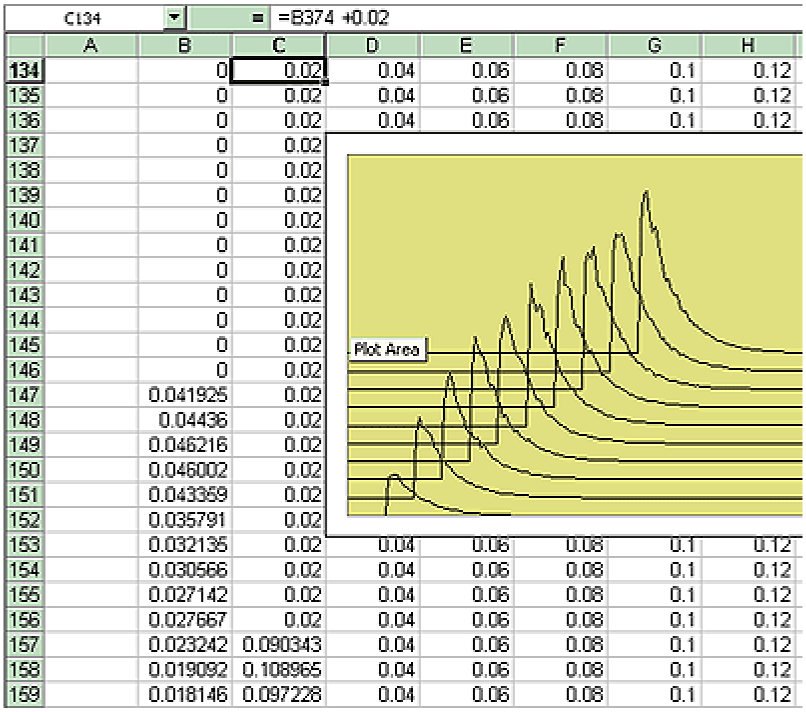
This example shows how you can place multiple traces in one picture, with only two operations in an Excel sheet. Depicted below is an example from an Excel spreadsheet.



Here is an original instrument trace  .

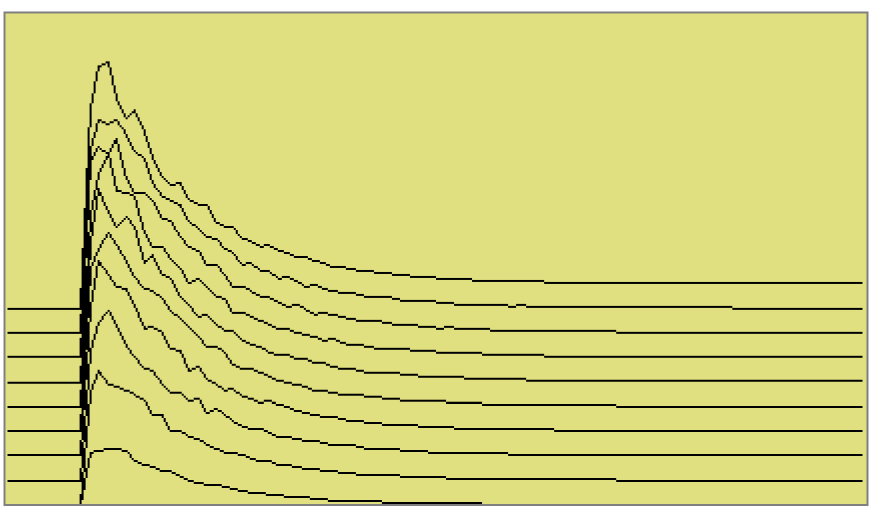
The method is very simple. First, the waveform is transferred to an Excel spreadsheet by means of an instrument Excel call. Second, two operations are needed in Excel: placing a simple formula in one cell, and copying that formula into a range of cells.

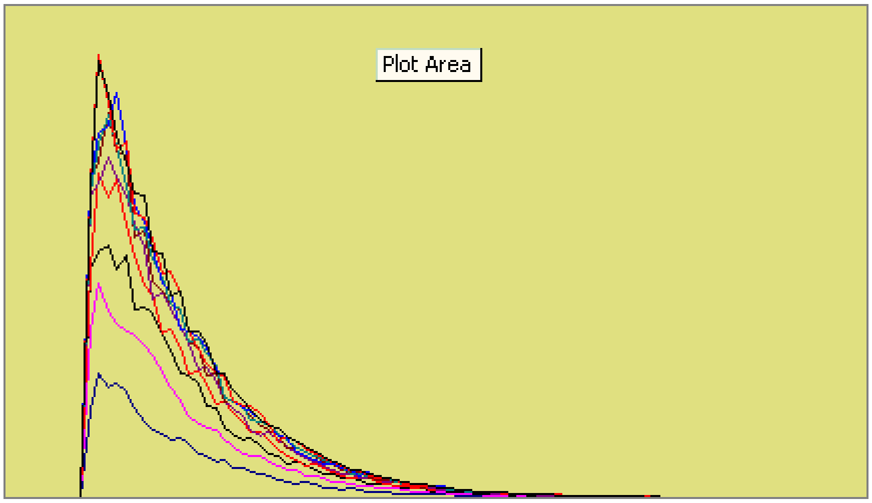
Depicted below is the required Excel formula.



	A	B	C	D	E	F	G	H
134		0	0.02	0.04	0.06	0.08	0.1	0.12
135		0	0.02	0.04	0.06	0.08	0.1	0.12
136		0	0.02	0.04	0.06	0.08	0.1	0.12
137		0	0.02					
138		0	0.02					
139		0	0.02					
140		0	0.02					
141		0	0.02					
142		0	0.02					
143		0	0.02					
144		0	0.02					
145		0	0.02					
146		0	0.02					
147		0.041925	0.02					
148		0.04436	0.02					
149		0.046216	0.02					
150		0.046002	0.02					
151		0.043359	0.02					
152		0.035791	0.02					
153		0.032135	0.02	0.04	0.06	0.08	0.1	0.12
154		0.030566	0.02	0.04	0.06	0.08	0.1	0.12
155		0.027142	0.02	0.04	0.06	0.08	0.1	0.12
156		0.027667	0.02	0.04	0.06	0.08	0.1	0.12
157		0.023242	0.090343	0.04	0.06	0.08	0.1	0.12
158		0.019092	0.108965	0.04	0.06	0.08	0.1	0.12
159		0.018146	0.097228	0.04	0.06	0.08	0.1	0.12

In fact, the simple expression B374 + 0.02 comprises several components. The original instrument trace is in column B, and the plot is required to start at cell B134. The traces repeat at intervals of 250 cells. Let us call this interval R. If we require a horizontal displacement D, then in cell CN we write B(N + R – D). In this example D is 10. Finally we may want a vertical displacement V, and we write B(N + R – D) + V. In this example, V is 0.02. D and V can be zero if required, as depicted below. All that remains is to copy the formula to the required range of cells.



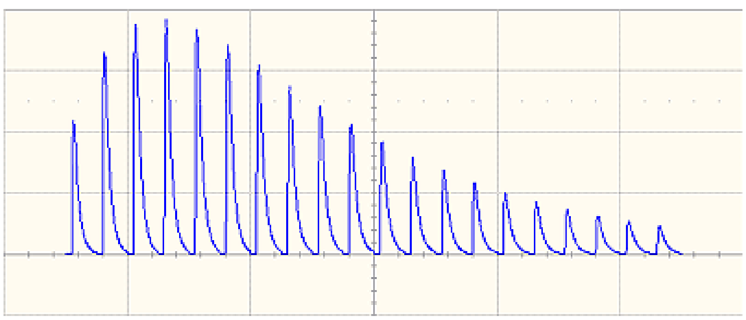


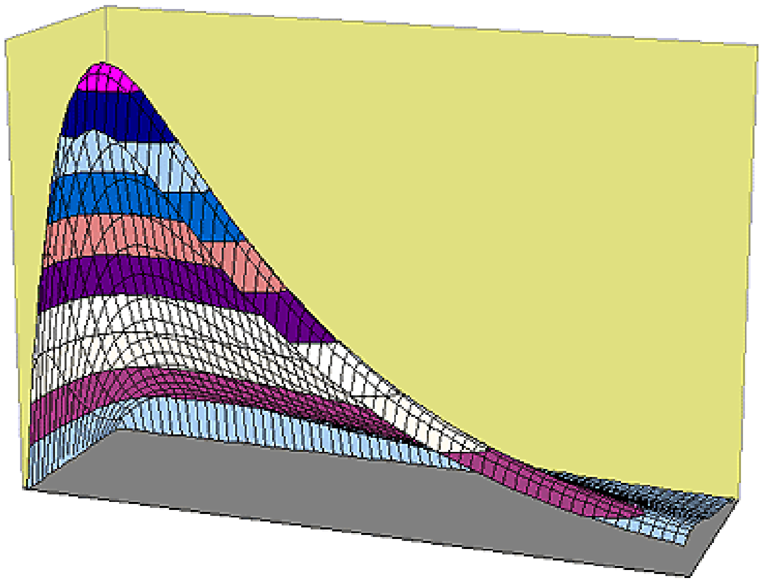
Required files:

F1 is needed only as a simulator of signals.

Instrument setup:	LaserStartup25Apr.lss
Function F1:	LaserStartupApr25.txt
Function F2:	LaserStartupPulses.xls No offset LaserStartupPulses2.xls Vertical offset LaserStartupPulses3.xls Vertical and horizontal offset

Using a Surface Plot





Required files:

Setup: LaserSurface1May.lss

Function F1 Generator: LaserSurface2May.txt

Function F2 Excel: LaserSurface2May.xls

Loading and Saving VBScripts

From the editing panel you can save your script and you can load a previous one. Should you forget to save a script, please note that when you save your setup, it has your current scripts embedded in it. Therefore it is a good idea to save your setup frequently. It is worth saving the script separately as well, because it is saved in a suitable format for printing or off-line editing with Notepad. Note that in both these examples the input data are referred to as InResult.DataArray. You can also write InResult1.DataArray and InResult2.DataArray, which refer to the two input traces. InResult.DataArray always refers to input trace 1. These remarks hold for any script that you write.

Example Waveform Function Script: Square of a Waveform

```
' Example script to produce a waveform
This example calculates the square of
the input waveform.
OutResult.Samples = InResult.Samples ' Visible trace length + 1
' Note that a trace of nominal length 1000 comprises data numbered from
' 0 to 1001. The 1001st point is not visible, so you
' normally use points 0 to 1000,
' giving 1001 points and 1000 intervals between points.
startData = 0
endData = OutResult.Samples
LastPoint = endData - 1 ' because the last point is invisible.
ReDim newArray(OutResult.Samples) ' to store the results
unscaledData = InResult.DataArray(False)
' InResult.DataArray(False) provides
' integer data from -32768 to 32767.
' InResult.DataArray(True) provides real data
' in the same physical unit as the vertical scale of the input trace.
ScaleFactor = 1.0 / 32768 ' to make the trace fill the screen.
       For i = 0 To LastPoint
```

```
        newArray(i) = ScaleFactor * (unscaledData(i)) ^ 2
        Next
OutResult.DataArray(False) = newArray ' signed long integer data output
```

Example Parameter Function Script: RMS of a Waveform

```
' Example script to produce a parameter.
' This script calculates the root mean square
' of the input waveform.
' Note that a trace of nominal length 1000 has data from
' 0 to 1001. The 1001st point is not visible, so you
' normally use points 0 to 1000,
' giving 1001 points and 1000 intervals between points.
startData = 0
endData = InResult.Samples
LastPoint = endData - 1 ' because the last point is invisible.
ReDim newArray(InResult.Samples) ' to store the results
unscaledData = InResult.DataArray(True)
' InResult.DataArray(False) provides
' integer data from -32768 to 32767.
' InResult.DataArray(True) provides real data
' in the same unit as the vertical scale of the trace.
Total = 0
        For i = 0 To LastPoint
        Total = Total + (unscaledData(i)) ^ 2
        Next
NewArray(0) = Sqr (Total / (LastPoint + 1) Place the result in the zeroth element.
OutResult.ValueArray(True) = newArray ' integer data output
```

The Default Waveform Function Script: Explanatory Notes

InResult.Samples is the number of points in the incoming waveform.

InResult.DataArray(Boolean) (or InResult1.DataArray or InResult2.DataArray) is the array of input data. If the Boolean is True you get scaled real data in the units of the trace. If the Boolean is false you get unscaled integer data in the range -32768 to + 32767.

The value of InResult.Samples is the total number of data in a trace. It is two more than the nominal value given on the screen. The first point DataArray(0), coincides with the left edge of the screen, apart from the wobble caused by the trigger-to-sample clock difference. If the trace length is nominally 500, the right edge of the screen coincides with DataArray(500), which is the 501st point. The last point, number 502, is just off the right of the screen, and is never seen. That is why the loop in the script runs only to endData - 1.

OutResult.Samples is the number of data in the output trace, and is set to be the same as the number of data in the input trace. If you set the output length less than the input length, you get a shorter trace, the remainder being made of zeroes. If you try to set the output values to something illegal, you may find that a part of the trace retains the values from a previous acquisition.

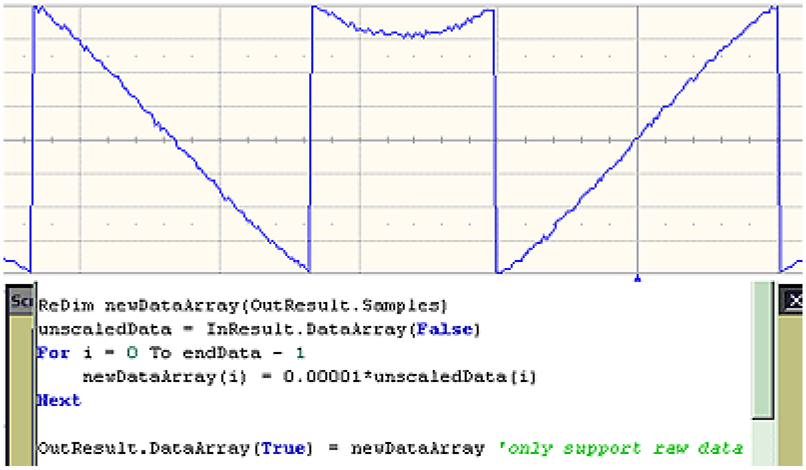
If you try to set something outside the bounds of an array, or you make some other error, or something overflows, or you ask for something impossible, such as log(-13), the instrument tells you the line number, and the nature of the problem. Other types of error may not be given the correct line number, for example, if "Next" or "End If" is omitted, because VBS does not know where it should have been.

UnscaledData is simply a copy of the input data set.

ReDim newDataArray(OutResult.Samples) defines an array of data for use as a scratch pad. Dim is short for Dimension, which is used in Visual Basic to declare a variable (even if it only has one element, in which case you omit the size of the array).

InResult.DataArray(False) means that the data are signed integers in the range -32768 to 32767. False is a Boolean value applying to the property Scaled. Scaled data are specified in the units of the vertical scale, such as volts. You get these by putting "True" instead of "False". If you want to make a section of the output trace invisible, you simply set the data values to full scale or bigger, top or bottom.

You can start with the unscaled data (False) as input, and then set the output data to scaled data (True), and you can go from scaled to unscaled. Using scaled data, an overflow will make a picture like this:



You can also start with True and convert to False, but in this case overflows will cause an error message.

Anything after a single quotation mark on a line will not be used by the instrument. This feature is intended for comments, for example

```
' This is a comment.
A = Amp * Sin(Omega * T) Calculate the output.
```

InResult.DataArray and OutResult.DataArray are only to be used as shown in the default scripts and in the example scripts: you cannot refer directly to individual elements of these arrays. You have to use your own arrays, in this example, unscaledData and newDataArray. You are not allowed to write statements like the following:

```
Y = InResult.DataArray (17)
OutResult.DataArray (257) = Z
```

Some parts of the default script must not be changed because they are a part of the interface. These are **highlighted** in the following script .

`' TODO add your custom code here accessing OutResult and InResult objects`

`' Here's a small example that just inverts the waveform.`

`OutResult.Samples = InResult.Samples`

`startData = 0`

`endData =` **`OutResult.Samples`**

`newNumPoints = endData - startData`

`ReDim newDataArray (`**`OutResult.Samples`**`)`

`unscaledData =` **`InResult.DataArray`** `(False)`

`For i = 0 To endData - 1`

`newDataArray (i) = - unscaledData (i)`

`Next`

`OutResult.DataArray` `(False) = newDataArray 	' only support raw data`

The four highlighted quantities are parts of the interface. The names must be retained. Furthermore, InResult.Samples and InResult.DataArray are inputs, and their values cannot be changed. OutResult.Samples and OutResult.DataArray are outputs, and can be changed, but not directly through their individual elements.

Default Parameter Function Script

The default parameter script is similar to the default waveform script, but there are subtle differences.

First, the size of the data array is the same as the nominal value: you cannot use or see the extra two points. So "500 points" means just that: 500 points.

Second, the output looks like an array, but only element zero is currently used. You must copy your parameter result into newValueArray(0). As with the arrays of the Waveform Script, you cannot refer directly to elements of the input and output arrays. You may not write something like

```
OutResult.ValueArray (0) = P.
```

Note that the unit of the parameter is displayed as the same as the vertical unit of the trace, even if you have squared the data, for example, unless you change the unit yourself.

To find out how to edit a parameter script, click here.

The default parameter script is shown below.

```
' TODO add your custom code here accessing OutResult and InResult objects
' Here's a small example that just inverts the waveform
numParam = InResult.Samples
ReDim newValueArray(numParam)
scaledData = InResult.DataArray
     For i = 0 To numParam-1
     newValueArray(i) = -scaledData(i)&#9;' Change this to do something useful.
     Next
OutResult.ValueArray = newValueArray 'only support raw data
```

Your parameter script should include something like this:

A. Do calculation to obtain your parameter value from the input data array.

B. newValueDataArray (0) = ParameterValue

C. OutResult.ValueArray = newValueArray

You can test this script using setup MeanDemoScriptApr2.lss.

You can edit scripts using Notepad, but you will not get any notification of errors.

You are not allowed to write OutResult.ValueArray(0) = MeanParameter.

InResult.DataArray and OutResult.DataArray are only to be used as shown in the default scripts and in the example scripts. You cannot refer to, or modify, any individual element in these arrays.

Hints and Tips for VBScripting

- Set the trigger to Single or Stopped if you need to do a lot of editing: it is faster.
- Before starting a script, remove any existing scripts that you do not need. This is because errors in an existing script will give you error messages, even if your current script is perfect. And an existing good script may develop a fault if you change the setup. For example, you might change the vertical scale or the memory length and get an overflow if you did not guard against it in the script.
- When starting a script, make sure that you have chosen the right kind: function or parameter. You can get some very frustrating problems if you are in the wrong mode. You can cut and paste the VBS statements if you discover this error.
- If your calculation requires a long memory, development might be quicker if you test the principles on a shorter trace at first.
- Note that the pseudo-random number generator is reset at the start of a script. If you want a different set of pseudo-randoms every time, put Randomize Timer in the program, to be run once, before any pseudo-randoms are generated. You can use this instruction to re-seed the generator at any time during execution.
- Do not put the final statement in a loop, hoping that you can see a progressive result as some parameter changes. No output will be seen on the screen of the instrument until the script has been completely run and quitted, so only the final result will appear. If the loop runs many times, you will think that the scope has hung up.
- If you want a For loop, end it with "Next" and not "Next X".

- If you make a script that takes a long time to run, go back to the default setup before quitting or powering down, or you will have a long wait next time you power up.
- Always use a recursive calculation when this will speed things up.
- Keep everything outside a loop that does not have to be inside, to speed things up.
- Make your scripts clear, not only by indenting and commenting, but by structuring neatly as well.
- Sometimes it might be easier to develop your script in Excel VBA (remembering that VBA is not identical to VBS), so that you can display intermediate results. If you do this, note that you can read from a cell or write to it using statements like these:

```
A = Worksheets("Sheet1").Cells(Row, Column).Value

Worksheets("Sheet1").Cells(Row, Column).Value = B
```

- Note that in VBS, after you have corrected an error and clicked on "Apply," the error message may go on flashing for a few seconds, or a few acquisitions, before being erased. Look for the "Script OK" message. Be patient before assuming that you still have a bug.
- If your calculation requires data to be used at some other horizontal positions than their original ones, make sure that your algorithm does not try to send data to non-existent array positions, that is, beyond the edges of the screen. You may have to truncate your output trace, as happens with the instrument's Enhanced Resolution and Boxcar functions.
- No output will emerge from a script until you press Apply.
- No output will emerge from a script until it has received an input. This includes the case where the input data are not used in calculating the output data. So you must have had at least one acquisition before you see anything.
- Because you can introduce undeclared variables at any point in a calculation, VBS does not check your spelling.
- You can make a portion of a trace disappear if you set the values to 32767 or -32768.
- You can highlight a section of a trace by making the points alternately too high and too low by a suitable amount. Providing the memory length is not too short, the compaction algorithm will give the effect of a thicker trace.
- The lengths of the output trace and the input trace need not be the same. You can even make the output trace longer than the input trace, but you will need to unzoom it to see it all. This feature can be used to avoid compaction problems with non-linear horizontal scales. It can also be used to show several versions of a function at the same time, without having to set up a separate script for each one.
- If your program structure is complicated, consider typing all the IFs, ELSEIF's, ENDIF's, FOR's, NEXT's, etc and then clicking Apply. You won't get any output, but the system will tell you if the structure is acceptable. Then you can insert the actual program statements.
- Always try to make the script as independent as possible of variables such as V/Div, T/Div, and memory length, unless that would make it harder to understand. If so, give some values as examples, and explain how the script would have to change if the variables changed.

Errors

The instrument VBS tries hard to help you when errors occur.

Errors may be of two main types:

- The script may not be usable because the interpreter cannot construct a logical structure from it.
- The script may be usable, but may fail while running because an incomputable function has been requested.

Sometimes the line number given for an error is wrong. This can happen when the error is of this general type:

```
Missing "Next" Missing "End If"

Extra "Next" Missing "Until" etc.
```

This happens because VBS cannot know where you should have put the statement.

If at some point during the calculation of an output array, a value goes outside the allowed range, the calculation will stop, and you will see the new values up to the point of the stoppage. To the right of that point, the trace will display the previous values. In fact, if you deliberately recalculate only a part of a trace, you can have a mixture of new and old values.

In the figure below is a type of error message that you may see if one of your calculations has tried to set a value outside the range -32768 to +32767. It takes extra time to guard against this, but unless you are sure that it will not happen, you need some kind of check. In the example on the next page, the red trace has gone outside the allowed range at the beginning, resulting in the message at the bottom of the instrument screen: This array is fixed or temporarily locked: `OutResult.DataArray`.

Error Handling

Note that the construction OnError GoTo Label: is not allowed in VBS. In fact no GoTo's or labels are allowed. Therefore there is no way for you to provide handlers to deal with errors and exceptions. You must be aware of all possibilities at all points in your program, and you must either be certain that errors will not occur, or you must take action to ensure that they do not.

Examples:

Function:	Description:
Sqr	You cannot take the square root of a negative number.
Log	You cannot take the log of zero or of a negative number.
A / B	You cannot divide by zero.
Array	You cannot use an index outside the bounds of an array.
Size	Unscaled data cannot go outside the range -32768 to 32767.

If there is any possibility that any of these might occur, take steps to deal with this before it can happen.

For example, you may write some kind of generator of pseudo-random statistical values. If these belong to a distribution that in principle has an infinite range, or a finite range which is wider than the signed 16-bits allowed, check each value. If a value falls outside the range, you could set it to the maximum or generate another example.

You can, however, use one of the following:

```
On Error Resume Next
```

followed by some code that may make some attempt to deal with the problem, or at least to allow execution to continue.

```
On Error GoTo 0
```

This cancels On Error Resume Next

Speed of Execution

To maximize the speed of execution of a script, the most important thing you can do is to minimize the number of operations that are performed inside loops. Anything done once only is unlikely to be an important source of delay. Please note that VBS is much slower than the internal computations of the instrument, so do everything you can to save time, unless time is irrelevant to the application.

Using an array element takes longer than using a single variable. Here is an example:

```
For K = 1 to Total
    If X (K) > X (K - 1) Then
    Y = Cos (X (K) ) * Sin (X (K) ) * Sqr (X (K) )
    End If
Next
```

To do the same thing we could also write this, using the index only once:

```
OldXK = X (0)
    For K = 1 To Total
    XK = X (K)
        If XK > OldXK Then
        Y = Cos (XK) * Sin (XK) * Sqr (XK)
        OldXK = XK
        End If
    Next
```

VBS runs slower than the "internal" calculations, because the scripts are interpreted. This could be serious for calculations where many operations are needed on each sample, such as convolution, correlation, and long digital filters.

Scripting Ideas

What can we do in a VBS script that we cannot do with the normal instrument functions? Here are some possibilities.

- Create a new function that acts on waveform values.
- Create a new parameter.
- Create a new form of non-linear vertical scale.
- Create a new form of non-linear horizontal scale.
- Move some or all data horizontally, including reflections.
- Combine data to form digital filters.
- Show several function results side by side.
- Show several function results interleaved.

You can even create output data that are not related to the input. The output data need not even be in the same domain as the input data, because the system treats them as pure numbers. So you can create your own transforms into the frequency domain, for example.

Example Waveform Script

Creating a window function for FFT calculations.

Example Parameter Script

Calculating the rate of decay of a damped sine.

Finding pulses in a pulse train.

Debugging Scripts

Until we have integrated a more comprehensive debugger for VBScript there is a workaround.

1. Download the Windows Scripting Debugger for Windows 2000 from here:

   http://download.microsoft.com/download/winscript56/Install/1.0a/NT45XP/EN-US/scd10en.exe

2. Enable JIT (Just In Time) debugging by setting the following registry key

   HKCU\Software\Microsoft\Windows Script\Settings\JITDebug = to 1 (DWORD value)

3. Place a **Stop** statement in your script.

Now, when the Stop statement is executed the debugger will open and allow single-stepping, variable examination, etc.

Using VBA or Visual Basic to debug VBScripts is not recommended since the language syntax for these three variants of basic is slightly different.

Calling MATLAB from the Scope

> **Note:** Load MATLAB version 6.5 just as you would on any PC. Once it is loaded, open MATLAB from the desktop, then close it again, before you attempt to open it from the instrument application. This is to update the registry.

MATLAB can be directly called from the instrument in two ways:

Calling MATLAB:	Description:	Result:
Using a function	F1 through Fx[The number of math traces available depends on the software options loaded on your scope. See Specifications.]	MATLAB returns a waveform
Using a parameter	P1 through Px	MATLAB returns a parameter

In both cases, one call to MATLAB can use two separate waveforms as input, providing much greater computing power than is available by calling MATLAB from a VBScript.

> Note: If you do not place a semicolon ";" at the end of a line, MATLAB will show the calculated value in the result window, significantly slowing down the processing rate. This feature is best kept for diagnostics.

Selecting a Waveform Function Call

The MATLAB Waveform functions are selected from the **Select Math Operator** menu. Please note that once you have clicked on "MATLAB Wave" there will be a slight pause before MATLAB starts.

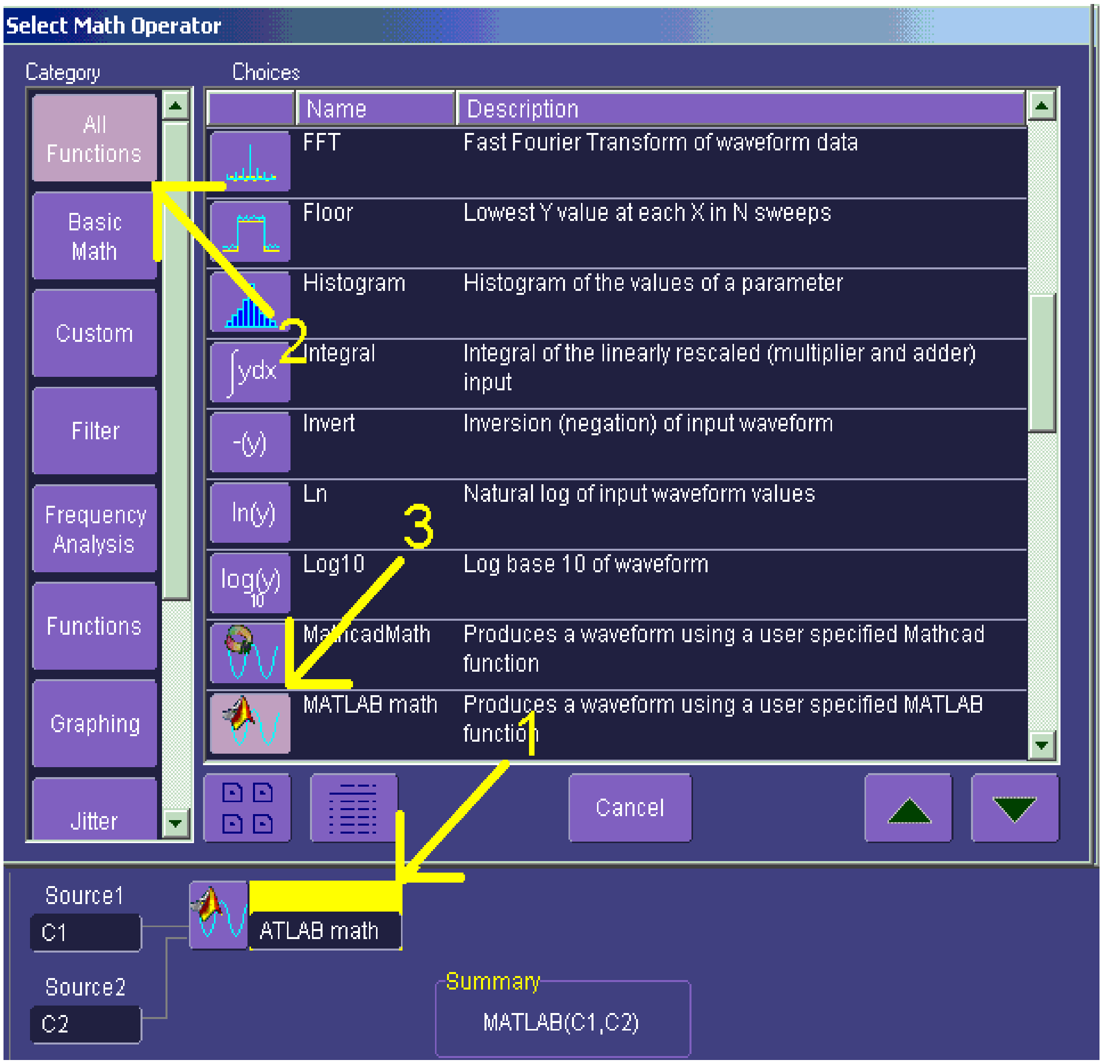
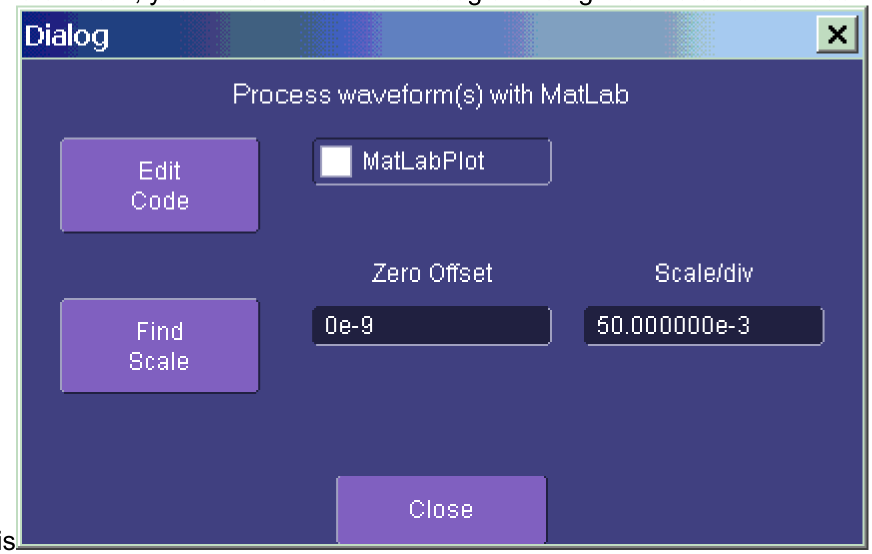


Figure 4-6. Source 1 and Source 2 are the waveforms that MATLAB will use.

MATLAB Waveform Control Panel

Once you have invoked a MATLAB waveform call, you will see the zoom dialog at the right of the screen. Touch the MATLAB tab to see a panel like this

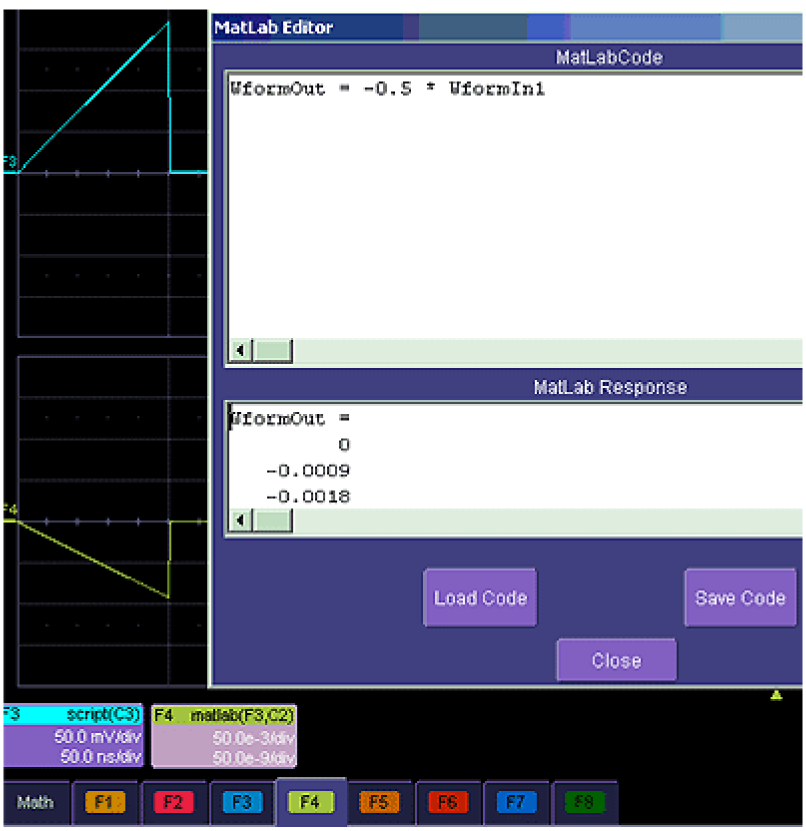


.

Touch **Find Scale** to make your output fit the grid, or use the text boxes to choose a scale.

MATLAB Waveform Function Editor

By touching **Edit Code**, you can reach the MATLAB Editor where you will see the default waveform function. If you are familiar with MATLAB, you might prefer to launch MATLAB and create a MATLAB function that performs your task. Your program in the instrument could then be a one-line call of your MATLAB function.



This is the default waveform function, with one important change – the semi-colon (;) has been removed from the end of the line. If the semicolon is present, your function will run much faster, because the output values will not be shown in MATLAB Response. With a long waveform, the time needed to display it could be quite long. The response values can be useful during development and debugging. Any line without a semicolon will produce a visible MATLAB Response.

From this panel you can save your code, load a previous code, and edit your function. A powerful feature of MATLAB is that you can refer to an entire waveform as a vector. The two input waveforms are WformIn1 and WformIn2, while the output is WformOut. You can also refer to individual samples, such as WformIn1(34), and sequences of samples, such as WformIn(55:89)

You can write statements such as these:

```
WformOut(5) = WformIn(5)
WformOut(89) = WformIn(144)
WformOut(34:55) = WformIn(34:55)
WformOut(233:377) = WformIn(100:244)
```

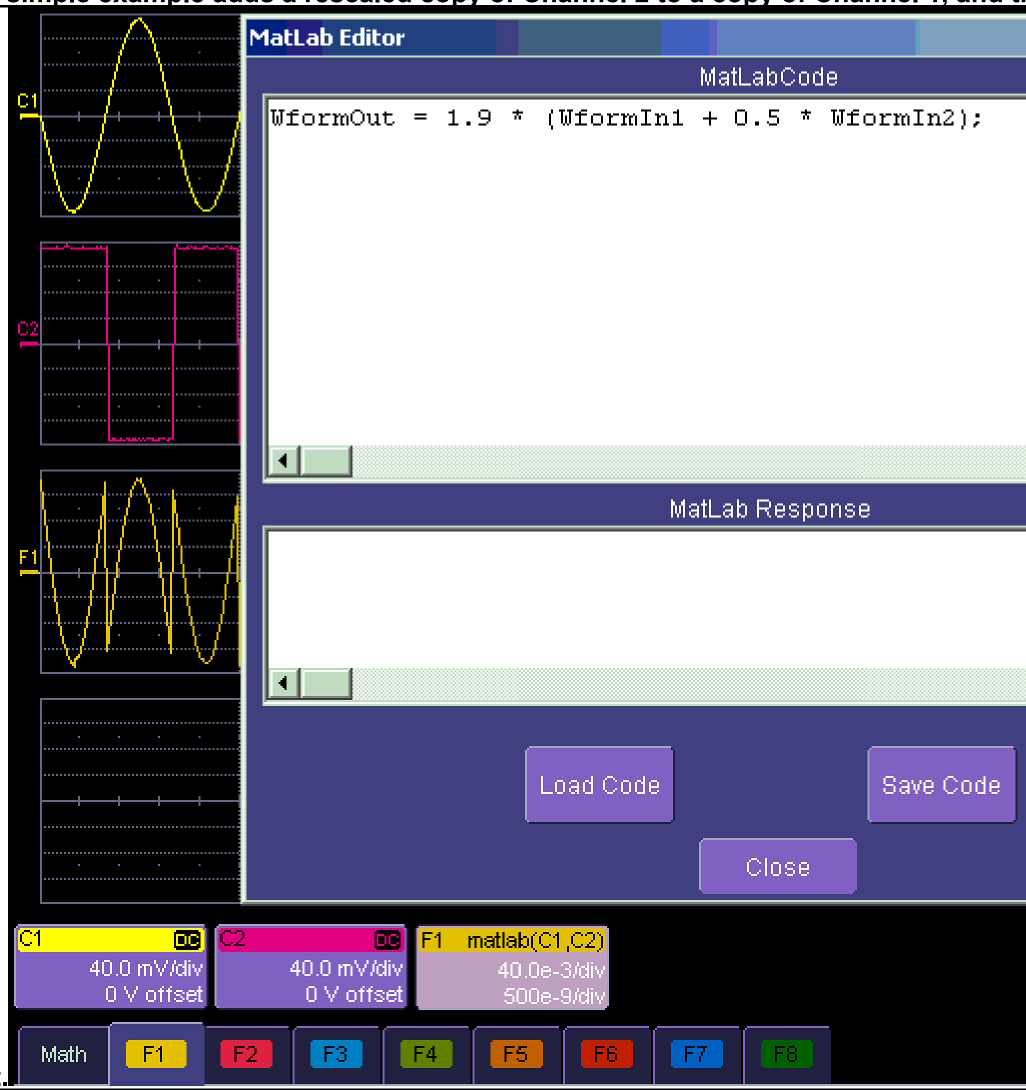
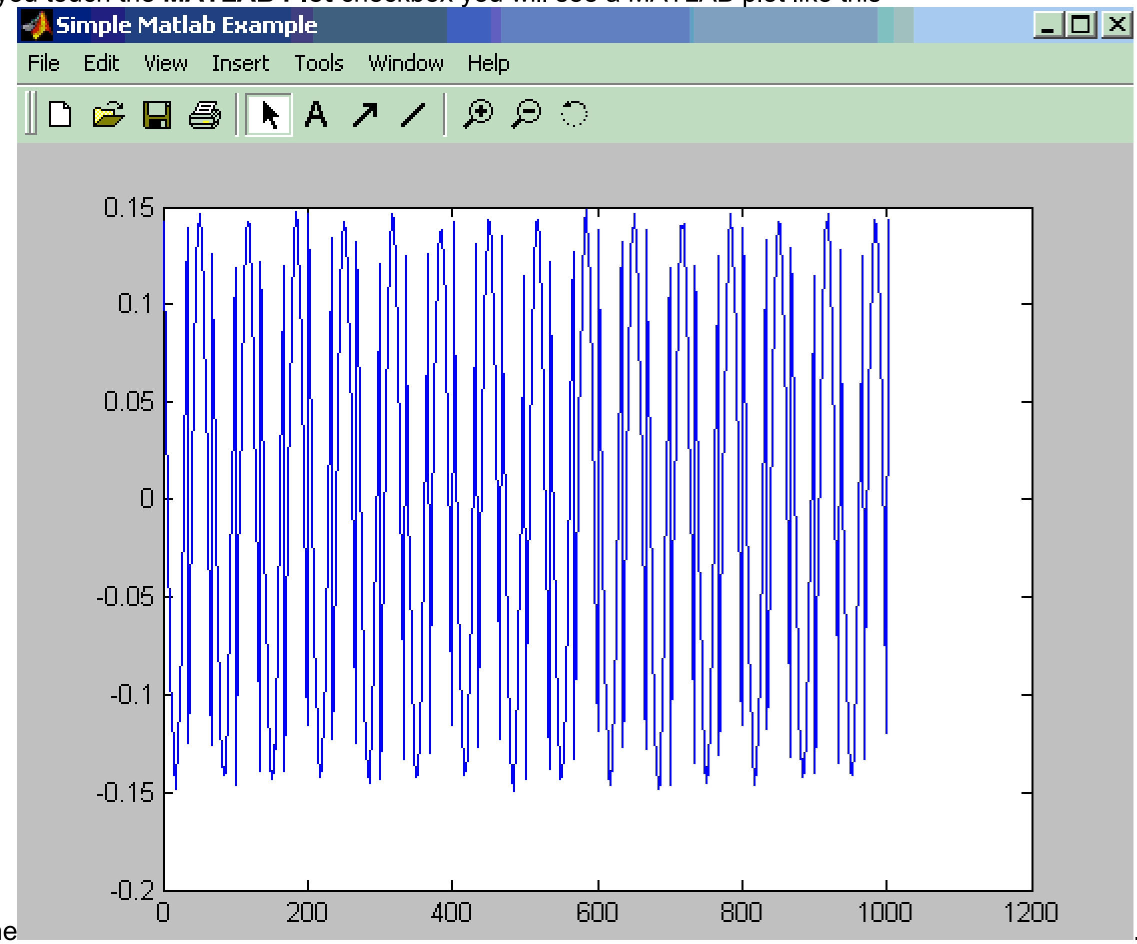


Figure 4-7. This very simple example adds a rescaled copy of Channel 2 to a copy of Channel 1, and then rescales the result.

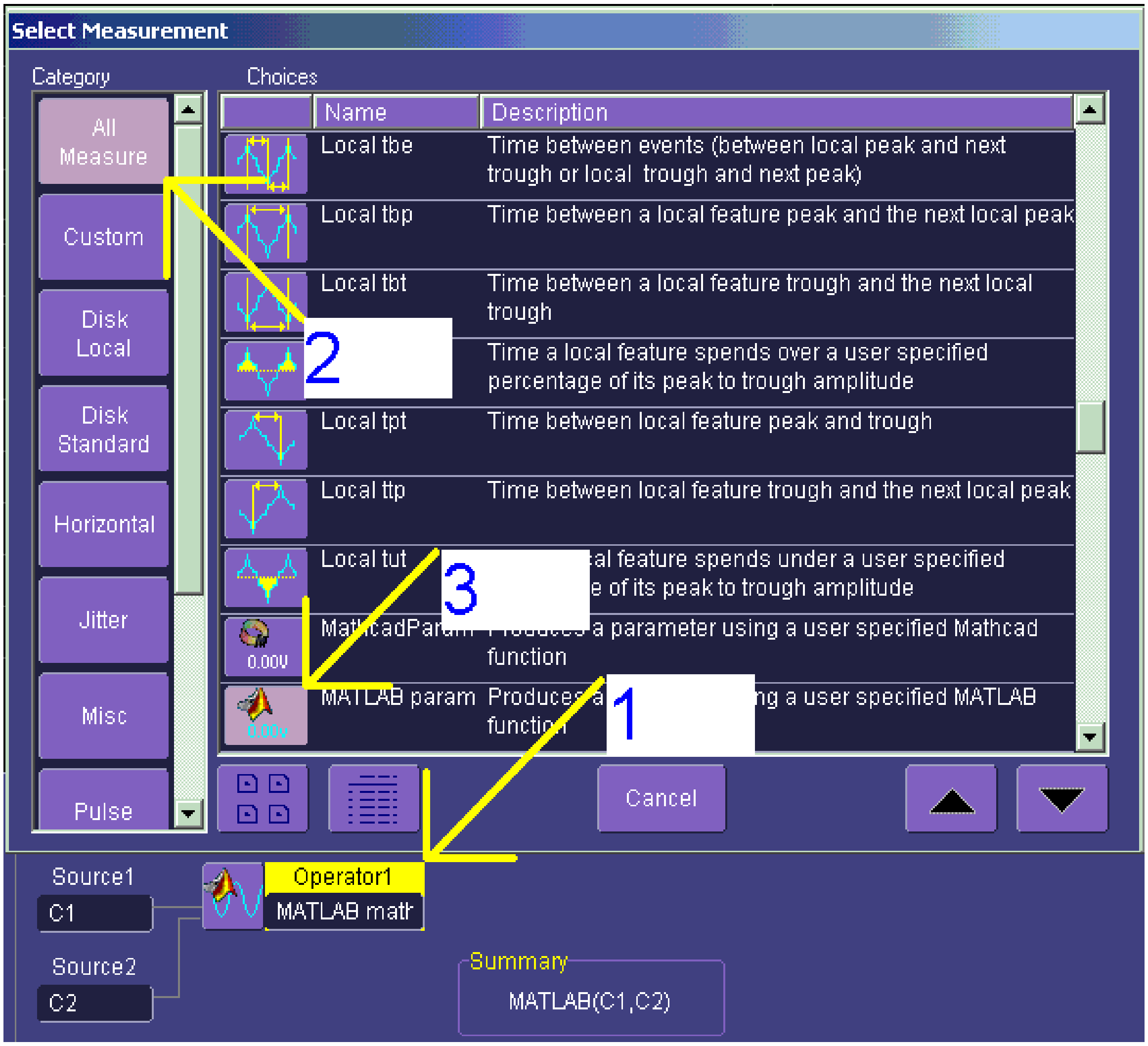
MATLAB Example Waveform Plot

If you touch the **MATLAB Plot** checkbox you will see a MATLAB plot like this

one 

.

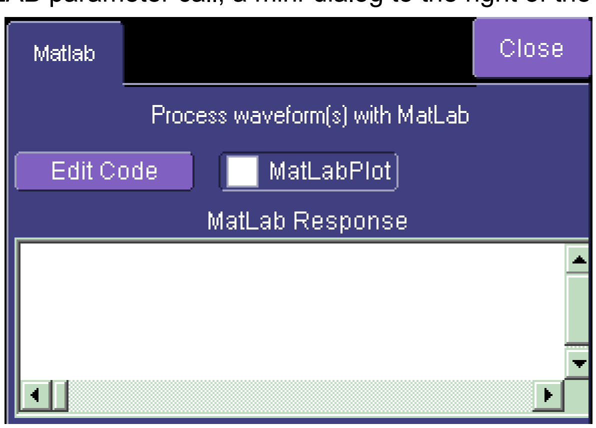
Selecting a MATLAB Parameter Call



Menu position for MATLAB parameter call in **Select Measurement** menu.

MATLAB Parameter Control Panel

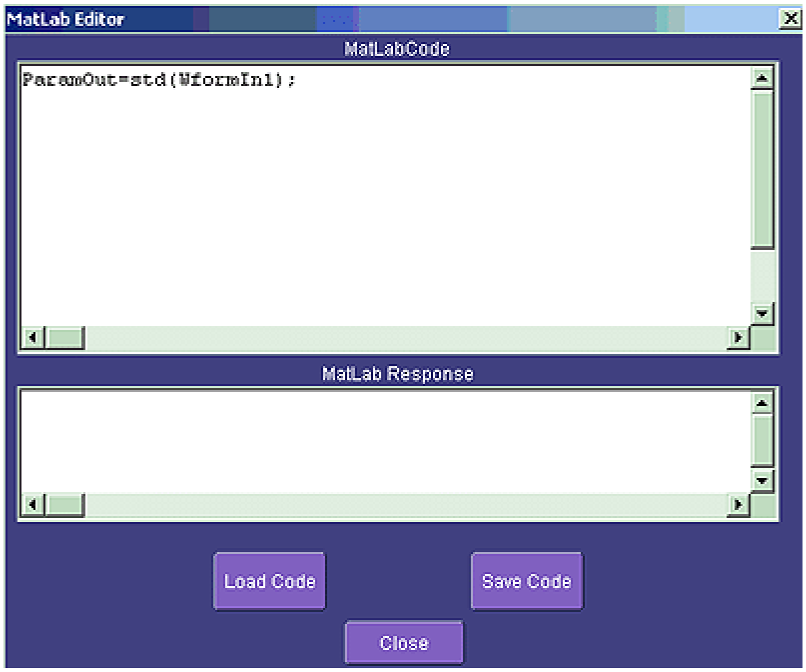
Once you have invoked a MATLAB parameter call, a mini-dialog to the right of the main dialog will appear:



You can touch the **MATLAB Plot** checkbox if you want to see a plot in MATLAB as well as getting a result in the instrument.

MATLAB Parameter Editor

By touching **Edit Code**, you can reach the MATLAB Editor:

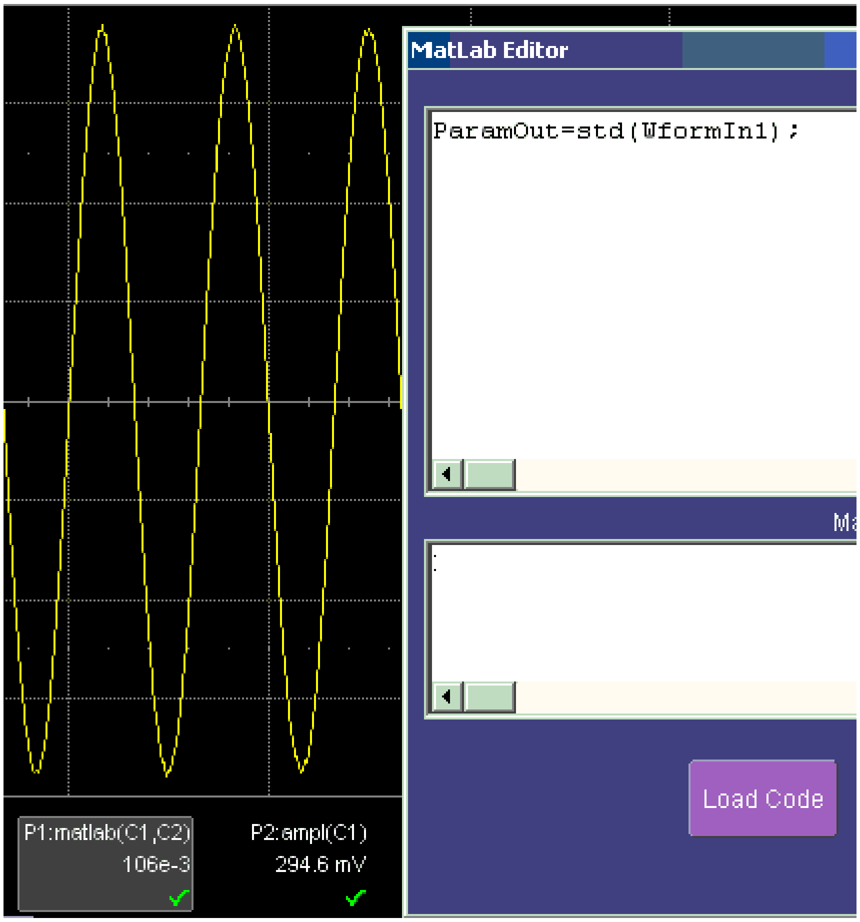


This simple example shows the MATLAB function Standard Deviation acting on input channel 1, and the result would be shown in the **MATLAB Response** pane for an amplitude of 0.15 volt.

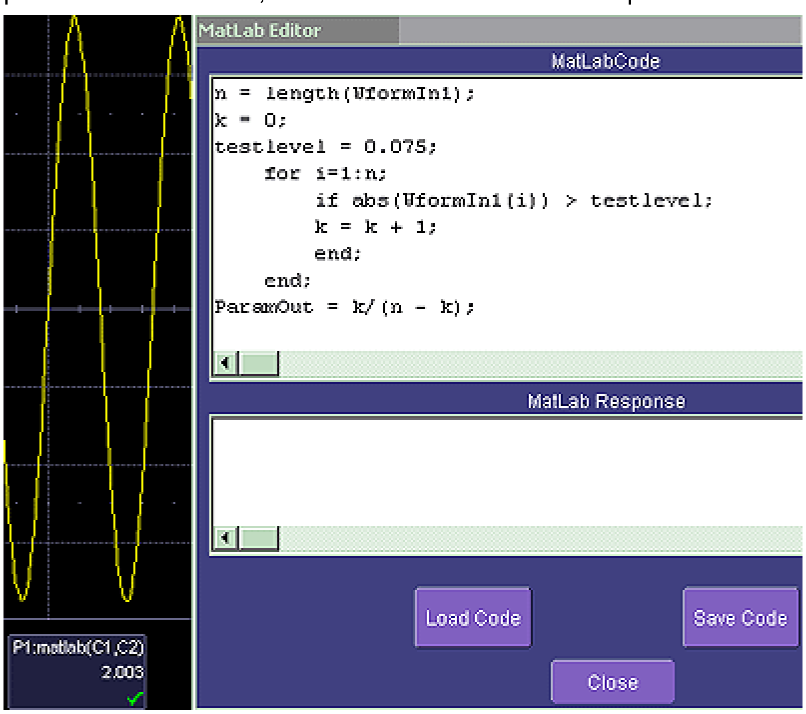
You can load an existing MATLAB program, using the **Load Code** button, and you can save the current program, using the **Save Code** button.

If you are familiar with MATLAB you might prefer to launch MATLAB and create a MATLAB function that performs your task. Your program in the instrument could then be a one-line call of your MATLAB function.

MATLAB Example Parameter Panel

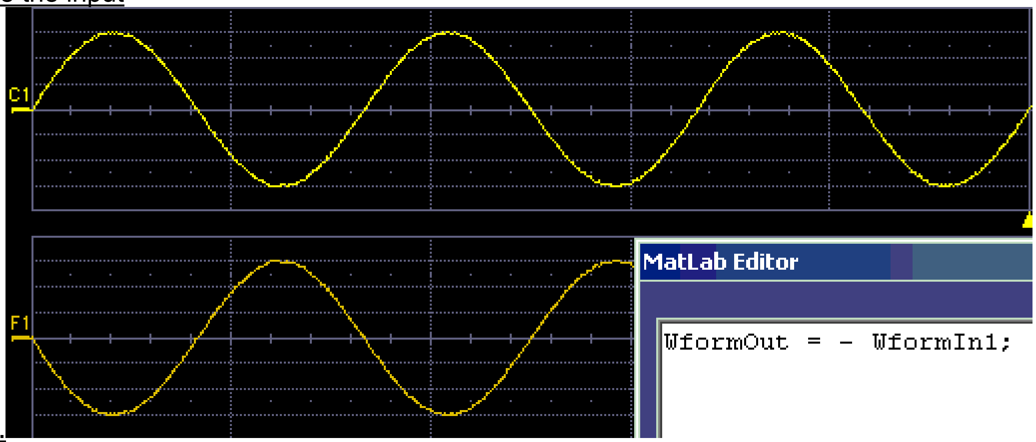


The next example calculates the ratio of the number of data points that are above a given level to the number of points below the level, in this case one half of the amplitude.

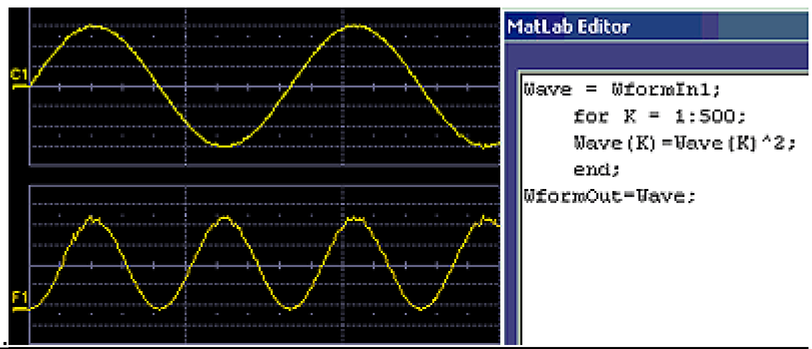


Further Examples of MATLAB Waveform Functions

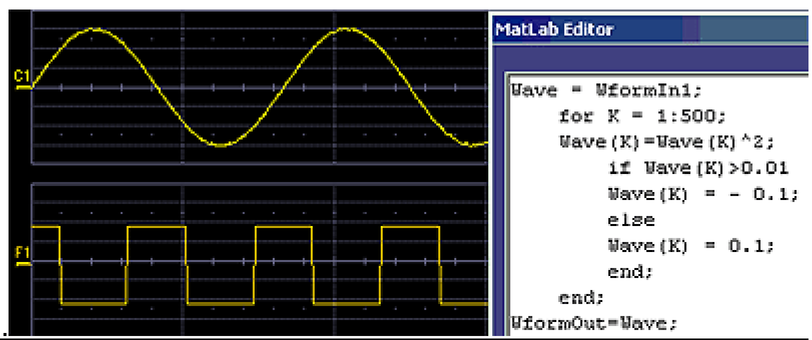
Negate the input signal.



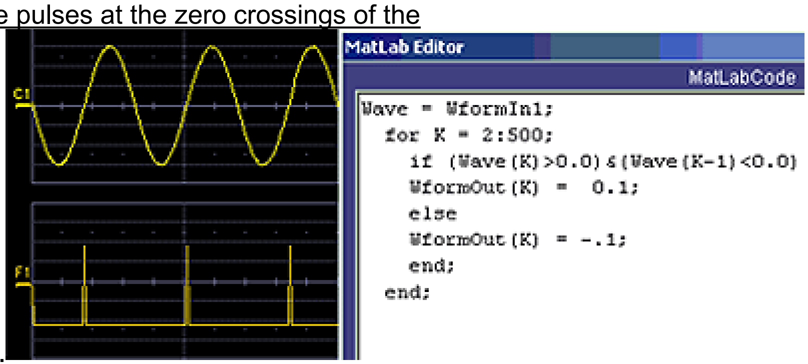
Square the input signal.



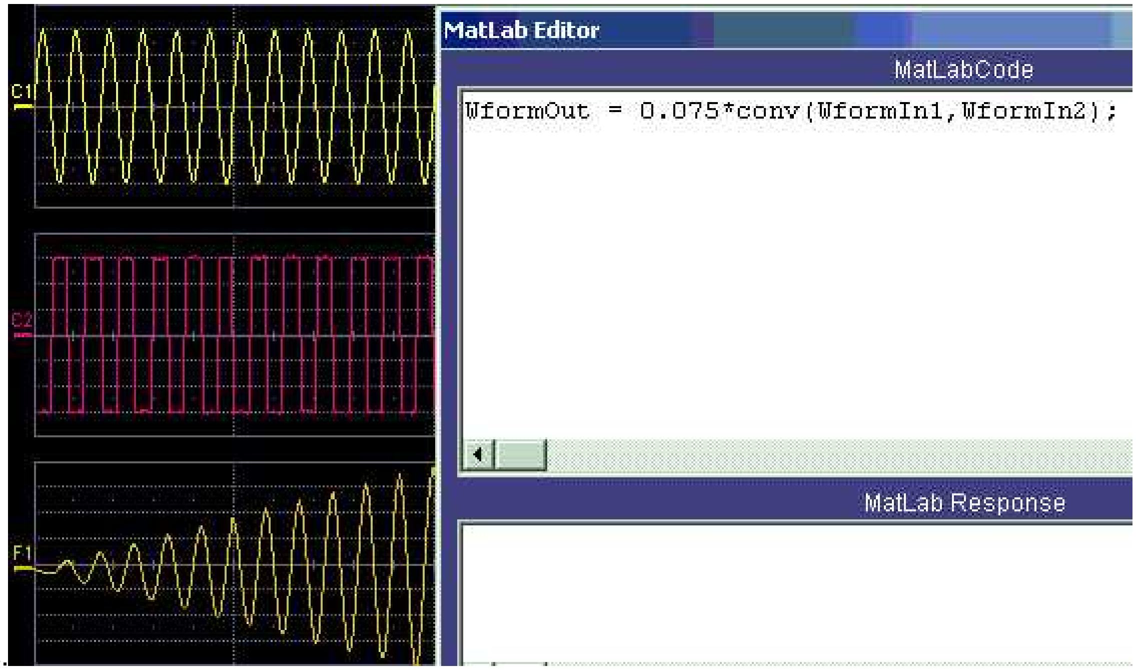
Create pulses from a sinusoid.



Create pulses at the zero crossings of the signal.



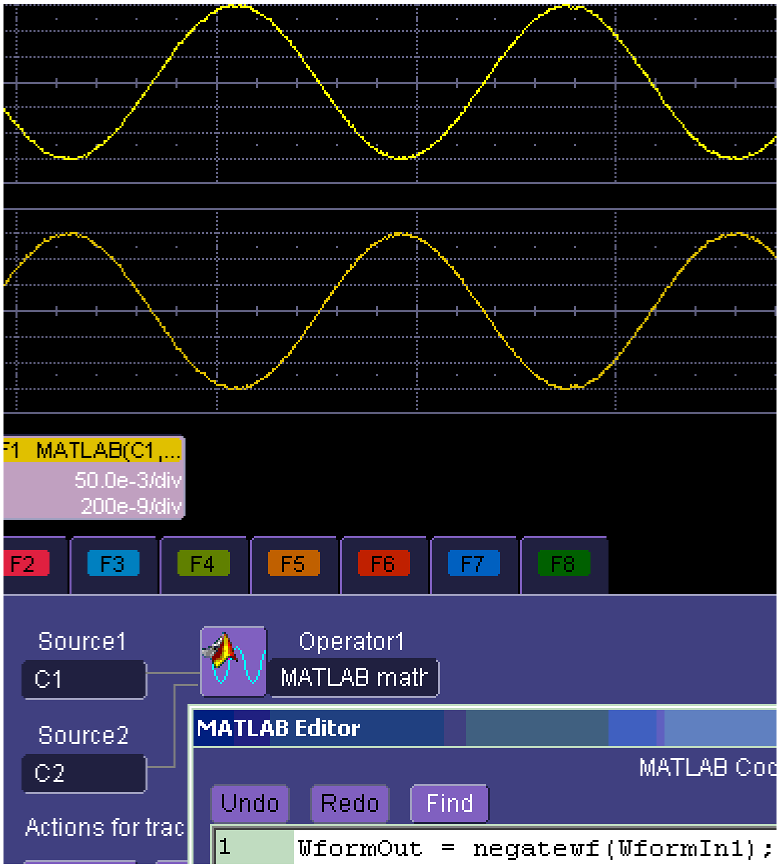
Convolve two signals.



Creating your Own MATLAB Function

The procedure is simple. Create a MATLAB function using any text editor, and save it as a MATLAB m-file by giving it a name of the form Filename.m. Call the function using the MATLAB math editor or the MATLAB parameter editor as appropriate. A simple example is shown below.

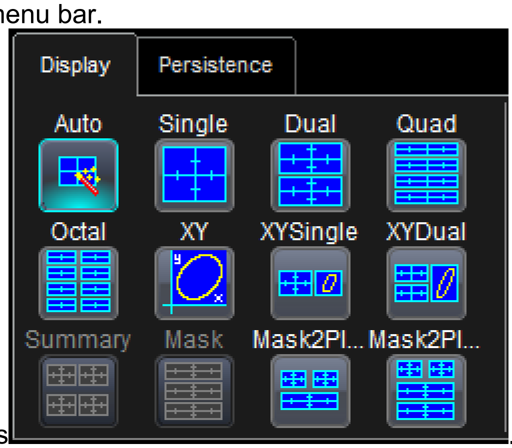
```
function out = negatewf(wf1)
      % NEGATEWF changes the sign of all the data.
out = -wf1;
```



Display Setup

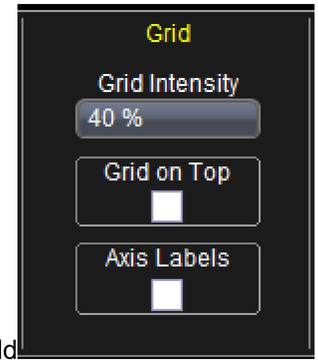
Note: Not all grid styles are available on all instruments.

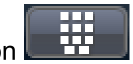
1. Touch **Display** → **Display Setup** on the menu bar.



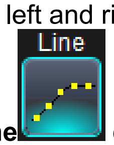
2. Touch one of the **Grid** combination buttons.

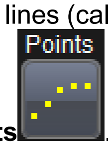
Note: **Autogrid** automatically adds or deletes grids as you select more or fewer waveforms to display.

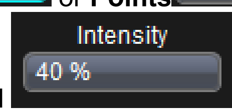


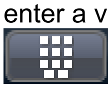
3. Touch inside the grid **Intensity** data entry field and enter a value from 0 to 100 using the slider bar at the bottom of the window. Click the keypad button  on the slider bar to enter a value using the pop-up numeric keypad.
4. Touch the **Grid on top** checkbox if you want to superimpose the grid over the waveform.

Note: Depending on the grid intensity, some of your waveform may be hidden from view when the grid is placed on top. To undo, simply uncheck **Grid on top**.

3. Touch the **Axis labels** checkbox to permanently display the values of the top and bottom grid lines (calculated from volts/div) and the extreme left and right grid lines (calculated from the timebase).
4. Choose a line style for your trace: solid **Line** 

or **Points** 

.
5. Touch inside the trace **Intensity** data entry field 

if you want to change the value and enter a value from 0 to 100 using the slider bar at the bottom of the window. Click the keypad button  on the slider bar to enter a value using the pop-up numeric keypad.

Note: Access the Monitor tab for external monitor display settings. Refer to the **External Monitor** topic for more details.

Sequence Mode Display

Note: Sequence mode is not available on all instruments.

To set up a Sequence Mode display, you must first have selected **Sequence** trigger mode in the **Timebase → Horizontal Setup** dialog. You must also have entered a **Num Segments** value. For more information on setting up Sequence Mode, see the **Sequence Sampling Mode – Working with Segments** topic.

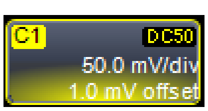
1. Touch **Display → Display Setup** on the menu bar.
2. Touch inside the **Display Mode** field and select a display mode from the pop-up menu.

Moving Traces from Grid to Grid

You can move traces from grid to grid at the touch of a button.

Moving a Channel or Math Trace

1. Touch the descriptor label for the waveform that you want to move.



2. Touch the **Next Grid** button 

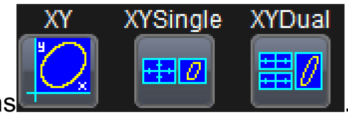
.

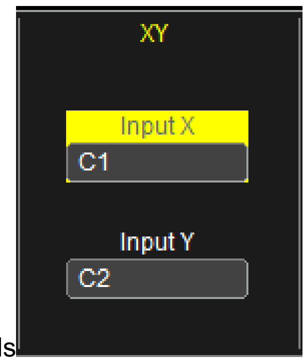
Note: If you have more than one waveform displayed on only one grid, a second grid will open automatically when you select **Next Grid**.

XY Display

Use XY displays to measure the phase shift between otherwise identical signals. You can display either voltage on both axes or frequency on both axes. The traces must have the same X-axis. The shape of the resulting pattern reveals information about phase difference and frequency ratio.

Setting Up XY Displays

1. Touch **Display** → **Display Setup** on the menu bar.
2. Choose an XY display by touching one of the XY display mode buttons 

. You can show the two waveforms on the XY grid only, or you can show the input waveforms on a single or dual grid.
3. Touch inside the **Input X** and **Input Y** data entry fields 

and select your input sources from the pop-up windows.

Note: The inputs can be any combination of channels, math functions, and memory locations.

Custom Grids

Some instruments and software packages offer the option to display side-by-side grids, instead of vertically stacked grids. These side-by-side grids are labeled **Custom** in the **Display** dialog. Included in the choices are a 3-grid display, single, dual, and quad displays.

Custom grids offer views of your waveform that are contracted horizontally and less vertically.

Zooming Waveforms

You can magnify a selected region of a waveform using the Zoom function. You can display up to four Zoom (Z1 - Z4) and eight Math Zoom traces (F1 - F8).You can zoom:

- a single channel (refer to **Zooming a Single Channel**)
- a math or memory trace (refer to **Zooming Memory or Math Function Traces**)
- multiple waveforms at once (refer to **Quickly Zooming Multiple Waveforms**)
- segments in a sequence (refer to **Sampling Modes** → **Sequence Sampling Mode – Working with Segments**

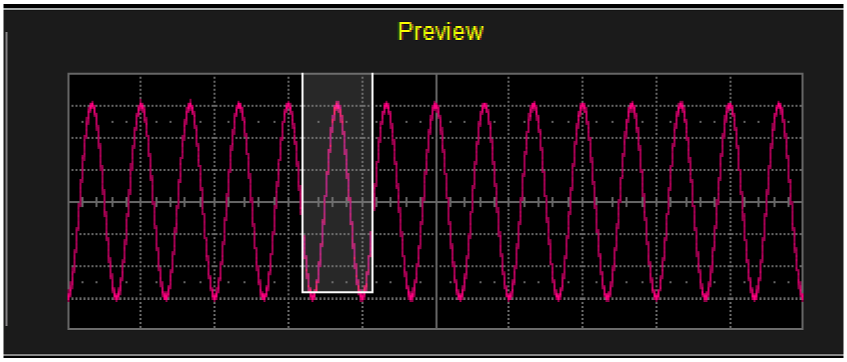
You can also use the Multi-Zoom Math function to create time-locked zoom traces for selected waveforms. For more information, refer to **Math** → **Multi-Zoom**).

At any time, you can zoom a portion of a channel waveform or Memory/Math function trace by touching and dragging a rectangle around any part of the input waveform, see the **Touch-and-Drag Zooming** topic. The zoom trace will size itself to fit the full width of the grid. The degree of magnification, therefore, will depend on the size of the rectangle that you draw.

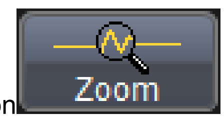
You can also use the front panel **QuickZoom** button  to create multiple zooms, one for each displayed input channel. For more information, see the **Quickly Zooming Multiple Waveforms** topic.

Previewing Zoomed Waveforms

When you zoom a waveform, a preview of the zoomed area will appear in the **Preview** section of the **Zx** dialog.



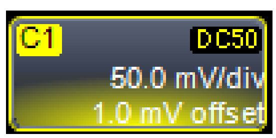
Zooming a Single Channel

The **Zoom** button 

appears as a standard button at the bottom of the channel **Cx Vertical Adjust** setup dialog for you to create a zoom trace of your input waveform.

1. On the menu bar, touch **Vertical** → **Channelx Setup**.

   OR

   Touch the channel trace label 

for a displayed channel.
2. Touch the **Zoom** button 

at the bottom of the **Cx Vertical Adjust** dialog. A zoom trace (one of Z1 to Z4) will be created of the selected channel.
3. To vary the degree of zoom, touch the newly created **Zx** trace label. The setup dialog for the zoom (Z1 to Z4) opens. It shows the current horizontal and vertical zoom factors.
4. If you want to increase or decrease your horizontal or vertical zoom in small increments, touch the **Var.** checkbox to enable variable zooming. Now with each touch of the zoom control buttons 

, the degree of magnification will change by a small increment.

   OR

   If you want to zoom in or out in large standard increments with each touch of the zoom control buttons, leave the **Var.** checkbox unchecked.

   OR

If you want to set exact horizontal or vertical zoom factors, touch inside the Horizontal **Scale/div** data entry field and enter a time-per-div value, using the pop-up numeric keypad. Then touch inside the Vertical **Scale/div** field and enter a voltage value.

PLEASE NOTE THE FOLLOWING:

- To reset the zoom to x1 magnification, touch **Reset Zoom** in the dialog.
- You can also use the Zoom and Math front panel knobs to adjust the horizontal and vertical zoom.
  - **Horizontal Position** - Press to reset the horizontal zoom position to zero. Turn to change the horizontal position of the selected math or zoom trace.
  - **Horizontal Ratio** - Press to toggle between fixed and variable horizontal zoom ratio adjustment. Turn to change the horizontal zoom ratio of the selected math trace.
  - **Quick Zoom** - Press to automatically display magnified views of up to four signal inputs on multiple grids. With four input signals, the signals are displayed along with four zoom traces, each on its own grid. Pressing this button also turns off all other traces.
  - **Vertical Position** - Press to reset the vertical zoom position to zero. Turn to change the vertical position of the selected math or zoom trace.
  - **Vertical Ratio** - Press to toggle between fixed and variable vertical zoom ratio adjustment. Turn to change the vertical zoom ratio of the selected math trace.

Touch-and-Drag Zooming

1. Touch and drag a rectangle around any part of an input channel waveform, math trace, or memory trace. The **Zoom** (Z1 to Z4) dialog opens.

Note: If you have enclosed a combination of channel and math or memory traces in the rectangle, a pop-up **Rectangle Zoom Wizard** will appear. Touch a **Zoom** button (Z1 to Z4)

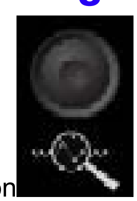
2. To turn off the zoom traces, touch the **Undo** button 

at the top-right corner of the screen.

   OR

   Uncheck the **Trace On** checkbox in the dialog for each zoom trace.

Quickly Zooming Multiple Waveforms

Press the **QUICKZOOM** button  on the front panel.

Turning Off Zoom

1. Touch the math function trace label (or **Zx** trace label) for the zoom you want to turn off.
2. Touch the **Trace On** checkbox to remove the check mark and disable the zoom trace.

Multi-Zoom

If available on your instrument, the Multi-zoom feature creates time-locked zoom traces for only the waveforms that you choose to include. The zooms are of the same X-axis section of each waveform. Thus, as you scroll through a waveform, all included zooms scroll in unison.

Setting Up Multi-zoom

1. On the menu bar, touch **Math** → **Math Setup**.
2. Verify that the math function selected for each **Fx** position you want to include is **zoom**. If you need to change the math function for any Fx position, simply touch the **Fx** button and select **Zoom** from the **Select Math Operator menu**.
3. Touch the **On** checkbox to display each zoom you want to include in the multi-zoom.
4. Touch the Multi-Zoom **Setup** button. The **Multi-Zoom** dialog opens.
5. Touch the Multi-zoom **On** checkbox to enable Multi-zoom. Then touch the **Include** checkbox for each zoom trace you want to include in the time-locked multi-zoom.

6. Use the **Auto-Scroll** buttons at the right of the Multi-Zoom dialog to control the zoomed section of your waveforms:

Turning Off Multi-Zoom

1. On the menu bar, touch **Math** → **Math Setup**.
2. Touch the Multi-Zoom **On** checkbox to turn off Multi-zoom.

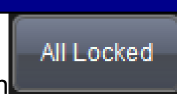
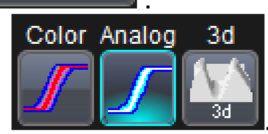
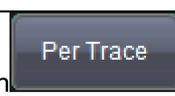
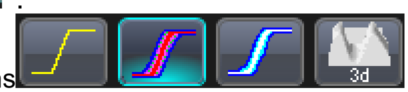
Persistence Setup

The Persistence feature helps you display your waveform and reveal its idiosyncrasies or anomalies for a repetitive signal. Use Persistence to accumulate on-screen points from many acquisitions to see your signal change over time. The instrument persistence modes show the most frequent signal path "three-dimensionally" in intensities of the same color, or graded in a spectrum of colors.

You can show persistence for up to eight inputs for any channel, math function, or memory location (M1 to M4).

Setting Up Persistence

1. Touch **Display** → **Persistence Setup** on the menu bar.
2. Touch the **Persistence On** checkbox.

If you want to set up persistence for:	Follow these steps:
all input channels at once	1. Touch the **All Locked** button  . 2. Touch one of the mode buttons  . 3. Touch the **Show last trace** checkbox if you want the last trace displayed. 4. Touch inside the **Saturation** data entry field and enter a whole number integer, using the slider bar at the bottom of the window. Click the keypad button  on the slider bar to enter a value using the pop-up numeric keypad. For more information on saturation, see the **Saturation Level** topic. 5. Touch inside the **Persistence time** data entry field and make a selection from the pop-up menu.
each input channel individually	1. Touch the **Per Trace** button  . 2. Touch one of the persistence mode buttons  next to an input channel. To turn off persistence for an individual channel, touch the left-most persistence mode button next to the channel. ***OR*** For each input channel, touch its tab to set up persistence for that channel.

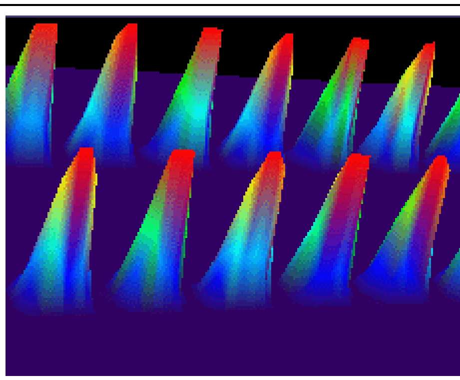
If you want to set up persistence for:	Follow these steps:
	3. Touch one of the mode buttons  . 4. Touch the **Show last trace** checkbox if you want the last trace displayed. 5. Touch inside the **Saturation** data entry field and enter a whole number integer, using the slider bar at the bottom of the window. Click the keypad button  on the slider bar to enter a value using the pop-up numeric keypad. For more information on saturation, see the **Saturation Level** topic. 6. Touch inside the **Persistence time** data entry field and make a selection from the pop-up menu. 7. If you want to return all input channel setups to their default settings, touch the **Reset All** button  .

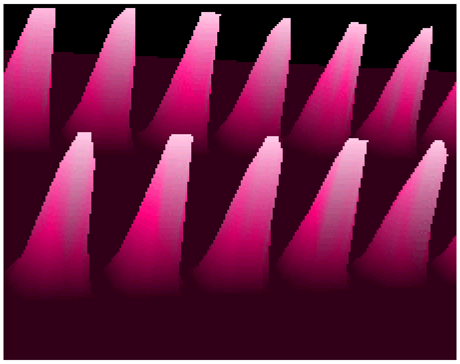
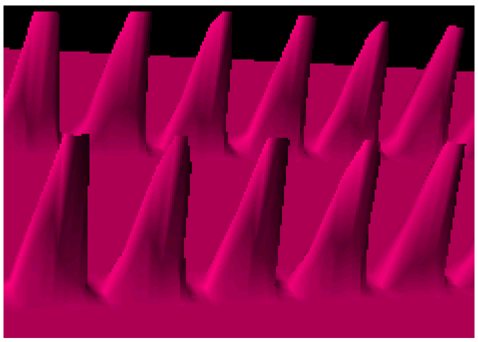
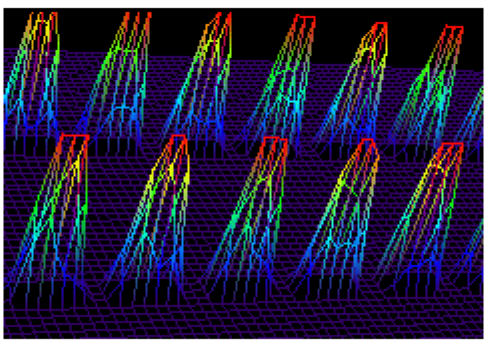
3-Dimensional Persistence - Not Available in All Scopes

By selecting **3d** 

, you can create a topographical view of your waveform from a selection of shadings, textures, and hues. The advantage of the topographical view is that areas of highest and lowest intensity are shown as peaks and valleys, in addition to color or brightness. The shape of the peaks (pointed or flat) can reveal further information about the frequency of occurrences in your waveform.

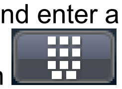
The instrument also gives you the ability to turn the X and Y axes of the waveform through 180° of rotation from -90° to +90°.

	Here is an example of a 3-dimensional view of a square wave using the **solid** view of color-graded persistence. Saturation is set at 50%, with red areas indicating highest intensity. The X-axis has been rotated 60%; the Y-axis has been rotated 15%.

	Here is a **monochrome** (analog) view of the same waveform. The lightest areas indicate highest intensity, corresponding to the red areas in the solid view.
	Here is a **shaded** (projected light) view of the same waveform. This view emphasizes the shape of the pulses.
	Here is a **wire frame** view of the same waveform in which lines of equal intensity are used to construct the persistence map.

Setting up 3-Dimensional Persistence

1. Touch the 3d button 

.
2. Touch inside the **Saturation** data entry field and enter a whole number integer, using the slider bar at the bottom of the window. Click the keypad button  on the slider bar to enter a value using the pop-up numeric keypad.
3. Touch inside the **Persistence time** data entry field and make a selection from the pop-up menu.
4. Under "3D settings," touch inside the **Quality** field and select an image quality from the pop-up menu: wire frame, solid, or shaded.
5. Touch the **MonoChrome** checkbox if you prefer a single-color representation.
6. For each axis, touch inside the data entry field and enter a value from -90° to +90°.

Show Last Trace

For most applications, you may not want to show the last trace because it will be superimposed on top of your persistence display. In those cases turn off **Show Last Trace** by touching the checkbox. However, if you are doing mask testing and want to see where the last trace is falling, turn **Show Last Trace** on.

Persistence Time

You can control the duration of persistence by setting a time limit, in seconds, after which persistence data will be erased: 0.5 s, 1 s, 2 s, 5 s, 10 s, 20 s, or infinity.

Locking Traces (Not Available in All Oscilloscopes)

The instrument gives you the choice of constraining all input channels to the same mode, saturation level, persistence time, and last trace display, or setting these for each input channel individually.

- Choose 

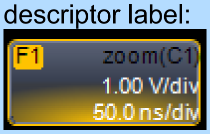
to constrain input channels.
- Choose 

to set up input channels individually.

Creating and Viewing a Histogram

Histogramming is not available on all instrument models.

Note: The number of sweeps comprising the histogram will be displayed in the bottom line of the trace descriptor label:



Note: The range of a histogram is limited to the portion of the trace that is visible on screen. That is, if you zoom in on a trace, the histogram will not contain data for that part of the original trace no longer visible.

Setting Up a Single Parameter Histogram

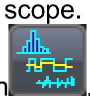
From Measure Dialog

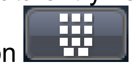
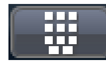
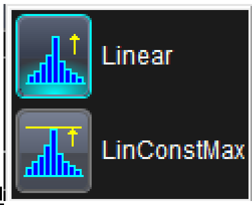
1. Touch **Measure** → **Measure Setup** on the menu bar.
2. Touch the **My Measure** button.
3. Touch one of tabs **P1** through **Px**.
4. Touch inside the **Source1** field and select an input waveform from the **Select Source** window.
5. Touch inside the **Measure** field and select a parameter from the **Select Measurement** window.
6. Touch the **Histogram** button at the bottom of the dialog.
7. Touch a math trace in which to place the resulting histogram, then close the pop-up menu.
8. Touch the trace descriptor label for the math trace you just created.
9. In the dialog to the right, touch the **Histogram** tab.
10. On the **Buffer** side of the **Histogram** tab, touch inside the **#Values** data entry field and enter a value.
11. On the **Scaling** side of the **Histogram** tab, touch inside the **#Bins** data entry field and enter a value.
12. To center the histogram, touch the **Find Center and Width** button.

    OR

    Touch inside the **Center**, then the **Width**, data entry fields and enter a value using the pop-up numeric keypad.

From Math Dialog

1. Touch **Math** → **Math Setup** on the menu bar.
2. Touch one of function tabs **F1** through **Fx** [The number of math traces available depends on the software options loaded on your scope. See specifications.] .
3. Touch the Graph button .
4. Touch inside the **Source1** field and select a source from the pop-up menu.

5. Touch inside the **Measurement** field and select a parameter from the pop-up menu.
6. Touch inside the **Graph with** field and select **Histogram** from the pop-up menu.
7. In the dialog to the right, touch the **Histogram** tab.
8. On the **Buffer** side of the **Histogram** tab, touch inside the **#Values** data entry field and enter a value or use the slider bar at the bottom of the window. Click the keypad button  on the slider bar to enter a value using the pop-up keypad..
9. On the **Scaling** side of the **Histogram** tab, touch inside the **#Bins** data entry field and enter a value or use the slider bar at the bottom of the window. Click the keypad button  on the slider bar to enter a value using the pop-up keypad..
10. Touch the **Find Center and Width** button to center the histogram. Or touch inside the **Center**, then the **Width**, data entry fields and enter a value or use the slider bar at the bottom of the window. Click the keypad button  on the slider bar to enter a value using the pop-up keypad.
11. Touch inside the Vertical Scale field and select **Linear** or **Linear Constant Max** from the pop-up menu 

.

Viewing Thumbnail Histograms (Histicons)

Histicons are miniature histograms of parameter measurements that appear below the grid. These thumbnail histograms let you see at a glance the statistical distribution of each parameter.

1. On the menu bar, touch **Measure**, then one of the Measure Mode buttons: **Std Vertical**, **Std Horizontal**, or **My Measure**.
2. Touch the **Histicons** checkbox to display thumbnail histograms below the selected parameters.

> **Note:** For measurements set up in My Measure, you can quickly display an enlarged histogram of a thumbnail histogram by touching the Histicon you want to enlarge. The enlarged histogram will appear superimposed on the trace it describes. This does not apply to "Std Vertical" or "Std Horizontal" measurements.

Persistence Histogram (JTA2 option)

You can create a histogram of a persistence display also by cutting a horizontal or vertical slice through the waveform. You also decide the width of the slice and its horizontal or vertical placement on the waveform.

This math operation is different than the "Histogram" math operation and is not affected by **Center** and **Width** settings made there.

Setting Up Persistence Histograms

1. Touch **Math** → **Math Setup** on the menu bar.
2. Touch one of function tabs **F1** through **Fx** [The number of math traces available depends on the software options loaded on your scope. See specifications.] .
3. Touch inside the **Source1** field and select a source from the pop-up menu.
4. Touch inside the **Operator1** field and select Phistogram  from the **Select Math Operator** window.
5. Touch the **Phistogram** tab, then touch inside the **Slice Direction** field and select **Horizontal** or **Vertical** slice from the pop-up menu.
6. Touch inside the **Slice Center** field and enter a value, using the pop-up keypad.
7. Touch inside the **Slice Width** field and enter a value, using the pop-up keypad.

Persistence Trace Range

Using this math operation, you can enter the percent of the persistence trace population to use in creating a new waveform.

Persistence Sigma

Using this math operation, you can enter a scale, measured in standard deviations, by which to create a new waveform.

Histogram Theory of Operation

An understanding of statistical variations in parameter values is needed for many waveform parameter measurements. Knowledge of the average, minimum, maximum, and standard deviation of the parameter may often be enough, but in many cases you may need a more detailed understanding of the distribution of a parameter's values.

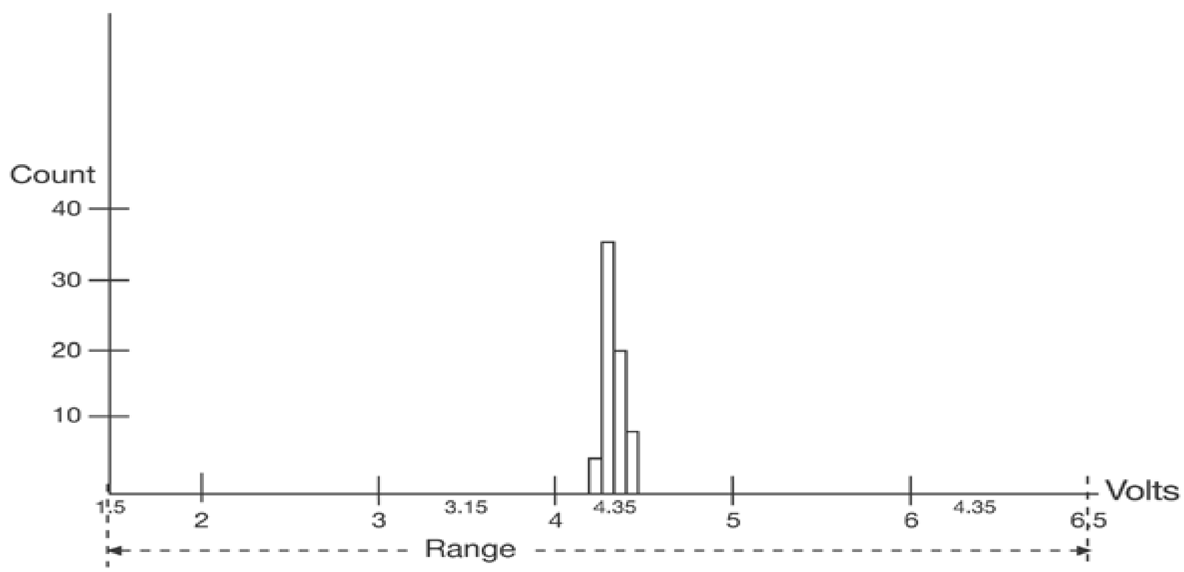
Histograms allow you to see how a parameter's values are distributed over many measurements. They do this by dividing a range of parameter values into sub-ranges called bins. A count of the number of parameter values (events) that fall within ranges of the bin itself is maintained for each bin.

While such a value range can be infinite, for practical purposes it need only be defined as large enough to include any realistically possible parameter value. For example, in measuring TTL high-voltage values a range of ±50 V is unnecessarily large, whereas one of 4 V ±2.5 V is more reasonable. It is the 5 V range that is then subdivided into bins. And if the number of bins used were 50, each would have a range of 5 V/50 bins or 0.1 V/bin. Events falling into the first bin would then be between 1.5 V and 1.6 V. While the next bin would capture all events between 1.6 V and 1.7 V, and so on.

After a process of several thousand events, the bar graph of the count for each bin (its histogram) provides a good understanding of the distribution of values. Histograms generally use the 'x' axis to show a bin's sub-range value, and the 'Y' axis for the count of parameter values within each bin. The leftmost bin with a non-zero count shows the lowest parameter value measurements. The vertically highest bin shows the greatest number of events falling within its sub-range.

Note: The range of the histogram is limited to the portion of the trace that is visible on screen. That is, if you zoom in on a trace, the histogram will not contain data for that part of the original trace no longer visible.

The number of events in a bin, peak or a histogram is referred to as its population. The following figure shows a histogram's highest population bin as the one with a sub-range of 4.3 to 4.4 V (which is to be expected of a TTL signal).



The lowest-value bin with events is that with a sub-range of 3.0 to 3.1 V. As TTL high voltages need to be greater than 2.5 V, the lowest bin is within the allowable tolerance. However, because of its proximity to this tolerance and the degree of the bin's separation from all other values, additional investigation may be required.

DSO Process

The instrument generates histograms of the parameter values of input waveforms. But first, you must define the following:

- The parameter to be histogrammed
- The trace on which the histogram is to be displayed
- The maximum number of parameter measurement values to be used in creating the histogram
- The measurement range of the histogram
- The number of bins to be used

Some of these are pre-defined but can be changed. Once they are defined, the oscilloscope is ready to make the histogram. The sequence for acquiring histogram data is as follows:

1. Trigger
2. Waveform acquisition
3. Parameter calculations
4. Histogram update
5. Trigger re-arm

If you set the timebase for non-segmented mode, a single acquisition occurs prior to parameter calculations. However, in Sequence mode an acquisition for each segment occurs prior to parameter calculations. If the source of histogram data is a memory, saving new data to memory effectively acts as a trigger and acquisition. Because updating the screen can take much processing time, it occurs only once a second, minimizing trigger dead time. Under remote control the display can be turned off to maximize measurement speed.

Parameter Buffer

The oscilloscope maintains a circular parameter buffer of the last 5,000 measurements made, including values that fall outside the set histogram range. If the maximum number of events to be used for the histogram is a number `N' less than 5,000, the histogram will be continuously updated with the last `N' events as new acquisitions occur. If the maximum number is greater than 5,000, the histogram will be updated until the number of events is equal to `N.' Then, if the number of bins or the histogram range is modified, the scope will use the parameter buffer values to redraw the histogram with either the last `N' or 5,000 values acquired - whichever is the lesser. The parameter buffer thereby allows histograms to be redisplayed, using an acquired set of values and settings that produce a distribution shape with the most useful information.

In many cases the optimal range is not readily apparent. So the scope has a powerful range finding function. If required it will examine the values in the parameter buffer to calculate an optimal range and redisplay the histogram using it. The instrument will also give a running count of the number of parameter values that fall within, below, or above the range. If any values fall below or above the range, the range finder can then recalculate to include these parameter values, as long as they are still within the buffer.

Capture of Parameter Events

The number of events captured per waveform acquisition or display sweep depends on the parameter type. Acquisitions are initiated by the occurrence of a trigger event. Sweeps are equivalent to the waveform captured and displayed on an input channel (1, 2, or 3 or 4). For non-segmented waveforms an acquisition is identical to a sweep. Whereas for segmented waveforms an acquisition occurs for each segment and a sweep is equivalent to acquisitions for all segments. Only the section of a waveform between the parameter cursors is used in the calculation of parameter values and corresponding histogram events.

The following table provides a summary of the number of histogram events captured per acquisition or sweep for each parameter, and for a waveform section between the parameter cursors.

Parameters	Number of Events Captured
duty, freq, period, width, time@lev, f@level, f80-20%, fall, r@level, r20-80%, rise	All events in the acquisition
ampl, area, base, cmean, cmedian, crms, csdev, cycles, delay, maximum, mean, minimum, nbph, nbpw, over+, over-, pkpk, npts, rms, sdev, Δdly	One event per acquisition

Histogram Parameters (XMAP and JTA2 Options)

Once a histogram is defined and generated, measurements can be performed on the histogram itself. Typical of these are the histogram's

- average value, standard deviation
- most common value (parameter value of highest count bin)
- leftmost bin position (representing the lowest measured waveform parameter value)
- rightmost bin (representing the highest measured waveform parameter value)

Histogram parameters are provided to enable these measurements. Available through selecting "Statistics" from the "Category" menu, they are calculated for the selected section between the parameter cursors:

fwhm – full width (of largest peak) at half the maximum bin

fwxx – full width (of largest peak) at xx% the maximum bin

hist ampl – histogram amplitude between two largest peaks

hist base – histogram base or leftmost of two largest peaks

hist max – value of the highest (right-most) populated bin in a histogram

hist mean – average or mean value of data in the histogram

hist median – value of the x-axis of a histogram that divides the population into two equal halves

hist min – value of the lowest (left-most) populated bin in a histogram

hist rms – rms value of data in histogram

hist sdev – standard deviation of values in a histogram

hist top – histogram top or rightmost of two largest peaks

max populate – population of most populated bin in histogram

mode – data value of most populated bin in histogram

percentile – data value in histogram for which specified `x'% of population is smaller

peaks – number of peaks in histogram

pop @ x – population of bin for specified horizontal coordinate

range – difference between highest and lowest data values

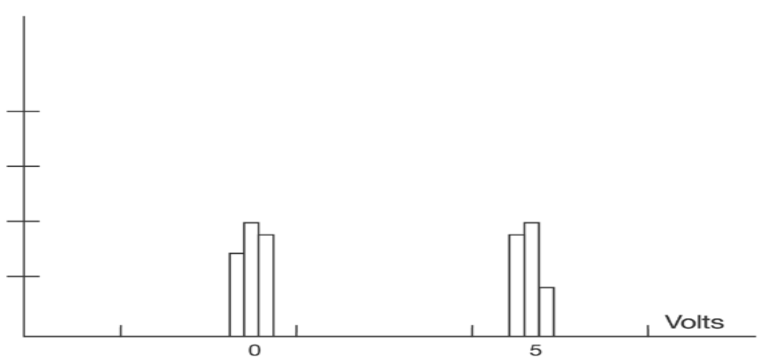
total pop – total population in histogram

x at peak – x-axis position of specified largest peak

Histogram Peaks

Because the shape of histogram distributions is particularly interesting, additional parameter measurements are available for analyzing these distributions. They are generally centered around one of several peak value bins, known, with its associated bins, as a histogram peak.

Example: In the following figure, a histogram of the voltage value of a five-volt amplitude square wave is centered around two peak value bins: 0 V and 5 V. The adjacent bins signify variation due to noise. The graph of the centered bins shows both as peaks.



Determining such peaks is very useful because they indicate dominant values of a signal.

However, signal noise and the use of a high number of bins relative to the number of parameter values acquired, can give a jagged and spiky histogram, making meaningful peaks hard to distinguish. The scope analyzes histogram data to identify peaks from background noise and histogram definition artifacts such as small gaps, which are due to very narrow bins.

Binning and Measurement Accuracy

Histogram bins represent a sub-range of waveform parameter values, or events. The events represented by a bin may have a value anywhere within its sub-range. However, parameter measurements of the histogram itself, such as average, assume that all events in a bin have a single value. The scope uses the center value of each bin's sub-range in all its calculations. The greater the number of bins used to subdivide a histogram's range, the less the potential deviation between actual event values and those values assumed in histogram parameter calculations.

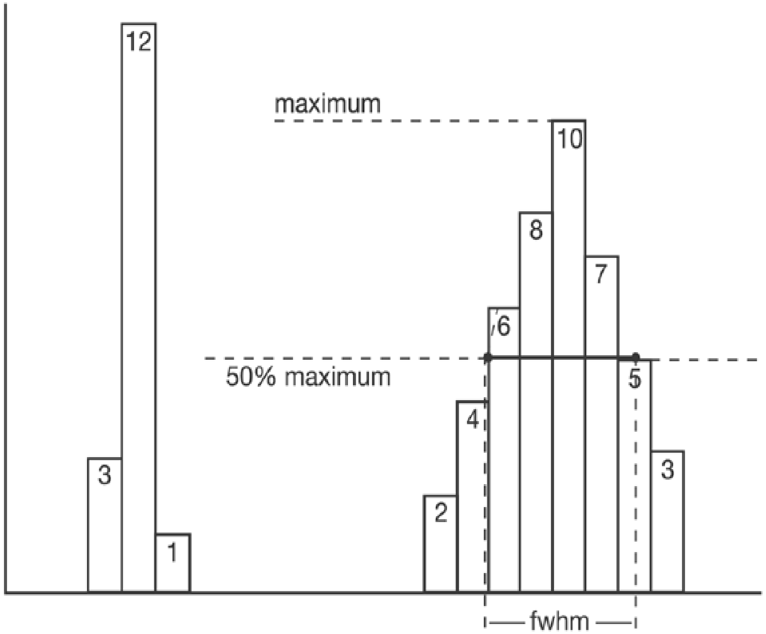
Nevertheless, using more bins may require that you perform a greater number of waveform parameter measurements, in order to populate the bins sufficiently for the identification of a characteristic histogram distribution.

In addition, very fine grained binning will result in gaps between populated bins that may make it difficult to determine peaks.

The oscilloscope's 5,000-parameter buffer is very effective for determining the optimal number of bins to be used. An optimal bin number is one where the change in parameter values is insignificant, and the histogram distribution does not have a jagged appearance. With this buffer, a histogram can be dynamically redisplayed as the number of bins is modified by the user. In addition, depending on the number of bins selected, the change in waveform parameter values can be seen.

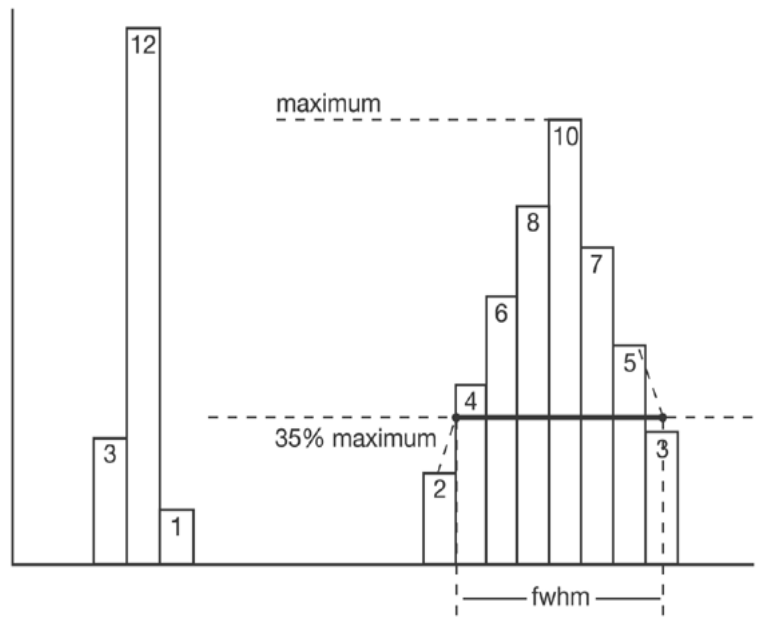
Full Width at Half Maximum

fwhm	Full Width at Half Maximum
Definition:	Determines the width of the largest area peak, measured between bins on either side of the highest bin in the peak that have a population of half the highest's population. If several peaks have an area equal to the maximum population, the leftmost peak is used in the computation.
Description:	First, the highest population peak is identified and the height of its highest bin (population) determined (for a discussion on how peaks are determined see the pks parameter Description:). Next, the populations of bins to the right and left are found, until a bin on each side is found to have a population of less than 50% of that of the highest bin's. A line is calculated on each side, from the center point of the first bin below the 50% population to that of the adjacent bin, towards the highest bin. The intersection points of these lines with the 50% height value is then determined. The length of a line connecting the intersection points is the value for fwhm.

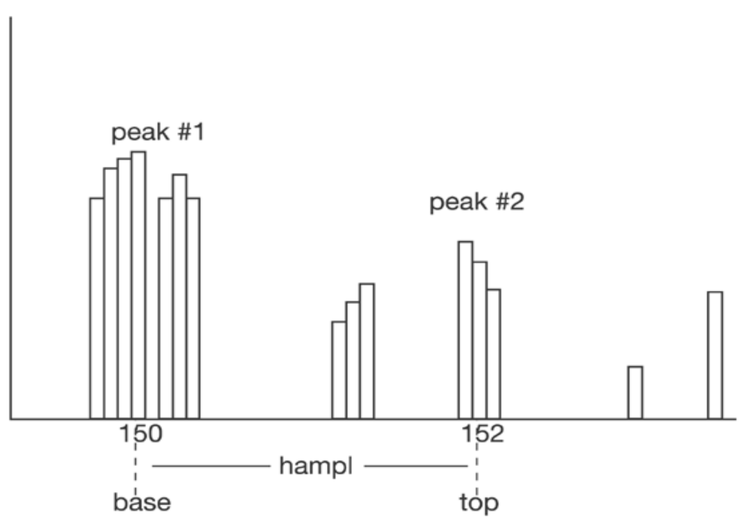
fwhm	Full Width at Half Maximum
Example:	

Full Width at xx% Maximum

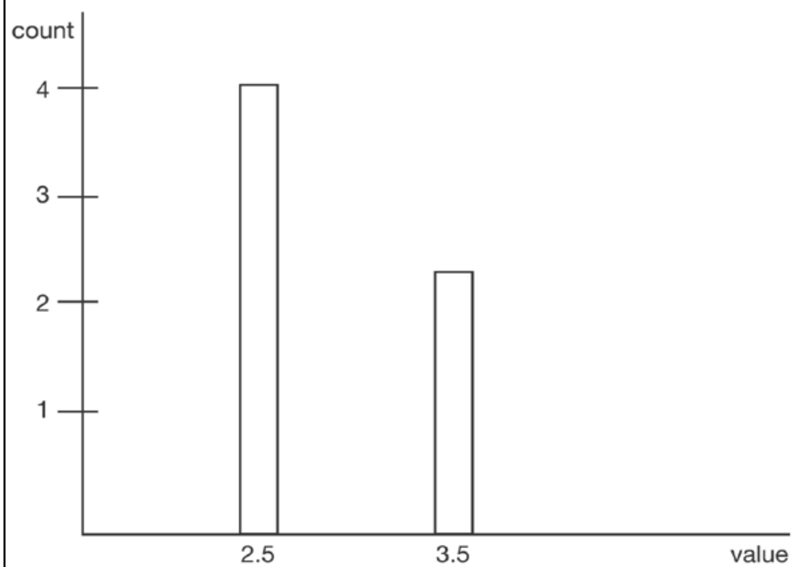
fwxx	Full Width at xx% Maximum
Definition:	Determines the width of the largest area peak, measured between bins on either side of the highest bin in the peak that have a population of xx% of the highest's population. If several peaks have an area equal to the maximum population, the leftmost peak is used in the computation.
Description:	First, the highest population peak is identified and the height of its highest bin (population) determined (see the pks description). Next, the bin populations to the right and left are found until a bin on each side is found to have a population of less than xx% of that of the highest bin. A line is calculated on each side, from the center point of the first bin below the 50% population to that of the adjacent bin, towards the highest bin. The intersection points of these lines with the xx% height value is then determined. The length of a line connecting the intersection points is the value for fwxx.
Example:	fwxx with threshold set to 35%:

fwxx	Full Width at xx% Maximum
	

Histogram Amplitude

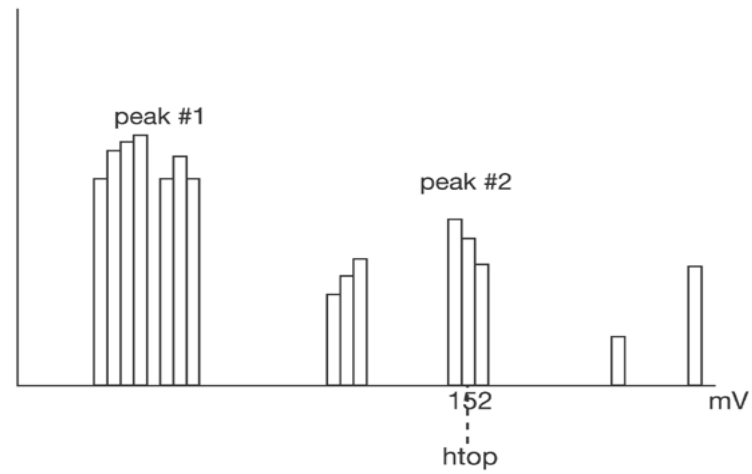
hist ampl	Histogram Amplitude
Definition:	The difference in value of the two most populated peaks in a histogram. This parameter is useful for waveforms with two primary parameter values, such as TTL voltages, where hampl would indicate the difference between the binary `1' and `0' voltage values.
Description:	The values at the center (line dividing the population of peak in half) of the two highest peaks are determined (see pks parameter description:). The value of the leftmost of the two peaks is the histogram base (see hbase). While that of the rightmost is the histogram top (see htop). The parameter is then calculated as: hampl = htop – hbase
Example:	 In this histogram, hampl is 152 mV – 150 mV = 2 mV.

Histogram Root Mean Square

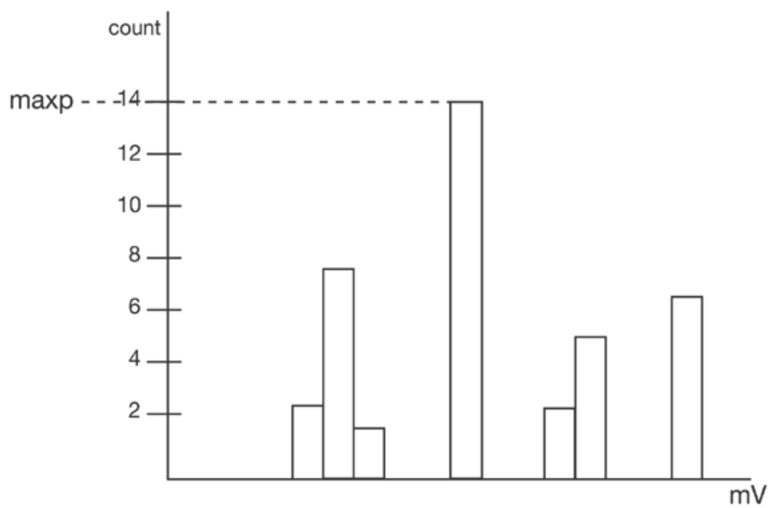
hist rms	**Histogram Root Mean Square**
Definition:	The rms value of the values in a histogram.
Description:	The center value of each populated bin is squared and multiplied by the population (height) of the bin. All results are summed and the total is divided by the population of all the bins. The square root of the result is returned as hrms.
Example:	Using the histogram shown here, the value for hrms is: hrms = $\sqrt{(3.5^2 * 2 + 2.5^2 * 4)/6}$ = 2.87
	

Histogram Top

hist top	**Histogram Top**
Definition:	The value of the rightmost of the two most populated peaks in a histogram. This parameter is useful for waveforms with two primary parameter values, such as TTL voltages, where htop would indicate the binary `1' voltage value.
Description:	The two highest histogram peaks are determined. The rightmost of the two identified peaks is then selected. The center of that peak is htop (center is the horizontal point where the population to the left is equal to the area to the right).

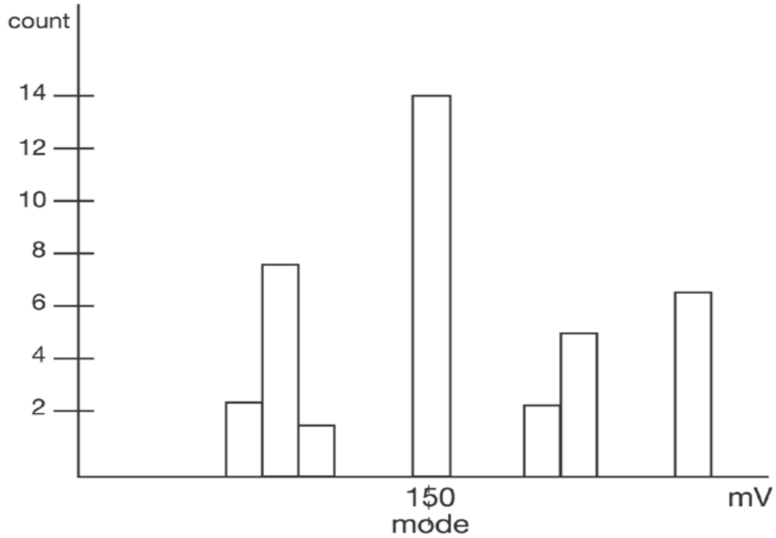
hist top	Histogram Top
Example:	

Maximum Population

maxp	Maximum Population
Definition:	The count (vertical value) of the highest population bin in a histogram.
Description:	Each bin between the parameter cursors is examined for its count. The highest count is returned as maxp.
Example:	 Here, maxp is 14.

Mode

mode	Mode
Definition:	The value of the highest population bin in a histogram.
Description:	Each bin between the parameter cursors is examined for its population count. The leftmost bin with the highest count found is selected. Its center value is returned as mode.

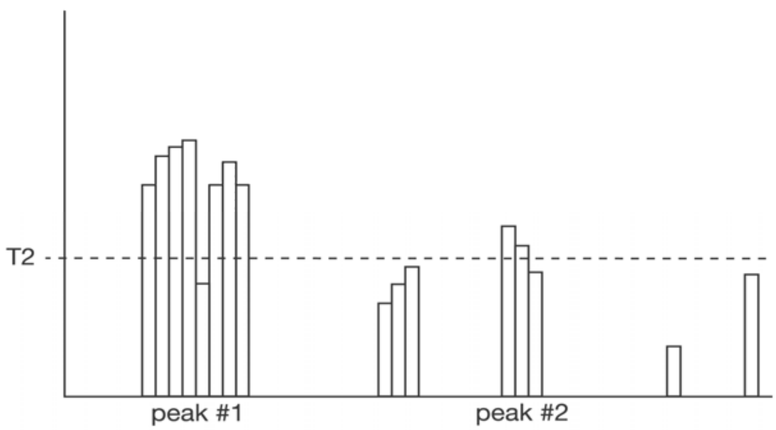
mode	Mode
Example:	 Here, mode is 150 mV.

Percentile

pctl	Percentile
Definition:	Computes the horizontal data value that separates the data in a histogram such that the population on the left is a specified percentage `xx' of the total population. When the threshold is set to 50%, pctl is the same as hmedian.
Description:	The total population of the histogram is determined. Scanning from left to right, the population of each bin is summed until a bin that causes the sum to equal or exceed `xx'% of the population value is encountered. A ratio of the number of counts needed for `xx'% population/total bin population is then determined for the bin. The horizontal value of the bin at that ratio point of its range is found, and returned as pctl.
Example:	The total population of a histogram is 100. The histogram range is divided into 20 bins and `xx' is set to 25%. The population sum at the sixth bin from the left is 22. The population of the seventh is 9 and its sub-range is 6.1 to 6.4 V. The ratio of counts needed for 25% population to total bin population is: 3 counts needed / 9 counts = 1/3. The value for pctl is: 6.1 volts + .33 * (6.4 – 6.1) volts = 6.2 volts.

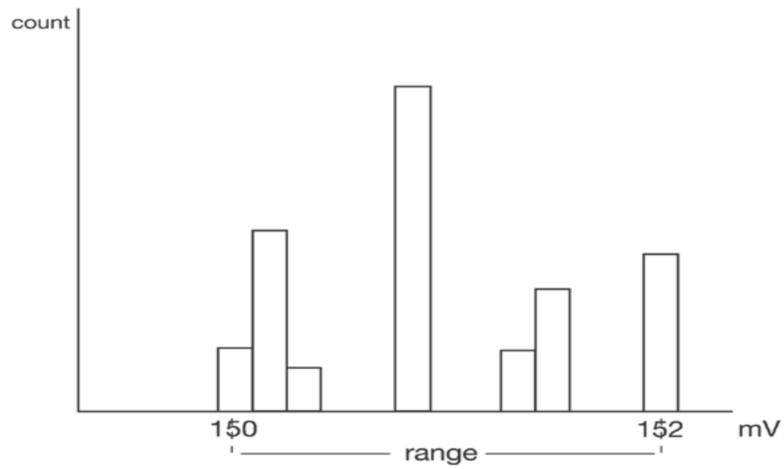
Peaks

pks	Peaks
Definition:	The number of peaks in a histogram.
Description:	The instrument analyzes histogram data to identify peaks from background noise and histogram binning artifacts such as small gaps. Peak identification is a 3-step process: 1. The mean height of the histogram is calculated for all populated bins. A threshold (T1) is calculated from this mean, where: ***T1= mean + 2 sqrt (mean)*** 2. A second threshold is determined based on all populated bins under T1 in height, where:

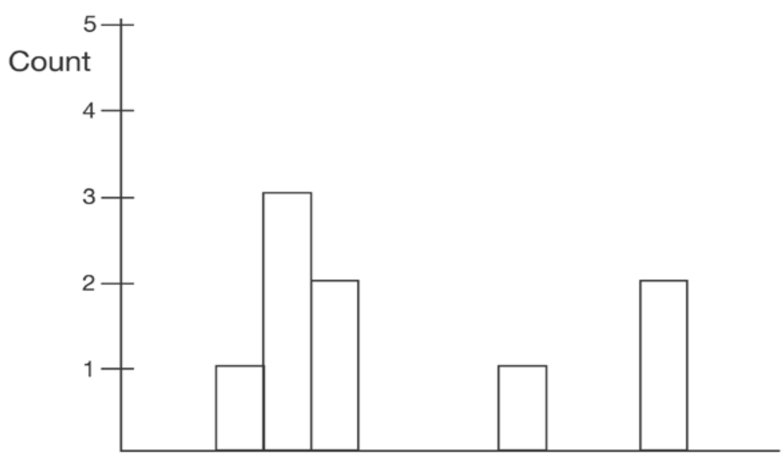
pks	Peaks
	T2 = mean + 2 * sigma, and where sigma is the standard deviation of all populated bins under T1. 3. Once T2 is defined, the histogram distribution is scanned from left to right. Any bin that crosses above T2 signifies the existence of a peak. Scanning continues to the right until one bin or more crosses below T2. However, if the bins cross below T2 for less than a hundredth of the histogram range, they are ignored, and scanning continues in search of peaks that cross under T2 for more than a hundredth of the histogram range. Scanning goes on over the remainder of the range to identify additional peaks. Additional peaks within a fiftieth of the range of the populated part of a bin from a previous peak are ignored. **NOTE:** If the number of bins is set too high, a histogram may have many small gaps. This increases sigma and, thereby, T2. In extreme cases, it can prevent determination of a peak, even if one appears to be present to the eye.
Example:	Here the two peaks have been identified. The peak with the highest population is peak #1. 

Range

range	Range
Definition:	Computes the difference between the value of the rightmost and that of the leftmost populated bin.
Description:	The rightmost and leftmost populated bins are identified. The difference in value between the two is returned as the range.

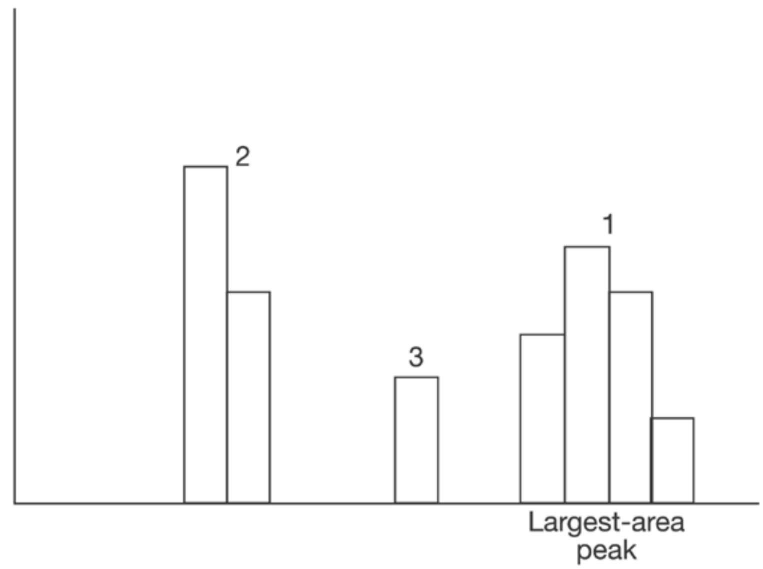
range	Range
Example:	 In this example, range is 2 mV.

Total Population

totp	Total Population
Definition:	Calculates the total population of a histogram between the parameter cursors.
Description:	The count for all populated bins between the parameter cursors is summed.
Example:	 The total population of this histogram is 9.

X Coordinate of xxth Peak

xapk	X Coordinate of xxth Peak
Definition:	Returns the value of the xxth peak that is the largest by area in a histogram.
Description:	First the peaks in a histogram are determined and ranked in order of total area (for a discussion on how peaks are identified see the description for the pks parameter). The center of the nth ranked peak (the point where the area to the left is equal to the area to the right), where n is selected by you, is then returned as xapk.

xapk	X Coordinate of xx[th] Peak
Example:	The rightmost peak is the largest, and is thus ranked first in area (1). The leftmost peak, although higher, is ranked second in area (2). The lowest peak is also the smallest in area (3). 

Restoring Software

System Recovery for Oscilloscopes Running Windows XP and Vista

Your oscilloscope is designed to operate reliably for many years. However, the application software operating the instrument runs on a Windows platform. The loading or incomplete removal of additional Windows applications may eventually cause problems in the stability of the operating system. Severe cases may require a reloading of the base operating system and oscilloscope application. This is done using a recovery routine that restores a clean copy of the image originally installed on the C: drive.

Note: Any user and calibration data located on the D: partition is not affected by the recovery process.

LeCroy provides you with a recovery application and a backup image in an extra partition on the instrument's hard drive. The recovery process is easy to perform, using the following instructions . After the recovery procedure is done, you must activate Windows, either by Internet connection to Microsoft's Web site or by telephone. Have your Windows Product Key number (located on the rear of the oscilloscope) handy during Widows reactivation.

Note: The recovery process produces a replica of the operating system and oscilloscope application software at the current revision levels when the oscilloscope was manufactured. Any further revisions of the application software, Windows operating system, and virus scan definition files are not automatically upgraded. Therefore, after completion of the disk image recovery, it is highly recommended to search vendor Web sites and upgrade the individual components to current revision levels.
The current oscilloscope application software can be downloaded directly from the LeCroy website at www.lecroy.com. Since the calibration data for the oscilloscope is stored in the D: drive, current calibration constants are not overwritten during the recovery process.

Recovery Procedure

1. Connect a network cable to the LAN port on the rear of the oscilloscope if you intend to activate windows through the Internet.
2. Connect a keyboard and a mouse to the oscilloscope.
3. Apply power to the oscilloscope.

4. As soon as the LeCroy logo appears on the screen, press and hold down the F4 key until the recovery software logo appears momentarily:

5. Then the cME console End User License Agreement is displayed. Read the agreement, and click **Accept**:

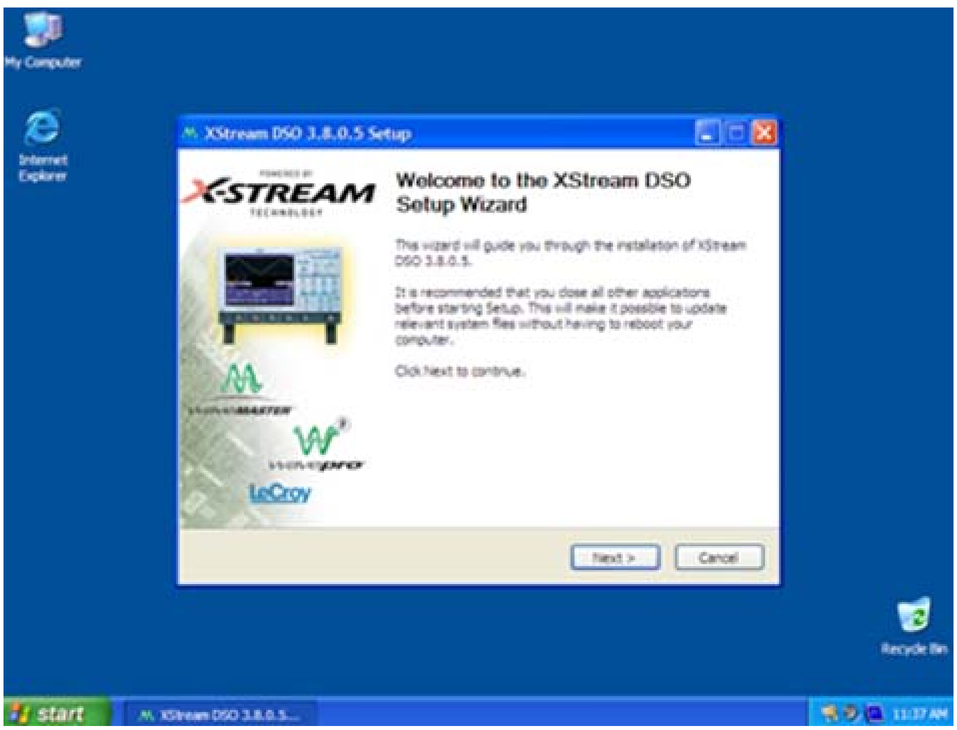
6. The Phoenix cME Console main page is displayed. Click **Click here to start recover**:

7. The FirstWare Recover splash screen is displayed momentarily:

8. The recovery starts, and the FirstWare Progress screen is displayed. No further selections are required. The recovery takes about 10 minutes.

Note: The screen goes blank on occasion for prolonged periods. This is normal and is not an indication of any malfunctioning of the recovery process.

9. After the recovery is completed, the X-Stream software installer screen appears. Click Next to continue:



10. When the X-Stream installation is completed, reboot the oscilloscope (as follows).

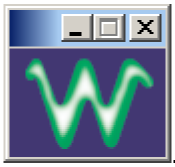
Now you must activate Windows by Internet connection to Microsoft's Web site or by telephone. When activating, have the Windows Product Key number handy (it is affixed to the rear of the oscilloscope).

Restarting the Application

Upon initial power-up, the oscilloscope automatically loads the instrument application software. If you exit the application and want to reload it, touch the shortcut icon on the desktop:

.

If you minimize the application, touch the appropriate task bar or desktop icon to maximize it:


.

Restarting the Operating System

If you need to restart the Windows® operating system, reboot the oscilloscope by pressing and holding in the power switch for 10 seconds, then turning the power on again.

Removable Hard Drive

Not available for all scope models, the removable hard drive option replaces the standard internal hard drive with a removable hard drive that is installed at the rear of the scope, in the slot normally occupied by the CD-ROM drive. The kit includes two hard drives, which can be used interchangeably. It also includes a USB CD-ROM for loading of new software.

Caution – The Removable Hard Drive Is Not Hot Swapable

To avoid damage to the drive or the oscilloscope, shut off power to the instrument before inserting or removing the hard drive. Make sure the protective cover is on the drive at all times.

Figure 4-9. Proper Orientation of Drive

Introduction to LabNotebook

LeCroy's LabNotebook feature extends the documentation capabilities of your scope. It allows you to create an annotated notebook entry containing all displayed waveforms, the setup of the DSO, and user-supplied annotation. The notebook entry can then be converted to hardcopy format -- pdf, rtf, or html -- and printed or e-mailed. You can also use the default report layout or configure your own, and even substitute your own company logo in the header.

Notebook entries are stored in an internal database and are available for recall at any time. Besides storing the waveform data, LabNotebook also stores your panel setups and parameter measurements. You have the capability to back up the database to external media.

The Flashback feature allows you to recall the state of the DSO at a later date, including the saved waveforms and the DSO setup, so that you can make additional measurements. A keyword filter makes it easy to find the correct notebook entry to recall.

You can choose which notebook to use for your entries, and label the notebook by project or user. If the scope is shared among several users, for example, or used for different projects, the data can be kept separately. Similarly, hardcopy reports can be stored in different folders.

Note: If your external monitor is connected, the LabNotebook automatically opens on the external monitor.

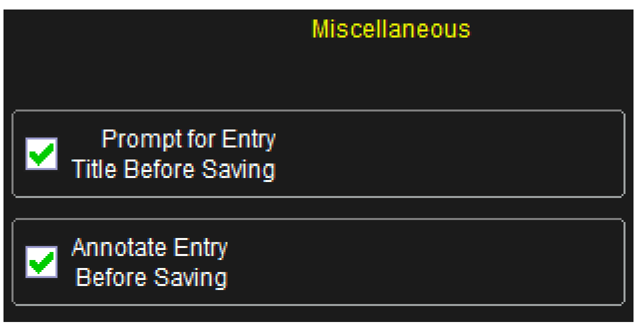
Preferences

You should set your preferences before creating notebook entries.

Setting Preferences

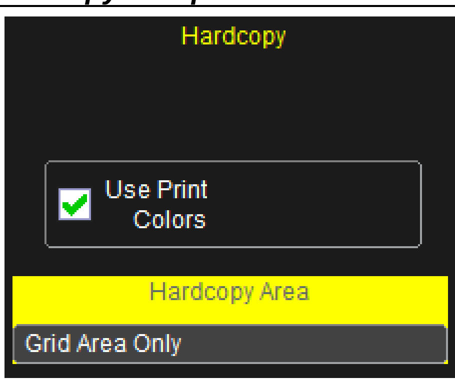
1. On the menu bar, touch **File** → **Lab Notebook**.
2. In the **Lab Notebook** dialog, click the **Preferences** tab.

Miscellaneous Settings



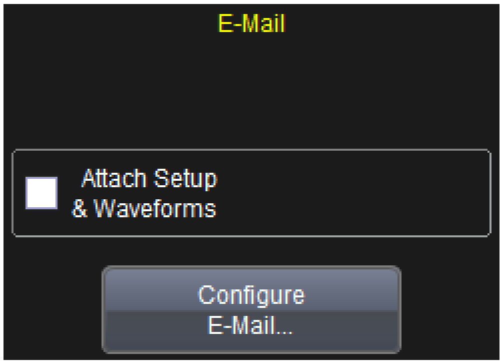
You can elect to name notebook entries with the default date and time by leaving the top box unchecked. Check the box if you want the opportunity to rename the notebook entry as soon as it is created.
Check the middle box if you want to be able to annotate a notebook entry as soon as it is created.

Hardcopy Setup

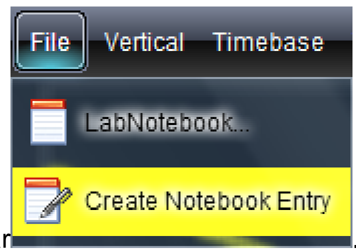


Check the **Use Print Colors** checkbox to place your waveforms on a white background in the notebook entry. This will save printer ink later when you print the hardcopy report.
Touch inside **Hardcopy Area** to determine how much of the screen image to include in the report: grid area only, grid area plus dialog, whole screen.

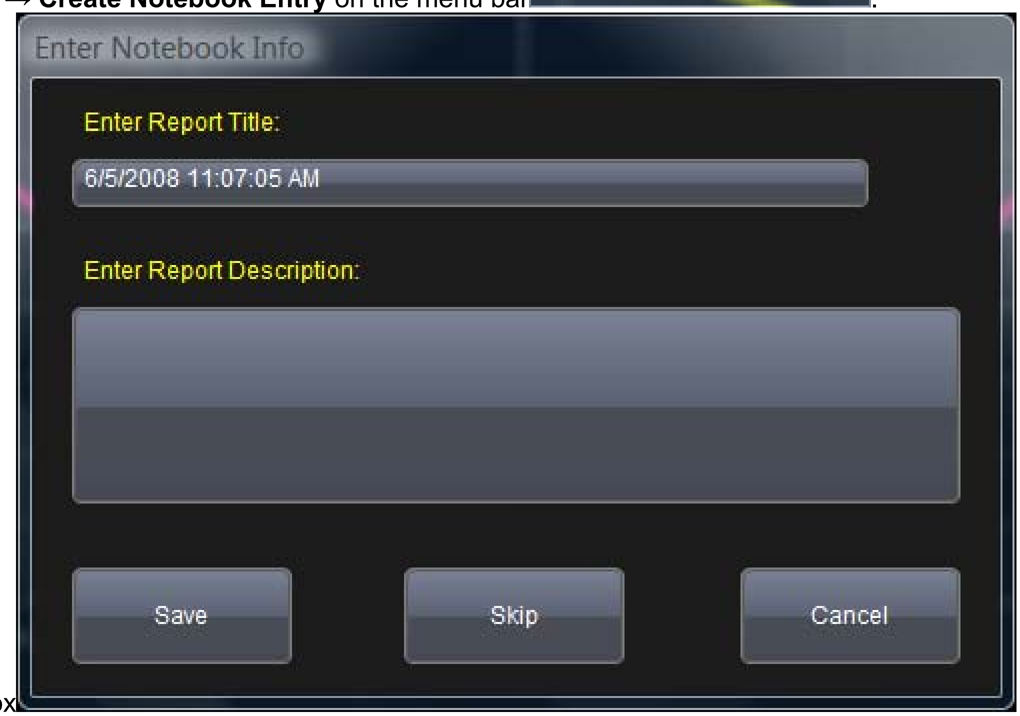
E-mail Setup

	You can e-mail just the pdf or html report; or, you can include additional files: trace data (.trc) for each waveform in the report, a screen dump, a scope setup file, and an xml report record. Touch the checkbox to enable the extra report segments.
	Touch the **Configure E-Mail** button to set the recipient address and server information.

Creating a Notebook Entry

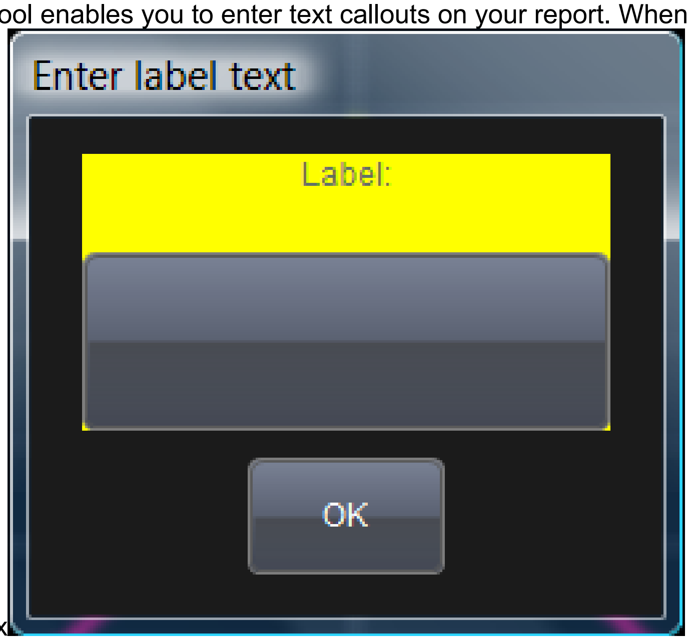
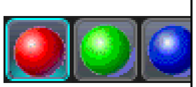
1. Touch **File** → **Create Notebook Entry** on the menu bar 

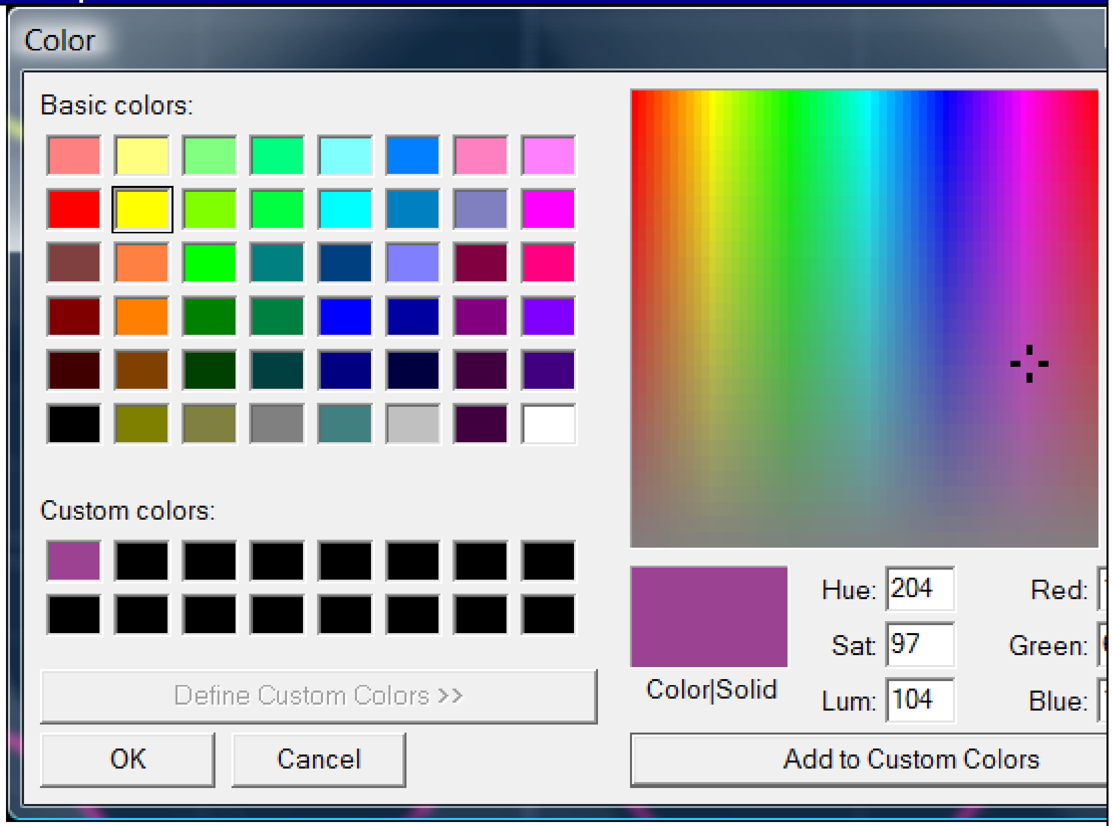
.

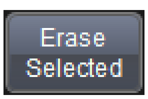
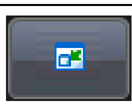
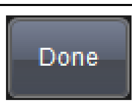
   A dialog box 

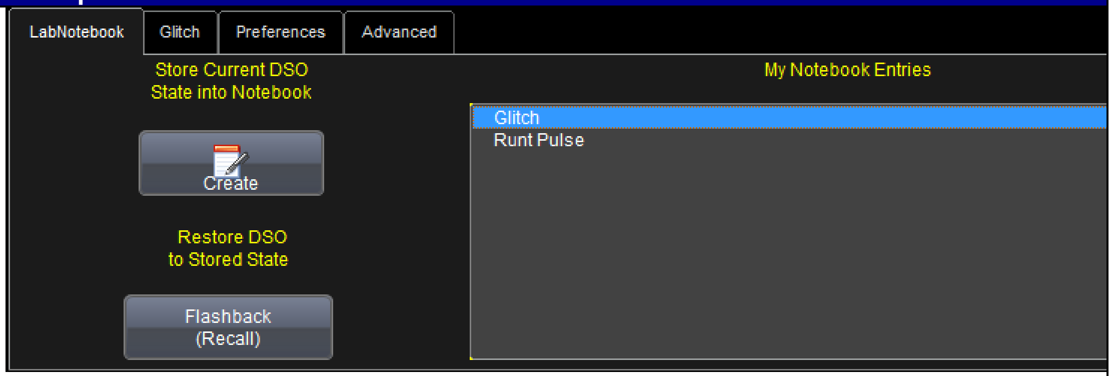
is displayed in which to enter a title and comments for the entry. By default, the entry is titled with the current date and time.
2. Touch inside the **Title** field and enter a title, using the pop-up keyboard. Then touch inside the **Description** field and enter a description, if desired, and touch **Close**.
3. The notebook entry will display your waveforms in "print colors," that is, on a white background to save printer ink, if you selected that option in notebook Preferences. Otherwise, the waveforms will appear on a black background.

A drawing toolbar appears at top:

Icon:	Description:
	The pen tool enables you to write or draw in freehand. You can use a mouse, or a stylus to do this using the touch screen. Once you click off, you can drag your note anywhere on your waveform.
	The circle tool enables you to create a circle around a waveform feature that you want to point out. Once you click off, the circle is drawn and you can drag it anywhere on the screen.
	The arrow tool enables you to draw lines with arrowheads for callouts. You can rotate these lines through 360 degrees and drag them to any location on the screen.
	The text tool enables you to enter text callouts on your report. When you touch this tool, a dialog box  opens in which to enter text by means of a pop-up keyboard. After you touch **Close**, your text will appear on the display as a draggable object.
	These are the three default colors that you can select for shapes, lines, and text. To use additional colors, touch **More**.
	When you touch **More**, a Custom box opens with the default color yellow displayed. Touch the yellow button to open the full color palette

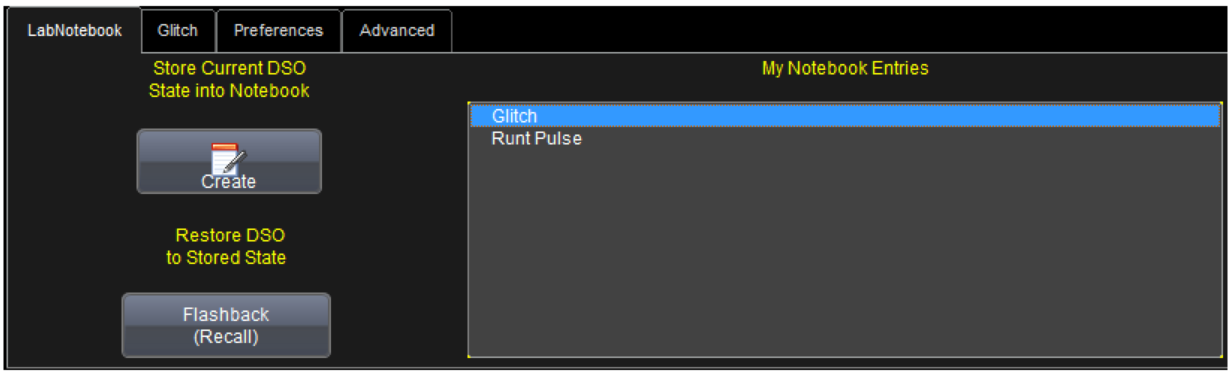
<table>
<tr><th>Icon:</th><th>Description:</th></tr>
<tr><td></td><td>

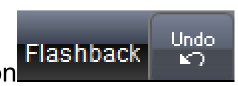
.
When you have chosen a custom color, touch Add to Custom Colors; the color will appear in the Custom Colors palette. Then touch the color to enable it, and touch OK. The next object that you create will be in that color.</td></tr>
<tr><td>
</td><td>If you want to erase a drawing object, touch it to select it, then touch Erase Selected.</td></tr>
<tr><td>
</td><td>Touch Erase All to erase all drawn objects and text.</td></tr>
<tr><td>
</td><td>Touch Undo to discard the last object drawn.</td></tr>
<tr><td></td><td>The Move Toolbar button enables you to place the toolbar anywhere on the screen. Touch the button a second time to return it to its original fixed location.</td></tr>
<tr><td>
</td><td>Touch Done when you are finished annotating the notebook entry. The name of the entry will appear in the list
box</td></tr>
</table>

Icon:	Description:
	 in the **LabNotebook** dialog. You can now create a hardcopy report of it, and email or print it out.

Recalling Notebook Entries

After a notebook entry is made, you can recall it at any time. The recall includes waveforms and scope settings.



1. Select the notebook entry from the list box.
2. Touch **Flashback**.
3. To exit Flashback, touch the **Undo Flashback** button  in the top-right corner of the screen, or press the **Auto** trigger button.

Note: The flashback feature currently recalls the DSO Setup, and all displayed waveforms. Some forms of 'result data' are not recalled, including:

a. Persistence data. This will be saved in the hardcopy, and will be printed in the report, but will not be recalled during Flashback.

b. Histogram data. Histograms internally have a 32-bit resolution, but when stored into a trace file and recalled during flashback they are clipped to 16-bits.

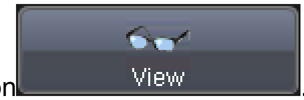
c. Floating point waveforms. Certain math operations result in the creation of floating point waveforms with much higher resolution than can be stored in a 16-bit waveform file. This extra resolution will not be preserved when traces are recalled using flashback.

d. Cumulative Measurements. Any measurements that are on when the Lab Notebook entry is created are not saved individually in the database, other than being embedded in the hardcopy image. This means that when flashback is used, the measurements will be recomputed using the waveform data that was recalled. Normally this will not pose a problem, but if cumulative measurements were on, which accumulated data from multiple acquired waveforms, they will lose their history and show instead only the results from the stored waveforms.

Creating a Report

Once the notebook entry is created, you can easily generate a hardcopy report for e-mailing or printing.

Previewing a Report

Before creating a report, you can preview it by simply touching the **View** button 

. To exit the preview, touch the **Close** button at the right of the dialog.

Locating a Notebook Entry

A search filter is provided to help you locate the notebook entry you want to make a report of. You can search by date or keyword.

1. Touch the **Filter** button 

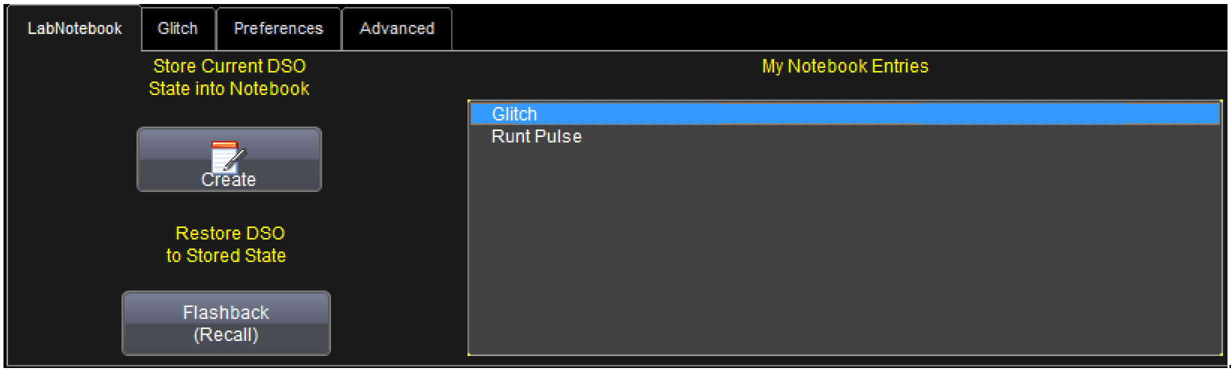
. A search dialog box opens.
2. Touch inside the **Day**, **Month**, and **Year** fields and enter a date.

   OR

   Touch inside the **Keyword** field and enter a keyword or phrase.
3. Touch **Find Now**. Only the entries fitting the date or keyword criteria will now appear in the list box.

Creating the Report

1. Select a notebook entry in the list box



.

2. Touch inside the **Format** field and select a report format from the pop-up menu.
3. Touch the **Create Report** button.
4. In the **Create Report** window, select a folder to contain the report. Touch inside the **File name** field and enter a name using the pop-up keyboard. Click **OK** to create the report.

Formatting the Report

LeCroy provides a default report format (template); however, you can use your own format, including company logo.

1. Touch the **Advanced** tab.
2. Touch inside the **Directory** field and navigate to a folder to contain the reports.
3. Touch the **Browse** button next to Template to navigate to an existing report format that you want to use. Or touch inside the **Template** field and enter the name and path to the template, using the pop-up keyboard. Otherwise, touch the **Use Default** checkbox to use LeCroy's format.
4. To use a logo other that the one provided, which indicates the scope that produced the report, browse to the bit map file or touch inside the **Logo** field and enter the name and path to the file, using the pop-up keyboard. Otherwise, touch the **Use Default** checkbox to use LeCroy's logo.

Note: If you elect to use your own logo bit map, do not use a bit map larger than 180 pixels (height) x 100 pixels (width).

Managing Notebook Entry Data

Adding Annotations

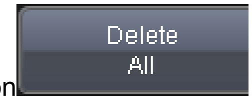
You can add annotations to your notebook entry at any time.

1. Touch the **LabNotebook** tab.

2. Touch the notebook entry you want to annotate in the scroll list box. A new tab appears bearing the name of the selected notebook entry.
3. Touch the new tab, then the **Scribble** button 

. The notebook entry opens again with the drawing toolbar, described in Creating a Notebook Entry.

Deleting Notebook Entries

1. Touch the **LabNotebook** tab.
2. To clear the database, touch the **Delete All** button 

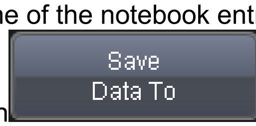
.

   OR
   To delete the selected entry only, select a notebook entry in the list box, then touch the **Delete** button 

.

Saving Notebook Entries to a Folder

You can save notebook entries to a folder other than the default.

1. Touch the tab bearing the name of the notebook entry.
2. Touch the **Save Data to** button 

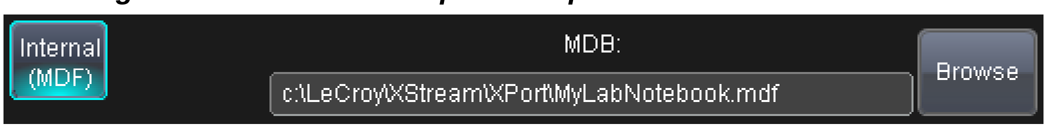
. In the **Save Data To** window, select the location where you want to save the data.
3. Touch the **Zip** checkbox 

 if you want to compress the data before archiving.

Managing the Database

You can begin a new database for your notebook entries at any time, back up the current one, or compress the data.

Selecting a Database for Backup or Compression

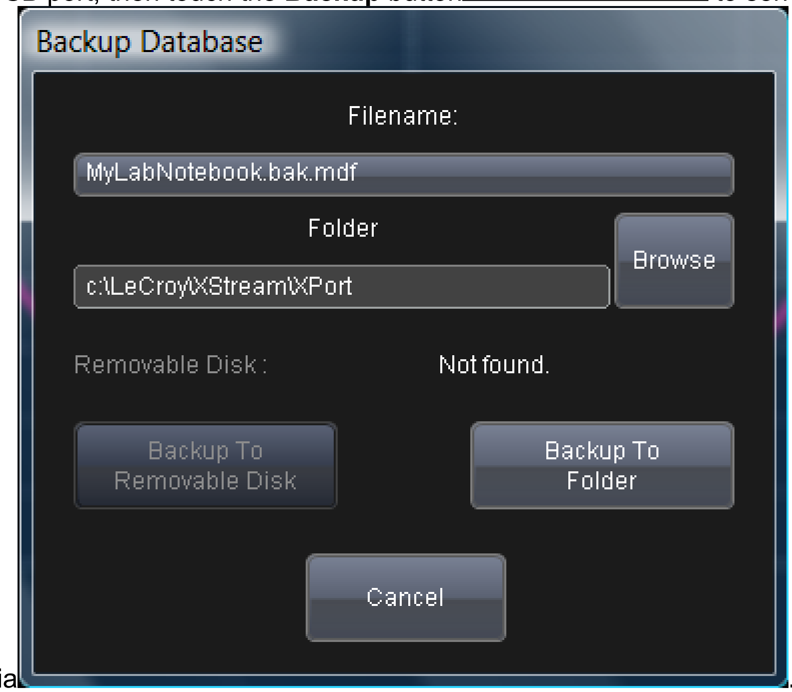


1. Touch the **Advanced** tab.
2. Touch the **Browse** button. A navigation window opens. Navigate to a database.
3. Touch the **Compact** button 

 to reduce the size of a database. This function "defragments" the notebook after a large amount of entries have been deleted.

   OR

Insert a memory stick into a USB port, then touch the **Backup** button 

to send the database to the external media 

.

Starting a New Database

Touch the **Start New** button. The name of the notebook database will be incremented by 1.

Introduction to Math Traces and Functions

With the instrument’s math tools you can perform mathematical functions on a waveform displayed on any channel, or recalled from any of the four reference memories M1 to M4. You can also set up traces F1 to Fx[The number of math functions that can be performed at the same time depends on the software options loaded on your scope.] to do math on parameter measurements P1 to Px[The number of parameters that can be measured at the same time depends on the software options loaded on your scope.] .

Instruments that offer more than one math trace provide enhanced capabilities. For example: you could set up Trace F1 as the difference between Channels 1 and 2, Trace F2 as the average of F1, and Trace F3 as the integral of F2. You could then display the integral of the averaged difference between Channels 1 and 2.

Any trace and function can be chained to another trace and function. For example, you could make Trace F1 an average of Channel 1, Trace F2 an FFT of F1, and Trace F3 a zoom of F2.

Note: Math traces F5-F8 are available only if you have loaded software option package XMATH or XMAP on WaveMaster or WavePro scopes, but are standard on Disk Drive Analyzers and Serial Data Analyzers.

Math Made Easy

With the instrument's math tools you can perform mathematical functions on a waveform displayed on any channel C1 to C4, or recalled from any of the four reference memories M1 to M4. To do computations in sequence, you can also use math functions F1 to Fx as a source input waveform. Or you can use Parameters P1 through Px

For example: you could set up F1 as the difference between Channels 1 and 2, F2 as the average of F1, and F3 as the integral of F2. You could then display the integral of the averaged difference between Channels 1 and 2. Any trace and function can be chained to another trace and function. For example, you could make F1 an average of Channel 1, F2 an FFT of F1, and F3 a zoom of F2.

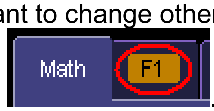
Refer to the Specifications to find out which math tools are available in each optional package.

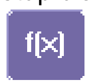
Setting Up a Math Function

Math Setup

This setup mode allows you to quickly apply frequently used math functions.

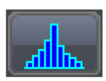
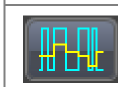
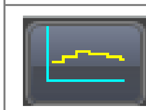
1. Touch **Math** → **Math Setup** on the menu bar.
2. If there are math functions already assigned to **F1** through **Fx**[The number of math traces available depends on the software options loaded on your scope. See Specifications.] , touch the checkbox for the function you want to enable.
3. To assign a new math function to a trace, touch the Fx button for that trace, for example 

. The math function menu appears.
4. Touch a menu selection; your new function is automatically assigned, with the same setups as were in place for the last function in that Fx position.
5. If you want to change other setup items, like the source waveform, touch the appropriate Fx tab, for example 

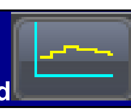
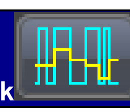
. The setup dialog for that Fx position appears.
6. Touch the **Single** function button 

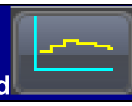
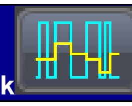
if you want to perform just one math function on the trace, or touch the **Dual** function button 

to perform math on math.
7. Touch the **Graph** button, then touch inside the **Graph with** field to select a graph mode. The Graph modes are as follows:

Mode:	Description:
	Histogram of the values of a parameter
	Track of each value of a parameter
	Trend of the values of a parameter over time

Both Track and Trend are tools that can be used to plot measurement data and observe variations with respect to time. Differences between Track and Trend are summarized in the table below:

Characteristic	Trend 	Track 
Representation	Parameter value vs. event	Parameter value vs. time
Behavior	Cumulative over several acquisitions up to 1 million events	Non-cumulative (resets after every acquisition). Unlimited number of events
Time Correlation to Other Data	No	Yes
Monitors an Evolution in the Frequency Domain	No. Trend points are not evenly spaced in time and therefore cannot be used for an FFT.	Yes
Monitors the Evolution of a Measurement Parameter over Several Acquisitions	Yes	No. Track resets after every acquisition.

Characteristic	Trend 	Track 
Ensures No Lost Measurement Data	No. Since data can be accumulated over many acquisitions, and since the oscilloscope takes time to calculate measurement values and to display data before the trigger is re-armed, data can be missed.	Yes. Maximum time period that can be captured is limited by acquisition memory and sampling rate.

In general, Track will be the tool to use if you want to capture a continuous stream of data that is spaced closely together. Trend can be used if your data is spaced widely apart and longer than the “dead-time” of the oscilloscope between acquisitions.

Resampling to Deskew

Deskew whenever you need to compensate for different lengths of cables, probes, or anything else that might cause timing mismatches between signals. Resample a signal on one channel and adjust it in time relative to a signal on another channel.

Deskewing

1. Touch **Math** → **Math Setup** on the menu bar.
2. Touch a math function trace tab **F1** through **Fx**[The number of math traces available depends on the software options loaded on your scope. See Specifications.]
3. Touch the **single** function button.
4. Touch inside the **Source1** field and select a source: channel, math trace, memory location.
5. Touch inside the **Operator1** field and select **Deskew** from the **Functions** category.
6. In the dialog on the right, touch the **Deskew** tab.
7. Touch inside the **Delay by** data entry field and type in a time value, using the pop-up keypad.

Rescaling and Assigning Units

This feature allows you to apply a multiplication factor (*a*) and additive constant (*b*) to your waveform: *a*X + *b*. You can do it in the unit of your choice, depending on the type of application.

Allowable unit **abbreviations** and **descriptions** are as follows:

Abbreviations and Descriptions				
1. **(blank)** - No units	11. **DIV** - Divisions	21. **L** - Liter	31. **POISE** - Poise	41. **T** - Tesla
2. **A** - Ampere	12. **Event** - Events	22. **M** - Meter	32. **PPM** - Parts per million	42. **UI** - Unit interval
3. **C** - Coulomb	13. **F** - Farad	23. **FT** - Foot	33. **RAD** - Radian	43. **V** - Volt
4. **CYCLE** - Cycles	14. **G** - Gram	24. **IN** - Inch	34. **DEG** - Degree (of arc)	44. **VA** - Volt amps
5. **DB** - Decibel	15. **H** - Henry	25. **YARD** - Yard	35. **MNT** - Minute (of arc)	45. **W** - Watt
6. **DBC** - Decibel referred to carrier	16. **HZ** - Hertz	26. **MILE** - Mile	36. **SAMPLE** - Sample	46. **WB** - Weber
7. **DBM** - Decibel Milliwatt	17. **J** - Joule	27. **N** - Newton	37. **SWEEP** - Sweeps	47. **MIN** - Min
8. **DBV** - Decibel Volts	18. **K** - Degree Kelvin	28. **OHM** - Ohm	38. **SEC** - Second (of arc)	48. **HOUR** - Hour
9. **DBUZ** - Decibel Microamp	19. **CEL** - Degree Celsius	29. **PAL** - Pascal	39. **S** - Second	49. **DAY** - Day
10. **DEC** - Decade	20. **FAR** - Degree Fahrenheit	30. **PCT** - Percent	40. **SIE** - Siemens	50. **WEEK** - Week

You can also enter combinations of the above units following the SI rules:

- for the quotient of two units, the character **/** should be used
- for the product of two units, the character **.** should be used
- exponents can be represented by a digit appended to the unit without a space

For example,

- acceleration can be entered as M/S2 for meters per second squared
- volts seconds can be entered as V.S.

In some cases, the units entered may be converted to simple units. For example entering V.A will display W (watts)

Rescaling Setup

1. Touch **Math** → **Math Setup** on the menu bar.
2. Touch a math function trace tab **F1** through **Fx**[The number of math traces available depends on the software options loaded on your scope. See Specifications.]
3. Touch the **single** function button.
4. Touch inside the **Source1** data entry field and select a source: channel, math trace, memory location.
5. Touch inside the **Operator1** data entry field and select **Rescale** from the **Functions** category.
6. In the dialog on the right, touch the **Rescale** tab.
7. Touch inside the **First multiply by:** checkbox and enter a value for *a*, the multiplication factor.
8. Touch inside the **then add:** data entry field and enter a value for *b*, the additive constant.
9. Touch inside the **Override units** checkbox to disregard the source waveform's units.

Averaging Waveforms

Summed vs. Continuous Averaging

For Summed averaging, you specify the number of acquisitions to be averaged. The averaged data is updated at regular intervals and presented on the screen.

On the other hand, Continuous averaging (the system default) helps to eliminate the effects of noise by continuously acquiring new data and adding the new waveforms into the averaging buffer. You determine the importance of new data vs. old data by assigning a weighting factor. Continuous averaging allows you to make adjustments to a system under test and to see the results immediately.

Note: Continuous Averaging is accessible from the channel "Vertical Adjust" dialog under "Pre-Processing," and from the math function menu.

Summed Averaging

Summed Averaging is the repeated addition, with equal weight, of successive source waveform records. If a stable trigger is available, the resulting average has a random noise component lower than that of a single-shot record. Whenever the maximum number of sweeps is reached, the averaging process stops.

An even larger number of records can be accumulated simply by changing the number in the dialog. However, the other parameters must be left unchanged or a new averaging calculation will be started. You can pause the averaging by changing the trigger mode from NORM/AUTO to STOP. The instrument resumes averaging when you change the trigger mode back to NORM/AUTO.

You can reset the accumulated average by pushing the CLEAR SWEEPS button or by changing an acquisition parameter such as input gain, offset, coupling, trigger condition, timebase, or bandwidth limit. The number of current averaged waveforms of the function, or its zoom, is shown in the acquisition status dialog. When summed averaging is performed, the display is updated at a reduced rate to increase the averaging speed (points and events per second).

Continuous Averaging

Continuous Averaging, the default setting, is the repeated addition, with unequal weight, of successive source waveforms. It is particularly useful for reducing noise on signals that drift very slowly in time or amplitude. The most recently acquired waveform has more weight than all the previously acquired ones: the continuous average is dominated by the statistical fluctuations of the most recently acquired waveform. The weight of ‘old’ waveforms in the continuous average gradually tends to zero (following an exponential rule) at a rate that decreases as the weight increases.

The formula for continuous averaging is

new average = (new data + weight * old average)/(weight + 1)

This is also the formula used to compute summed averaging. But by setting a "sweeps" value, you establish a fixed weight that is assigned to the old average once the number of "sweeps" is reached. For example, for a sweeps (weight) value of **4**:

1st sweep (no old average yet): new average = (new data +0 * old average)/(0 + 1) = new data only

2nd sweep: new average = (new data + 1*old average)/(1 + 1) = 1/2 new data +1/2 old average

3rd sweep: new average = (new data + 2 * old average)/(2 + 1) = 1/3 new data + 2/3 old average

4th sweep: new average = (new data + 3 * old average)/(3 + 1) = 1/4 new data + 3/4 old average

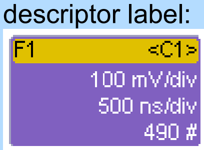
5th sweep: new average = (new data + 4 * old average)/(4 + 1) = 1/5 new data + 4/5 old average

6th sweep: new average = (new data + 4 * old average)/(4 + 1) = 1/5 new data + 4/5 old average

7th sweep: new average = (new data + 4 * old average)/(4 + 1) = 1/5 new data + 4/5 old average

In this way, for sweeps > 4 the importance of the old average begins to decrease exponentially.

Note: The number of sweeps used to compute the average will be displayed in the bottom line of the trace descriptor label:



Setting Up Continuous Averaging

1. Touch **Math** → **Math Setup** on the menu bar.
2. Select a function tab from **F1** through **Fx**[The number of math traces available depends on the software options loaded on your scope. See Specifications.]
3. Touch inside the **Source1** field and select a source waveform from the pop-up menu.
4. Touch inside the **Operator1** field and select **Average** from the **Select Math Operator** menu.
5. Touch the **Average** tab in the dialog to the right of the "Fx" dialog, touch the **Continuous** button.
6. Touch inside the **Sweeps** data entry field and enter a value using the pop-up keypad. The valid range is 1 to 1,000,000 sweeps.

Setting Up Summed Averaging

1. Touch **Math** → **Math Setup** on the menu bar.
2. Select a function tab from **F1** through **Fx**[The number of math traces available depends on the software options loaded on your scope. See Specifications.]
3. Touch inside the **Source1** field and select a source waveform from the pop-up menu.
4. Touch inside the **Operator1** field and select **Average** from the **Select Math Operator** menu.
5. Touch the **Average** tab in the dialog to the right of the "Fx" dialog, then touch the **Summed** button.
6. Touch inside the **Sweeps** data entry field and type in a value using the pop-up keypad. The valid range. is 1 to 1,000,000 sweeps.

Enhanced Resolution

ERES (Enhanced Resolution) filtering increases vertical resolution, allowing you to distinguish closely spaced voltage levels. The functioning of the instrument's ERES is similar to smoothing the signal with a simple, moving-average filter. However, it is more efficient concerning bandwidth and pass-band filtering. Use ERES on single-shot waveforms, or where the data record is slowly repetitive (when you cannot use averaging). Use it to reduce noise when your signal is noticeably noisy, but you do not need to perform noise measurements. Also use it when you perform high-precision voltage measurements: zooming with high vertical gain, for example.

How the Instrument Enhances Resolution

The instrument's enhanced resolution feature improves vertical resolution by a fixed amount for each filter. This real increase in resolution occurs whether or not the signal is noisy, or your signal is single-shot or repetitive. The signal-to-noise ratio (SNR) improvement you gain is dependent on the form of the noise in the original signal. The enhanced resolution filtering decreases the bandwidth of the signal, filtering out some of the noise.

The instrument's constant phase finite impulse response (FIR) filters provide fast computation, excellent step response in 0.5 bit steps, and minimum bandwidth reduction for resolution improvements of between 0.5 and 3 bits. Each step corresponds to a bandwidth reduction factor of two, allowing easy control of the bandwidth resolution trade-off. The parameters of the six filters are given in the following table.

Resolution increased by	-3 dB Bandwidth (x Nyquist)	Filter Length (Samples)
0.5	0.5	2
1.0	0.241	5
1.5	0.121	10
2.0	0.058	24
2.5	0.029	51
3.0	0.016	117

With low-pass filters, the actual SNR increase obtained in any particular situation depends on the power spectral density of the noise on the signal.

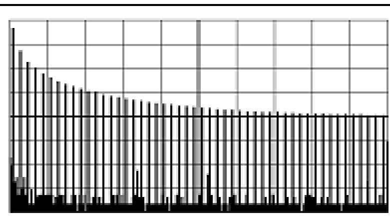
The improvement in SNR corresponds to the improvement in resolution if the noise in the signal is white -- evenly distributed across the frequency spectrum.

If the noise power is biased towards high frequencies, the SNR improvement will be better than the resolution improvement.

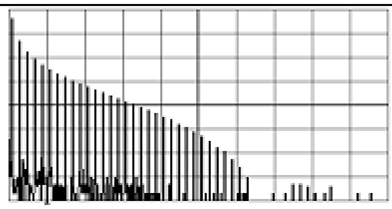
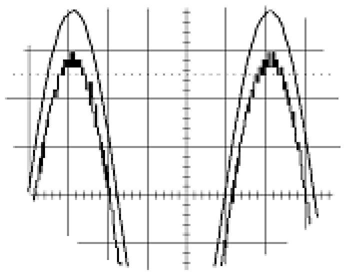
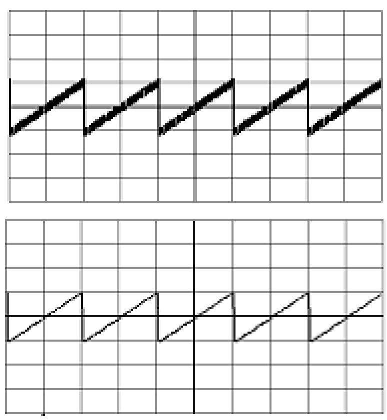
The opposite may be true if the noise is mostly at lower frequencies. SNR improvement due to the removal of coherent noise signals - feed-through of clock signals, for example - is determined by the fall of the dominant frequency components of the signal in the passband. This is easily ascertained using spectral analysis. The filters have a precisely constant zero-phase response. This has two benefits. First, the filters do not distort the relative position of different events in the waveform, even if the events' frequency content is different. Second, because the waveforms are stored, the delay normally associated with filtering (between the input and output waveforms) can be exactly compensated during the computation of the filtered waveform.

The filters have been given exact unity gain at low frequency. Enhanced resolution should therefore not cause overflow if the source data is not overflowed. If part of the source trace were to overflow, filtering would be allowed, but the results in the vicinity of the overflowed data -- the filter impulse response length - would be incorrect. This is because in some circumstances an overflow may be a spike of only one or two samples, and the energy in this spike may not be enough to significantly affect the results. It would then be undesirable to disallow the whole trace.

The following examples illustrate how you might use the instrument's enhanced resolution function.



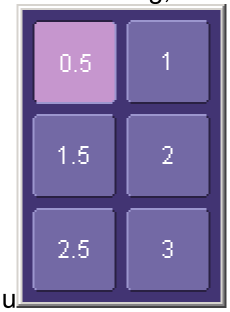
In low-pass filtering: The spectrum of a square signal before (left top) and after (left bottom) enhanced resolution processing. The result clearly illustrates how the filter rejects high-frequency components from the signal. The higher the bit enhancement, the lower the resulting bandwidth.

	
	**To increase vertical resolution:** In the example at left, the lower ("inner") trace has been significantly enhanced by a three-bit enhanced resolution function.
	**To reduce noise:** The example at left shows enhanced resolution of a noisy signal. The original trace (left top) has been processed by a 2-bit enhanced resolution filter. The result (left bottom) shows a "smooth" trace, where most of the noise has been eliminated.

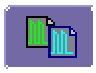
Note: Enhanced resolution can only improve the resolution of a trace; it cannot improve the accuracy or linearity of the original quantization. The pass-band will cause signal attenuation for signals near the cut-off frequency. The highest frequencies passed may be slightly attenuated. Perform the filtering on finite record lengths. Data will be lost at the start and end of the waveform: the trace will be slightly shorter after filtering. The number of samples lost is exactly equal to the length of the impulse response of the filter used: between 2 and 117 samples. Normally this loss (just 0.2 % of a 50,000 point trace) is not noticed. However, you might filter a record so short there would be no data output. In that case, however, the instrument would not allow you to use the ERES feature.

Setting Up Enhanced Resolution (ERES)

1. Touch **Math** → **Math Setup** on the menu bar.
2. Touch a function tab **F1** through **Fx**[The number of math traces available depends on the software options loaded on your scope. See Specifications.]
3. Touch inside the **Operator1** data entry field.
4. Select **ERES** from the **All Functions** or **Filter** group of Math functions.
5. Touch the **Trace On** checkbox.

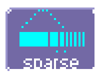
6. Touch the **ERES** tab in the right-hand dialog, then touch inside the **bits** field and make an **Enhance** by selection from the pop-up menu 

Waveform Copy

The **Copy** math function  makes a copy of your present waveform in its unprocessed state. While processing may continue on the original waveform, the copy enables faster throughput in some cases by preserving the original data. That is, no calculations need to be undone on the copy before additional math can be calculated.

This benefit of faster throughput, however, comes at the expense of memory usage.

Waveform Sparser

The Sparse math function 

allows you to thin out an incoming waveform by skipping points at regular intervals, and by starting acquisition at a particular "offset" (point). The **Sparsing factor** specifies the number of sample points to reduce the input waveform by. A sparsing factor of 4, for example, tells the scope to retain only one out of every 4 samples. A **Sparsing offset** of 3, on the other hand, tells the scope to begin on the third sample, then skip the number of samples specified by the sparsing factor (4). In this way, the sample rate is effectively reduced.

For the sparsing factor (interval), you can set a value from 1 to 1,000,000 points. For the sparsing offset you can set a value from 0 to 999,999.

Note: The maximum sparsing offset that can be entered for any sparsing factor equals Sparsing Factor 1.

Waveform Sparser Setup

1. Touch **Math** → **Math setup** on the menu bar.
2. Touch the tab for the function (F1 to Fx[The number of math traces available depends on the software options loaded on your scope. See specifications.]) you want to assign the Sparse operation to.
3. Touch inside the **Source1** field and select an input waveform.
4. Touch inside the **Operator1** field and select **Sparse** from the **Select Math Operator** menu.
5. Touch inside the **Sparsing factor** field and enter a value, using the pop-up keypad.
6. Touch inside the **Sparsing offset** field and enter a value, using the pop-up keypad.

Interpolation

Linear interpolation, which inserts a straight line between sample points, is best used to reconstruct straight-edged signals such as square waves. (Sinx)/x interpolation, on the other hand, is suitable for reconstructing curved or irregular waveshapes, especially when the sampling rate is 3 to 5 times the system bandwidth. The instrument also gives you a choice of Cubic interpolation.

For each method, you can select a factor from 2 to 50 points by which to interpolate (upsample).

Setting Up Interpolation

1. Touch **Math** → **Math setup** on the menu bar.
2. Touch the tab for the function (F1 to FxThe number of math traces available depends on the software options loaded on your scope. See Specifications.) you want to assign the Interpolate operation to.
3. Touch inside the **Source1** field and select an input waveform.
4. Touch inside the **Operator1** field, then touch the **Filter** button in the **Select Math Operator** menu.

5. Select **Interpolate** from the Filter submenu.
6. Touch the **Interpolate** tab in the mini setup dialog to the right of the main dialog.
7. Touch inside the **Algorithm** field and select an interpolation type.
8. Touch inside the **Upsample by** (Upsampling is the factor by which sampling is increased) field and enter a value, using the pop-up numeric keypad, if you want to enter a specific value. Otherwise, use the **Up/Down** buttons to increment the displayed value in a 1-2-5 sequence.

Demodulation

The DMOD math function adds seven new processing functions:

- **Amplitude Demodulation** - For a given bandwidth, the instantaneous magnitude or amplitude of a specific carrier frequency sinusoid.
- **Complex Demodulation** - For a given bandwidth, the instantaneous complex vector (Cartesian) of a specific carrier frequency sinusoid.
- **Frequency Demodulation** - This is based on the phase of the sequential vectors compared with the expected phase since the last vector (the derivative of the phase error) or the rate of phase movement (in Hz).
- **Imaginary Component Demodulation** - “Quadrature-phase” component "Q".
- **Phase Demodulation** - This is based on the phase of sequential vectors, as compared to the “expected ideal” phase. Closely related to Time Interval Error, but with a change in sign since positive phase error implies early edge arrival (negative TIE).
- **Real Component Demodulation** - the “In-phase” component "I".
- **Wideband AM** - Wideband amplitude demodulation

These demodulated waveforms are useful for many purposes:

- Analyzing FM or PM modulated signals
- Studying unwanted variations in any of these domains (frequency or phase “noise” analysis)
- Studying unwanted systematic “pickup” or interference on either transmitted RF, or on oscillators, phase-locked loops etc.
- Studying instantaneous power
- Studying envelopes of amplitude modulated signals with variable bandwidth.

Only two of the result types are rescalable, Frequency (FM) and Phase (PM), since the magnitudes of these projections cannot be known beforehand. All of the other results are scaled on the vertical scale of the original digitized carrier waveform.

There are no parameters specific to the Demodulation math function.

Theory of Operation

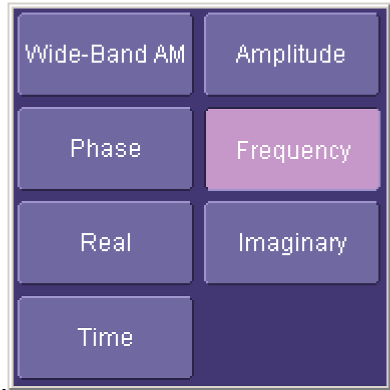
The basic procedure is a localized (by windowing) complex finite impulse response (FIR), which amounts to no less than a local (windowed) discrete complex Fourier “term” (or sum) at the carrier frequency.

The length of the complex FIR is determined by the selection of Bandwidth. The result of each convolution with the FIR at a given time offset is then translated into a single complex vector, defining the instantaneous phase and amplitude of the modulated carrier. The user selects one of several “views” on this vector (including its amplitude, phase, rate of change, real, imaginary, or complex). The appropriate values from the vector constitute the samples of the resulting processed waveform.

Using the same window and bandwidth criteria, there is also a “wide-band” AM demodulation function. This function doesn’t produce a complex vector for each data point of the resulting function, but simply uses the instantaneous rms signal to produce a scaled output. Basically there is no complex FIR, only the window. The scaling is such as to show (for practically any carrier frequency) an envelope of the same magnitude as the sinusoid.

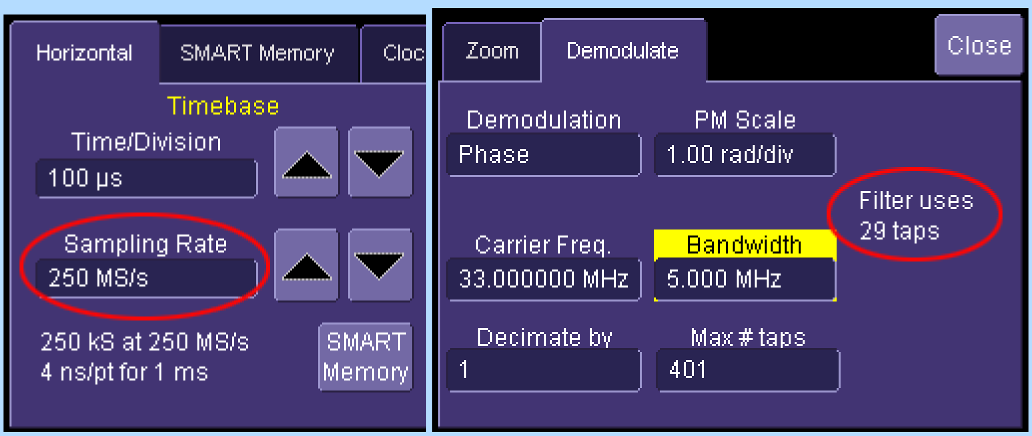
Setting Up Demodulation

1. Touch **Math** → **Math Setup** on the menu bar.
2. Touch a Math function tab **F1** to **F8**.
3. Touch inside the **Source1** field and select an input waveform from the pop-up menu.
4. Touch inside the **Operator1** field and select **Demodulate** from the pop-up menu.
5. Touch the **Demodulate** tab in the right-hand dialog.



6. Touch inside the **Demodulation** field and select a modulation mode.
7. Touch inside the **Carrier Freq.** field and enter the frequency of the carrier wave, using the pop-up keypad.
8. Touch inside the **Decimate by** field and enter a value by which to "thin out" your input waveform. Decimation will speed up processing.
9. If you selected **Phase** or **Frequency** modulation from the **Demodulation** menu, touch inside the **Scale** field and enter a value to rescale the waveform to. The units for phase modulation (PM) will be in radians per division, and Hz per division for frequency modulation (FM).
10. Touch inside the **Bandwidth** field and enter the bandwidth of your source waveform.
11. Touch inside the **Max # taps** field and enter a value for the maximum allowable filter taps.

Note: The actual number of taps used by the filter depends on the sampling rate and bandwidth selected. With the sampling rate held constant, the number of filter taps decreases as the bandwidth increases. With the bandwidth held constant, the number of filter taps increases as sampling rate increases.



Fast Wave Port Introduction

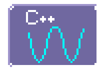
Fast Wave Port is a processing function for the LeCroy X-Stream family of digital oscilloscopes that enables you to insert your own custom processing algorithm, written in the C/C++ language, into the DSO's processing stream. Fast Wave Port maximizes data throughput from the acquisition system to your processing function. It also makes it simple to create these custom processing functions.

The technology that makes this system possible is the ability of two processes in a Windows system to share a region of memory. This enables the transfer of data at high-speed between the acquisition software and the custom processing function, which runs in a separate process from the DSO application. A major benefit of Fast Wave Port is that your application may be implemented and, more importantly, debugged independently of the main application.

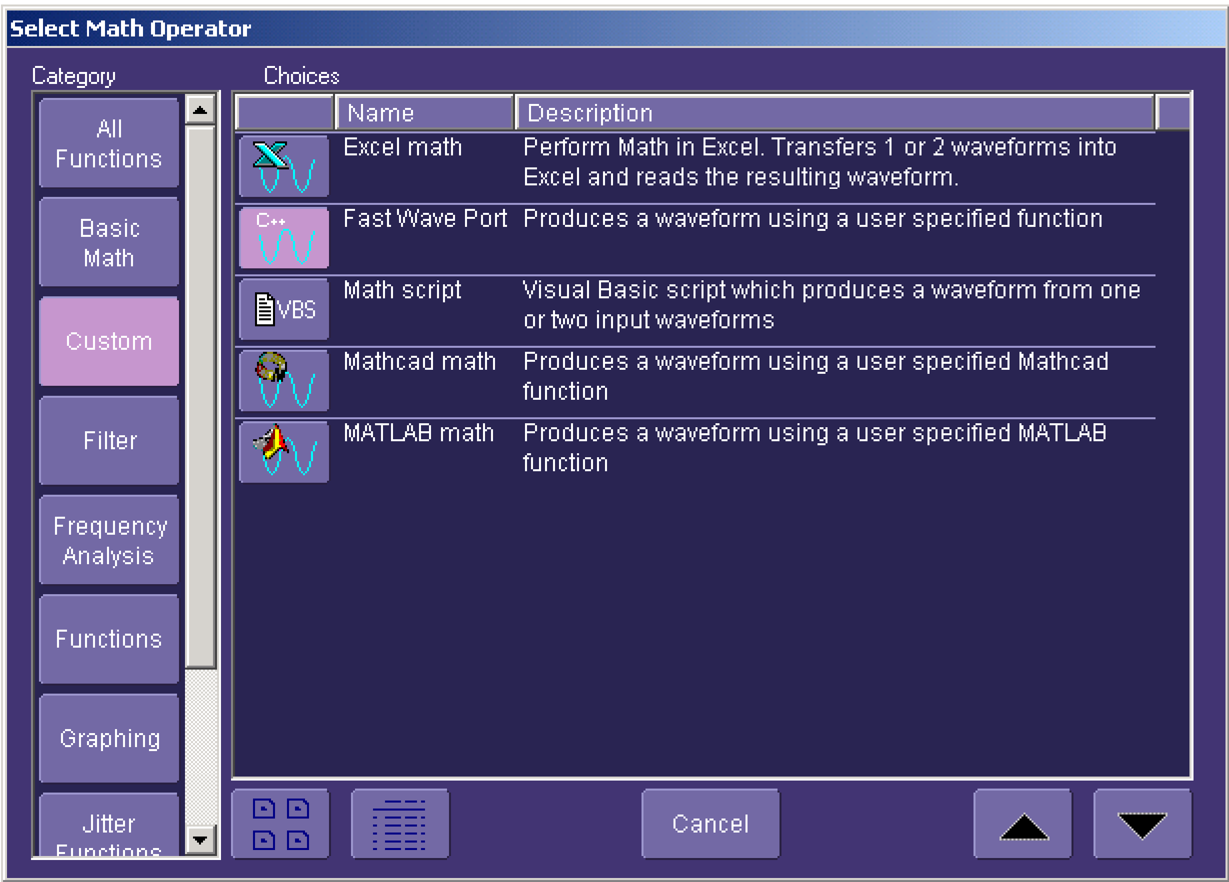
It is important to note that the transfer of the results of your processing function back into the X-Stream processing stream is optional. If performance is the primary goal, and display or further processing of the results within the DSO software is not required, then this may be skipped.

Fast Wave Port Setup

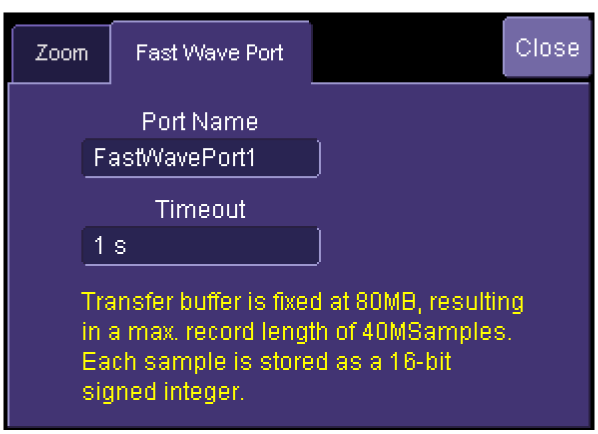
1. Touch **Math** → **Math Setup** on the menu bar.
2. Touch one of the Math function tabs, 

for example.
3. Touch inside the **Source1** field, and select a signal source from the pop-up menu. The source can be a channel waveform, math or memory trace, or a parameter.
4. Touch inside the **Operator1** field and select **FastWavePort** 

from the **Custom** menu:



5. In the right-hand mini-dialog, touch the **Fast Wave Port** tab:



6. Touch inside the "Timeout" field and enter a suitable value.

Setup - Case 1

This scenario assumes that you have developed your application on a PC.

1. Compile your application on your PC
2. Copy the compiled file onto the scope, using a memory stick or network drive.
3. Open the Command Prompt window (Start --> Programs --> Accessories --> Command Prompt) and run your application.

Setup - Case 2

This scenario assumes that you have Visual C++ loaded on the scope.

- Use the Visual C++ editor to develop and run your application.

Setup - Case 3

This scenario assumes that you are using a compiler other than Visual C++ (such as GNU's MinGW). It should be noted that the optimizer in the GNU C Compiler (GCC) is less efficient than that in Visual C++ and will result in lower performance.

1. Save your application in a text file, and copy it onto memory stick or network drive.
2. Using Windows Explorer, copy the text file to the scope.
3. Download and install the compiler onto the scope.
4. Configure Environmental Variables as follows:

   A. Open Start --> Settings --> Control Panel --> System.

   B. Click the "Advanced" tab, then the **Environmental Variables** button.

   C. In the "System variables" window, click **Path**, then the **Edit** button.

   D. At the end of the "Variable Value" string, append **;C:\MinGW\bin** for the case of the GNU C Compiler (GCC) for example.

   E. Click OK.

5. Open the Command Prompt window (Start --> Programs --> Accessories --> Command Prompt) and compile your application.
6. Run your application.

Fast Wave Port Operational Notes

Once FastWavePort is selected, the right-hand dialog shows the current settings. The first of these is critical, and indicates the base name of the memory window and the two events, which are global within the Windows O/S. This should be left at its default value and only changed if multiple FastWavePort functions are used in parallel. Note that this name must match the base name used in the client application.

The full names of these global objects are:

Object	Full Name
Memory Mapped File	"FastWavePort1File"
Data Available Event	"FastWavePort1MutexDataAvailable"
Processing Complete Event	"FastWavePort1MutexProcessingComplete"

The "Timeout" field specifies the amount of time that the DSO will wait for the custom processing function to complete. This prevents the DSO from waiting indefinitely for a potentially unforthcoming custom processing function. Be careful to set this value to something reasonable, which means a time that is longer by a reasonable margin than the custom processing is ever expected to take.

Data Length Limitations

The size of the memory window is fixed at 80 Mbytes, which equates to 40 MSamples.

Performance

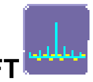
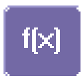
This is by far the fastest way to process data using a user-defined algorithm on an X-Stream DSO.

Choice of Programming Language

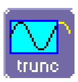
The system was designed for use with the C/C++ programming language. However, it is theoretically possible for the processing to be implemented in any language that supports Windows named events (Mutex) and can open a named memory-mapped file. No guarantee can be given, however, as to the behavior of the system using anything but C/C++.

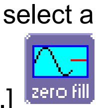
FFT Setup

Setting Up an FFT

1. Touch **Math** → **Math Setup** on the menu bar.
2. Touch a Math function trace button: **F1** through **Fx**[The number of math traces available depends on the software options loaded on your scope. See Specifications.] a pop-up menu appears. Select **FFT**  from the menu.
3. Touch the **Single** 

 or **Dual** (function of a function) 

 button if the FFT is to be of the result of another math operation.
4. Touch inside the **Source1** field and select a channel, memory, or math trace on which to perform the FFT.
5. Touch inside the **Operator1** field:
   Select **FFT** from the pop-up menu if you selected **Single** function.
   Select another math function if you selected **Dual** function. Then touch inside the **Operator2** field and select **FFT** from the pop-up menu.
6. In the right-hand dialog, touch the **FFT** tab.
7. Choose whether to <u>Truncate</u>[When the FFT transform size does not match the record length, you can truncate the record and perform an FFT on the shorter record. This will increase the resolution bandwidth.] 

 or Zero-fill[Zero-fill is useful when the source data for the FFT comes from a math operation that shortens the record. This is commonly encountered in filtering operations like enhanced resolution. The missing data points are replaced by data values, whose amplitudes are interpolated to fit between the last data point and the first data point in the record. This guarantees that there is not a first-order discontinuity in the filled data. Since the data at the end of the record is "filled" data, it is advisable to select a weighting window other than rectangular to minimize the effect of the fill on the resulting spectrum.] 

 the trace display.
8. Touch the **Suppress DC** checkbox if you want to make the DC bin go to zero. Otherwise, leave it unchecked.
9. Touch inside the **Output** type field, and make a selection from the pop-up menu.
10. Touch inside the **Window** field, select a window type.
11. Touch inside the **Algorithm** field and select either <u>Least Prime</u>[The default algorithm is a least primes algorithm that computes FFTs on transform sizes having lengths that can be expressed as factors of 2^N*5^K. This is very compatible with the record lengths encountered in the oscilloscope, which are often multiples of 1, 2, 4, 5, or 10.] or <u>Power of 2</u>[The other choice is a power of two algorithm where the record lengths are in the form of 2^N. The power of 2 algorithm generally runs faster than the least primes algorithm. The price that is paid is a record length that is not the same as the acquired signal. The power-of-two FFT will truncate to the nearest power-of-two less than record length if truncate is chosen or fill data to nearest power-of-two greater than the record length if zero fill is selected.] from the pop-up menu.

Why Use FFT?

For a large class of signals, you can gain greater insight by looking at spectral representation rather than time description. Signals encountered in the frequency response of amplifiers, oscillator phase noise and those in mechanical vibration analysis, for example, are easier to observe in the frequency domain.

If sampling is done at a rate fast enough to faithfully approximate the original waveform (usually five times the highest frequency component in the signal), the resulting discrete data series will uniquely describe the analog signal. This is of particular value when dealing with transient signals because, unlike FFT, conventional swept spectrum analyzers cannot handle them.

Spectral analysis theory assumes that the signal for transformation is of infinite duration. Since no physical signal can meet this condition, a useful assumption for reconciling theory and practice is to view the signal as consisting of an infinite series of replicas of itself. These replicas are multiplied by a rectangular window (the display grid) that is zero outside of the observation grid.

An FFT operation on an N-point time domain signal can be compared to passing the signal through a comb filter consisting of a bank of N/2 filters. All the filters have the same shape and width and are centered at N/2 discrete frequencies. Each filter collects the signal energy that falls into the immediate neighborhood of its center frequency. Thus it can be said that there are N/2 frequency bins. The distance in Hz between the center frequencies of two neighboring bins is always the same: Delta f.

Power (Density) Spectrum

Because of the linear scale used to show magnitudes, lower amplitude components are often hidden by larger components. In addition to the functions offering magnitude and phase representations, the FFT option offers power density and power spectrum density functions. These latter functions are even better suited for characterizing spectra. The power spectrum (V^2) is the square of the magnitude spectrum (0 dBm corresponds to voltage equivalent to 1 mW into 50 ohms.) This is the representation of choice for signals containing isolated peaks — periodic signals, for instance.

The power density spectrum (V^2/Hz) is the power spectrum divided by the equivalent noise bandwidth of the filter associated with the FFT calculation. This is best employed for characterizing broadband signals such as noise.

Memory for FFT

The amount of acquisition memory available will determine the maximum range (Nyquist frequency) over which signal components can be observed. Consider the problem of determining the length of the observation window and the size of the acquisition buffer if a Nyquist rate of 500 MHz and a resolution of 10 kHz are required. To obtain a resolution of 10 kHz, the acquisition time must be at least:

T = 1/Delta f = 1/10 kHz = 100 ms

For a digital oscilloscope with a memory of 100 kB, the highest frequency that can be analyzed is:

Delta f x N/2 = 10 kHz x 100 kB/2 = 500 MHz

FFT Pitfalls to Avoid

Take care to ensure that signals are correctly acquired: improper waveform positioning within the observation window produces a distorted spectrum. The most common distortions can be traced to insufficient sampling, edge discontinuities, windowing or the "picket fence" effect.

Because the FFT acts like a bank of band-pass filters centered at multiples of the frequency resolution, components that are not exact multiples of that frequency will fall within two consecutive filters. This results in an attenuation of the true amplitude of these components.

Picket Fence and Scallop

The highest point in the spectrum can be 3.92 dB lower when the source frequency is halfway between two discrete frequencies. This variation in spectrum magnitude is the picket fence effect. The corresponding attenuation loss is referred to as scallop loss. LeCroy scopes automatically correct for the scallop effect, ensuring that the magnitude of the spectra lines correspond to their true values in the time domain.

If a signal contains a frequency component above Nyquist, the spectrum will be aliased, meaning that the frequencies will be folded back and spurious. Spotting aliased frequencies is often difficult, as the aliases may ride on top of real harmonics. A simple way of checking is to modify the sample rate and observe whether the frequency distribution changes.

Leakage

FFT assumes that the signal contained within the time grid is replicated endlessly outside the observation window. Therefore if the signal contains discontinuities at its edges, pseudo-frequencies will appear in the spectral domain, distorting the real spectrum. When the start and end phase of the signal differ, the signal frequency falls within two frequency cells, broadening the spectrum.

The broadening of the base, stretching out in many neighboring bins, is termed leakage. Cures for this are to ensure that an integral number of periods is contained within the display grid or that no discontinuities appear at the edges. Another is to use a window function to smooth the edges of the signal.

Choosing a Window

The choice of a spectral window is dictated by the signal's characteristics. Weighting functions control the filter response shape, and affect noise bandwidth as well as side lobe levels. Ideally, the main lobe should be as narrow and flat as possible to effectively discriminate all spectral components, while all side lobes should be infinitely attenuated. The window type defines the bandwidth and shape of the equivalent filter to be used in the FFT processing.

In the same way as one would choose a particular camera lens for taking a picture, some experimenting is generally necessary to determine which window is most suitable. However, the following general guidelines should help.

Rectangular windows provide the highest frequency resolution and are thus useful for estimating the type of harmonics present in the signal. Because the rectangular window decays as a (sinx)/x function in the spectral domain, slight attenuation will be induced. Alternative functions with less attenuation (Flat Top and Blackman-Harris) provide maximum amplitude at the expense of frequency resolution. Whereas, Hamming and Von Hann are good for general purpose use with continuous waveforms.

Window Type	Applications and Limitations
Rectangular	These are normally used when the signal is transient (completely contained in the time-domain window) or known to have a fundamental frequency component that is an integer multiple of the fundamental frequency of the window. Signals other than these types will show varying amounts of spectral leakage and scallop loss, which can be corrected by selecting another type of window.
Hanning (Von Hann)	These reduce leakage and improve amplitude accuracy. However, frequency resolution is also reduced.
Hamming	These reduce leakage and improve amplitude accuracy. However, frequency resolution is also reduced.
Flat Top	This window provides excellent amplitude accuracy with moderate reduction of leakage, but with reduced frequency resolution.
Blackman–Harris	It reduces the leakage to a minimum, but with reduced frequency resolution.

FFT Window Filter Parameters				
Window Type	**Highest Side Lobe (dB)**	**Scallop Loss (dB)**	**ENBW (bins)**	**Coherent Gain (dB)**
Rectangular	-13	3.92	1.0	0.0
von Hann	-32	1.42	1.5	-6.02
Hamming	-43	1.78	1.37	-5.35
Flat Top	-44	0.01	2.96	-11.05
Blackman-Harris	-67	1.13	1.71	-7.53

Improving Dynamic Range

Enhanced resolution uses a low-pass filtering technique that can potentially provide for three additional bits (18 dB) if the signal noise is uniformly distributed (white). Low-pass filtering should be considered when high frequency components are irrelevant. A distinct advantage of this technique is that it works for both repetitive and transient signals. The SNR increase is conditioned by the cut-off frequency of the ERES low-pass filter and the noise shape (frequency distribution).

LeCroy digital oscilloscopes employ FIR digital filters so that a constant phase shift is maintained. The phase information is therefore not distorted by the filtering action.

Record Length

Because of its versatility, FFT analysis has become a popular analysis tool. However, some care must be taken with it. In most instances, incorrect positioning of the signal within the display grid will significantly alter the spectrum. Effects such as leakage and aliasing that distort the spectrum must be understood if meaningful conclusions are to be arrived at when using FFT.

An effective way to reduce these effects is to maximize the acquisition record length. Record length directly conditions the effective sampling rate of the scope and therefore determines the frequency resolution and span at which spectral analysis can be carried out.

FFT Algorithms

A summary of the algorithms used in the oscilloscope's FFT computation is given here in a few steps:

1. The data are multiplied by the selected window function.
2. FFT is computed, using a fast implementation of the DFT (Discrete Fourier Transform):

$$X_n = \frac{1}{N}\sum_{k=0}^{k=N-1} x_k \times W^{nk}$$

where: $x_k$is a complex array whose real part is the modified source time domain waveform, and whose imaginary part is 0; $X_n$is the resulting complex frequency-domain waveform; $W = e^{-2\pi j/N}$; and N is the number of points in x_k and X_n.

The generalized FFT algorithm, as implemented here, works on N, which need *not* be a power of 2.

3. The resulting complex vector X_n is divided by the coherent gain of the window function, in order to compensate for the loss of the signal energy due to windowing. This compensation provides accurate amplitude values for isolated spectrum peaks.
4. The real part of X_n is symmetric around the Nyquist frequency, that is

$R_n = R_{N-n}$

while the imaginary part is asymmetric, that is

$I_n = -I_{N-n}$

The energy of the signal at a frequency *n* is distributed equally between the first and the second halves of the spectrum; the energy at frequency 0 is completely contained in the 0 term.

The first half of the spectrum (Re, Im), from 0 to the Nyquist frequency is kept for further processing and doubled in amplitude:

$R'_n = 2 \times R_n$	$0 \le n < N/2$

$I'_n = 2 \times I_n$		$0 \le n < N/2$

5. The resultant waveform is computed for the spectrum type selected.

If "Magnitude" is selected, the magnitude of the complex vector is computed as:

$$M_n = \sqrt{R'^2_n + I'^2_n}$$

Steps 1–5 lead to the following result:

An AC sine wave of amplitude 1.0 V with an integral number of periods N_p in the time window, transformed with the rectangular window, results in a fundamental peak of 1.0 V magnitude in the spectrum at frequency N_p x Delta f. However, a DC component of 1.0 V, transformed with the rectangular window, results in a peak of 2.0 V magnitude at 0 Hz.

The waveforms for the other available spectrum types are computed as follows:

Phase: angle = arctan (I_n/R_n)	$M_n > M_{min}$

	angle = 0	 $M_n \le M_{min}$

Where M_{min} is the minimum magnitude, fixed at about 0.001 of the full scale at any gain setting, below which the angle is not well defined.

The dBm Power Spectrum:

$$dBm\ PS = 10 \times \log_{10}\left(\frac{M_n^2}{M_{ref}^2}\right) = 20 \times \log_{10}\left(\frac{M_n}{M_{ref}}\right)$$

where M_{ref}= 0.316 V (that is, 0 dBm is defined as a sine wave of 0.316 V peak or 0.224 V rms, giving 1.0 mW into 50 ohms).

The dBm Power Spectrum is the same as dBm Magnitude, as suggested in the above formula.

dBm Power Density:

$$dBm\ PD = dBm\ PS - 10 \times \log_{10}\ (ENBW \times \Delta f)$$

where *ENBW* is the equivalent noise bandwidth of the filter corresponding to the selected window, and *Delta f* is the current frequency resolution (bin width).

6. The FFT Power Average takes the complex frequency-domain data R'_n and I'_n for each spectrum generated in Step 5, and computes the square of the magnitude:

   $M_n^2 = R'^2_n + I'^2_n$,

   then sums M_n2 and counts the accumulated spectra. The total is normalized by the number of spectra and converted to the selected result type using the same formulas as are used for the Fourier Transform.

Glossary

This section defines the terms frequently used in FFT spectrum analysis and relates them to the oscilloscope.

Aliasing If the input signal to a sampling acquisition system contains components whose frequency is greater than the Nyquist frequency (half the sampling frequency), there will be less than two samples per signal period. The result is that the contribution of these components to the sampled waveform is indistinguishable from that of components below the Nyquist frequency. This is **aliasing** .

The timebase and transform size should be selected so that the resulting Nyquist frequency is higher than the highest significant component in the time-domain record.

Coherent Gain The normalized coherent gain of a filter corresponding to each window function is 1.0 (0 dB) for a rectangular window and less than 1.0 for other windows. It defines the loss of signal energy due to the multiplication by the window function. This loss is compensated for in the oscilloscope. The following table lists the values for the implemented windows.

Window Frequency Domain Parameters				
Window Type	**Highest Side Lobe (dB)**	**Scallop Loss (dB)**	**ENBW (bins)**	**Coherent Gain (dB)**
Rectangular	-13	3.92	1.0	0.0
Hanning (Von Hann)	-32	1.42	1.5	-6.02
Hamming	-43	1.78	1.37	-5.35
Flattop	-44	0.01	2.96	-11.05
Blackman–Harris	-67	1.13	1.71	-7.53

ENBW Equivalent Noise BandWidth (ENBW) is the bandwidth of a rectangular filter (same gain at the center frequency), equivalent to a filter associated with each frequency bin, which would collect the same power from a white noise signal. In the table on the previous page, the ENBW is listed for each window function implemented, given in bins.

Filters Computing an N-point FFT is equivalent to passing the time-domain input signal through N/2 filters and plotting their outputs against the frequency. The spacing of filters isDelta f = 1/T, while the bandwidth depends on the window function used (see Frequency Bins).

Frequency Bins The FFT algorithm takes a discrete source waveform, defined over N points, and computes N complex Fourier coefficients, which are interpreted as harmonic components of the input signal.

For a real source waveform (imaginary part equals 0), there are only N/2 independent harmonic components.

An FFT corresponds to analyzing the input signal with a bank of N/2 filters, all having the same shape and width, and centered at N/2 discrete frequencies. Each filter collects the signal energy that falls into the immediate neighborhood of its center frequency. Thus it can be said that there are N/2 "frequency bins."

The distance in hertz between the center frequencies of two neighboring bins is always:

Delta f = 1/T

where T is the duration of the time-domain record in seconds.

The width of the main lobe of the filter centered at each bin depends on the window function used. The rectangular window has a nominal width at 1.0 bin. Other windows have wider main lobes (see table).

Frequency RangeThe range of frequencies computed and displayed is 0 Hz (displayed at the left-hand edge of the screen) to the Nyquist frequency (at the rightmost edge of the trace).

Frequency Resolution In a simple sense, the frequency resolution is equal to the bin width Delta f. That is, if the input signal changes its frequency by Delta f, the corresponding spectrum peak will be displaced by Df. For smaller changes of frequency, only the shape of the peak will change.

However, the effective frequency resolution (that is, the ability to resolve two signals whose frequencies are almost the same) is further limited by the use of window functions. The ENBW value of all windows other than the rectangular is greater than Delta f and the bin width. The table of Window Frequency-Domain Parameters lists the ENBW values for the implemented windows.

Leakage In the power spectrum of a sine wave with an integral number of periods in the (rectangular) time window (that is, the source frequency equals one of the bin frequencies), the spectrum contains a sharp component whose value accurately reflects the source waveform's amplitude. For intermediate input frequencies this spectral component has a lower and broader peak.

The broadening of the base of the peak, stretching out into many neighboring bins, is termed *leakage*. It is due to the relatively high side lobes of the filter associated with each frequency bin.

The filter side lobes and the resulting leakage are reduced when one of the available window functions is applied. The best reduction is provided by the Blackman–Harris and Flattop windows. However, this reduction is offset by a broadening of the main lobe of the filter.

Number of Points TheFFT is computed over the number of points (Transform Size) whose upper bounds are the source number of points, and by the maximum number of points selected in the menu. The FFT generates spectra of N/2 output points.

Nyquist Frequency The Nyquist frequency is equal to one half of the effective sampling frequency (after the decimation): Delta f x N/2.

Picket Fence Effect If a sine wave has a whole number of periods in the time domain record, the power spectrum obtained with a rectangular window will have a sharp peak, corresponding exactly to the frequency and amplitude of the sine wave. Otherwise the spectrum peak with a rectangular window will be lower and broader.

The highest point in the power spectrum can be 3.92 dB lower (1.57 times) when the source frequency is halfway between two discrete bin frequencies. This variation of the spectrum magnitude is called the *picket fence effect* (the loss is called the scallop loss).

All window functions compensate for this loss to some extent, but the best compensation is obtained with the Flattop window.

Power Spectrum The power spectrum (V^2) is the square of the magnitude spectrum.

The power spectrum is displayed on the dBm scale, with 0 dBm corresponding to:

$$V_{ref}^2 = (0.316\ V_{peak})^2,$$

where V_{ref} is the peak value of the sinusoidal voltage, which is equivalent to 1 mW into 50 ohms.

Power Density Spectrum The power density spectrum (V^2/Hz) is the power spectrum divided by the equivalent noise bandwidth of the filter, in hertz. The power density spectrum is displayed on the dBm scale, with 0dBm corresponding to (V_{ref}^2/Hz).

Sampling Frequency The time-domain records are acquired at sampling frequencies dependent on the selected time base. Before the FFT computation, the time-domain record may be decimated. If the selected maximum number of points is lower than the source number of points, the effective sampling frequency is reduced. The effective sampling frequency equals twice the Nyquist frequency.

Scallop Loss This is loss associated with the picket fence effect.

Window Functions All available window functions belong to the sum of cosines family with one to three non-zero cosine terms:

$$W_k = \sum_{m=0}^{m=M-1} a_m \cos\left(\frac{2 p k}{N} m\right) \qquad 0 \leq k < N$$

where: $M = 3$ is the maximum number of terms, a_m are the coefficients of the terms, N is the number of points of the decimated source waveform, and k is the time index.

The table of Coefficients of Window Functions lists the coefficients a_m. The window functions seen in the time domain are symmetric around the point k = N/2.

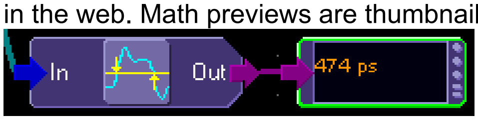
Coefficients of Window Functions			
Window Type	a_0	a_1	a_2
Rectangular	1.0	0.0	0.0
Hanning (Von Hann)	0.5	-0.5	0.0
Hamming	0.54	-0.46	0.0
Flattop	0.281	-0.521	0.198
Blackman–Harris	0.423	-0.497	0.079

Processing Web (XWEB)

The Processing Web provides a graphical way to quickly and easily set up math functions and parameter measurements. Using the Processing Web, you can chain together many more math-on-math functions than you can using the Math Setup dialog, where you are limited to two functions. In addition, you can insert a parameter measurement for any math output waveform anywhere in the web.

The "web" analogy derives from the nodes and connecting lines used to construct the web. Nodes are math functions selected from the **Add Math Processor** menu, parameters selected from the **Add Measure Processor** menu, or parameter math functions from the **Add Parameter Math Processor** menu.

Another key feature of the Processing Web is that you can preview your waveform at any math or parameter node in the web. Math previews are thumbnail images of the waveform. For parameters, the statistic



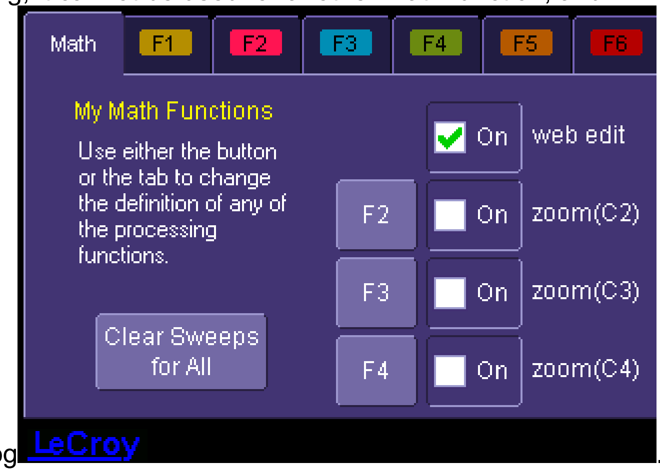
Statistic for Width Parameter displayed is the value of the last acquisition.

Once you have created a Processing Web setup, you can save and recall it for future use, the same as for any panel setup.

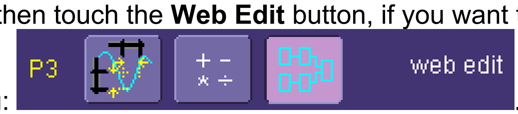
To Use the Web Editor

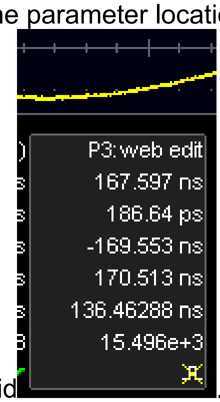
1. In the menu bar, touch **Display**, then Web Editor 

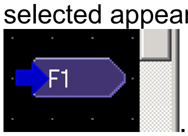
in the drop-down menu.
2. Touch the **Math** tab and select a math location (F1 to Fx[The number of math traces available depends on the software options loaded on your scope. See Specifications.]) for the new math function that you are about to create by touching the **Web Edit** button: 

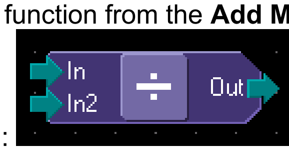
.
Once you select a math location for web editing, it cannot be used for another math function, and will appear as unavailable in the Math Setup dialog 

.
However, you can cancel web processing within the "Math Setup" dialog by touching the single function, double function, or graph button.

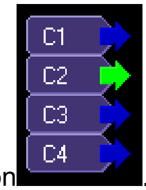
Touch the **Measure** tab, then touch the **Web Edit** button, if you want to dedicate a parameter location (P1 to Px) for web processing: 

.
The parameter location you choose will display "Web Edit" under the waveform display grid 

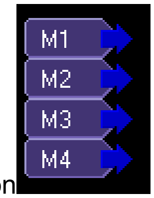
.
3. Touch the **Web Editor** tab to return to the web setup dialog. The math and parameter locations you selected appear as outputs at the far right:


.
You may have to scroll up or down to see it.
4. Touch the **Add Math** button and select a math function from the **Add Math Processor** menu. The math function icon will appear on the web setup field: 

.
Touch and drag the icon to the desired location.

5. If you are using channel inputs, touch the arrow of a channel input icon 

. Then drag a line from the channel to the input of the math function icon. If your math function is a dual input function (such as ratio), select a second input and drag another line to the second math input.

   If you are using a memory location 

(M1 to M4) as an input, drag a line to the math function in the same way as for channel inputs.

> **Note:** You can use a combination of channel input and memory input to your math function.

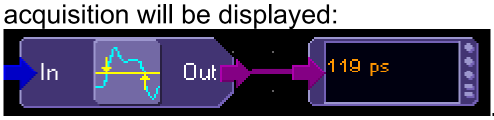
6. Touch the output arrow of the math function icon and drag a line to the Fx output on the right-hand side of the setup field. Your math function is complete.

Adding Parameters

Add parameter measurements in the same way as for math functions. Parameters can be connected to any math function in the web.

Adding Previews

1. Touch the **Add Preview** button: 

. A scope-like icon will appear: .
2. Touch the output arrow on the math function or parameter icon and drag a line to the input arrow of the preview icon. A thumbnail view of your signal will appear if the preview icon is connected to a channel output or math function output. If it is connected to a parameter output, a numeric value of the last acquisition will be displayed:


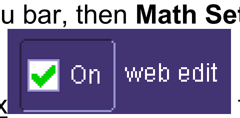
.

Exiting the Web Editor

To exit, touch the **Close** tab; or, in the menu bar, touch **Display** then Scope Display 

in the drop-down menu. The scope display will return to the normal waveform display grid.

Viewing the Output

1. Touch **Math** in the menu bar, then **Math Setup...** in the drop-down menu.
2. Touch the **On** checkbox 

for the function you want to view.

Measuring with Cursors

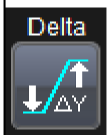
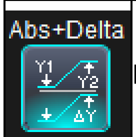
Cursors are important tools that aid you in measuring signal values. Cursors are markers — lines, cross-hairs, or arrows — that you can move around the grid or the waveform itself. Use cursors to make fast, accurate measurements and to eliminate guesswork. There are two basic types:

- **Horiz(ontal)** (generally Time or Frequency) cursors are markers that you move horizontally along the waveform. Place them at a desired location along the time axis to read the signal's amplitude at the selected time.
- **Vert(ical)** (Voltage) cursors are lines that you move vertically on the grid to measure the amplitude of a signal.

Cursor Measurement Icons

Note: Not all of these measure modes are available on all scope models.

Each cursor locates a point on the waveform. The cursor values can be read in the descriptor label for the trace. Use the **Position** data entry fields at the right side of the dialog to place the cursors precisely. The **Readout** icons depict what is being measured for each measurement mode.

Icon:	Description:
	Displays the absolute Y position.
	Displays the difference in Y values. The value can be read in the descriptor label for the trace.
	Displays absolute and delta cursors together.
	Displays the slope between cursors.

If there are non–time-domain waveforms displayed, there will also be a menu offering choices of x-axis units: **s** or **Hz**, for example.

Cursors Setup

Note: Setup procedures will vary depending on the measure modes available in your scope model.

Quick Display

At any time, you can change the display of cursor types (or turn them off) without invoking the **Cursors Setup** dialog as follows:

1. On the menu bar, touch **Cursors**, then **Off**, **Abs Horizontal**, **Rel Horizontal**, **Abs Vertical**, or **Rel Vertical**.

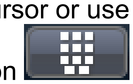
   OR

   Press the CURSOR TYPE button on the front panel repeatedly to scroll through each cursor type. Turn the CURSORS knobs to adjust the position of the cursor.

Note: The bottom cursor knob adjusts relative cursors. It does not work with absolute cursor types.

2. The cursors displayed will assume the positions previously set up. If you want to change their position or measurement mode, in the menu bar touch **Cursors**, then **Cursors Setup** in the drop-down menu.

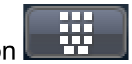
Setting Up Absolute Cursors

1. Touch **Cursors** → **Cursors Setup** on the menu bar. The **Standard Cursors** dialog opens.
2. In the dialog area, touch the **Cursors On** check box to display them.
3. Touch the **Horizontal Abs** or **Vertical Abs** mode button.
4. Touch inside the **Position 1** data entry field and type in a value for the cursor or use the slider bar at the bottom of the window to adjust the cursor position. Click the keypad button  on the slider bar to enter a value using the pop-up keypad.

   OR

   Use the CURSORS knobs on the front panel to place the cursor.

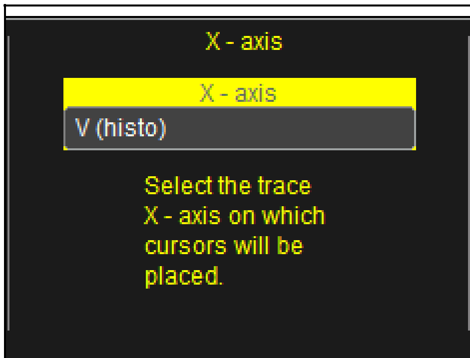
Setting Up Relative Cursors

1. Touch **Cursors** → **Cursors Setup** on the menu bar. The **Standard Cursors** dialog opens.
2. In the dialog area, touch the **Cursors On** check box to display them.
3. Touch the **Horizontal Rel** or **Vertical Rel** mode button.
4. Touch a readout parameter button: **Absolute**, **Delta**, **Abs+Delta**, or **Slope**.
5. Touch inside the **Position 1** and **Position 2** data entry fields and type in a value for each cursor or use the slider bar at the bottom of the window to adjust the cursor positions. Click the keypad button  on the slider bar to enter a value using the pop-up keypad.

   OR

   Use the CURSORS knobs on the front panel to place the cursors.
6. If you want both cursors to move in unison as you adjust the position, touch the **Track** check box to enable tracking.

Cursors on Math Functions



You can place cursors on a math function whose X-axis has a dimension other than time (FFT, for example). On the **Cursors Setup** dialog, an **X-Axis** menu will become available from which to choose the units. For example, if you are placing cursors on a histogram, select **V (histo)**.

Overview of Parameters

Parameters are measurement tools that determine a wide range of waveform properties. Use them to automatically calculate many attributes of your waveform, like rise-time, rms voltage, and peak-to-peak voltage, for example.

There are parameter modes for the amplitude and time domains, custom parameter groups, and parameters for pass and fail testing. You can make common measurements on one or more waveforms.

Turning On Parameters

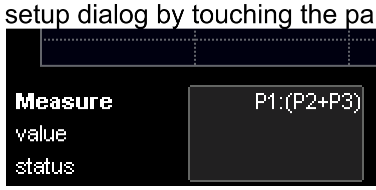
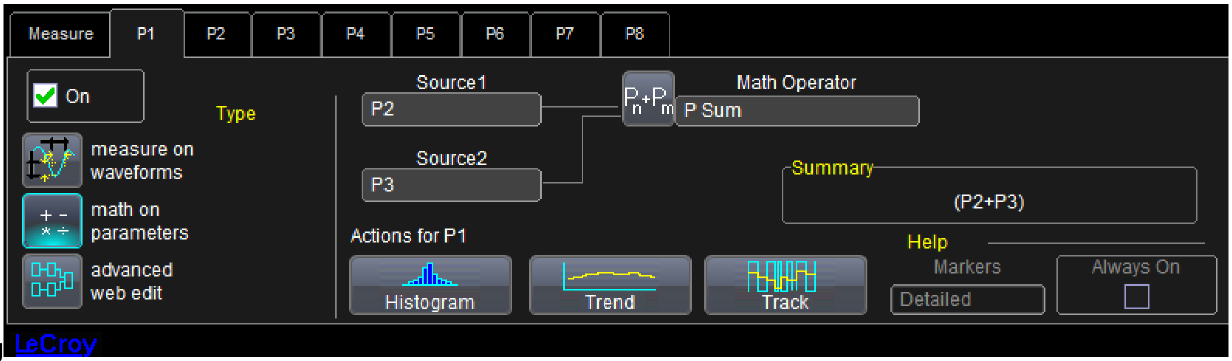
1. Touch **Measure** → **Measure Setup** on the menu bar.
2. Touch the **On** checkbox for each parameter you want to enable.

Note: Some instruments do not have this option and will display all parameters together.

3. Touch the **Show Table** checkbox to display the parameters below the grid.

Quick Access to Parameter Setup Dialogs

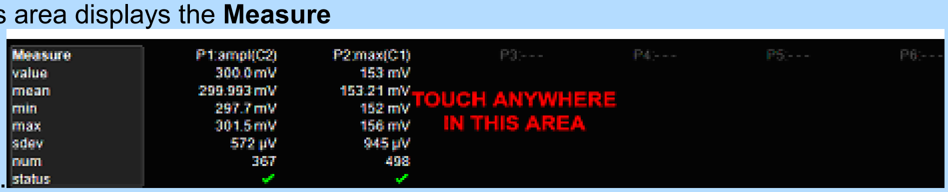
You can quickly gain access to a parameter setup dialog by touching the parameter list box below the grid.

For example, touch within **P1** below the grid  to display the setup dialog 

for P1.

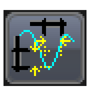
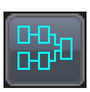
Touch the row titles 

to display the top **Measure** dialog.

Note: Some instruments have only one **Measure** dialog. In this case, touching anywhere in the parameter results area displays the **Measure** dialog. 

Parameter Setup

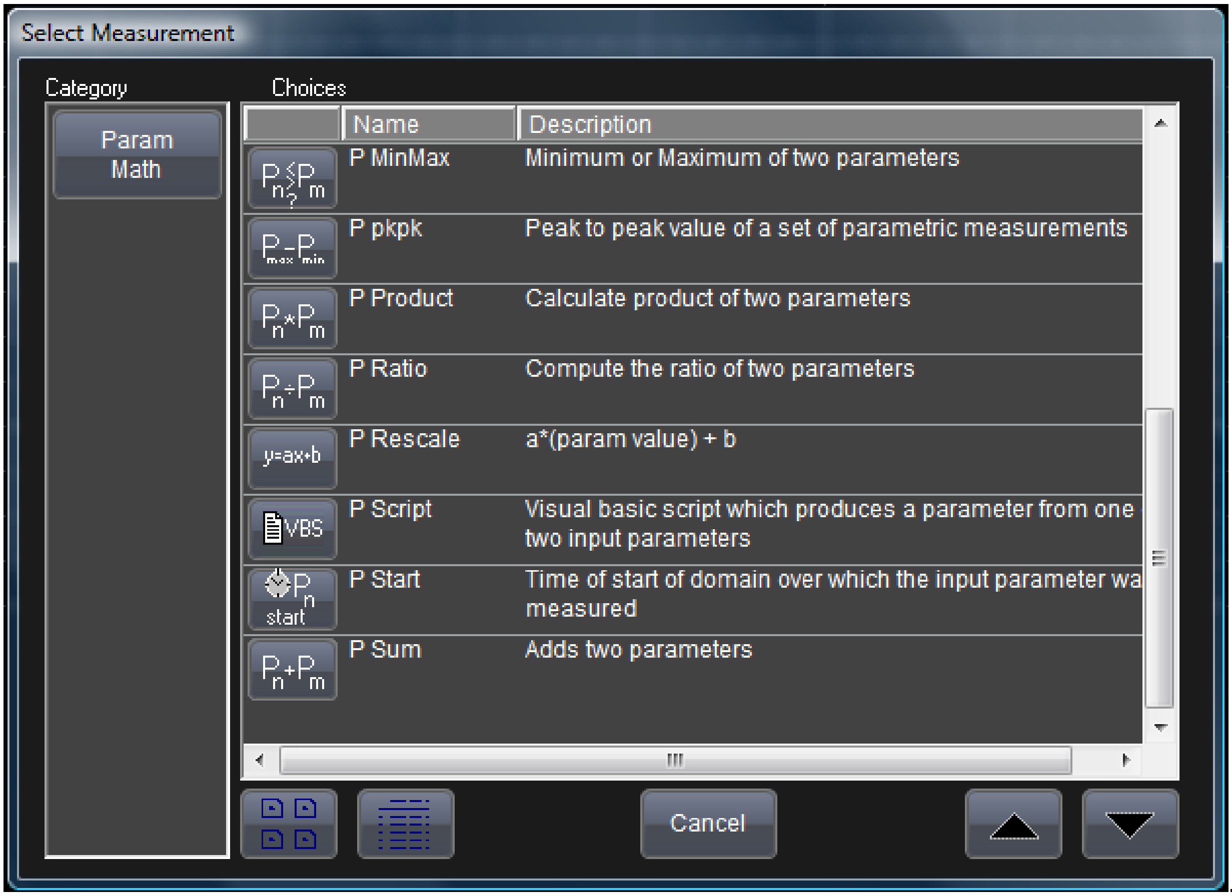
1. Touch the parameter tab (**Px**) of an unused location or one that you want to change.
2. Under **Type**, select a measurement type:

	**Measure On Waveforms** - Applies measurements directly on the waveform denoted in the **Source1** field.
	**Math On Parameters** - Performs math (addition, subtraction, multiplication, division) on parameters denoted in the **Source1** and **Source2** fields. This feature is available with some optional math packages.
	**Advanced Web Edit** - Uses LeCroy's Processing Web to set up the measurement. This feature, which is available with the XWEB option, allows you to chain practically unlimited math functions together to operate on your waveform measurements.

3. Touch inside the **Source1** field and select a channel, math trace, memory trace, or other waveform to be measured.
4. If you selected **Measure On Waveforms**, touch inside the **Measure** field and select a parameter from the pop-up menu

OR

If you selected **Math On Parameters**, touch inside the **Math Operator** data entry field and select a math function from the pop-up menu

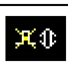


5. In the mini-dialog to the right of the main setup dialog, touch the Gate tab to narrow the span of measurement.
6. Touch the Accept tab to define parameter values to be used in the measurement.

Parameter Status

Status Symbols

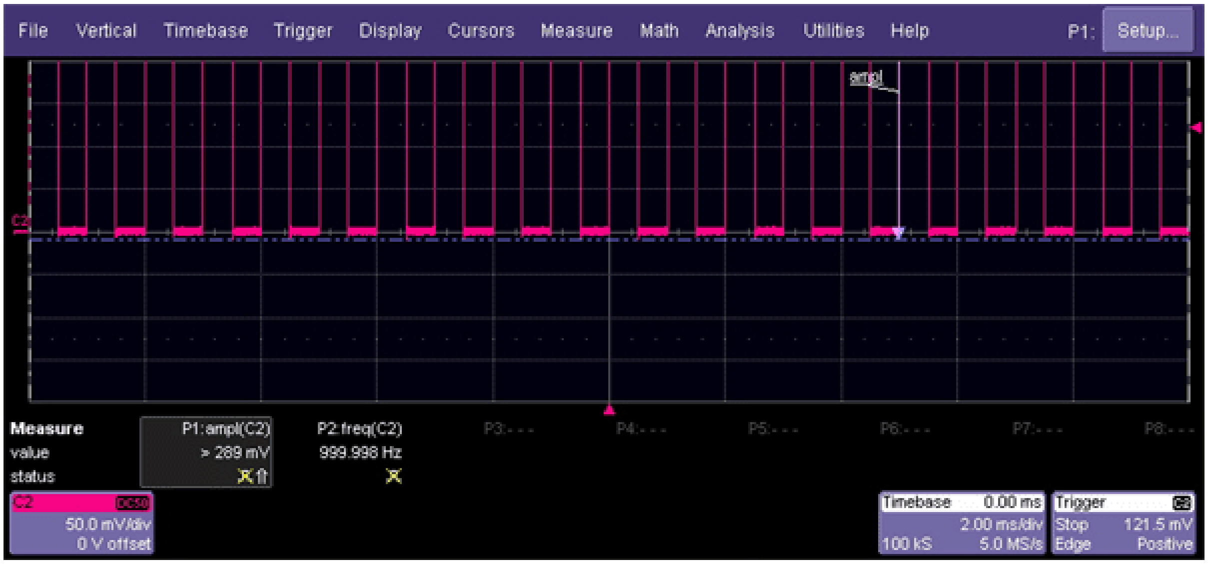
Below each parameter appears a symbol that indicates the status of the parameter, as follows:

Symbol:	Description:
	A warning symbol indicates that there is something wrong with the signal or the setup. Touch the parameter list box and read the explanation in the message line at the bottom of the screen.
	A green check mark means that the scope is returning a valid value.
	A crossed-out pulse means that the scope is unable to determine top and base; however, the measurement could still be valid.
	A downward pointing arrow indicates an underflow condition.
	An upward pointing arrow indicates an overflow condition.
	An upward-and-downward pointing arrow indicates an underflow and overflow condition.

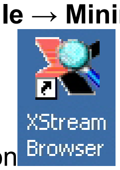
Using X-Stream Browser to Obtain Status Information

Example:

Here is a case of an overflow condition, in which the amplitude of the waveform cannot be determined:

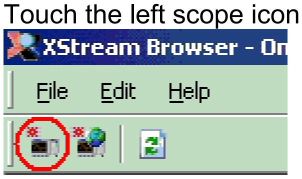


1. Minimize the scope display by selecting **File** → **Minimize**.

2. Touch the **X-Stream Browser** desktop icon 

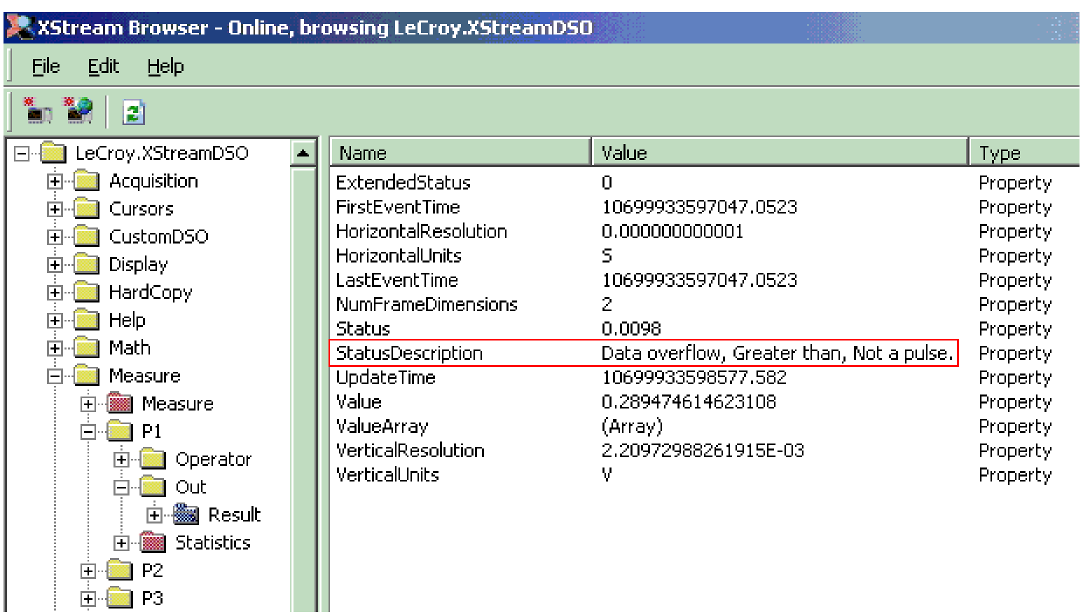
to open the browser.

3. Touch the left scope icon ("Connect to a local X-Stream DSO device") in the X-Stream Browser toolbar:



4. Select

Measure
Parameter in error (P1)
Out
Result



5. Read the status information in line **StatusDescription**.

Statistics

Touch the **Statistics On** checkbox in the **Measure** dialog to display statistics for standard vertical or horizontal parameters, or for custom parameters. The statistics that are displayed are as follows:

Statistics:
value (last)
mean
min.
max.
sdev
num

The value displayed in the **num** row is the number of measurements computed. For any parameter that computes on an entire waveform (like edge@level, mean, minimum, maximum, etc.) the value displayed represents the number of sweeps.

For any parameter that computes on every event, the value displayed is equal to the number of events per acquired waveform. If *x* waveforms were acquired, the value represents *x* times the number of cycles per waveform. Also, the "value" is equal to the measurement of the last cycle on the last acquisition.

Measure Modes

The selections for Measure Mode allow you to quickly apply parameters for standard vertical and standard horizontal setups, and custom setups.

Standard Vertical Parameters

These are the default Standard Vertical Parameters:

Vertical
mean
sdev
max.
min.
ampl
pkpk
top
base

Standard Horizontal Parameters

These are the default Standard Horizontal Parameters:

Horizontal
freq
period
width
rise
fall
delay
duty
num points

Selecting Measure Modes

1. Touch **Measure** → **Measure Setup** on the menu bar.
2. Under **Measure Mode**, select the **Std Vertical** or **Std Horizontal** buttons.

Note: You can choose to customize up to 12 parameters by touching **My Measure**.

3. Touch inside the **Source** data entry field and choose a source for which the measurements should apply.

Parameter Math

The instrument gives you the ability to perform arithmetic operations (addition, subtraction, multiplication, division) on the results of two parameter measurements. Alternatively, you can apply math to a single parameter (for example, invert). By customizing parameters in this way, you can effectively extend the range of parameter measurements based on your particular needs.

Logarithmic Parameters

The parameter math feature prevents multiplication and division of parameters that return logarithmic values. These parameters are as follows:

- auto-correlation signal-to-noise ratio (ACSN)
- narrow-band power (NBPW)
- media signal-to-noise ratio (MSNR)
- residual signal-to-noise ratio (RSNR)

- top-to-base ratio when the units are in dB (TBR)

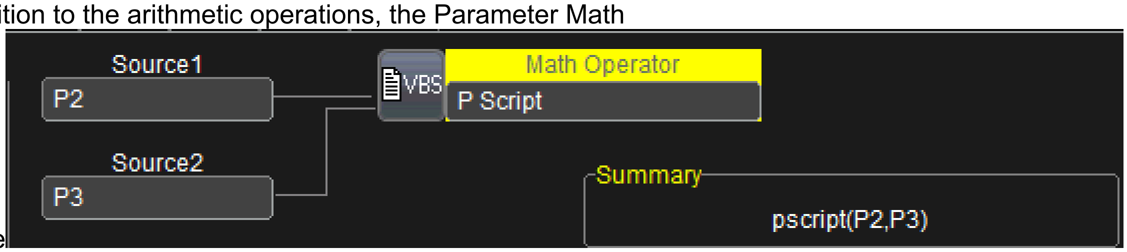
Excluded Parameters

Parameters that are already the result of parameter math operations are excluded. If they are included in a remote control setup command, an error message is generated and the setup canceled.

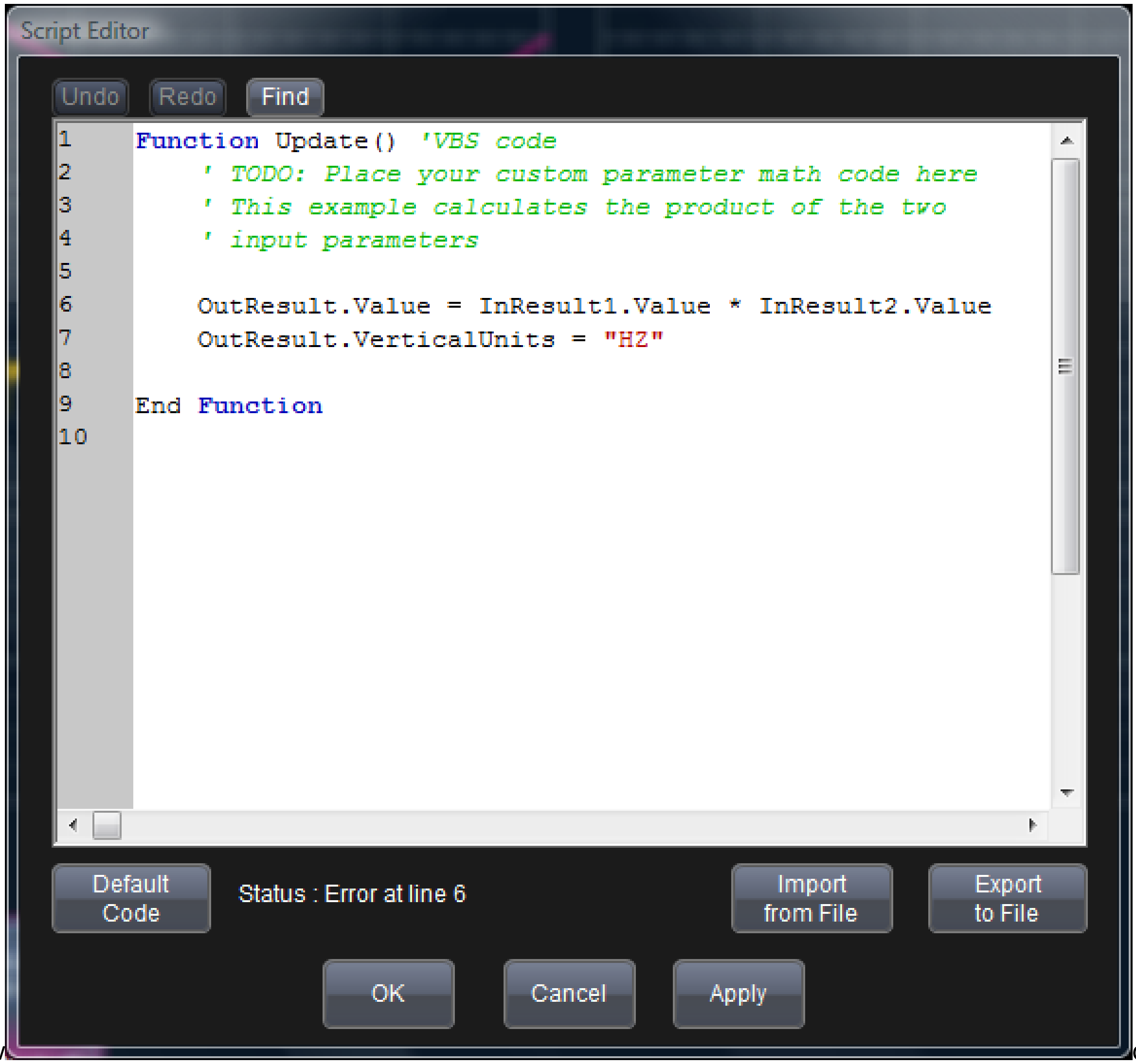
- Excluded parameters are as follows:
- delta clock-to-data near (DC2D)
- delta clock-to-data next (DC2DPOS)
- delta clock-to-data previous (DC2DNEG)
- delta delay (DDLY)
- delta time at level (DTLEV)
- phase (PHASE)
- resolution (RES)
- mTnTmT shift (BEES)
- mTnTmT shift sigma (BEESS)
- mTnTmT shift sigma – list (BEESS)

Parameter Script Parameter Math

In addition to the arithmetic operations, the Parameter Math feature



allows you to use VBScript or JavaScript to write your own script for one or two measurements and produce a result that suits your needs. Code entry is done in the Script Editor

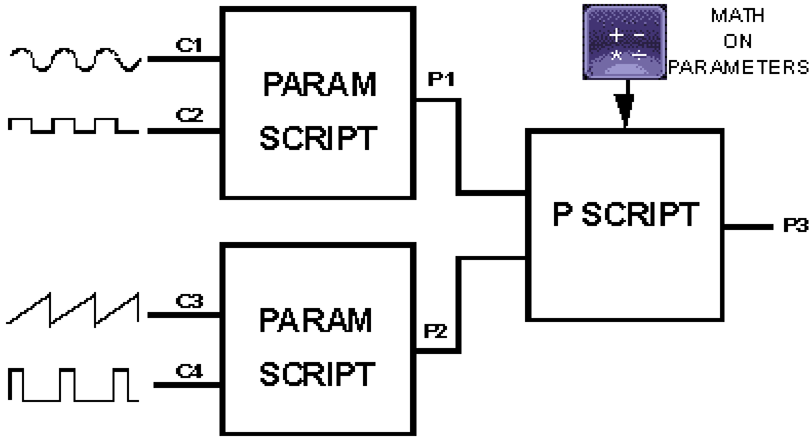


window directly on the instrument. You can also import an existing script.

Param Script vs. P Script

Param Script is a VBScript or JavaScript that operates on one or two *waveforms* and outputs a parameter measurement, as shown in the figure below. P Script, on the other hand, is another VBScript or JavaScript that takes as input one or two *parameters* and performs a math operation on them to produce another parameter output.

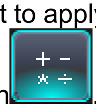
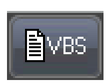
The inputs to Param Script can also be math (F1-Fx) or memory (M1-Mx) traces. The inputs to P Script can be the results of any parameter measurement, not necessarily Param Script.



Setting Up Parameter Math

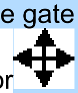
1. Touch **Measure** → **Measure Setup** on the menu bar.
2. Touch the **My Measure** button in the **Measure** dialog.
3. Touch the **Px** tab for the parameter to which you want to apply parameter math.
4. In the **Px** dialog, touch the math on parameters button . The **Source** field expands to two fields.
5. Touch inside the **Source1** and **Source2** fields and select the parameters you want to apply math to (P1 to Px). If you are applying math to a single parameter (for example, invert), just touch inside the **Source1** field and select a parameter (P1 to Px).
6. Touch inside the **Math Operator** field and select a math operation from the **Select Measurement** menu. If you select an operation that requires two input parameters, the **Source** field will expand to two fields.

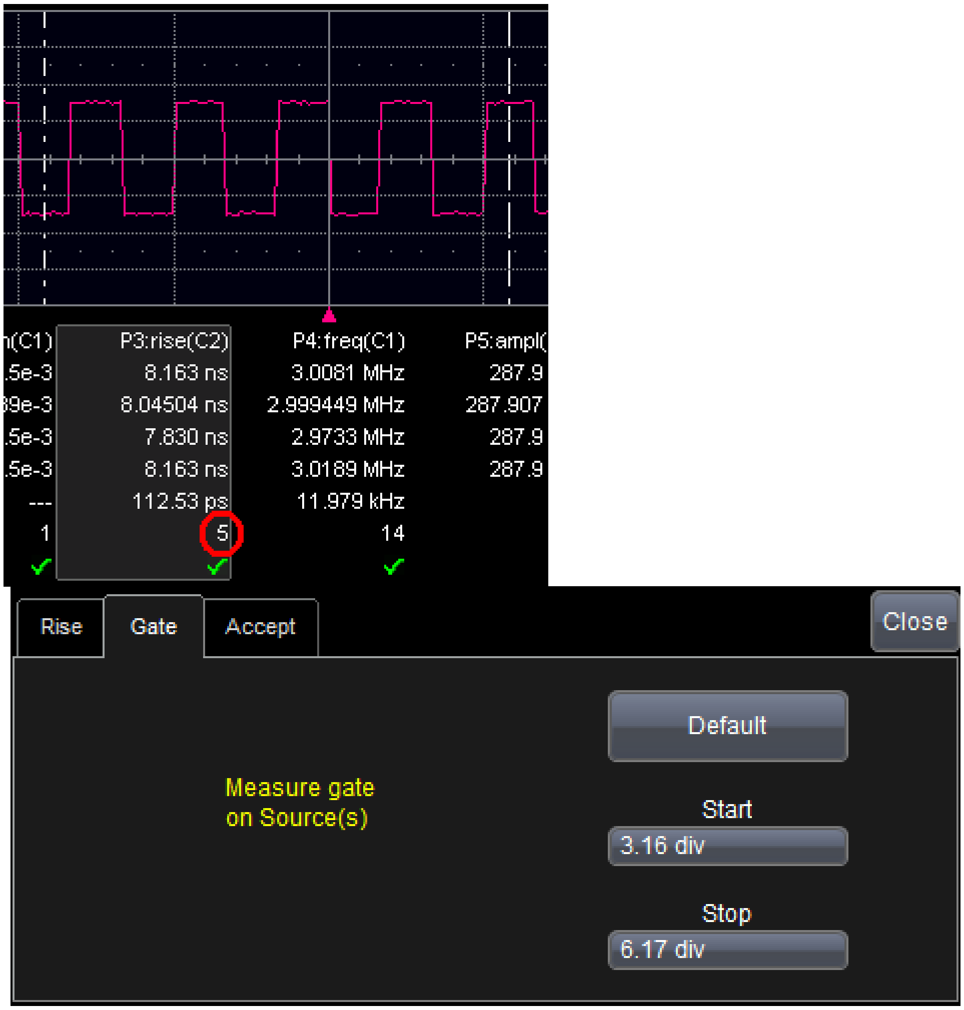
Setting Up Parameter Script Math

1. Touch **Measure** → **Measure Setup** on the menu bar.
2. Touch the **My Measure** button in the **Measure** dialog.
3. Touch the **Px** tab for the parameter to which you want to apply parameter math.
4. In the **Px** dialog, touch the math on parameters button . The **Source** field expands to two fields.
5. Touch inside the **Source1** and **Source2** fields and select the parameters you want to apply math to (P1 to Px). If you are applying math to a single parameter (for example, invert), just touch inside the **Source1** field and select a parameter (P1 to Px).
6. Touch inside the **Math Operator** field and select P Script  from the **Select Measurement** menu.
7. In the **Script Math** dialog, touch inside the Script Language field and select either **VBScript** or **JScript** from the pop-up menu.
8. Touch the **Edit Code** button. The **Script Editor** window opens. You can enter code in this window or call up an existing script from a file storage location. If you create your script in this window, you can then export it and save it to file.

Measure Gate

Using Measure Gate, you can narrow the span of the waveform on which to perform parameter measurements, allowing you to focus on the area of greatest interest. You have the option of dragging the gate posts horizontally along the waveform, or specifying a position down to hundredths of a division. The default starting positions of the gate posts are 0 div and 10 div, which coincide with the left and right ends of the grid. The gate, therefore, initially encloses the entire waveform.

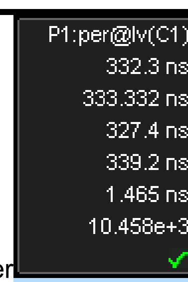
Note: If you have **Grid On Top** enabled, you will not see the gate posts in their default position at each end of the grid. But if you touch either end of the grid, a drag cursor  will appear, indicating that you have control of the post and can now drag it.



In this example, you can see that the Measure Gate includes only five rising edges. Therefore, parameter calculations for rise time are performed only on the five pulses bounded by the gate posts. The position of the gate posts is shown in the **Start** and **Stop** fields in the accompanying dialog.

Setting Up Measure Gate

1. Touch **Measure** → **Measure Setup** on the menu bar.
2. Touch the **Px** tab for the parameter you want to gate. A **Gate** dialog to the right of the main setup dialog opens.



Note: If you already have the parameter of interest set up, you can simply touch the parameter Example Parameter Readout directly below the grid.

3. Touch inside the **Start** data entry field and enter a value using the slider bar at the bottom of the window. Click the keypad button  on the slider bar to enter a value using the pop-up numeric keypad. Or, you can simply touch the leftmost grid line and drag the gate post to the right.
4. Touch inside the **Stop** data entry field and enter a value using the slider bar at the bottom of the window. Click the keypad button  on the slider bar to enter a value using the pop-up numeric keypad. Or, you can simply touch the rightmost grid line and drag the gate post to the left.

Help Markers

Help Markers clarify parameter measurements by displaying movable cursors and a visual representation of what is being measured. For the "at level" parameters, Help Markers make it easier to see where your waveform intersects the chosen level. This feature also displays the hysteresis band that you have set about that level.

You also have the option, by means of an **Always On** checkbox, to leave the Help Markers displayed after you have closed the Help Markers setup dialog.

You have a choice of **Simple** or **Detailed** views of the markers:

- The **Simple** selection produces cursors and Measure Gate gate posts. The gate posts are independently placeable for each parameter.
- The **Detailed** selection produces cursors, Measure Gate gate posts, a label identifying the parameter being measured, and a level indicator and hysteresis band for "at level" parameters (not part of Standard Horizontal or Standard Vertical parameters).

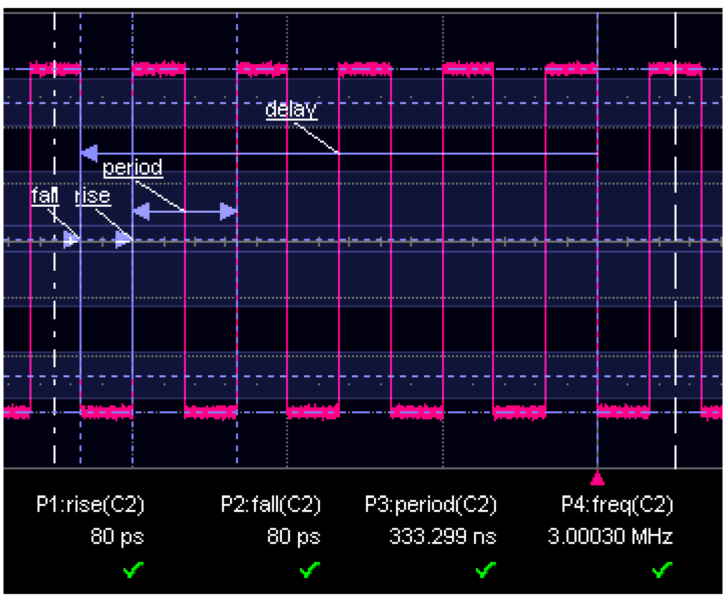


Figure 4-10. Standard Horizontal Parameter Help Markers

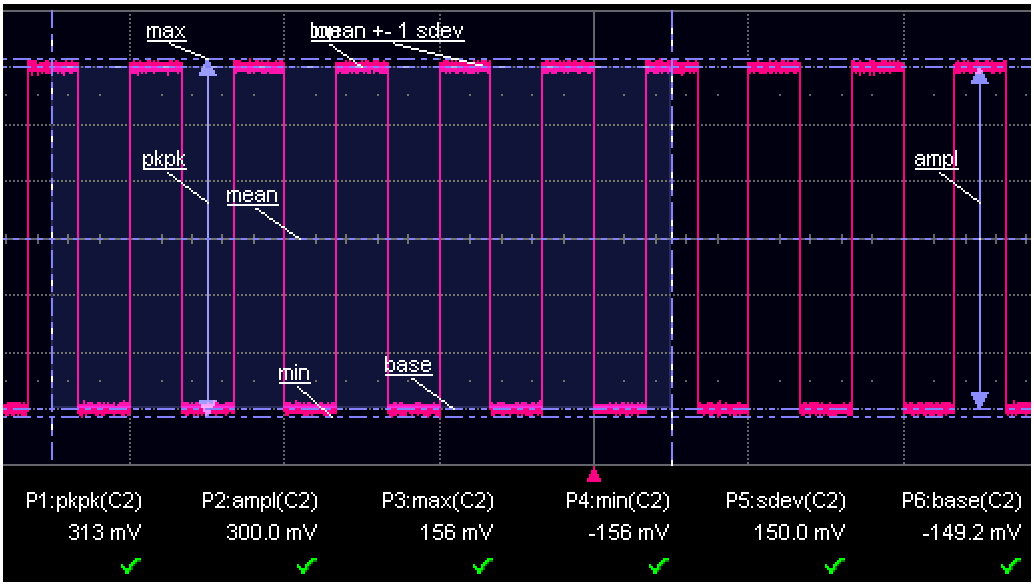


Figure 4-11. Standard Vertical Parameter Help Markers

Setting Up Help Markers

1. Touch **Measure** → **Measure Setup** on the menu bar.
2. Select a Measure Mode: **Std Vertical**, **Std Horizontal**, or **My Measure**.
3. Touch the **Show All** button 

to display Help Markers for every parameter being measured on the displayed waveform (C2 in the examples above).
4. Touch a **P1** to **Px** tab 

for any parameter listed; it doesn't matter which. Touch inside the **Help Markers** field and select **Simple**. The **Simple** selection produces cursors and Measure Gate gate posts. The gate posts are independently placeable for each parameter. The **Detailed** selection produces cursors, Measure Gate gate posts, a label identifying the parameter being measured, and a level indicator and hysteresis band for "at level" parameters. .

Note: The choice of **Simple** or **Detailed** is applied to all parameters at the same time. That is, if you choose Simple markers for one parameter, all parameters will be displayed in this mode.

5. Touch the **Always On** checkbox if you want to continuously display Help Markers for this parameter.

Turning Off Help Markers

1. Touch the **Clear All** button 

to turn off Help Markers for all parameters.
2. To turn off Help Markers for individual parameters, touch the **Px** tab for the parameter in question. Then uncheck the **Always On** checkbox. When you close this dialog, the Help Markers for this parameter will no longer be displayed.

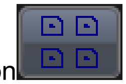
Customizing a Parameter

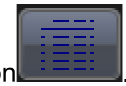
From the Measure Dialog

1. Touch **Measure** → **Measure Setup on the menu bar**.
2. Touch the **My Measure** button in the **Measure** dialog. The dialog presents you with a panel of eight preset parameters.
3. For each parameter, touch the **On** check box to enable the parameter listed.

4. If you want to change the parameter listed, or a measurement characteristic, touch the parameter button 

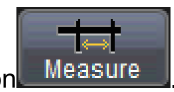
(**P1** for example) alongside the check box. A pop-up menu of parameters categorized by type appears.

To display parameter icons only, touch the icon button  at the bottom of the menu.

To display the icons in list form, along with an explanation of each parameter, touch the list button .

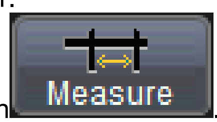
Use the **Up/Down** buttons to scroll through the list of icons.
4. When you make a selection from the parameter icon menu, the setup dialogs for that parameter appear. You can then change the waveform source and other conditions of the parameter.
5. If you are setting up an "@level" parameter, make selections for **Level type** (percent or absolute), **Slope** (positive, negative, both), and **Hysteresis** level.
6. Touch the **Gate** tab, and set the position of the gate posts.

From a Vertical Setup Dialog

1. Touch **Vertical** → **Channelx Setup** on the menu bar.
2. In the **Cx Vertical Adjust** dialog, touch the Measure button 

.
3. Select a parameter from the pop-up menu. (The **Actions for trace** source defaults to the channel or trace whose dialog is open. If a parameter, it goes into the next "available" parameter, or the last one if all are used.)
4. Select another parameter or touch **Close**.

From a Math Setup Dialog

1. Touch **Math** → **Fx Setup** on the menu bar.
2. In the **Fx** dialog, touch the Measure button 

.
3. Select a parameter from the pop-up menu. (The **Actions for trace** source defaults to the channel or trace whose dialog is open. If a parameter, it goes into the next "available" parameter, or the last one if all are used.)
4. Select another parameter or touch **Close**.

List of Parameters

Instrument parameters are described as follows. Parameter availability varies based on the options installed on the oscilloscope (specific references provided in corresponding Notes).

Parameter	Description	Definition	Notes
100BT Fall (fall)	Fall time between 2 levels (upper-base, base-lower) of a 3-level signal (100BT)		Available with ENET option.
100BT Rise (rise)	Rise time between 2 levels (Lower-base, base-upper) of a 3-level signal (100BT)		Available with ENET option.
100BT TIE (TIE)	Difference between the measured and ideal times at level between base and upper or lower levels of 100BT signal.		Available with ENET option.
100BT Tj	Total jitter from a TIE at level		Available with ENET option.

Parameter	Description	Definition	Notes
(Tj)	between base and upper or lower levels of 100BT signal.		
ACSN (acsn)	Auto-correlation Signal-to-Noise provides a signal-to-noise ratio for periodic waveforms.		Available with DDM2 option. Standard in DDA-5005A.
AltNCycle	Alternate N-Cycle Plot. Timing of the transitions in the data waveform is measured for each transition and plotted as a function of the number of unit intervals over which the timing is measured. The N-cycle plot displays the mean or standard deviation of the edge placement in the waveform relative to each other (data to data) or to a reference clock (clock to data).		Available with ASDA option.
Amplitude (ampl)	Measures the difference between upper and lower levels in two-level signals. Differs from pkpk in that noise, overshoot, undershoot, and ringing do not affect the measurement.	top - base	On signals not having two major levels (such as triangle or saw-tooth waves), returns same value as pkpk. Standard parameter.
Ampl Asym (aasym)	Amplitude asymmetry between taa+ and taa-	1- |(taa+ - taa-)|/(taa+ - taa-)	Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
Apparent Pwr	Product of the RMS values of the voltage and current		Available with PMA2 option.
Area	Integral of data: Computes area of waveform between cursors relative to zero level. Values greater than zero contribute positively to the area; values less than zero negatively.	Sum from first to last of data multiplied by horizontal time between points	Standard parameter.
Avg Power (avgpwr)	Average power of the waveform		Available with SDA and SDM options. Standard in SDA100G scopes.
Base	Lower of two most probable states (higher is top). Measures lower level in two-level signals. Differs from min in that noise, overshoot,	Value of most probable lower state	On signals not having two major levels (triangle or saw-tooth waves, for example), returns same value as min.

Parameter	Description	Definition	Notes
	undershoot, and ringing do not affect measurement.		Standard parameter.
Bit Rate (bitrate)	One over duration of one UI measured on an eye		Available with SDA option. Standard in SDA, SDA100G, and WaveExpert scopes.
CMACp	PCI Express V TX-CM-ACp and V RX-CM-Acp		Available with PCIE option.
Cycles (cycles)	Determines number of cycles of a periodic waveform lying between cursors. First cycle begins at first transition after the left cursor. Transition may be positive- or negative-going.	Number of cycles of periodic waveform	Standard parameter.
cyclic Mean (mean)	Cyclic mean: Computes the average of waveform data. Contrary to mean, computes average over an integral number of cycles, eliminating bias caused by fractional intervals.	Average of data values of an integral number of periods	Choose this parameter by selecting **Mean** from the parameter table, then touching the **Cyclic** checkbox. Standard parameter.
cyclic Median (median)	Cyclic median: Computes average of base and top values over an integral number of cycles, contrary to median, eliminating bias caused by fractional intervals.	Data value for which 50% of values are above and 50% below	Choose this parameter by selecting **Median** from the parameter table, then touching the **Cyclic** checkbox. Standard parameter.
cyclic RMS (rms)	Cyclic root mean square: Computes square root of sum of squares of data values divided by number of points. Contrary to rms, calculation is performed over an integral number of cycles, eliminating bias caused by fractional intervals.	$\sqrt{\frac{1}{N} \sum_{i=1}^{N} (v_i)^2}$ Where: v_i denotes measured sample values, and N = number of data points within the periods found.	Choose this parameter by selecting **RMS** from the parameter table, then touching the **Cyclic** checkbox. Standard parameter.
cyclic Std dev (sdev)	Cyclic standard deviation: Standard deviation of data values from mean value over integral number of periods. Contrary to sdev, calculation is performed over an integral number of cycles, eliminating bias caused by fractional intervals.	$\sqrt{\frac{1}{N}\sum_{i=1}^{N} (v_i - mean)^2}$ Where: v_i denotes measured sample values, and N = number of data points within the periods found.	Standard parameter. Choose this parameter by selecting **Std dev** from the parameter table, then touching the **Cyclic** checkbox.

Parameter	Description	Definition	Notes
DCD	Amount of jitter due to duty cycle distortion		Available with SDA and ENET options. Standard in SDA and WaveExpert scopes.
DDj	Amount of data dependent jitter in a signal		Available with SDA option.
Delay	Time from trigger to transition: Measures time between trigger and first 50% crossing after left cursor. Can measure propagation delay between two signals by triggering on one and determining delay of other.	Time between trigger and first 50% crossing after left cursor	Standard parameter.
Delta Delay (ddelay)	Computes time between 50% level of two sources.	Time between midpoint transition of two sources	Standard parameter.
Dj Effective (Dje)	Amount of deterministic jitter (estimated) in a signal		Available with SDA option.
DOV	Differential Output Voltage of a 100Base-T signal		Available with ENET option.
Dperiod@level (dper@lv)	Adjacent cycle deviation (cycle-to-cycle jitter) of each cycle in a waveform		Reference levels and edge-transition polarity can be selected. Hysteresis argument used to discriminate levels from noise in data. Available with JTA2, SDA, and XMAP options. Standard in SDA100G scopes.
Droop FG (drpfg)	1000Base-T test mode 1 droop from F to G		Available with ENET option.
Droop HJ (drphg)	1000Base-T test mode 1 droop from H to J		Available with ENET option.
Dtime@level (dt@lv)	Computes transition between selected levels or sources.	Time between transition levels of two sources, or from trigger to transition level of a single source This measurement gives the time of the source 2 edge minus the time of the source 1 edge.	Reference levels and edge-transition polarity can be selected. Hysteresis argument used to discriminate levels from noise in data. Standard parameter.
Dtrig Time (dtrig)	Time from last trigger to this trigger		Standard in WaveRunner 6000A, WavePro 7000A, WaveMaster, and sampling scopes.
Duration (dur)	For single sweep waveforms, dur is 0; for sequence waveforms: time from first to last segment's trigger; for	Time from first to last acquisition: for average, histogram or sequence waveforms	Standard parameter.

Parameter	Description	Definition	Notes
	single segments of sequence waveforms: time from previous segment's to current segment's trigger; for waveforms produced by a history function: time from first to last accumulated waveform's trigger.		
Duty@level (duty@lv)	Percent of period for which data are above or below a specified level.		Reference levels and edge-transition polarity can be selected. Hysteresis argument used to discriminate levels from noise in data. Available with JTA2, SDA, and XMAP options. Standard in SDA100G and WavePro 7000A scopes.
Duty Cycle (duty)	Duty cycle: Width as percentage of period.	width/period	Standard parameter.
Dwidth@level (dwid@lv)	Difference of adjacent width above or below a specified level.		Reference levels and edge-transition polarity can be selected. Hysteresis argument used to discriminate levels from noise in data. Available with JTA2, SDA, and XMAP options. Standard in SDA100G scopes.
Edge@level (edge@lv)	Number of edges in waveform.		Reference levels and edge-transition polarity can be selected. Hysteresis argument used to discriminate levels from noise in data. Available with JTA2, USB2, SDA, and XMAP options. Standard in SDA100G and WavePro 7000A scopes.
Excel param (excel)	Performs measurements in Excel by transferring one or two waveforms and reading the resulting parameter value.		Available with XMAP and XDEV options. Standard on DDA-5005A scope. Excel must be loaded on the instrument.
Ext Ratio (ER)	Ratio of the power levels of an eye diagram		Available with SDA and SDM options. Standard in SDA, SDA100G, and WaveExpert scopes.
Eye AC RMS (erms)	Root mean square of data within one UI		Standard in SDA and WaveExpert scopes.
Eye Amplitude	Difference of the levels of an		Available with SDA and SDM

Parameter	Description	Definition	Notes
(eampl)	eye diagram		options. Standard in SDA, SDA100G, and WaveExpert scopes.
Eye BER	Bit Error Rate estimated from an eye diagram		Available with SDA and SDM options. Standard in SDA, SDA100G, and WaveExpert scopes.
Eye Bit Rate (ebitr)	Reciprocal of the duration of one UI measured on an eye		Standard in SDA and WaveExpert scopes.
Eye Bit Time (ebitt)	Duration of one UI measured on an eye		Standard in SDA and WaveExpert scopes.
Eye Crossing (ecross)	Level of the crossing in an eye diagram		Available with SDA and SDM options. Standard in SDA, SDA100G, and WaveExpert scopes.
Eye CrossN (ecrn)	Time of first 50% crossing level with negative edge of an eye relative to trigger or eye reference		Standard in SDA and WaveExpert scopes.
Eye CrossP (ecrp)	Time of first 50%crossing level with positive edge of an eye relative to trigger or eye reference		Standard in SDA and WaveExpert scopes.
Eye Cyc Area (ecyca)	The area under the mean persistence trace under first UI		Standard in SDA and WaveExpert scopes.
Eye Delay (edly)	Time of first crossing of an eye relative to trigger or eye reference		Standard in SDA and WaveExpert scopes.
Eye Delt Dly (eddly)	Delay of crossing times between two eyes		Standard in SDA and WaveExpert scopes.
Eye FallTime(efall)	Fall time of the mean of persistence data		Standard in SDA and WaveExpert scopes.
Eye Height(ehght)	Size of the vertical opening of an eye diagram		Available with SDA and SDM options. Standard in SDA, SDA100G, and WaveExpert scopes.
Eye Mean (emean)	Mean level of an eye		Standard in SDA and WaveExpert scopes.
Eye Open Fac (eofac)	Eye opening factor measured within the eye aperture		Standard in SDA and WaveExpert scopes.
Eye OverN (eovn)	Negative overshoot measured on an eye		Standard in SDA and WaveExpert scopes.
Eye OverP	Positive overshoot measured		Standard in SDA and WaveExpert

Parameter	Description	Definition	Notes
(eovp)	on an eye		scopes.
Eye Pk Noise (eppn)	Peak-to-peak noise of a level of an eye diagram		Standard in SDA and WaveExpert scopes.
Eye PkPk Jit (eppj)	Peak-to-peak jitter measured on eye persistence		Standard in SDA and WaveExpert scopes.
Eye Pulse Wid (epwid)	The width of the eye measured at mid level		Standard in SDA and WaveExpert scopes.
Eye Q Factor (Q)	Q factor measured within the eye aperture		Available with SDA and SDM options. Standard in SDA, SDA100G, and WaveExpert scopes.
Eye RiseTime (erise)	Rise time of the mean of persistence data		Standard in SDA and WaveExpert scopes.
Eye RMS Jit (ermsj)	Root mean square jitter of an eye		Standard in SDA and WaveExpert scopes.
Eye SD Noise (esdev)	The standard deviation of data on one eye level		Standard in SDA and WaveExpert scopes.
Eye SgToNoise (estn)	Signal to noise of an eye diagram		Standard in SDA and WaveExpert scopes.
Eye SupRatio (esupr)	Suppression ratio of an eye		Standard in SDA and WaveExpert scopes.
Eye Width (ewdth)	Size of the horizontal opening of an eye diagram		Available with SDA and SDM options. Standard in SDA, SDA100G, and WaveExpert scopes.
Fall 80-20% (fall8020)	Fall 80-20%: Duration of pulse waveform's falling transition from 80% to 20% of the amplitude, averaged for all falling transitions between the cursors.	Average duration of falling 80-20% transition	On signals not having two major levels (triangle or saw-tooth waves, for example), top and base can default to maximum and minimum, giving, however, less predictable results. Standard parameter.
Fall time (fall)	Fall time: Duration of falling edge from 90-10% of amplitude.	Time at upper threshold minus Time at lower threshold averaged over each falling edge	On signals not having two major levels (triangle or saw-tooth waves, for example), top and base can default to maximum and minimum, giving, however, less predictable results. Standard parameter.
Fall@level (fall@lv)	Fall at level: Duration of pulse waveform's falling edges between user-specified transition levels. See also Rise@level.	Transition time for % or absolute levels of all falling edges. Enhanced version sets measurement calculations to use one of the following:	On signals not having two major levels (triangle or saw-tooth waves, for example), top and base can default to maximum and minimum, giving, however, less predictable results. Standard parameter.

<table>
<tr><th>Parameter</th><th>Description</th><th>Definition</th><th>Notes</th></tr>
<tr><td></td><td><table><tr><th>Thresh.</th><th>Remote</th><th>Lower Limit</th><th>Upper Limit</th><th>Default</th></tr><tr><td>Lower</td><td>Low</td><td>1%</td><td>45%</td><td>10%</td></tr><tr><td>Upper</td><td>High</td><td>55%</td><td>99%</td><td>90%</td></tr></table>Threshold arguments specify two vertical values on each edge used to compute fall time. Formulas for upper and lower values:
lower = lower thresh. x amp/100 + base
upper = upper thresh. x amp/100 + base</td><td>Base & Top (% or absolute)
Peak-Peak (%)
0V-Min (%)</td><td>Enhanced parameter available with EMC option.</td></tr>
<tr><td>First</td><td>Indicates value of horizontal axis at left cursor.</td><td>Horizontal axis value at left cursor</td><td>Indicates location of left cursor. Cursors are interchangeable: for example, the left cursor may be moved to the right of the right cursor and first will give the location of the cursor formerly on the right, now on left.
Standard parameter.</td></tr>
<tr><td>Frequency (freq)</td><td>Frequency: Period of cyclic signal measured as time between every other pair of 50% crossings. Starting with first transition after left cursor, the period is measured for each transition pair. Values then averaged and reciprocal used to give frequency.</td><td>1/period</td><td>Standard parameter.</td></tr>
<tr><td>Freq@level (freq@lv)</td><td>Frequency at a specific level and slope for every cycle in waveform.</td><td></td><td>Reference levels and edge-transition polarity can be selected. Hysteresis argument used to discriminate levels from noise in data.
Available with JTA2, SDA, and XMAP options.
Standard in SDA100G and WavePro 7000A scopes.</td></tr>
<tr><td>FWHM</td><td>Measures the width of the largest area histogram peak at half of the population of the highest peak.</td><td></td><td>Available with DDM2, JTA2, XMATH, XWAV, CAN02, SDA, and XMAP options.
Standard in DDA-5005A, SDA100G, and WaveExpert scopes.</td></tr>
<tr><td>FWxx</td><td>Measures the width of the largest area histogram peak at xx% of the population of the highest peak.</td><td></td><td>Available with DDM2, JTA2, XMATH, XWAV, CAN02, SDA, and XMAP options.
Standard in DDA-5005A, SDA100G, WaveExpert, and sampling scopes.</td></tr>
</table>

Parameter	Description	Definition	Notes
Half Period (hper)	Half period of a waveform.		Reference levels and edge-transition polarity can be selected. Hysteresis argument used to discriminate levels from noise in data. Available with JTA2, SDA, and XMAP options. Standard in SDA100G scopes.
Hist Ampl (hampl)	Difference in value between the two most populated peaks in a histogram.		Available with DDM2, JTA2, XMATH, XWAV, CAN02, SDA, and XMAP options. Standard in DDA-5005A, SDA100G, WaveExpert, and sampling scopes.
Hist Base (hbase)	Value of the left-most of the two most populated histogram peaks.		Available with DDM2, JTA2, XMATH, XWAV, CAN02, SDA, and XMAP options. Standard in DDA-5005A, SDA100G, WaveExpert, and sampling scopes.
Hist Maximum (hmax)	Value of the highest (right-most) populated bin in a histogram.		Available with DDM2, JTA2, XMATH, XWAV, CAN02, SDA, and XMAP options. Standard in DDA-5005A, SDA100G, WaveExpert, and sampling scopes.
Hist Max Pop (hmaxp)	Peak with maximum population in a histogram.		Available with DDM2, JTA2, XMATH, XWAV, CAN02, SDA, and XMAP options. Standard in DDA-5005A, SDA100G, WaveExpert, and sampling scopes.
Hist Mean (hmean)	Average or mean value of data in the histogram.		Available with DDM2, JTA2, XMATH, XWAV, CAN02, SDA, and XMAP options. Standard in DDA-5005A, SDA100G, and WaveExpert scopes.
Hist Median (hmedi)	Value of the horizontal axis of a histogram that divides the population into two equal halves.		Available with DDM2, JTA2, XMATH, XWAV, CAN02, SDA, and XMAP options. Standard in DDA-5005A, SDA100G, WaveExpert, and sampling scopes.
Hist Minimum (hmin)	Value of the lowest (left-most) populated bin in a histogram.		Available with DDM2, JTA2, XMATH, XWAV, CAN02, SDA, and XMAP options. Standard in DDA-5005A, SDA100G, and WaveExpert scopes.
Hist Mode (mode)	Position of the highest histogram peak.		Available with DDM2, JTA2, XMATH, XWAV, CAN02, SDA, and XMAP options. Standard in DDA-5005A, SDA100G,

Parameter	Description	Definition	Notes
			WaveExpert, and sampling scopes.
Hist Pop@X (pop@x)	Population at bin for specified horizontal coordinate. You can place the cursor at any bin and use either Absolute, Reference, or Difference cursor shape.		Available with DDM2, JTA2, SDM, XMATH, XWAV, CAN02, SDA, and XMAP options. Standard in DDA-5005A, SDA100G, and sampling scopes.
Hist Range (range)	Calculates range (max - min) of a histogram.		Available with DDM2, JTA2, ENET, XMATH, XWAV, CAN02, SDA, and XMAP options. Standard in DDA-5005A, SDA100G, WaveExpert, and sampling scopes.
Hist RMS (hrms)	Root mean square of the values in a histogram.		Available with DDM2, JTA2, XMATH, XWAV, CAN02, SDA, and XMAP options. Standard in DDA-5005A, SDA100G, and sampling scopes
Hist Sdev (hsdev)	Standard deviation of values in a histogram.		Available with DDM2, JTA2, XMATH, XWAV, CAN02, SDA, and XMAP options. Standard in DDA-5005A, SDA100G, WaveExpert, and sampling scopes.
Hist Top (htop)	Value of the right-most of the two most populated histogram peaks.		Available with DDM2, JTA2, XMATH, XWAV, CAN02, SDA, and XMAP options. Standard in DDA-5005A, SDA100G, WaveExpert, and sampling scopes.
Hist X@peak (x@pk)	The value of the nth highest histogram peak.		Applies only to histograms. Available with JTA2, XMATH, XWAV, CAN02, DDM2, SDA, and XMAP options. Standard in DDA-5005A, SDA100G, and WaveExpert scopes.
Hold Time (hold)	Time from the clock edge to the data edge. You can set levels, slope, and hysteresis independently for Hold Clock and Hold Data. See also Setup parameter.		Reference levels and edge-transition polarity can be selected. Hysteresis argument used to discriminate levels from noise in data. Available with JTA2, ENET, USB2, SDA, and XMAP options. Standard in SDA100G scopes.
Hparam Script (hscript)	Visual Basic (or Java) script which produces a measurement from one or two input histogram results		Available with XMAP, ASDA, and XDEV options. Standard in DDA-5005A.
Jitter Filter	Jitter in the specified frequency band. Generates a time sequence of jitter		Available with ASDA option.

Parameter	Description	Definition	Notes
	measurements that are filtered by the selected band-pass filter.		
Last	Time from trigger to last (rightmost) cursor.	Time from trigger to last cursor	Indicates location of right cursor. Cursors are interchangeable: for example, the right cursor may be moved to the left of the left cursor and first will give the location of the cursor formerly on the left, now on right. Standard parameter.
Level@X (lvl@x)	Gives the vertical value at the specified *x* position. If the *x* position is between two points, it gives the interpolated value. When the **Nearest point** checkbox is checked, it gives the vertical value of the nearest data point.		Standard parameter.
Local Base (lbase)	Value of the baseline for a local feature.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
Local Bsep (lbsep)	Local baseline separation, between rising and falling slopes.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
Local Max (lmax)	Maximum value of a local feature.	If 25% of the way down toward min is more than 10,000 points, the extremal value is used as the local max.	Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
Local Min (lmin)	Minimum value of a local feature.	If 25% of the way up toward max is more than 10,000 points, the extremal value is used as the local min.	Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
Local Number (lnum)	Number of local features (peak/trough pairs).		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
Local pkpk (lpp)	Vertical difference between the peak and trough of a		Hysteresis argument used to discriminate levels from noise in

Parameter	Description	Definition	Notes
	local feature (lmax - lmin).		data. Available with DDM2 option. Standard in DDA-5005A.
Local tbe	Time between events (between local peak and next trough or local trough and next peak).		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
Local tbp	Time between a local feature peak and the next local peak.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
Local tbt	Time between a local feature trough and the next local trough.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
Local t@max (ltmx)	Time of the maximum value of a local feature.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
Local t@min (ltmn)	Time of the minimum value of a local feature.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
Local tot (ltut)	Time a local feature spends over a user specified percentage of its peak-to-trough amplitude.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
Local tpt (ltpt)	Time between local feature peak and trough.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
Local ttp (lttp)	Time between local feature trough and the next local peak.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
Local tut	Time a local feature spends under a user specified		Hysteresis argument used to discriminate levels from noise in

Parameter	Description	Definition	Notes
(ltut)	percentage of its peak-to-trough amplitude.		data. Available with DDM2 option. Standard in DDA-5005A.
Mathcad (mcad)	Produces a parameter using a user-specified Mathcad function.		Available with XMAP and XDEV option. Standard in DDA-5005A. Mathcad 2001i or later must be loaded on the instrument.
MATLAB	Produces a parameter using a user-specified MATLAB function.		Available with XMAP and XDEV option. Standard in DDA-5005A, WaveRunner 6000A, WaveMaster, WavePro 7000A, and sampling scopes MATLAB must be loaded on the instrument.
Maximum (max)	Measures highest point in waveform. Unlike top, does not assume waveform has two levels.	Highest value in waveform between cursors	Gives similar result when applied to time domain waveform or histogram of data of same waveform. But with histograms, result may include contributions from more than one acquisition. Computes horizontal axis location of rightmost non-zero bin of histogram -- not to be confused with maxp. Standard parameter.
Mean	Average of data for time domain waveform. Computed as centroid of distribution for a histogram of the data values.	Average of data	Gives similar result when applied to time domain waveform or histogram of data of same waveform. But with histograms, result may include contributions from more than one acquisition. Standard parameter.
Median	The average of base and top values.	Average of Base and Top.	Standard parameter.
Minimum (min)	Measures the lowest point in a waveform. Unlike base, does not assume waveform has two levels.	Lowest value in waveform between cursors	Gives similar result when applied to time domain waveform or histogram of data of same waveform. But with histograms, result may include contributions from more than one acquisition. Standard parameter.
Nb Phase (nbph)	Provides a measurement of the phase at a specific frequency of a waveform (narrow band).		Available with DDM2, XMATH, SDA, and XMAP options. Standard in DDA-5005A and SDA100G scopes.
Nb Power (nbpw)	Provides a measurement of the power at a specific		Available with DDM2, XMATH, PMA2, SDA, and XMAP options.

Parameter	Description	Definition	Notes
	frequency of a waveform (narrow band).		Standard in DDA-5005A and SDA100G scopes.
N-cycle Jitter	Peak-to-peak jitter between edges spaced *n* UI apart.	Compares the expected time to the actual time of leading edges n bits apart.	Available in SDA analyzers.
NLTS	Provides a measurement of the nonlinear transition shift for a prml signal.		Available with DDM2 option. Standard in DDA-5005AA.
None	Disable parameter calculation		Standard parameter.
Num Points (npoints)	Number of points in the waveform between the cursors.		Standard parameter.
One Level (one)	One level of an eye diagram		Available with SDA and SDM options. Standard in SDA, SDA100G, and WaveExpert scopes.
Overshoot-	Overshoot negative: Amount of overshoot following a falling edge, as percentage of amplitude.	(base - min.)/ampl x 100	Waveform must contain at least one falling edge. On signals not having two major levels (triangle or saw-tooth waves, for example), may not give predictable results. Standard parameter.
Overshoot+	Overshoot positive: Amount of overshoot following a rising edge specified as percentage of amplitude.	(max. - top)/ampl x 100	Waveform must contain at least one rising edge. On signals not having two major levels (triangle or saw-tooth waves, for example), may not give predictable results. Standard parameter.
Overwrite (owrite)	Ratio of residual-to-original power of a low frequency waveform overwritten by a higher frequency.		Available with DDM2 option. Standard in DDA-5005A.
Param Script (script)	Visual Basic or Java script that produces a measurement from one or two input waveforms.		Available with XMAP, XDEV, and ASDA options. Standard in DDA-5005A.
Pattern Time (Ptime)	Detects a bit pattern in a digital sequence and outputs the start and stop time of the pattern.		Available with SDA and PCIE options. Standard in SDA100G scope.
Peak Mag (pkmag)	Peak mag away from a baseline. Note: the measure gate must include more of the baseline than any other single level.		Available with ENET option.

Parameter	Description	Definition	Notes
Peaks	Number of peaks in a histogram.		Available with DDM2, JTA2, XMATH, XWAV, CAN02, SDA, and XMAP options. Standard in DDA-5005A, SDA100G, and WaveExpert scopes.
Peak to Peak (pkpk)	Peak-to-peak: Difference between highest and lowest points in waveform. Unlike ampl, does not assume the waveform has two levels.	maximum - minimum	Gives a similar result when applied to time domain waveform or histogram of data of the same waveform. But with histograms, result may include contributions from more than one acquisition. Standard parameter.
Percentile (pctl)	Horizontal data value that divides a histogram so the population to the left is xx% of the total.		Available with DDM2, JTA2, XMATH, XWAV, CAN02, SDA, and XMAP options. Standard in DDA-5005A, SDA100G, WaveExpert, and sampling scopes.
Per DCD (pdcd)	Amount of jitter due to duty cycle distortion measured on a persistence trace.		Standard in SDA and WaveExpert scopes.
Per Duty Cyc (pduty)	Duty cycle measured on a persistence trace.		Standard in SDA and WaveExpert scopes.
Period	Period of a cyclic signal measured as time between every other pair of 50% crossings. Starting with first transition after left cursor, period is measured for each transition pair, with values averaged to give final result.	$\frac{1}{Mr}\sum_{i=1}^{Mr}(Tr_i^{50} - Tr_i^{50})$ Where: Mr is the number of leading edges found, Mf the number of trailing edges found, Tr_i^x the time when rising edge i crosses the x% level, and Tf_i^x the time when falling edge i crosses the x% level.	Standard parameter.
Period@level (per@lv)	Period at a specified level and slope for every cycle in waveform.		Reference levels and edge-transition polarity can be selected. Hysteresis argument used to discriminate levels from noise in data. Available with JTA2, AORM, ENET, SDA, and XMAP options. Standard in DDA-5005A, SDA100G, and WavePro 7000A scopes.
Per Pulse Sym	Symmetry of RZ pulse around eye aperture center.		Standard in SDA and WaveExpert scopes.
Persist Area (parea)	Area under mean persistence trace.		Standard in SDA and WaveExpert scopes.

Parameter	Description	Definition	Notes
Persist Max (pmax)	Highest vertical value of input persistence trace.		Standard in SDA and WaveExpert scopes.
Persist Mean (pmean)	Average of persistence data.		Standard in SDA and WaveExpert scopes.
Persist Mid (pmid)	Mid level between Maximum and Minimum data of a persistence trace.		Standard in SDA and WaveExpert scopes.
Persist Min (pmin)	Lowest vertical value of input persistence trace.		Standard in SDA and WaveExpert scopes.
Persist PkPk (pPkPk)	Difference between maximum and minimum data values		Standard in SDA and WaveExpert scopes.
Persist RMS (prms)	Root mean square of persistence data		Standard in SDA and WaveExpert scopes.
Phase	Phase difference between signal analyzed and signal used as reference. You can set the output type to percent, degrees, or radians. After setting up the reference, touch the **More** tab for signal setups.	Phase difference between signal and reference	Standard parameter.
Pj	Periodic component of jitter		Available with SDA option.
Power Factor	Ratio of real to apparent power		Available with PMA2 option.
PW50	Average pulse width at the 50% point between the local baseline and the local peak or trough.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
PW50–	Average pulse width at the 50% point between the local baseline and the local trough.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
PW50+	Average pulse width at the 50% point between the local baseline and the local peak.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
Real Power (res)	Mean of the product of voltage and current waveform (or mean of the instantaneous power)		Available with PMA2 option.
Resolution	Ratio of taa for a high and	taa (HF) / mean taa	Hysteresis argument used to

<table>
<tr><th>Parameter</th><th>Description</th><th>Definition</th><th>Notes</th></tr>
<tr><td></td><td>low frequency waveform</td><td>(LF)*100</td><td>discriminate levels from noise in data.
Available in DDM2.
Standard in DDA-5005A.</td></tr>
<tr><td>Ring</td><td>Ringback (high or low)</td><td></td><td>Available with SDA parameter.</td></tr>
<tr><td>Rj Effective</td><td>Amount of random jitter (estimated) in a signal</td><td></td><td>Available with SDA parameter.</td></tr>
<tr><td>Rise</td><td>Rise time: Duration of rising edge from 10-90% of the signal amplitude.</td><td>Time at lower threshold minus
Time at upper threshold
averaged over each rising edge</td><td>On signals not having two major levels (triangle or saw-tooth waves, for example), top and base can default to maximum and minimum, giving, however, less predictable results.
Standard parameter.</td></tr>
<tr><td>Rise 20-80% (rise2080)</td><td>Rise 20% to 80%: Duration of pulse waveform's rising transition from 20% to 80% of the signal amplitude, averaged for all rising transitions between the cursors.</td><td>Average duration of rising 20-80% transition</td><td>On signals not having two major levels (triangle or saw-tooth waves, for example), top and base can default to maximum and minimum, giving, however, less predictable results.
Standard parameter.</td></tr>
<tr><td>Rise@level (rise@lv)</td><td>Rise at level: Duration of pulse waveform's rising edges between user-defined transition levels.
<table>
<tr><th>Thresh.</th><th>Remote</th><th>Lower Limit</th><th>Upper Limit</th><th>Default</th></tr>
<tr><td>Lower</td><td>Low</td><td>1%</td><td>45%</td><td>10%</td></tr>
<tr><td>Upper</td><td>High</td><td>55%</td><td>99%</td><td>90%</td></tr>
</table>
Threshold arguments specify two vertical values on each edge used to compute rise time.
Formulas for upper and lower values:
lower = lower thresh. x amp/100 + base
upper = upper thresh. x amp/100 + base</td><td>Slew rate for % or absolute levels of rising or falling edges.
Enhanced version sets measurement calculations to use one of the following:
Base & Top (% or absolute)
Peak-Peak (%)
0V-Max (%)</td><td>On signals not having two major levels (triangle or saw-tooth waves, for example), top and base can default to maximum and minimum, giving, however, less predictable results.
Standard parameter.
Enhanced parameter available with EMC option.</td></tr>
<tr><td>RMS</td><td>Root Mean Square of data between the measure gate.</td><td>$\sqrt{\frac{1}{N}\sum_{i=1}^{N}(v_i)^2}$
Where: v_i denotes measured sample values, and N = number of data points within the periods</td><td>Gives similar result when applied to time domain waveform or histogram of data of same waveform. But with histograms, result may include contributions from more than one acquisition.
Standard parameter.</td></tr>
</table>

Parameter	Description	Definition	Notes
		found up to maximum of 100 periods.	
SAS	Signal Amplitude Symmetry of a 100Base-T signal		Available with ENET option.
SD2Skew (sdskw)	Calculates the time skew between 2 serial data lanes		Available with SDA and PCIE options.
Setup	Time from the data edge to the clock edge.		Reference levels and edge-transition polarity can be selected. Hysteresis argument used to discriminate levels from noise in data. Available with JTA2, SDA, and XMAP options. Standard in SDA and SDA100G scopes.
Skew	Time of clock1 edge minus time of nearest clock2 edge.		Reference levels and edge-transition polarity can be selected. Hysteresis argument used to discriminate levels from noise in data. Hysteresis on a measurement (if set to 500 mdiv) requires that the signal must transition one way 1/2 division (total swing) across the threshold for the measurement to be valid. Available with JTA2, SDA, and XMAP options. Standard in SDA100G and WaveSurfer scopes.
Slew Rate (slew)	Slew rate or local dV/dt in a transition zone		Available in SDA and JTA2 options. Standard in SDA100G scopes.
SSC Diff	Calculates difference between average SSC frequencies.		Available with PCIE option.
SSC Frequency	Frequency of Spread Spectrum Clock signal.		Available with PCIE option.
SSC Ratio	Calculates the ratio between the maximum and minimum SSC frequencies.		Available with PCIE option.
SSC Track	Tracks Spread Spectrum Clock. Filtered track of frequency at level.		Available with ASDA option.
Std Dev (sdev)	Standard deviation of the data between the measure gate-- equivalent to the rms for a zero-mean waveform.	$\sqrt{\frac{1}{N}\sum_{i=1}^{N}(v_i - mean)^2}$ Where: v_i denotes measured sample values,	Gives similar result when applied to time domain waveform or histogram of data of same waveform. But with histograms, result may include contributions from more than one acquisition.

Parameter	Description	Definition	Notes
		and N = number of data points within the periods found up to maximum of 100 periods.	Standard parameter.
TAA	Average peak-to-trough amplitude for all local features.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
TAA-	Average local baseline-to-trough amplitude for all local features.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
TAA+	Average local baseline-to-peak amplitude for all local features.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
TIE@level (tie@lv)	Difference between the measured times of crossing a given slope and level and the ideal expected time. For Slope you can choose positive, negative, or both. For output units you can choose time or unit interval (UI). A unit interval equals one clock period. The Virtual Clock setup gives you a choice of Standard (1.544 MHz) or Custom reference clocks. You can also use a mathematically derived Golden PLL to filter low frequency jitter. The cutoff frequency is user selectable.	Cutoff Freq = (1/1.667e3) x Clock Freq	Reference levels and edge-transition polarity can be selected. Hysteresis argument used to discriminate levels from noise in data. Available with JTA2, ENET, SDA, and XMAP options. Standard in SDA100G, WavePro 7000A, WaveExpert, and sampling scopes.
Time@edge	Measures time at each edge on each digital line		Available with MS-32 option.
Time@level (time@lv)l	Time at level: Time from trigger (t=0) to crossing at a user-specified level.	Time from trigger to crossing level	Reference levels and edge-transition polarity can be selected. Hysteresis argument used to discriminate levels from noise in data. Standard parameter.
Top	Higher of two most probable states, the lower being base; it is characteristic of	Value of most probable higher state	Gives similar result when applied to time domain waveform or histogram of data of same waveform. But with

Parameter	Description	Definition	Notes
	rectangular waveforms and represents the higher most probable state determined from the statistical distribution of data point values in the waveform.		histograms, result may include contributions from more than one acquisition. Standard parameter.
Total Jitter (Tj)	Total jitter at a user-specified bit error ratio.		Available with ENET and SDA options.
Total Pop (totp)	Total population of a histogram.		Available with DDM2, JTA2, XMATH, XWAV, CAN02, SDA, and XMAP options. Standard in DDA-5005A, SDA100G, WaveExpert, and sampling scopes
tUpS	Upsamples a time parameter by a user-specified factor.		Available with SDA and SDM options.
TxCmD	PCI Express: V TX-CM-DC-LINE-DELTA Absolute delta of DC common mode voltage between D+ and D-		Available with PCIE option.
TxFall	Transition time from 80% to 20% for all falling edges as a fraction of UI.		Available with PCIE option.
TxRise	Transition time from 20% to 80% for all rising edges as a fraction of UI.		Available with PCIE option.
Vcross	Voltage at which two signals cross.	Voltage of either signal at the time when difference is zero.	Available with SDA and PCIE options.
Vdiff	Used for V TX-DIFFp-p and V RX-DIFFp-p for PCI-Express.		Available with PCIE option.
VTxDeRatio	Ratio between transition and de-emphasized bits.		Available with PCIE option.
Width	Width of cyclic signal determined by examining 50% crossings in data input. If first transition after left cursor is a rising edge, waveform is considered to consist of positive pulses and width the time between adjacent rising and falling edges. Conversely, if falling edge, pulses are considered negative and width the time between adjacent falling and rising edges. For both cases, widths of all waveform pulses are averaged for the final	Width of first positive or negative pulse averaged for all similar pulses	Similar to fwhm, though, unlike width, that parameter applies only to histograms. Standard parameter.

Parameter	Description	Definition	Notes
	result.		
Width@level (wid@lv)	Width measured at a user-specified level.	Time between two transitions of opposite slope at a specified level. (Slope specified for 1st transition.) Enhanced version sets measurement calculations to use one of the following: Base & Top (% or absolute) Peak-Peak (%) 0V-Max (%) 0V-Min (%)	Reference levels and edge-transition polarity can be selected. Hysteresis argument used to discriminate levels from noise in data. Available with JTA2, USB2, EMC, SDA, and XMAP options. Standard in SDA100G and WavePro 7000A scopes. Enhanced parameter available with EMC option.
WidthN (widn)	Width measured at the 50% level and negative slope.		Standard parameter.
X@max	Determines the horizontal axis location of the maximum value between the measure gate.		Restricted to time and frequency waveforms only. Standard parameter.
X@min	Determines the horizontal axis location of the minimum value between the measure gate.		Restricted to time and frequency waveforms only. Standard parameter.
Zero Level (zero)	Zero level of an eye diagram.		Available with SDA and SDM options. Standard in SDA, SDA100G, and WaveExpert scopes.

Qualified Parameters

Some LeCroy instruments and software packages give you the ability to constrain parameter measurements to a vertically or horizontally limited range, or to occurrences gated by a second waveform. Furthermore, both constraints can operate together. This capability enables you to exclude unwanted characteristics from your measurements. It is much more restrictive than Measure Gate, which is used only to narrow the span of the waveform for analysis, along the horizontal axis.

> **Note:** Since this feature operates on only a subset of the data, possible alerts or status indicators concerning the measurement (such as " Data range too low") are not displayed.

Range Limited Parameters

Setting Up Range Qualifiers

1. Touch **Measure** → **Measure Setup** on the menu bar.
2. Touch a **Px** tab to open the setup dialog.
3. Touch inside the **Source** field and select a source from the pop-up menu.
4. Touch inside the **Measure** field and select a parameter from the pop-up menu.
5. Touch the **Accept** tab of the dialog on the right, then touch the **Values In Range** checkbox.

Note: Depending on whether you select a vertical or horizontal parameter, the correct units will be automatically displayed (V, s, Hz, dB) in the **Between** and **And** fields. Or, if you select a simple ratio parameter (such as power factor) that yields a dimensionless number, no units will be displayed.

6. Touch the **Find Range** button to quickly display the most recent value of the parameter measurement.

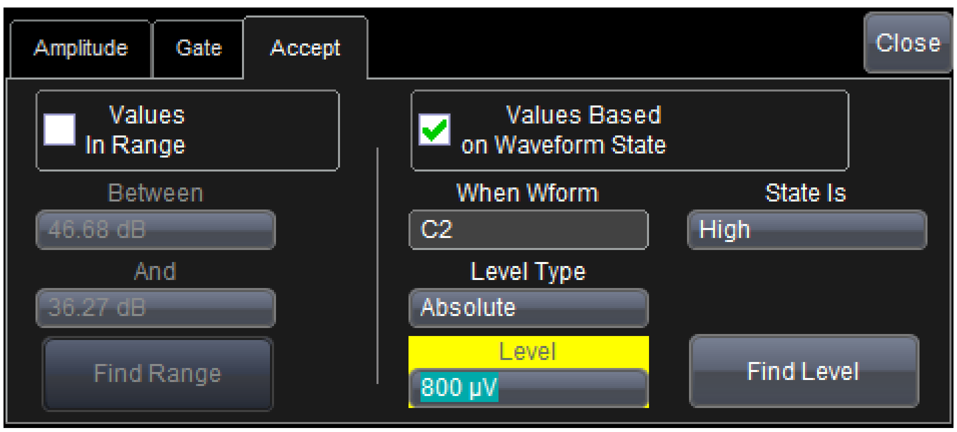
Waveform Gated Parameters

Setting Up Waveform Qualifiers

1. Touch **Measure** → **Measure Setup**.
2. Touch a **Px** tab to open the setup dialog.
3. Touch inside the **Source** field and select a source from the pop-up menu.
4. Touch inside the **Measure** field and select a parameter from the pop-up menu.
5. Touch the **Accept** tab of the dialog on the right, then touch the **Values Based on Waveform State** checkbox.
6. Touch inside the **When Wform** field and select the gating source.
7. Touch inside the **State Is** field and select **High** or **Low** from the pop-up menu. Parameter measurements on the subject waveform will only be taken when the gating waveform is in the selected state.
8. Touch inside the **Level Type** field and select **Absolute** or **Percent** from the pop-up menu.
9. Touch inside the **Level** field and enter the crossing level value at which you want measurements to begin using the scroll bar at the bottom of the window. Touch the keypad button  to enter a value using the pop-up keypad.

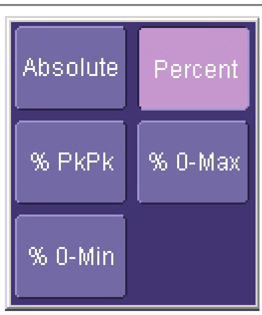
   OR

   Touch the **Find Level** button to automatically select the 50% level of your gating waveform.



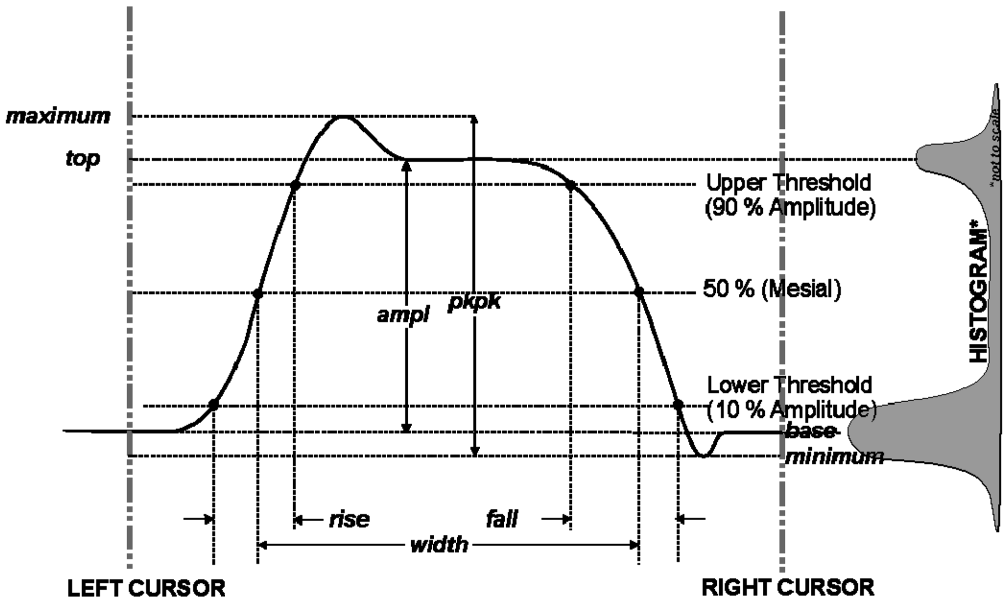
EMC Parameters

Not available for all scopes, the EMC Pulse Parameter software package includes enhanced Rise@level, Fall@level, Width@level, Time@level, and DTime@level parameters. The functionality includes user definable thresholds for accurate pulse measurements.

	**% pk-to-pk** is for ESD pulses that do not start at 0; they might go from negative to positive or positive to negative voltage. The risetime is measured as a percentage of peak to peak not base to top.
	% 0-to-max is for ESD pulses that start at 0 V and reach a peak positive voltage. The risetime is measured as a percentage of 0 to max instead of base to top of a traditional IEEE pulse.
	% 0-to-min is for ESD pulses that start at 0 V and reach a peak negative voltage. The risetime is measured as a percentage of 0 to min instead of top to base.

Determining Top and Base Lines

Proper determination of the top and base reference lines is fundamental for ensuring correct parameter calculations. The analysis begins by computing a histogram of the waveform data over the time interval spanned by the left and right time cursors. For example, the histogram of a waveform transitioning in two states will contain two peaks (see figure). The analysis will attempt to identify the two clusters that contain the largest data density. Then the most probable state (centroids) associated with these two clusters will be computed to determine the top and base reference levels: the top line corresponds to the top and the base line to the bottom centroid.



Determining Rise and Fall Times

Once top and base are estimated, calculation of the rise and fall times is easily done (see figure). The 90% and 10% threshold levels are automatically determined by the instrument, using the amplitude (ampl) parameter.

Threshold levels for rise or fall time can also be selected using absolute or relative settings (r@level, f@level) if these parameters are included in your scope. If absolute settings are chosen, the rise or fall time is measured as the time interval separating the two crossing points on a rising or falling edge. But when relative settings are chosen, the vertical interval spanned between the base and top lines is subdivided into a percentile scale (base = 0 %, top = 100 %) to determine the vertical position of the crossing points.

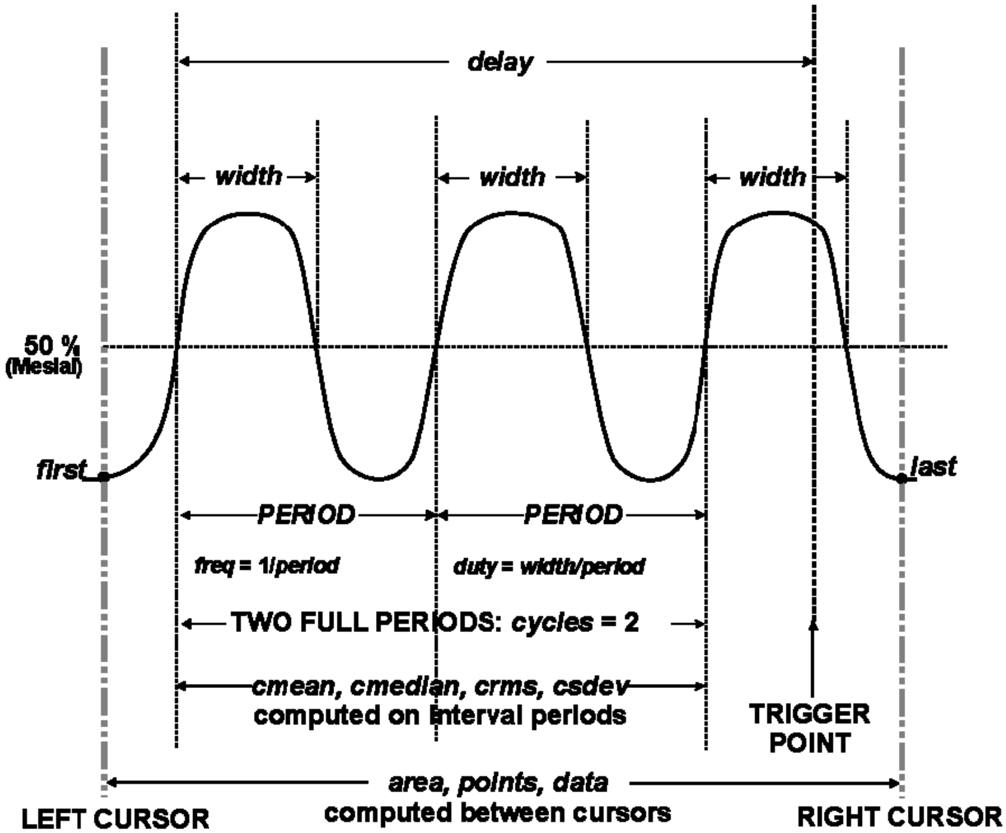
The time interval separating the points on the rising or falling edges is then estimated to yield the rise or fall time. These results are averaged over the number of transition edges that occur within the observation window.

Rising Edge Duration	$\frac{1}{Mr}\sum_{i=1}^{Mr}(Tr_i^{90}-Tr_i^{10})$
Falling Edge Duration	$\frac{1}{Mf}\sum_{i=1}^{Mf}(Tf_i^{10}-Tf_i^{90})$
Where *Mr* is the number of leading edges found, *Mf* the number of trailing edges found, Tr_i^x the time when rising edge *i* crosses the x% level, Tf_i^x and the time when falling edge *i* crosses the x% level.	

Determining Time Parameters

Time parameter measurements such as width, period and delay are carried out with respect to the mesial reference level (see figure), located halfway (50%) between the top and base reference lines.

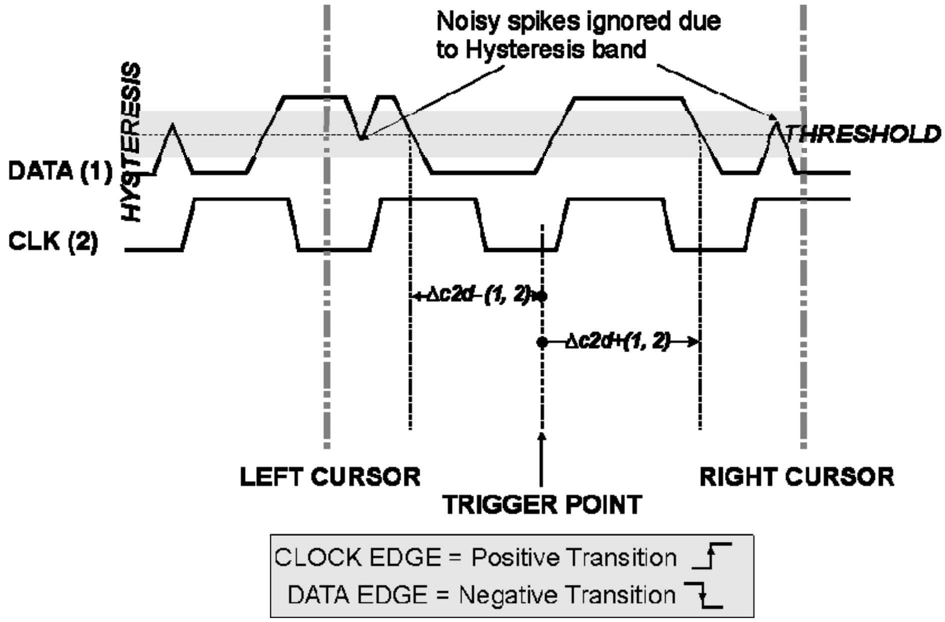
Time-parameter estimation depends on the number of cycles included within the observation window. If the number of cycles is not an integer, parameter measurements such as rms or mean will be biased. However, only the last value is actually displayed, the mean being available when statistics are enabled. To avoid these bias effects, cyclic parameters can be chosen, including crms and cmean, that restrict the calculation to an integer number of cycles.



Determining Differential Time Measurements

The instrument enables accurate differential time measurements between two traces: for example, propagation, setup and hold delays (see figure).

If included in your oscilloscope, parameters such as Delta c2d± require the transition polarity of the clock and data signals to be specified.



Moreover, a hysteresis range may be specified to ignore any spurious transition that does not exceed the boundaries of the hysteresis interval. In the figure, Delta c2d- (1, 2) measures the time interval separating the rising edge of the clock (trigger) from the first negative transition of the data signal. Similarly, Delta c2d+ (1, 2) measures the time interval between the trigger and the next transition of the data signal.

Level and Slope

For several time-based measurements, you can choose positive, negative, or both slopes to begin parameter measurements if these parameters are included in your scope. For two-input parameters, such as Dtime@level, you can specify the slope for each input, as well as the level and type (percent or absolute).

Print, Plot, or Copy

The instrument gives you the ability to output files to a printer or plotter, to print to file, or to e-mail your files. Any Microsoft Windows® supported printer is supported by your instrument.

Printing

Setting Up the Printer

1. Touch **File** → **Print Setup** on the menu bar. The Utilities **Hardcopy** dialog opens.

   OR

   Press the PRINT front panel button. Then, touch the **Print Setup** button on the **Print** flyout menu.

2. On the Hardcopy dialog, touch the **Printer** icon .
3. Under **Colors**, touch the **Use Print Colors** checkbox if you want the traces printed on a white background.

Note: A white background saves printer toner. (You can change the printer colors in the **Utilities** → **Preference Setup** → **Color** dialog.)

4. Touch inside the **Select Printer** field. Choose a printer from the pop-up menu.
5. Touch the **Properties** button to see your printer setup.
4. Touch the icon for the layout **Orientation** you want: portrait or landscape.

5. Touch the **Hardcopy Area** field to choose which part of the screen you want to print from the pop-up menu 

. Choose **Grid Area Only** if you do not need to print the dialog area and you only want to show the waveforms and grids. Choose **DSO window** if you want to print the dialogs with the waveforms and grids. Choose **Full Screen** if you want to print the entire screen.

Printing a Screen Image

You can print in one of three ways:

- Press the PRINT button  on the front panel. Then, touch the **Print Now** button on the **Print** flyout menu.
- Touch **File** → **Print** on the menu bar.
- Touch the **Print Now** button  in the **Utilities** → **Hardcopy** dialog

> **Note**: The instrument uses the Print settings on the **Utilities** → **Hardcopy** dialog to print the screen image.

Adding Printers and Drivers

> **Note:** If you want to add a printer driver, the driver must first be loaded on the scope.

1. Touch **File** → **Print Setup** on the menu bar.

   OR

   Press the PRINT front panel button. Then, touch the **Print Setup** button on the **Print** flyout menu.
2. On the Utilities **Hardcopy** dialog, touch the **Printer** icon .
3. Touch the **Add Printer** button. A Microsoft Windows® Printer window opens where you can add a printer.

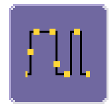
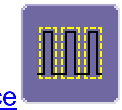
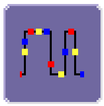
Managing Files

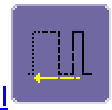
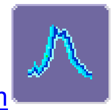
Use the instrument's **Utilities** menu to save the screen image as a file to storage media, such as a USB drive or hard drive. You can give your files custom names and create directories for them.

Hard Disk Partitions

The instrument's hard disk is partitioned into drive **C:** and drive **D:**. Drive C: contains the Windows operating system and the instrument application software. Drive D: is intended for data files.

Sampling Modes

Depending on your timebase, you can choose either Single-shot (Real Time) , Sequence , or RIS  mode sampling.

Some instruments also offer two other modes: Roll  and WaveStream .

Selecting a Sampling Mode

1. Touch **Timebase** → **Horizontal Setup** from the menu bar.
2. In the **Timebase** dialog, touch a **Sample Mode** button.

Single-shot Sampling Mode

Basic Capture Technique

A single-shot acquisition is a series of digitized voltage values sampled on the input signal at a uniform rate. It is also a series of measured data values associated with a single trigger event. The acquisition is typically stopped a defined number of samples after this event occurs: a number determined by the selected trigger delay and measured by the timebase. The waveform's horizontal position (and waveform display in general) is determined using the trigger event as the definition of time zero.

You can choose either a pre- or post-trigger delay. Pre-trigger delay is the time from the left-hand edge of the display grid forward to the trigger event, while post-trigger delay is the time back to the event. You can sample the waveform in a range starting well before the trigger event up to the moment the event occurs. This is 100% pre-trigger, and it allows you to see the waveform leading up to the point at which the trigger condition was met and the trigger occurred. (The instrument offers up to the maximum record length of points of pre-trigger information.) Post-trigger delay, on the other hand, allows you to sample the waveform starting at the equivalent of 10,000 divisions after the event occurred.

Because each instrument input channel has a dedicated ADC (Analog-to-Digital Converter), the voltage on each is sampled and measured at the same instant. This allows very reliable time measurements between the channels.

On fast timebase settings, the maximum single-shot sampling rate is used. But for slower timebases, the sampling rate is decreased and the number of data samples maintained.

The relationship between sample rate, memory, and time can be simply defined as:

$$Capture\ Interval = \frac{1}{Sample\ Rate} \times Memory$$

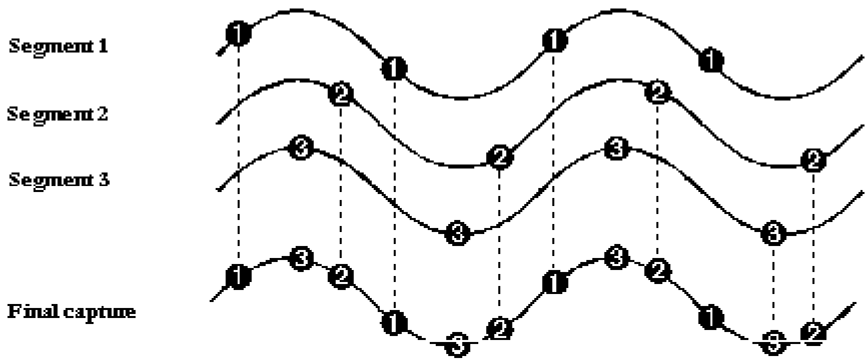
and

$$\frac{Capture\ Interval}{10} = Time\ Per\ Division$$

RIS Sampling Mode - For Higher Sampling Rates

RIS (Random Interleaved Sampling) is an acquisition technique that allows effective sampling rates higher than the maximum single-shot sampling rate. It is used on repetitive waveforms with a stable trigger. The maximum effective sampling rate of 50 GS/s can be achieved with RIS by making 100 single-shot acquisitions at 500 MS/s. The bins thus acquired are positioned approximately 20 ps apart. The process of acquiring these bins and satisfying the time constraint is a random one. The relative time between ADC sampling instants and the event trigger provides the necessary variation, measured by the timebase to 5 ps resolution.

The instrument requires multiple triggers to complete an acquisition. The number depends on the sample rate: the higher the sample rate, the more triggers are required. It then interleaves these segments (see figure) to provide a waveform covering a time interval that is a multiple of the maximum single-shot sampling rate. However, the real-time interval over which the instrument collects the waveform data is much longer, and depends on the trigger rate and the amount of interleaving required. The oscilloscope is capable of acquiring approximately 40,000 RIS segments per second.



> **Note:** RIS mode is not available when the scope is operating in Fixed Sample Rate mode.

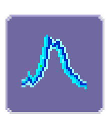
Roll Mode

If available on your instrument, Roll mode is invoked automatically when the time per division is 500 ms/div or greater. However, you can cancel Roll Mode and return to Real Time mode at any time.

Roll mode displays, in real time, incoming points in single-shot acquisitions that have a sufficiently low data rate. The oscilloscope rolls the incoming data continuously across the screen until a trigger event is detected and the acquisition is complete. The parameters or math functions connected to each channel are updated every time the roll mode buffer is updated, as if new data is available. This resets statistics on every step of Roll mode that is valid because of new data.

> **Note:** If the processing time is greater than the acquire time, the data in memory gets overwritten. In this case, the instrument issues the warning: **Channel data is not continuous in ROLL mode!!!** and rolling will start over again.

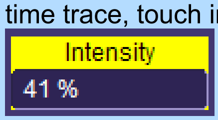
WaveStream Mode

	Not available on all instruments, this fast viewing mode provides brightness-graded intensity with a decay time similar to the action of phosphor on an analog screen. WaveStream mode operates at up to 10 GS/s with an update rate up to 8000 waveforms/second for better capture of higher frequency abnormal events.

Adjusting Trace Intensity

The **Intensity** knob adjusts the brightness of your trace, with the intensity value displayed in the bottom right-hand corner of the screen:

Pushing the knob toggles between WaveStream fast-viewing mode and real-time mode.

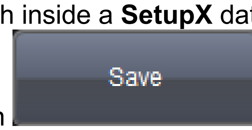
Note: The **Intensity** knob controls trace intensity only for WaveStream traces. Turning the knob when the sampling mode is Real Time will also show a change at the bottom of the screen, but the trace intensity of the real-time trace will not change; i.e., the value displayed is for the WaveStream trace. To change the trace intensity of a real-time trace, touch inside the **Intensity** field 

of the Display dialog and use the pop-up keypad or turn the front panel **Adjust** knob.

Saving and Recalling Scope Settings

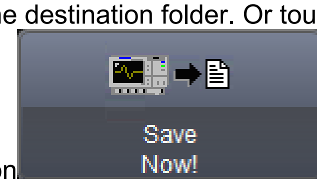
You can quickly save and recall up to six scope panel settings internally. These internally saved settings are easy to recall later. You can also save and recall your scope panel settings as a file (.lss) on the hard disk, a network location, USB drive, or another location.

Saving Scope Settings

1. Touch **File** →**Save Setup** on the menu bar.
2. To save the scope settings internally, under **Save to Internal Setup**, touch inside a **SetupX** data entry field and use the pop-up keyboard to enter a file name. Touch the Save button 

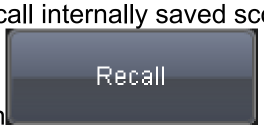
next to the **SetupX** data entry field. The file is saved in **D:\Internal Setups**, and the current date/time is displayed above the field.

   OR

   To save the scope settings to a file, under **Save To File**, touch inside the **Save panel to file** data entry field and use the pop-up keyboard to enter the path for the destination folder. Or touch **Browse** to navigate to the destination folder. Then, touch the Save Now button 

.

Recalling Scope Settings

1. Touch **File** → **Recall Setup** on the menu bar.
2. To recall internally saved scope settings, under **Recall from Internal Setup**, touch the Recall button 

next to the file you want to recall.

   OR

To recall scope settings from a saved file, under **Recall From File**, touch inside the **Recall panel from file** data entry field and use the pop-up keyboard to enter the path to the source folder. Or touch **Browse** to navigate to the source folder. Then touch the Recall Now button 

.

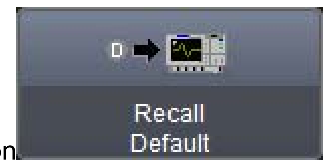
Recalling Default Settings

Some of the default settings for your WavePro 700Zi oscilloscope include the following configurations:

- Channels 1 and 2 are turned on at 50 mV/div, 0V offset, linear interpolation
- All other measurements are cleared for Math, Reference Waveforms, and Channels
- Trigger is set to Channel 1 with an Auto positive edge, DC coupled, and with a 0V level
- The Timebase set to 50 ns/div, 10ks, 20 GS/s, and with zero delay
- Smart memory is set to Set max memory with 100ks selected
- The Cursors are turned off
- The Grids are set to AutoGrid

If you want to reset the settings to the default configuration, press the **DEFAULT SETUP** front panel button.

OR

Touch **File** → **Recall Setup** on the menu bar. Touch the Recall Default button 

under **Recall Default Setup**.

Saving and Recalling Waveforms

Saving Waveforms

1. Touch **File** → **Save Waveform** on the menu bar.
2. In the **Save Waveform** dialog, touch the **Save To** Memory button 

or File button 

.
3. Touch inside the **Source** field and select a source from the **Select Source** pop-up window. The source can be any trace; for example, a channel (C1–C4), math function (F1–F4), or a waveform stored in non-volatile RAM (M1–M4).
4. Touch inside the **Trace Title** data entry field if you want to change the default name of your waveforms. Use the pop-up keyboard to type the new name.

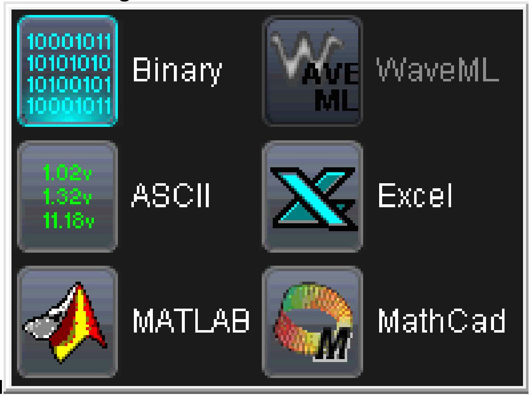
Note: You can change the name but not the sequence number.

CAUTION

If you use a name that ends in a number instead of a letter, the instrument may truncate the number. This is because, by design, the first waveform is automatically numbered 0, the second 1, etc. For example, if you want to use waveform name "XYZ32" but it is not preceded by waveforms XYZ0 through XYZ31, the waveform will be renumbered with the next available number in the sequence.

If you need to use a number in your waveform's name, it is recommended that you append an alpha character at the end of the number : "XYZ32a" for example.

5. If you are saving to file, touch the **Data Format** field and select a format type from the pop-up menu



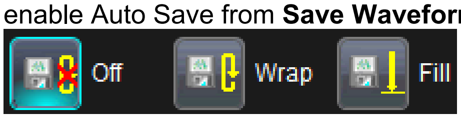
Depending on your selection, you may need to touch the **SubFormat** field and select a subformat. If you select **ASCII**, touch the **Delimiter** field and select a delimiter character from the pop-up menu: comma, space, semicolon, or tab.

Note: The **WaveML** format, which enables XML output, is used for persistence traces.

6. Touch the **Browse** button next to the **Save file in directory** field and browse to the location where you want the file saved. The file name is assigned automatically and is shown below the field.
7. Touch the Save Now button .

Auto Save

You can use the Auto Save feature to automatically save a waveform to disk after each new trigger. You can enable Auto Save from **Save Waveform** dialog by touching one of the Auto Save buttons



Select **Wrap** (old files overwritten) or **Fill** (no files overwritten).

CAUTION

If you select Fill, you can quickly use up all disk space on your hard disk.

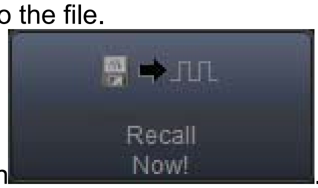
Recalling Waveforms

Only .trc files that were saved in binary format can be recalled into the oscilloscope.

1. Touch **File** → **Recall Waveform** on the menu bar.
2. To recall a waveform from memory, in the **Recall Waveform** dialog, touch the **Recall From** Memory button 

. Touch inside the **Source** field and select a memory location in which to store the file: M1 to M4.

   OR
   To recall a waveform from a file, in the **Recall Waveform** dialog, touch the **Recall From** File button 

. Touch inside the **Destination** field and select a memory location in which to store the file: M1 to M4. Touch inside the **Show only files** field and select an area to limit the search to: channels, math functions, or memory. Touch inside the **Recall files from directory** data entry field and enter the path using the pop-up keyboard. Or, touch the **Browse** button to navigate to the file. Touch inside the **Next file will be recalled from data entry** field and enter the path using the pop-up keyboard. Or, touch the **Browse** button to navigate to the file.
3. Touch the Recall Now button 

.

Disk Utilities

Use the Disk Utilities dialog to delete files or create folders.

> **Note:** All of the Disk Utilities can also be accomplished using the standard file management tools in Microsoft Windows®.

Deleting a Single File

1. Touch **File** → **Disk Utilities** on the menu bar.
2. Touch the Delete button  in the **Disk Utilities** dialog.
3. Touch inside the **Current folder** data entry field and use the pop-up keyboard to enter the path to the folder that contains the file you want to delete.

   OR

   Touch the **Browse** button and navigate to the folder.
4. Touch inside the **File to be deleted** data entry field and use the pop-up keyboard to enter the name of the file.

   OR

   Touch the **Browse** button and navigate to the file.
5. Once you have located the file, touch the **Delete File** button.

Deleting All Files in a Folder

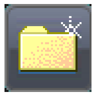
1. Touch **File** → **Disk Utilities** on the menu bar.
2. Touch the Delete button  in the **Disk Utilities** dialog.

3. Touch inside the **Current folder** data entry field and use the pop-up keyboard to enter the path to the folder that contains the file you want to delete.

   OR

   Touch the **Browse** button and navigate to the folder.
4. Once you have located the folder, touch the **Empty Folder** button.

Creating a Folder

1. Touch **File** → **Disk Utilities** on the menu bar.
2. Touch the Create button  in the **Disk Utilities** dialog.
3. Touch inside the **Current folder** data entry field and use the pop-up keyboard to enter the path to the directory where you want to create the folder, and then name of the folder.
4. Touch the **Create Folder** button.

Introduction to Serial Decode

Overview

A variety of Serial Data standards, such as Inter-IC (I^2C) and Serial Peripheral Interface (SPI), govern communication between microprocessors and peripherals. I^2C is a standardized protocol created by Philips with a documented technical specification. SPI was popularized by Motorola but is not standardized, per se - there are a variety of variants with the differences characterized by how data is clocked, whether data is MSB or LSB format, and whether it is multi-slave or single-slave. A summary of the differences is outlined in the following table:

Standard	Definition	Number of Lines	Data Rate	Synchronous or Asynchronous
I^2C	Inter IC	2	Up to 3.4 Mb/s	Synchronous
SPI	Serial Peripheral Interface	4	Up to ~50 Mb/s	Synchronous

I^2Cbus D, I2Cbus TD, SPIbus D and SPIbus TD are unique oscilloscope tools from LeCroy that greatly enhance your ability to debug and analyze embedded controllers that use I^2C or SPI Bus communications. The D products are Decode only, and the TD products include Serial Trigger and Decode. The serial triggers are integrated into the oscilloscope -- no external hardware is used -- and are selected through the normal oscilloscope trigger menus. I^2C and SPI signals are input to the oscilloscope through normal passive or active probes, such as LeCroy's ZS Series of high impedance active probes. Decoding is accessed from the **Analysis** pull-down menu in the menu bar. The decoding is overlaid on top of the appropriate channel, and is intuitively presented and color-coded for quick understanding. All packages contain Search capability for specific messages, and a Table to display protocol data in summary form underneath the oscilloscope grid.

I2Cbus TD and SPIbus TD are available for WaveRunner Xi and WaveSurfer Xs oscilloscopes.

I2Cbus D and SPIbus D are available for WaveRunner 6000, WavePro 7000 and WaveMaster 8000 oscilloscopes.

The I2C-BUS Specification published by Philips Semiconductors fully describes the I^2C standard. As of the date of printing of this manual, Version 2.1, January 2000 is the most recent version (though this does not include the recent FM+ update).

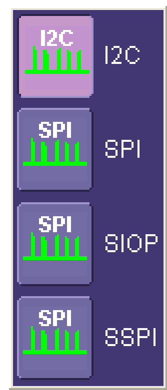
There is no formal SPI standard. Descriptions of SPI and its variants are usually included in the technical documentation for the microprocessor that offers support for SPI.

This manual presumes that you have a basic understanding of I^2C and SPI physical layer and protocol layer specifications, and knowledge of how I^2C and SPI are used in embedded controllers. It also presumes a basic understanding of oscilloscope operation, specifically the LeCroy oscilloscope that the I^2C or SPI option will be used with. Where practical or necessary, detail on specific oscilloscope features is included in this manual.

Note: LeCroy has a policy of frequently updating software. It is possible that screen images in this manual may not exactly match what you see on your oscilloscope display. However, functionality will be nearly identical.

TD Series Software

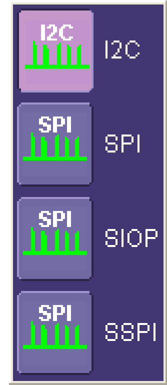
The TD option adds the following capability to the LeCroy oscilloscope software user interface dialogs:

Serial Trigger Selection	If this is the first serial trigger option you have installed on your scope, an additional icon will appear in your trigger dialog box. It allows a serial trigger condition to be set from within the oscilloscope using an easy-to-understand interface.	
Serial Decode	If this is the first serial decode option you have installed on your scope, an additional set of Serial Decode and Decode Setup dialog boxes will be provided for setup of protocol format (as necessary) and decoding. These can be accessed from the **Analysis** menu.	
Decode Protocol Selections	As serial decode options are added to your oscilloscope, additional protocol selections are available in a pop-up menu within the Decode Setup dialog box.	

D Series Software

The D option adds the following capability to the LeCroy oscilloscope software user interface dialogs:

Serial Decode	If this is the first serial decode option you have installed on your scope, an additional set of Serial Decode and Decode Setup dialog boxes will be provided for setup of protocol format (as necessary) and decoding.	

Decode Protocol Selections	As serial decode options are added to your oscilloscope, additional protocol selections are available in a pop-up menu within the Decode Setup dialog box.	

Technical Overview

LeCroy's offering of serial trigger and decode options utilize advanced trigger circuitry and advanced software algorithms to provide powerful capability for serial data triggering and decoding.

Serial Trigger

TD options contain advanced serial data triggering. This serial data triggering is implemented directly within the hardware of the oscilloscope acquisition system, and contains advanced algorithms to protocol decode, recognize, and trigger on user-defined serial data patterns. This allows a recognized serial data pattern to be used to trigger the oscilloscope at a pre-determined time, and other signals coincident with the desired serial data pattern can be captured simultaneously.

Serial Decode

Both the D and TD options contain powerful protocol decoding and annotation software algorithms. This algorithm is used in all LeCroy serial decoders sold with oscilloscopes, and differs slightly for serial data signals that have a clock embedded in data or a clock separate from data.

The software algorithm examines the embedded clock (CAN) or separate clock line (I^2C, SPI) for each message based on a default (or user set) vertical level. The algorithm is intelligent in that it applies a hysteresis to the rising and falling edge of the serial data signal to minimize the chance that perturbations or ringing on the edge will affect the decoding. The default level is usually set to 50% and is determined from a measurement of peak amplitude of the signals acquired by the oscilloscope. It can also be set to an (absolute) voltage level, if desired. For serial data signals with embedded clocks (i.e., CAN), the algorithm then performs an analysis of the serial data message to determine the nominal bit width. The clock rate/bit width is measured directly from the clock line for other serial data formats (i.e., I^2C and SPI). Once the clock signal is acquired and the decoding is completed for a serial data message with separate clock and data lines, the oscilloscope channel can be turned OFF to reduce screen clutter.

After determining bit width, a different algorithm performs a decoding of the serial data message into binary format after separation of the underlying data into logical groups (Header/ID, Data Length Codes, Data, CRC, Start Bits, Stop Bits, etc.). Finally, another algorithm provides the appropriate color coding of the message, and displays the protocol message data on the screen, as desired, overlaid on the source trace. Various compaction schemes are utilized to show the data during a long acquisition (many hundreds or thousands of serial data messages) or a short acquisition (one serial data message acquisition). In the case of the longest acquisition, only the most important information is highlighted. In the case of the shortest acquisition, all information is displayed (Header/ID, Data Length Codes, Data, CRC, Start Bits, Stop Bits, etc.) with additional highlighting of the complete message frame.

Note that although the decoding algorithm is based on a clock extraction software algorithm using a vertical level, the results returned are the same as those from a traditional protocol analyzer using sampling point-based decode. In addition, the clock extraction technique allows partial decoding of messages in the event of physical layer noise, in many cases, whereas a protocol analyzer usually cannot. This is a significant advantage for the LeCroy software algorithm.

If the sampling rate (SR) is insufficient to resolve the signal adequately, based on the bit rate (BR) setup or clock frequency, then the protocol decoding will be turned OFF in order to protect the operator from incorrect data. The minimum SR:BR ratio required is 4:1. It is suggested that you use a slightly higher SR:BR ratio if possible, and use significantly higher SR:BR ratios if you want to also view perturbations or other anomalies on your serial data analog signal.

Table Display

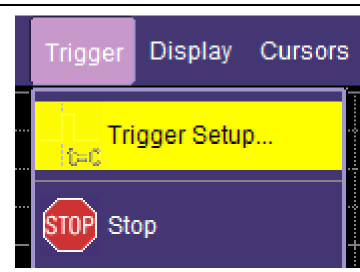
The tabular display of serial decoded data is a powerful feature to allow you to see all of your protocol messages, even if the compaction of serial data messages on the oscilloscope grid means that annotation is impractical. The table uses as its source the decoded data, extracted as described above. Therefore, if **View Decode** is not checked ON, then the table will not be displayed.

Accessing Overview

TD trigger and decoding tools are easily accessible in a variety of ways. The TD options add an additional **Serial** selection to the "Trigger Type" menu in the Trigger dialog, and a new set of dialogs for Decode setup. These dialogs are shared by all the low-speed serial protocols that LeCroy offers, so all serial trigger and serial decoding selections will be grouped into a common selection with nearly identical selection and setup. These dialogs are conveniently accessed with just one or two touches of the screen.

Trigger

Methods to access the Serial Trigger dialogs are listed below:

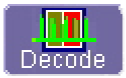
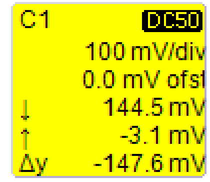
Touch the Trigger descriptor label. Then select the Serial  trigger type in the dialog box.	
Touch **Trigger** in the menu bar, and select **Trigger Setup...** from the drop-down menu. Then select the Serial  trigger button and choose the appropriate serial trigger.	

Serial Decode and Decode Setup

These dialogs provide the ability to set the oscilloscope up for protocol decoding of serial data messages, with display of the protocol data overlaid on the signal. They also allow quick and easy access to oscilloscope zooming, searching, table display, and table export.

The serial decode and decode setup dialogs are accessed in any the following ways:

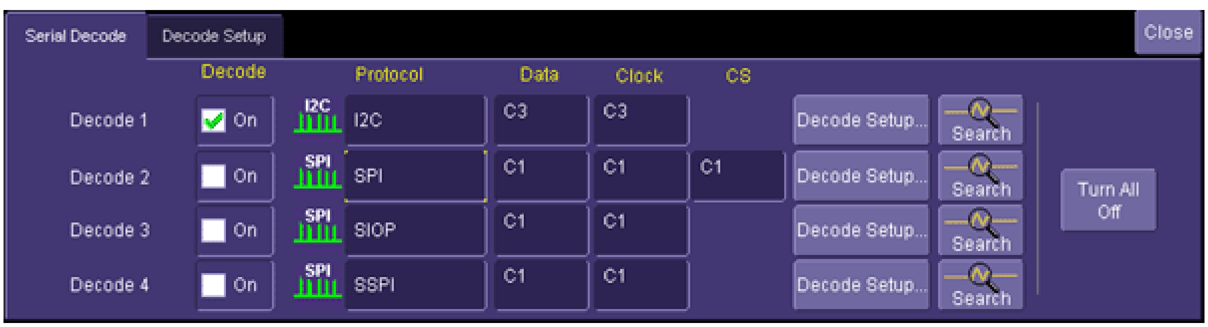
Touch **Analysis** in the menu bar, and select **Serial Decode...** from the drop-down menu. This will open the **Serial Decode** summary dialog, and provide access to **Decode Setup**.	

Touch the Channel or Memory Descriptor Box to open the respective dialog box, and touch the **Decode** button  in the bottom toolbar.	
Touch a Channel, Memory, or Math trace to open a pop-up dialog that displays a shortcut to the Decode Setup dialog box	

There is a great deal of commonality in decode setup among the various serial data standards. Therefore, the common areas are discussed next. Specifics about triggering and decoding of each protocol are discussed in later chapters.

Serial Decode (Summary) Dialog Box

The Serial Decode dialog box shows a summary of which decoders are ON, and how they are set up. In addition, there are shortcuts to Decode Setup and Search. A sample dialog box is shown below. This dialog box reflects an oscilloscope that has both the I^2C and SPI options.



There are four independent decoders. A user can operate up to four at a single time, though the limitation may be on the number of channels that can be accommodated at one time. Practically speaking, if a user were decoding signals with a clock and data line (and perhaps also a chip select or other third line), then two simultaneous decodes would be the maximum number.

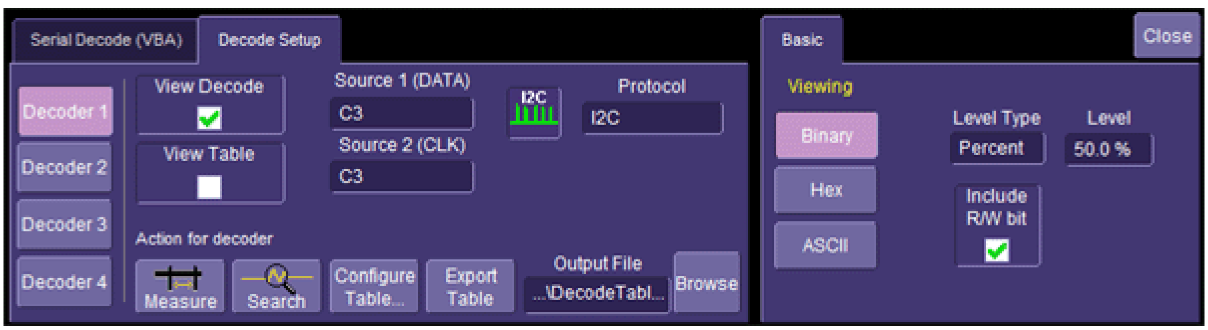
Details of the Serial Decode dialog box are described below:

Decoder # - There are four independent decoders; each can be set up a different way for a different protocol.	

Decode ON Checkbox - If checked, it means the decoder is turned ON and will decode, assuming the protocol is correctly set up in the **Decode Setup** dialog box.	
Protocol - This pop-up dialog allows selection of a specific serial protocol. In some cases where the protocol is not completely standardized, or where there are higher-level definitions of the protocol, multiple selections may be provided. For example, SPI has variants with no chip select. Examples are Simplified SPI (SSPI), and Simple Synchronous Serial I/O Port, (SIOP). Each of these has a selection in this pop-up dialog.	
Data and Clock Selection - This pop-up dialog allows you to select a channel or other source for decoding. Some protocols may require a third selection (for instance, SPI also requires a Chip or Slave Selection).	
Decode Setup Shortcut Button - This provides quick access to the second tab (Decode Setup) where there are quick buttons for Search, and Table (Configure Table and Export Table).	
Search - Push this button to open a Zoom (Zx trace) that has as its source the corresponding Channel (Cx trace). In addition, the right-hand dialog in the Zoom trace has Search options specific to the serial protocol that the Source is assigned to.	

Decode Setup Dialog

The Decode Setup dialog box is where the details of a specific protocol decode are entered. It appears as follows:

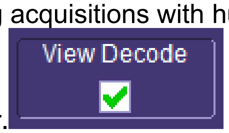


This is a single tab with an indicator on the left side that describes which of the four decoders the setup information pertains to. The left-hand side of this dialog box is described as follows (the right side is described in the protocol-specific sections):

1. **Decoder # Buttons** - This describes which of the four decoders to which the currently entered information

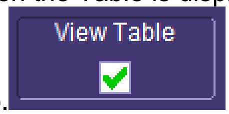
pertains.

2. **View Decode Checkbox** - Check this box to turn on decoding. Decoding ON will provide a highlight of each message frame, with color-coded highlighting and decoding of the various portions of the protocol message. Note that for very long acquisitions with hundreds or thousands of messages, decoding of the entire acquisition will take longer.

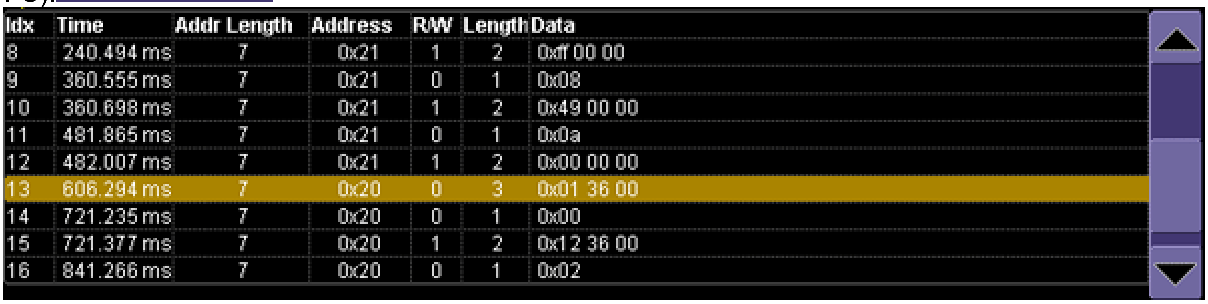


3. **View Table Checkbox** - Checking this box turns the Table ON. Un-checking it turns the Table OFF.

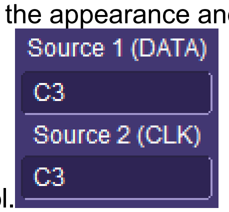
> **Note:** If the View Decode checkbox is not checked (i.e., decode is not ON), then the View Table checkbox will be grayed out. The Table cannot be displayed unless decode is occurring.

When the Table is displayed, it appears similar to that shown to the right (the example shown is for 

I^2C).



Idx	Time	Addr Length	Address	R/W	Length	Data
8	240.494 ms	7	0x21	1	2	0xff 00 00
9	360.555 ms	7	0x21	0	1	0x08
10	360.698 ms	7	0x21	1	2	0x49 00 00
11	481.865 ms	7	0x21	0	1	0x0a
12	482.007 ms	7	0x21	1	2	0x00 00 00
13	606.294 ms	7	0x20	0	3	0x01 36 00
14	721.235 ms	7	0x20	0	1	0x00
15	721.377 ms	7	0x20	1	2	0x12 36 00
16	841.266 ms	7	0x20	0	1	0x02

4. **Source Selection** - Touch inside the Source fields and choose a source to use for Clock, Data, and (for some protocols) a third line (e.g., Chip Select for SPI). The Source selection is dynamically linked to the Protocol selection, so the appearance and number of sources to choose from will change depending on your selected Protocol. 

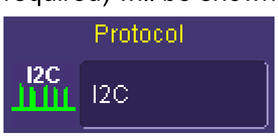
This source can be either a Channel (C1 to C4), a Memory Trace (M1 to M4), or a Math Function (F1 to F4).

A Channel would be used for a new, real-time acquisition.

A Memory would be used if you had saved data from a previous acquisition and were recalling it to do further analysis. Reference the oscilloscope's on-line Help for information about how to Store and Recall Waveforms.

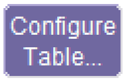
A Math Function would be used to view decoded data on Sequence mode acquisitions. Sequence Mode is a unique capability whereby you can utilize oscilloscope memory to capture events widely spaced in time, and view them sequentially. Reference the chapter on Isolating and Analyzing Serial Bus Activity for more information on setting the oscilloscope up in this mode.

5. **Protocol Selection** - Touch this selection to open a pop-up dialog box that allows you to select protocol decoders. Depending on the decoder selected, the correct inputs (i.e., Clock, Data, and a third line, if required) will be shown to the left.



6. **Action for Decoder Toolbar** - Various buttons on this toolbar provide context-sensitive shortcuts for decoding.
   - **Search** allows quick creation of a zoom trace and changes the dialog box to the zoom/search dialog box. Acquire long records of message data, and use Search to look through the record for a particular message. Upon finding a message that meets the search criteria, the complete message will be displayed with the Zoom Trace. Touch the arrow buttons to advance to the next message, or revert to the previous message. Unsuccessful searches will be noted in a text line.

- **Configure Table** displays a pop-up dialog box specific to a particular protocol. The pop-up dialog contains checkboxes for various columns in the table. Check the boxes to display that particular column in the table.

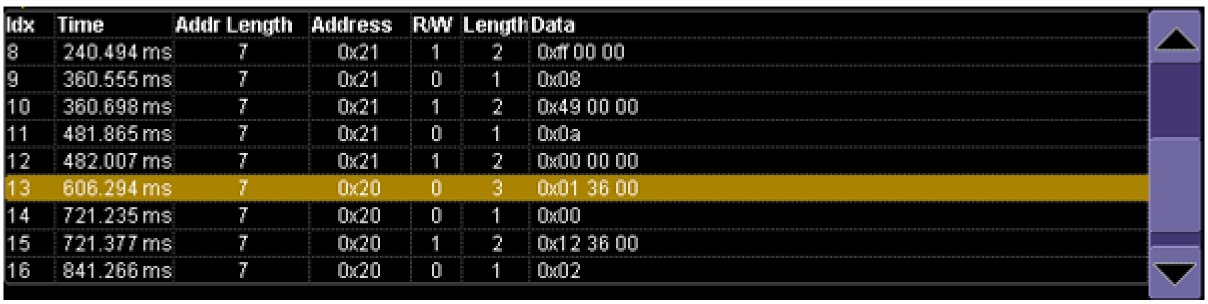


- **Export Table** exports the complete protocol table data to a user-defined file. The output file name and directory can be selected by the user, using the controls to the right.

Protocol Results Table

The protocol results table provides a quick and easy way to understand all of your protocol data as decoded by the oscilloscope, even when messages are too compacted to allow annotation on the display. In addition, the table provides a quick and easy method to look at decode results, and quickly zoom to a specific message.

When displayed, the protocol results table will appear under the waveform grid. It looks like the following example (this example is for I^2C – each protocol table will look slightly different):

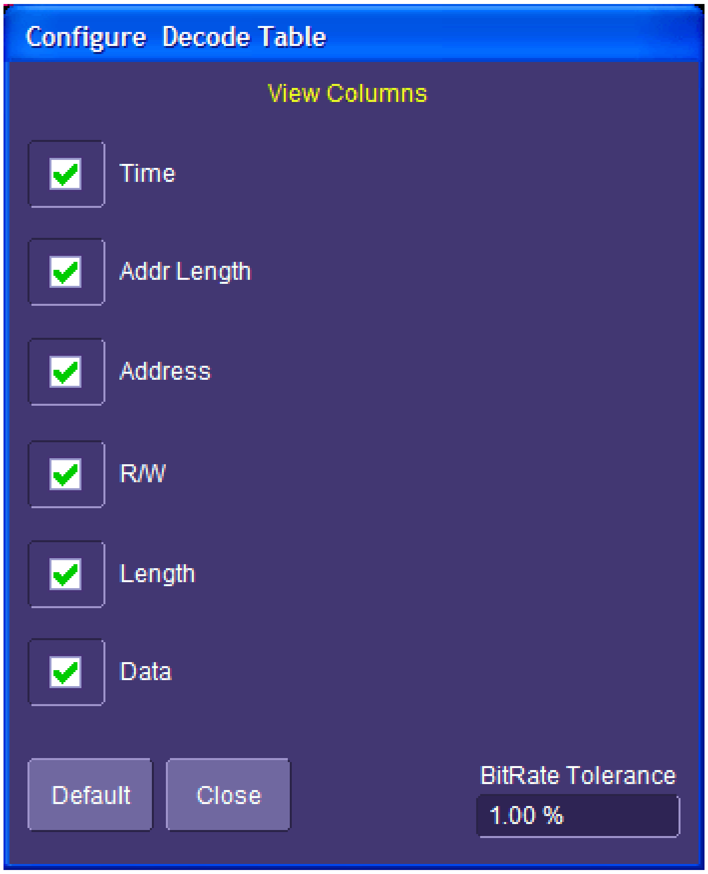


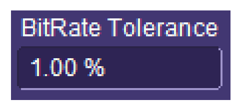
Idx	Time	Addr Length	Address	R/W	Length	Data
8	240.494 ms	7	0x21	1	2	0xff 00 00
9	360.555 ms	7	0x21	0	1	0x08
10	360.698 ms	7	0x21	1	2	0x49 00 00
11	481.865 ms	7	0x21	0	1	0x0a
12	482.007 ms	7	0x21	1	2	0x00 00 00
13	606.294 ms	7	0x20	0	3	0x01 36 00
14	721.235 ms	7	0x20	0	1	0x00
15	721.377 ms	7	0x20	1	2	0x12 36 00
16	841.266 ms	7	0x20	0	1	0x02

The slider bar on the right can be used to navigate through the protocol table. If the slider bar is yellow, then you can use the ADJUST knob on the oscilloscope front panel to move the slider bar.

If you touch a row, a decoded zoom trace will be created that will display that message trace in a zoom.

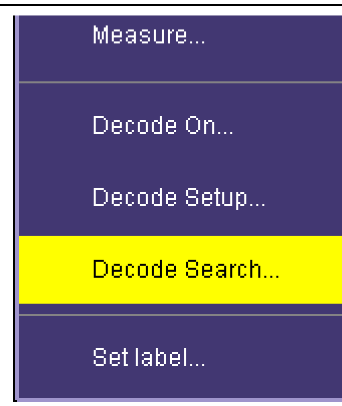
The table will only be displayed if the **View Table** checkbox is checked and decoded has occurred on the trace. Only one protocol table can be viewed at a time. As described in the preceding section, the protocol table can be configured or exported. If you press the **Configure Table** button  in the **Decode Setup** dialog box, a pop-up dialog similar to the one below will appear:

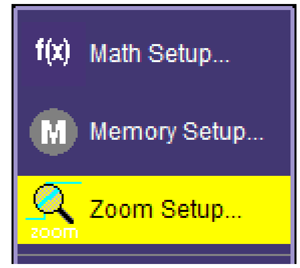


Default - Press this button to return to the default state of the table for that particular protocol.	
BitRate Tolerance - Some protocols have a Bit Rate Tolerance setting. This can be set to any value from 0.01% to 10%. If the bit rate is outside the tolerance range set, then the calculated bit rate will appear as red text in the table.	

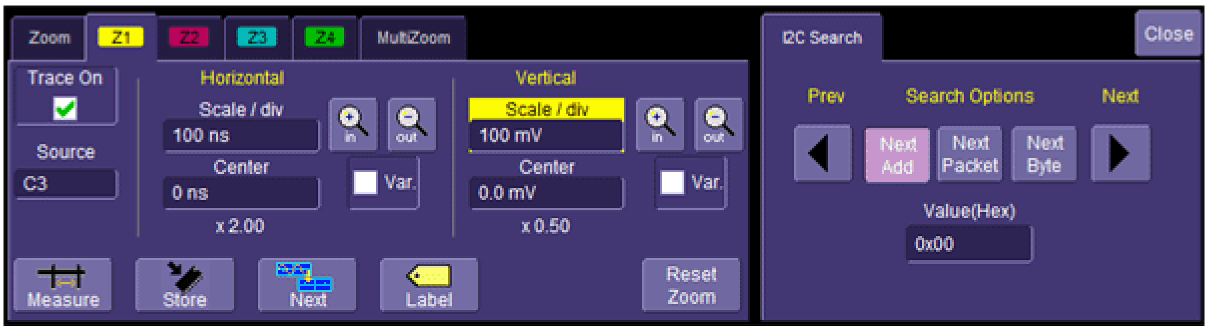
Searching for Messages

There are several ways to search for specific messages.

Touch the decoded waveform to open a pop-up dialog.	
Touch the **Search** button in the **Serial Decode** summary dialog box or in the **Decode Setup** dialog box.	

From the **File** menu, go to **Math**, and choose **Zoom Setup** to turn a zoom ON, define its source, and search directly.	

Any of these methods will take you to a Zoom dialog box with a right-hand dialog box with Search



The Search capability will differ by protocol. For instance, SPI has no Address, so there is no capability to Search by Address in SPI as there is in I^{2}C.

Use the Search Options buttons to define the type of Search you want, enter a value in Hexadecimal format, and use the left and right arrows to move your way from one message to the next.

Overview of I^{2}CBus Options

Both I2Cbus TD and D options contain powerful software algorithms to extract serial data information from physical layer waveforms measured on your oscilloscope. The extracted information is overlaid (annotated) on the actual physical layer waveforms, and color coded to provide fast, intuitive understanding.

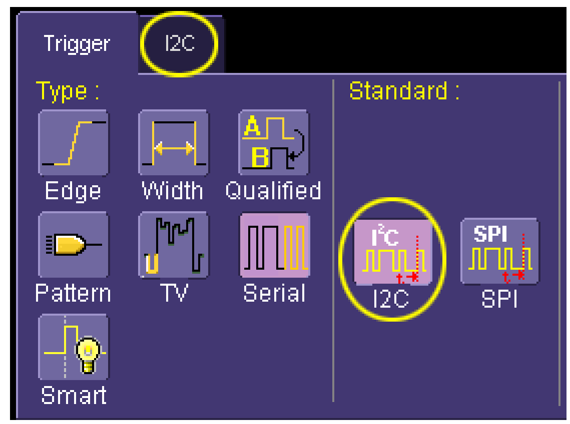
The I2Cbus TD option contains a very powerful and flexible trigger, but it is also very easy to set up for basic triggering. The I2Cbus TD option contains a conditional I2C DATA trigger to select a range of DATA values to trigger on, not just a single DATA value. Oftentimes, I^{2}C utilizes DATA bytes to specify sub-addresses for accessing memory locations in EEPROMs. Conditional DATA trigger allows triggering on a range of DATA bytes that correspond to reads or writes to specific sub-address memory blocks in the EEPROM. It can also aid in monitoring DATA outputs from I^{2}C-based sensors, such as analog-to-digital converters, and triggering when DATA is outside a safe operating range. In both cases, verifying proper operation becomes a simple task. Other powerful and user-friendly features included in I2Cbus TD trigger include:

- Ability to define an ADDR or DATA condition in either binary or hexadecimal formats.
- Ability to define an ADDR condition in binary with the DATA condition defined in hexadecimal so as to trigger on a range of ADDR values using Don't Care bits.
- FRAME LENGTH trigger setups
- EEPROM trigger setups to trigger on up to 96 bits (12 bytes) of DATA at any location within an I^{2}C frame or at a user-defined location in a 2048 byte window.
- All permutations of Read, Write, or R/W Don't Care conditional setup for 7 and 10-bit addresses.
- For any I^{2}C message trigger, select whether an ACK condition should be ACK, NO ACK, or DON'T CARE. You can choose to trigger on a NO ACK condition by itself, or as part of a more complex ADDR/DATA trigger.

If you are not familiar with or are just learning about I^{2}C, begin by using the simplest trigger conditions (Start, Stop, ReStart, NoAck) to gain confidence, then set up simple ADDR only conditions. When you are confident that you understand I^{2}C operation, set up an ADDR+DATA condition with a condition of "DATA =". Then, try different setups using other DATA conditions (>, <, INRANGE, etc.). Lastly, experiment with the EEPROM trigger setup, which provides the most flexibility by allowing location of data, with conditions, within specific bytes of a long sequence of DATA bytes.

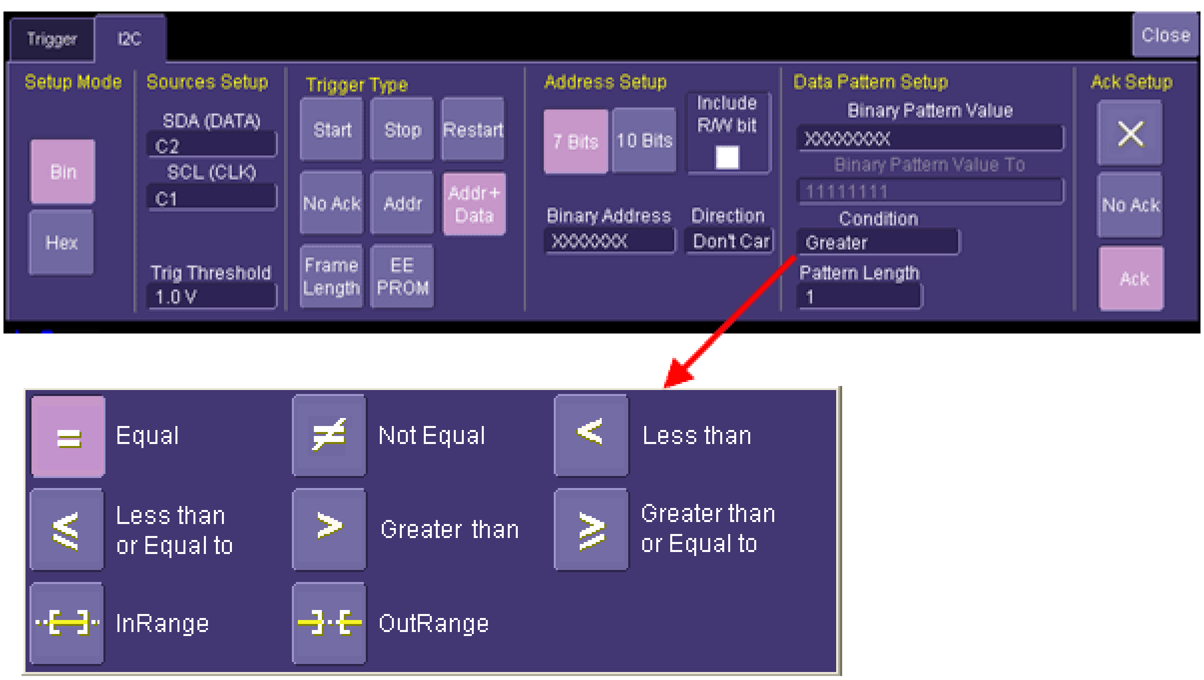
Accessing Serial Triggers

The I^{2}C and SPI serial triggers are accessed from the oscilloscope trigger dialog. This dialog can be accessed in the following ways:

Touch the Trigger descriptor label in the lower right hand corner of the oscilloscope display. Or, touch **Trigger** in the menu bar at the top of the scope display, then select **Trigger Setup...** from the drop-down menu.	
Touch **Serial** under the trigger "Type."	
Select the appropriate serial trigger. Then, the menu will automatically change to a different tab in the Trigger dialogs. The tab will be named after the serial trigger that you have selected.	

Creating an I^{2}C Trigger Condition

The I^{2}C Trigger dialog, with detail on some of the setup conditions, is shown in the images that follow:

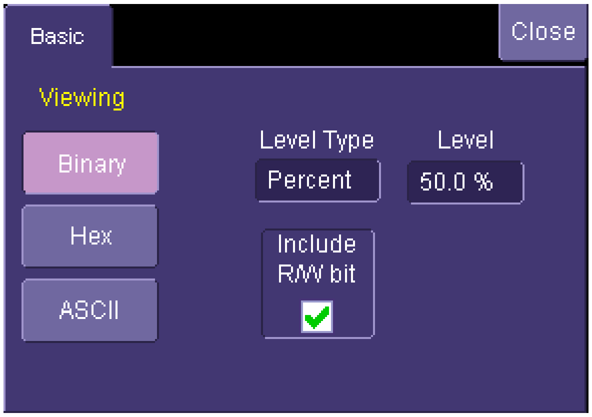


Selection of Trigger Type results in dynamic changes to the I^{2}C Trigger dialog. Simple I^{2}C triggers, such as Start, Stop, ReStart, and NoAck, require no additional setup, while frame-based triggers, such as ADDR, ADDR+DATA, FRAME LENGTH, and EEPROM require additional user-defined setup information.

To select a value for any of the conditions, touch the existing value to open the pop-up dialog box with a list of choices, and select one of the choices.

I^2C Decode Setup Detail

The I^2C Decode Setup Right-Hand Dialog (which is part of the Decode Setup when I^2C is selected as the Protocol to decode), with detail on the setup conditions, is shown in the images that follow:



Viewing - Select to view the protocol data in either Binary, Hexadecimal, or ASCII modes. **Note:** The trigger setup could be different than the decode setup.	
Level Type and **Level** - The message decoding algorithm setup is performed here. The level is normally set up in %, and defaults to 50%. To adjust the level, touch inside the number area to highlight the box title in yellow, then use the oscilloscope front panel **Adjust** knob to adjust. Or touch inside the number area twice and select a value using the pop-up numeric keypad. The set Level appears as a dotted horizontal line across the oscilloscope grid. If your initial decoding indicates that there are a number of error frames, make sure that the level is set to a reasonable value.	
Include R/W Bit - Some engineers think of the 7-bit address pattern as including the R/W bit (i.e., 8-bits) and others think of the address pattern as not including the R/W bit (i.e., 7-bits). If you decoded I^2C messages include 7-bit addresses, touch the checkbox if you want to include the R/W bit in the decoded Address value. **Note:** There is an identical checkbox selection in the I^2C trigger setup dialog. These two setups are dynamically linked, so selecting it one way in Decode setup will result in an identical selection in Trigger setup. This ensures that the decode address format matches trigger setup information.	

Setup Mode

Select either Binary (BIN) or Hexadecimal (Hex) setup mode. The mode selected will propagate through the entire I^2C trigger setup,

You can select BIN mode, and set up the address in binary format, then reselect HEX mode and set up the data in hexadecimal format. Toggling back and forth between the modes will not result in loss of information (binary is used internally as the core format for all triggering and decoding operations), though use of "don't care" bits in a binary setup will result in the display of an X (for a full nibble "don't care") or a $ (for a partial nibble "don't care").

Sources Setup

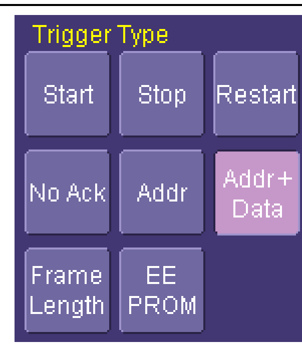
DATA and CLOCK - The pop-up dialog is used to select the appropriate channel or EXT input for each. Make sure that you have these selected correctly or your trigger may not function.

Trig(ger) Threshold - Adjust the vertical level for the trigger. Just as for an Edge trigger, you must specify the level used in order to process the incoming signals and determine whether the desired serial data pattern is meeting the set trigger condition. This value is used for both DATA and CLOCK signals.

Trigger Type Selection

The I^2C trigger can be configured to trigger on simple conditions (i.e., the presence of a START, STOP, RESTART bit, or the absence of an ACK bit (NO ACK). In addition, more complex trigger conditions can be created using ADDR, ADDR+DATA, FRAME LENGTH, or EEPROM setups.

If one of the more complex trigger conditions is selected, then reference the appropriate sections for information on Address and Data Pattern setup.

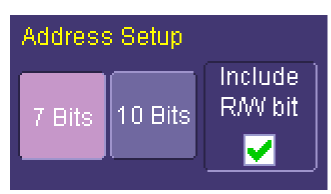


Address Setup

These setup selections are displayed if you select ADDR, ADDR+DATA, FRAME LENGTH, OR EEPROM trigger selections.

Address Length - I^2C utilizes either 7-bit or 10-bit formats for the address, depending on the device. Make the appropriate selection so as to be able to enter the correct address value.

If 7-bit address length is selected, another selection will appear for whether the Read/Write bit should be included as part of the address value entered. For instance, some engineers think of the address pattern as including the R/W bit (i.e., 8 bits) and others think of the address pattern as not including the R/W bit (i.e., 7 bits). Check the checkbox if you want to include the R/W bit in your Address value. If this is done, then the "Direction" value will auto select either **Read** or **Write** (as appropriate) and be grayed out as not-selectable by the user.



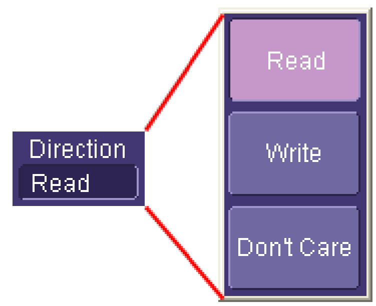
> **Note:** There is an identical checkbox selection in the I^{2}C decode setup dialog. These two setups are dynamically linked, so selecting it one way in trigger will result in an identical selection in decode. This ensures that the trigger address format matches decoded information on the display.

Address Value Setup - Enter the Address Value in binary or hex (depending on what was selected in the Setup Mode). The pattern condition for the Address is always “equal”.

Binary addresses allow use of "don’t care" conditions in any bit position (entered as X). Hexadecimal addresses allow use of "don’t care" conditions in any nibble position) also entered as an X. If an address is set up in Binary, then converted to Hex with a Setup Mode change, then any non-nibble length "don’t care" values will be shown as $.

> **Note:** Address values are always MSB format. Therefore, conversion of address values from binary to hex when "don’t care" values are used will be on that basis.

Direction - Enter a Direction (either **Read**, **Write**, or **Don’t Care**) for the Address value. If you have elected to use 7-bit addresses with the R/W bit included in the address value, then this selection will be grayed out and not selectable.



Data Setup

These setup selections are displayed if the Trigger Selection is ADDR+DATA or EEPROM.

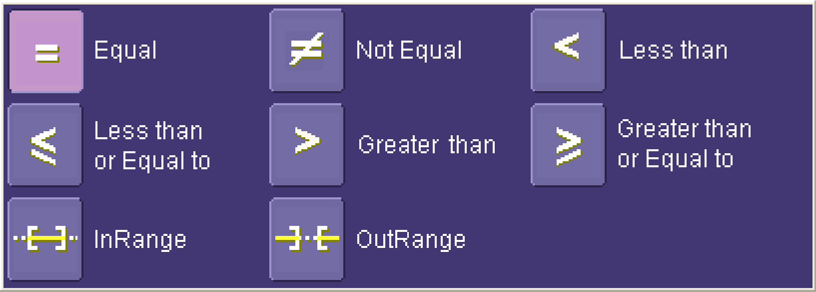
Data Pattern Value - The pattern value is entered in either Binary or Hexadecimal mode depending on the previous selection of Setup Mode. There are two selections for pattern value: “Pattern Value” and “Pattern Value To”. The second selection is exposed for data entry if the Condition is set to INRANGE or OUT(of)RANGE. Otherwise, it is grayed out. Up to 12 bytes of data can be entered as a pattern value.

If less than 12 bytes of data is entered for the pattern value, the data is assumed to begin at the 0 (i.e., first) data byte in the I^{2}C message. If this is not desired, then add preceding or trailing don’t care (X) nibbles to the pattern value.

When more than one byte of data is entered as a data pattern value, the data is treated as “Most Significant Byte (MSB) First”. This is especially important to remember when setting up <=, <, >, >=, INRANGE and OUTRANGE comparison.

> **Note:** In Hexadecimal format, data must be entered as full bytes even though the minimum required acceptable entry is a nibble. If less than a full byte is entered, then a "don’t care" X will precede the pattern values entered.

Condition - The DATA condition can be set many different ways. Possible conditions are <=, <, =, >, >=, not =, in a range, out of a range, or don’t care.



Oftentimes, I^2C utilizes DATA bytes to specify sub-addresses for accessing memory locations in EEPROMs. Conditional DATA trigger allows triggering on a range of DATA bytes that correspond to reads or writes to specific sub-address memory blocks in the EEPROM. It can also aid in monitoring DATA outputs from I^2C-based sensors, such as analog-to-digital converters, and triggering when DATA is outside a safe operating range. In both cases, verifying proper operation becomes a simple task.

Pattern Length - The pattern length value defaults to the length, in bytes, of the pattern set in the Pattern Value selection. If you were to change the length to be less than this value, it would truncate the beginning of the pattern value. If you were to increase the pattern length, it would add "don't care" XX byte values to the beginning of the pattern value.

Start Byte - This selection is present only when the Trigger Selection is EEPROM or ADDR+DATA with an INRANGE or OUTRANGE Data Condition. For EEPROM, use this to specify a particular location of data, such as a sub-address memory block, that the Pattern Value must occupy in order for triggering to occur. For ADDR+DATA, use this to specify a location where the data values should be placed without using don't care (X) values in the pattern value (X values would be nonsensical in an INRANGE or OUTRANGE conditional setup).

Values can be set from 0-2047 bytes. Setting to -1 creates a "Don't Care" condition for data value location.

Note: The first byte is counted as Byte 0, not Byte 1.

Note: Setting the Start Byte to -1 will result in a Don't Care setting for Start Byte. Trigger will occur no matter where the data is in the data string. This is a good trigger debugging tool if your trigger is failing to work as you expect.

Frame Length Setup

This setup selection is displayed if the Trigger Selection is FRAME LENGTH. It is used to trigger on a specific Address value with a defined length of data bytes.

Bytes Length - Specify a data length value between 0 and 2047. The default value is 1.

If the Data Length Condition (see below) is selected to be either INRANGE or OUT(of)RANGE, then this selection will be for the minimum data length value (i.e., the lower value of the range you wish to include or exclude).

Note: All values entered in this field are always in decimal format.	
Bytes Length Max - If the Data Length Condition (see below) is selected to be either INRANGE or OUT(of)RANGE, then you will also need to specify a maximum data length value (i.e., the upper value of the range you wish to include or exclude)	
Length Condition - The Data Length Condition can be set to many different values, such as <=, <, =, >, >=, not =, INRANGE, or OUT(of)RANGE. Select the correct condition for your needs.	 

Ack Setup

Use this setup to choose whether you want to add an Acknowledge bit condition to your ADDR, ADDR+DATA, FRAME LENGTH, or EEPROM trigger condition. DON'T CARE would be the most common setup, although ACK or NO ACK might be a useful condition to add for an unusual or hard-to-find I^2C problem. An example of this would be triggering on an EEPROM write (selected by an ADDR trigger) where the EEPROM failed to acknowledge a byte written.	

Overview of SPIbus

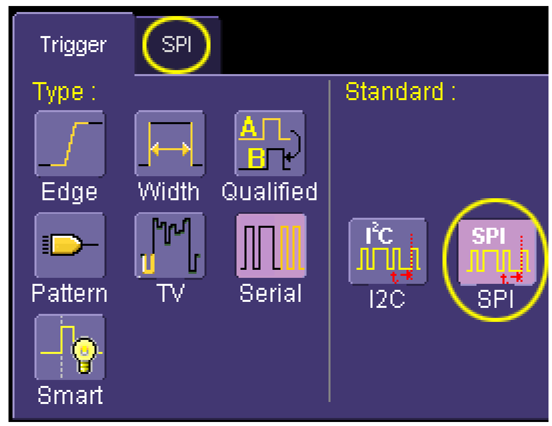
Both SPIbus TD and D options contain powerful software algorithms to extract serial data information from physical layer waveforms measured on your oscilloscope. The extracted information is overlaid (annotated) on the actual physical layer waveforms, and color-coded to provide fast, intuitive understanding.

The SPIbus TD option contains a data trigger that can be configured for the many variants of SPI, such as SSPI (single master and slave with predetermined format settings) and SIOP. The basic SPI Type is all-inclusive and the SSPI and SIOP types are just pre-selected settings in the basic SPI trigger.

The SPI trigger does not require use of a Chip Select line. In its place is the ability to set a Minimum Interframe Time that corresponds to a time that is (typically) 4x a single bit time and less than the interframe time between different message packets. By eliminating the requirement to have a Chip Select line present, an additional oscilloscope channel is preserved for use with other analog signals - a significant feature.

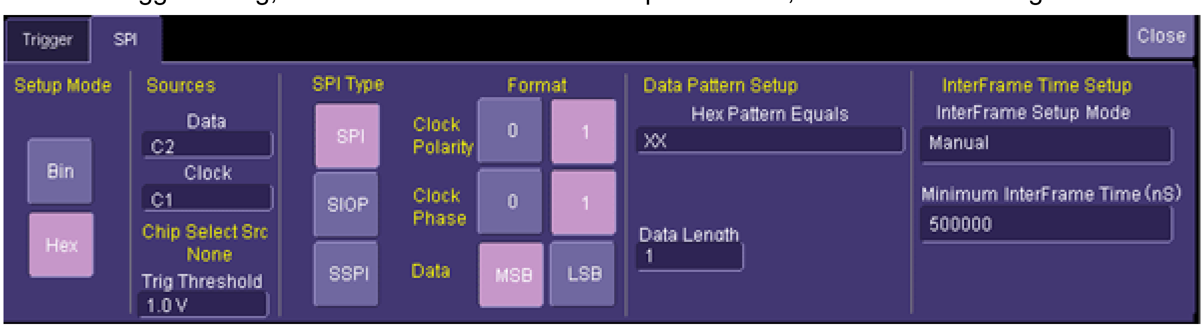
Accessing SPI Serial Triggers

The I²C and SPI serial triggers are accessed from the oscilloscope trigger dialog. This dialog can be accessed in the following ways:

<table>
<tr><td>Touch the Trigger descriptor label in the lower right-hand corner of the oscilloscope display. Or, touch Trigger in the menu bar, then Trigger Setup... in the drop-down menu.</td><td>
</td></tr>
<tr><td>Touch the Serial trigger "Type" selection.</td><td>
</td></tr>
<tr><td>Select the appropriate serial trigger. Then, the menu will automatically change to a different tab in the Trigger dialogs. The tab will be named after the serial trigger that you have selected.</td><td>
</td></tr>
</table>

Creating a SPI Trigger Condition

The SPI Trigger dialog, with detail on some of the setup conditions, is shown in the images that follow:

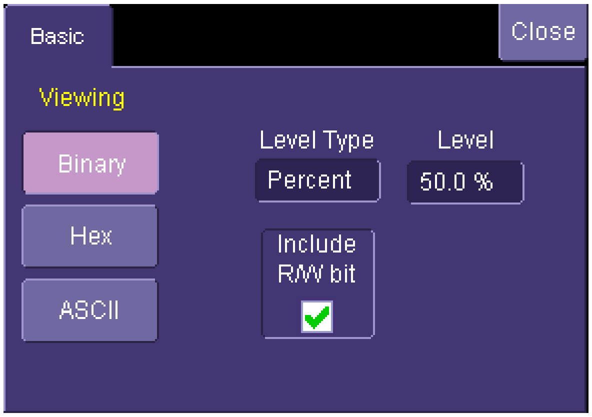


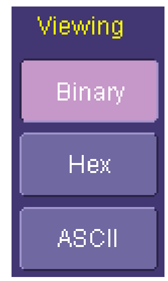
The SPI trigger dialog is very "flat" - there are few dynamic changes to the dialog based on selections within it. The one exception is the Setup Mode.

To select a value for any of the conditions, touch the existing value (using your finger, or use a mouse pointer) to open the pop-up dialog box with a list of choices, and select one of the choices.

SPI Decode Setup Detail

The SPI Decode Setup Right-Hand Dialog (which is part of the Decode Setup when SPI is selected as the Protocol to decode), with detail on the setup conditions, is shown in the images that follow:



Viewing - Select to view the protocol data in either Binary, Hexadecimal, or ASCII modes. **Note:** The trigger setup can be different than the decode setup.	
Level Type and **Level** - The message decoding algorithm setup is performed here. The level is normally set up in %, and defaults to 50%. To adjust the level, touch inside the number area to highlight the box title in yellow, then use the oscilloscope front panel ADJUST knob to adjust. Or touch inside the number area twice and select a value using the pop-up numeric keypad. The set **Level** appears as a dotted horizontal line across the oscilloscope grid. If your initial decoding indicates that there are a number of error frames, make sure that the level is set to a reasonable value.	
Include R/W Bit - Some engineers think of the 7-bit address pattern as including the R/W bit (i.e., 8 bits) and others think of the address pattern as not including the R/W bit (i.e., 7 bits). If you decoded I^2C messages include 7-bit addresses, Touch the checkbox if you want to include the R/W bit in the decoded Address value. **Note:** There is an identical checkbox selection in the I^2C trigger setup dialog. These two setups are dynamically linked, so selecting it one way for Decode will result in an identical selection for Trigger. This ensures that the decode address format matches trigger setup information.	

SPI Setup Mode

Select either Binary (Bin) or Hexadecimal (Hex) setup mode. The mode selected will propagate through the entire SPI trigger setup.

Toggling back and forth between the modes will not result in loss of information (binary is used internally as the core format for all triggering and decoding operations); though, use of "don't care" bits in a binary setup will result in the display of an X (for a full nibble "don't care") or a $ (for a partial nibble "don't care").

SPI Sources Setup

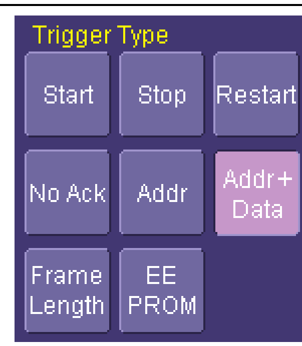
DATA and CLOCK - The pop-up dialog is used to select the appropriate channel or EXT input for each. Make sure that you have these selected correctly or your trigger may not function.

Trig(ger) Threshold - Adjust the vertical level for the trigger. Just as for an Edge trigger, you must specify the level used in order to process the incoming signals and determine whether the desired serial data pattern is meeting the set trigger condition. This value is used for both DATA and CLOCK signals.

SPI Trigger Type Selection

The SPI trigger can be configured to trigger on simple conditions (i.e., the presence of a START, STOP, RESTART bit, or the absence of an ACK bit (NO ACK). In addition, more complex trigger conditions can be created using ADDR, ADDR+DATA, FRAME LENGTH, or EEPROM setups.

If one of the more complex trigger conditions is selected, reference the sections below for information on Address and Data Pattern setup.

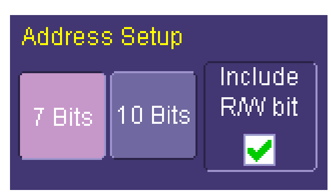


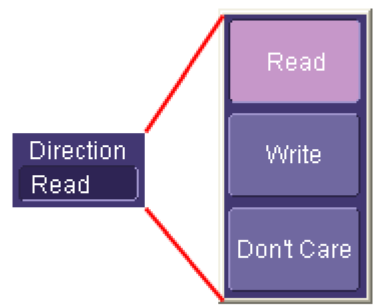
SPI Address Setup

This setup selections are displayed if you select ADDR, ADDR+DATA, FRAME LENGTH, OR EEPROM Trigger Selections.

Address Length - SPI utilizes either 7-bit or 10-bit formats for the address, depending on the device. Make the appropriate selection so as to be able to enter the correct address value.

If 7-bit address length is selected, another selection will appear for whether the Read/Write bit should be included as part of the address value entered. For instance, some engineers think of the address pattern as including the R/W bit (i.e., 8 bits) and others think of the address pattern as not including the R/W bit (i.e., 7 bits). Touch the checkbox if you want to include the R/W bit in your entered Address value. If this is done, then the "Direction" value



will auto select either **Read** or **Write** (as appropriate) and be grayed out as not-selectable by the user. **Note:** There is an identical checkbox selection in the I^2C decode setup dialog. These two setups are dynamically linked, so selecting it one way for Trigger will result in an identical selection for Decode. This ensures that the trigger address format matches decoded information on the display.	
Address Value Setup - Enter the Address Value in binary or hex (depending on what was selected in the Setup Mode). The pattern condition for the Address is always "equal". Binary addresses allow use of "don't care" conditions in any bit position (entered as X). Hexadecimal addresses allow use of "don't care" conditions in any nibble position) also entered as an X. If an address is set up in Binary, then converted to Hex with a Setup Mode change, then any non-nibble length "don't care" values will be shown as $. **Note:** Address values are always MSB format. Therefore, conversion of address values from binary to hex when "don't care" values are used will be on that basis.	
Direction - Enter a Direction (either Read, Write, or Don't Care) for the Address value. If you have elected to use 7-bit addresses with the R/W bit included in the address value, then this selection will be grayed out and not selectable.	

SPI Data Setup

These setup selections are displayed if the Trigger Selection is ADDR+DATA or EEPROM.

Data Pattern Value - The pattern value is entered in either Binary or Hexadecimal mode depending on the previous selection of Setup Mode. There are two selections for pattern value - "Pattern Value" and "Pattern Value To". The second selection is exposed for data entry if the Condition is set to INRANGE or OUT(of)RANGE. Otherwise, it is grayed out. Up to 12 bytes of data can be entered as a pattern value. If less than 12 bytes of data is entered for the pattern value, the data is assumed to begin at the 0 (i.e., first) data byte in the I^2C message. If this is not desired, then add leading or trailing "don't care" (X) nibbles to the pattern value. When more than one byte of data is entered as a data pattern value, the	

data is treated as “Most Significant Byte (MSB) First”. This is especially important to remember when setting up <=, <, >, >=, INRANGE and OUTRANGE comparison. **Note:** In Hexadecimal format, data must be entered as full bytes even though the minimum required acceptable entry is a nibble. If less than a full byte is entered, then a "don’t care" X will precede the pattern values entered.	
Condition - The DATA condition can be set many different ways. Possible conditions are <=, <, =, >, >=, not =, in a range, out of a range, or don’t care. Oftentimes, I^2C utilizes DATA bytes to specify sub-addresses for accessing memory locations in EEPROMs. Conditional DATA trigger allows triggering on a range of DATA bytes that correspond to reads or writes to specific sub-address memory blocks in the EEPROM. It can also aid in monitoring DATA outputs from I^2C-based sensors, such as analog-to-digital converters, and triggering when DATA is outside a safe operating range. In both cases, verifying proper operation becomes a simple task.	
Pattern Length - The pattern length value defaults to the length, in bytes, of the pattern set in the Pattern Value selection. If you were to change the length to be less than this value, it would truncate the beginning of the pattern value. If you were to increase the pattern length, it would add "don’t care" XX byte values to the beginning of the pattern value.	
Start Byte - This selection is present only when the Trigger Selection is EEPROM or ADDR+DATA with an INRANGE or OUTRANGE Data Condition. For EEPROM, use this to specify a particular location of data, such as a sub-address memory block, that the Pattern Value must occupy in order for triggering to occur. For ADDR+DATA, use this to specify a location where the data values should be placed without using "don’t care" (X) values in the pattern value (X values would be nonsensical in an INRANGE	

or OUTRANGE conditional setup). Values can be set from 0-2047 bytes. Setting to -1 creates a Don't Care condition for data value location. **Note:** The first byte is counted as Byte 0, not Byte 1. **Note:** Setting Start Byte to -1 will result in a Don't Care setting for Start Byte. Trigger will occur no matter where the data is in the data string. This is a good trigger debugging tool if your trigger is failing to work as you expect.	

SPI Frame Length Setup

This setup selection is displayed if the Trigger Selection is FRAME LENGTH. It is used to trigger on a specific Address value with a defined length of data bytes.

Bytes Length - Specify a data length value between 0 and 2047. The default value is 1. If the Data Length Condition (see below) is selected to be either INRANGE or OUT(of)RANGE, then this selection will be for the minimum data length value (i.e., the lower value of the range you wish to include or exclude). **Note:** All values entered in this field are always in decimal format.	
Bytes Length Max - If the Data Length Condition (see below) is selected to be either INRANGE or OUT(of)RANGE, then you will also need to specify a maximum data length value (i.e., the upper value of the range you wish to include or exclude)	
Length Condition - The Data Length Condition can be set to many different values, such as <=, <, =, >, >=, not =, INRANGE, or OUT(of)RANGE. Select the correct condition for your needs.	

SPI Ack Setup

Use this setup to choose whether you want to add an Acknowledge bit condition to your ADDR, ADDR+DATA, FRAME LENGTH, or EEPROM trigger condition. DON'T CARE would be the most common setup, although ACK or NO ACK might be a useful condition to add for an unusual or hard to find I^2C problem. An example of this would be triggering on an EEPROM write (selected by an ADDR trigger) where the EEPROM failed to acknowledge a byte written.	

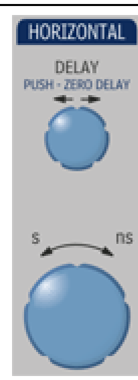
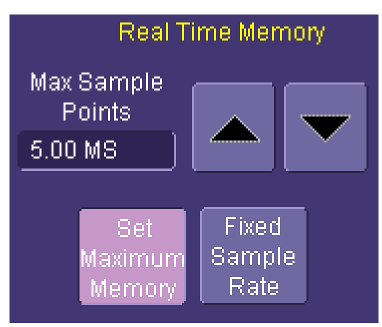
Overview of Serial Bus Activity

The combination of Serial Data Triggering, Decoding, and normal oscilloscope features is a powerful combination of tools that can make it very easy to find latent Serial Data hardware or software problems in your circuit. No longer is the oscilloscope a tool just for the hardware engineer. Now the software engineer can also easily visualize the Serial Data signals and relate it to programming code and operation. The TD options can enable the hardware engineer and software engineer to "speak the same language" when it comes to system debugging and performance checking.

Some common Serial Data analysis needs and methods are discussed next.

Capturing Long Pre-Trigger Time

LeCroy oscilloscopes are available with optional very long acquisition memory. For instance, the WaveRunner Xi Series oscilloscopes can capture up to 12.5 Mpts on 4 channels, or 25 Mpts on 2 channels. If your Serial Data signals are 1 Mb/s, and you sample at the minimum required and available sample rate (5 MS/s, you would be able to capture 5 seconds of Serial Data traffic. If you wish, this can be 100% pre-trigger, 100% post-trigger, or something in between.

1. Adjust Pre-Trigger and Post-Trigger time by adjusting the Delay knob on the oscilloscope's front panel.	
2. Optimize your Sample Rate or Memory Length by accessing the Horizontal Dialog in your oscilloscope and selecting either **Set Max Memory** mode or **Fixed Sample Rate** mode. 3. If you choose to Set Max Memory, you can decrease the memory usage so that you will not sample at too high a sampling rate (too high a sampling rate will slow down the decoding algorithm). Then adjust your timebase setting to as long as necessary to capture the event. Note that you must make sure that your timebase setting and memory length combined do not result in too low a sampling rate or adequate capture and decode will not be performed. 4. More commonly, you will probably choose to fix the sampling rate to a specific value that provides you with the	

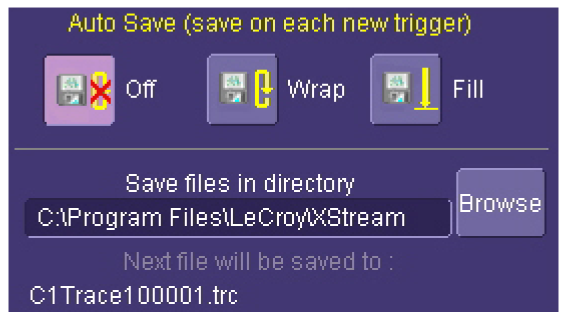
oversampling you need to capture your Serial Data messages (at least 4x the bit rate) and also allow you to have a high enough sample rate to capture any transients that you may want to see on your Serial Data and analog signals (at least 2x the frequency of any expected transients, preferably 10x).	

Reference SMART Memory for more information on these common settings.

Saving Data

You may wish to set up your oscilloscope to capture a short or long memory acquisition for a certain trigger condition, then save data to a hard drive or memory stick whenever the trigger condition is met. This can be easily done in most LeCroy oscilloscopes. However, you must realize that there is significant trigger "dead time" when using this method. To minimize dead time, use the method described under Storing Triggers (Sequence Mode).

1. First, set up your desired serial data (or other) trigger condition.

2. Then, from the menu bar, choose **File**, **Save Waveform**. This will open a dialog that allows you to set up the Save Waveform conditions. You can choose to have this OFF (no Auto Saving occurs), WRAP (Auto Save occurs until the hard drive is filled, then discards the oldest data to write the newest data), or FILL (Auto Save occurs until the hard drive is filled). 3. Be sure to choose a Binary file format if you wish to recall the traces into a LeCroy oscilloscope for later analysis.	

Even though the LeCroy oscilloscope hard drives are very large, it is a good idea to make sure that your trigger condition is set correctly before beginning your acquisitions.

Note: This method is not guaranteed to capture all of your trigger events, since there will be a large amount of "dead time" between triggers as the acquisition is captured, displayed, and stored to the hard drive before the scope is re-armed for a new trigger. To minimize dead time, use Sequence Mode.

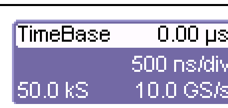
Storing Triggers

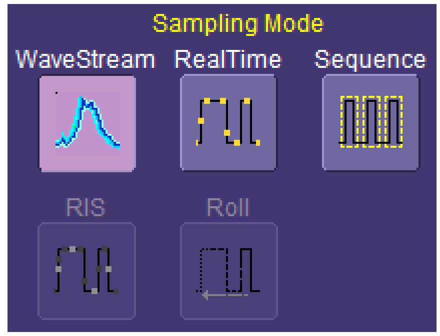
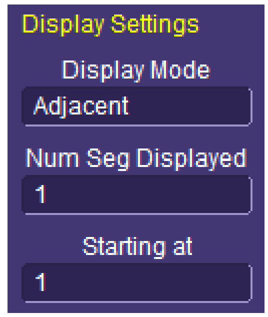
LeCroy oscilloscope's have a powerful capability called Sequence Mode that allows you to store all triggered events by minimizing the dead time between triggers to < 800 nanoseconds. This is ideal for finding repetitive causes of problems on your serial data buses or associated signals.

Sequence Mode uses long acquisition memory that is divided into "segments." As triggered events are acquired, they are stored in acquisition "segments" to be recalled at a later date. The length of each sequence mode acquisition segment and the total number of segments allowed is roughly determined by the total acquisition memory in the oscilloscope. For instance, for a WaveRunner Xi with VL memory, you can get 10,000 segments that are each a maximum of 625 samples long, or 10 segments that are each a maximum of 1.25 megasamples long, or something in between. Different acquisition memory lengths have different ranges of segments and segment lengths. You can define any number of segments from 2 to the maximum for that memory length, and any length of segment (so long as there is sufficient acquisition memory). After acquisition of all segments is complete, you can recall them one-by-one and view them in decoded format on the oscilloscope screen.

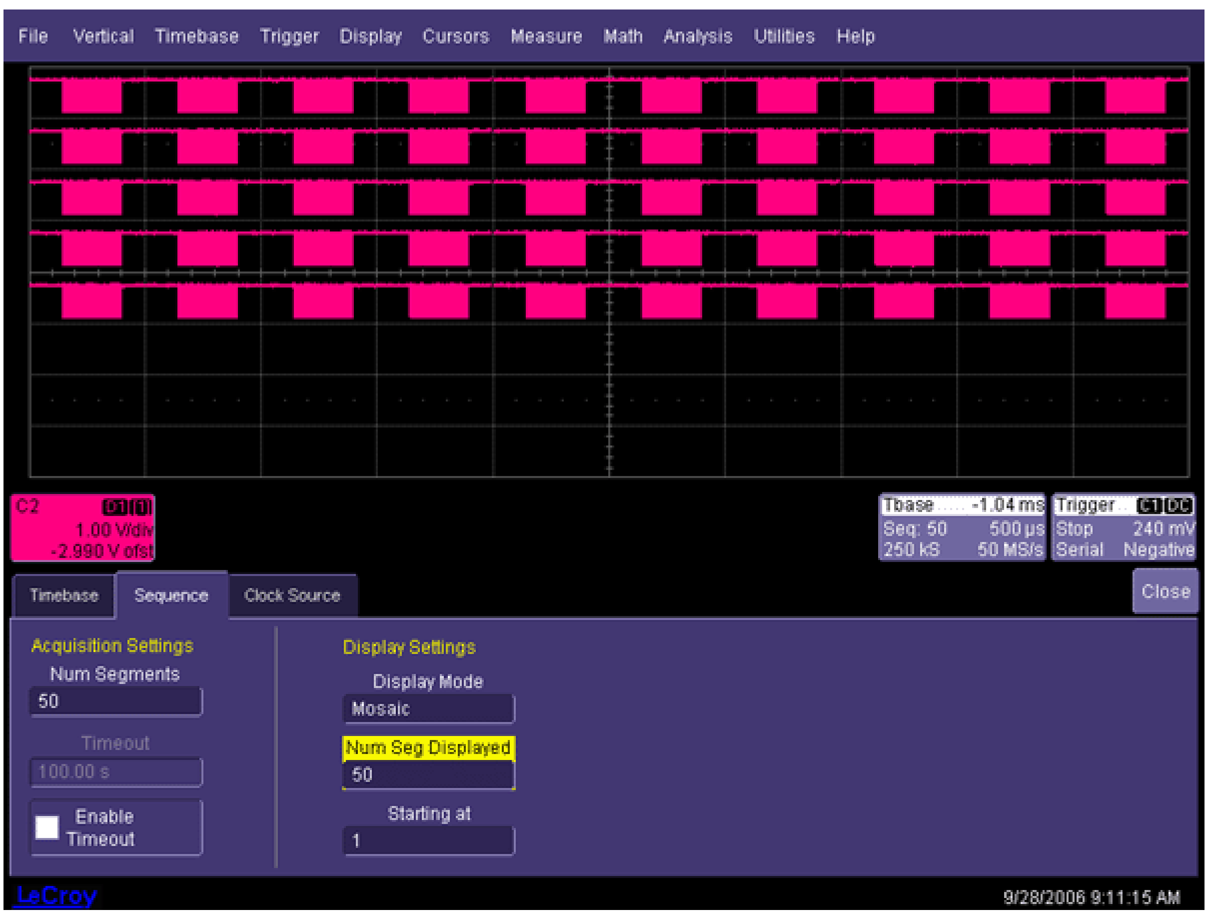
Acquisition dead time is kept to a minimum because there are no operations performed during the acquisition. All data for each triggered event is written only into high-speed acquisition memory. Until the entire sequence is completed, there is no updating of the oscilloscope display, or other operations that cause unnecessary dead time. This is ideal for situations when you cannot take a chance on losing data.

In the example shown below, we have only acquired Channel 1 (the CAN signal) in sequence mode. We could also acquire additional analog or other signals as desired or as necessary to do a proper analysis.

1. Touch the Timebase descriptor label to open the **Timebase** dialog.	

2. In the Sampling Mode area, select **Sequence** mode. A tab labeled "Sequence" will appear next to the Timebase tab. Touch that tab.	
3. In the Sequence tab, select a display mode. Select the number of segments to be displayed at one time. 4. If you have acquired more segments than you can display at one time, you can choose the segment to begin the display at.	

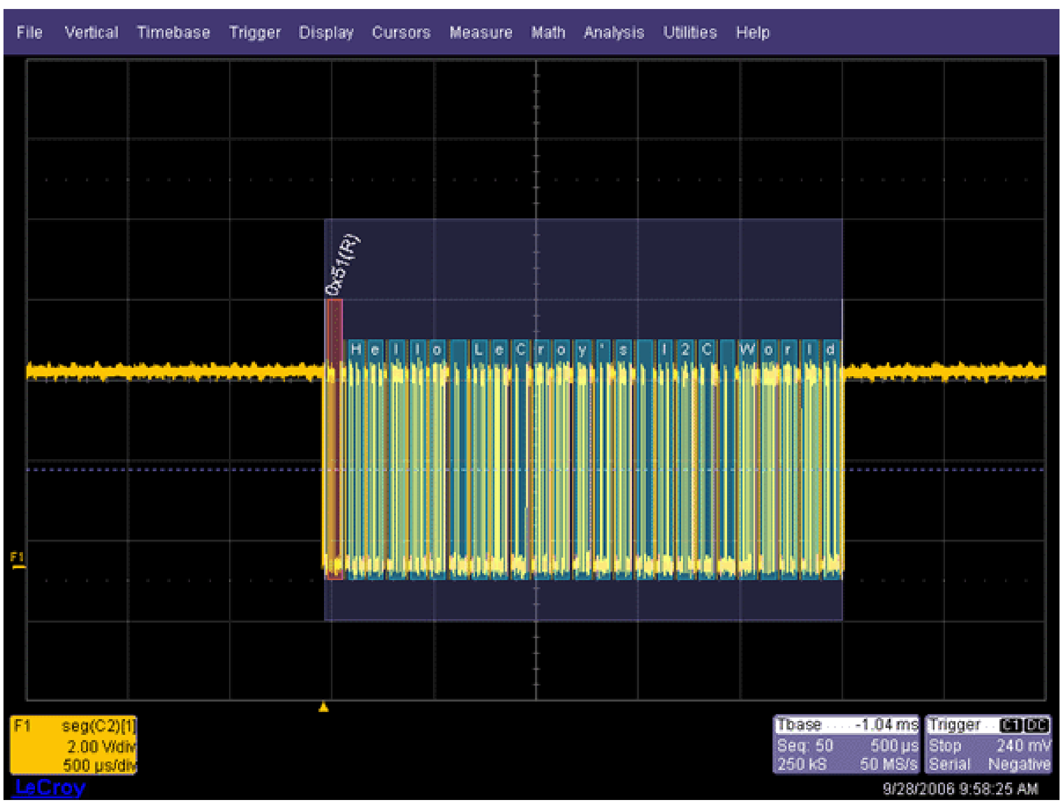
5. Set up the Serial Trigger to capture the event you desire. For instance, you might want to trigger on a specific address or data value, and capture long pre-trigger time to determine what precedes that message. In this example, we've used a simple I^2C Start trigger. To begin the sequence mode acquisition, press the front panel SINGLE trigger button. Each time the trigger condition is met, the TRIG'D light on the front panel will flash. When you've acquired the set number of segments, the trigger will STOP and the display will appear as below (this is a 50 segment acquisition in Mosaic display mode).



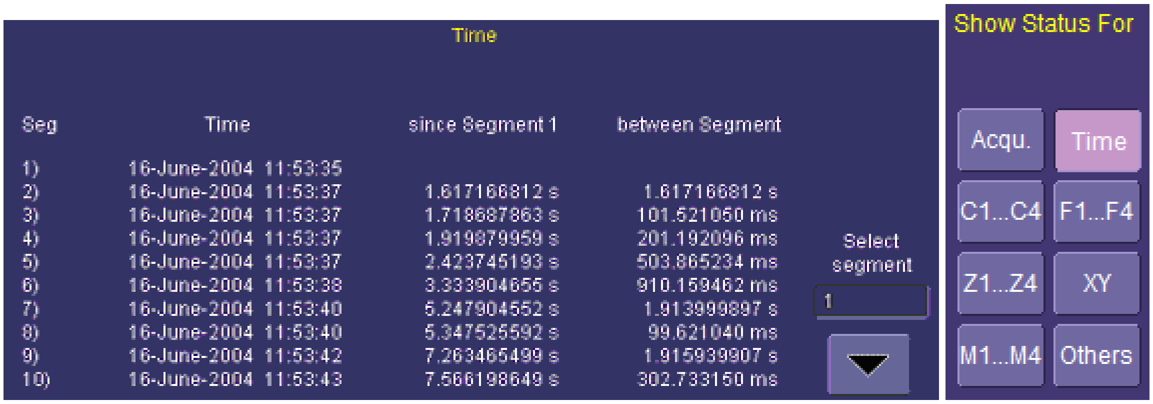
6. To display an individual segment separately from the main channel display, select **Math**, **Math Setup** from the menu bar, and choose a math trace to define as a Segment (in this case, we chose to define F1 as a Segment of C2). As a source, use the channel that your serial data was acquired on (in this case Channel 2). To display the trace, check the **Trace On** checkbox. To select the segment to view, touch the **Select** tab and select a segment using the pop-up keypad or the front panel adjust knob.

7. To view decoded data on the individual segment, set up the Decode to use the Math trace as the source for Data (in this case F4 is the source). If you wish to change the segment that is decoded, just select a new segment from the Math trace dialog (as shown in the preceding step).

8. To maximize display space, you could turn off the Channel and just select the segment that you wish to view.



9. To view the time stamps for each segment, select **Vertical**, **Channels Status** in the menu bar, then in the "Show Status For" portion of the dialog, select **Time** (as shown to the right). Then, you will see a display of time stamp information for each segment in the sequence acquisition, as shown below:



Seg	Time	since Segment 1	between Segment
1)	16-June-2004 11:53:35		
2)	16-June-2004 11:53:37	1.617166812 s	1.617166812 s
3)	16-June-2004 11:53:37	1.718687863 s	101.521050 ms
4)	16-June-2004 11:53:37	1.919879959 s	201.192096 ms
5)	16-June-2004 11:53:37	2.423745193 s	503.865234 ms
6)	16-June-2004 11:53:38	3.333904655 s	910.159462 ms
7)	16-June-2004 11:53:40	5.247904552 s	1.913999897 s
8)	16-June-2004 11:53:40	5.347525592 s	99.621040 ms
9)	16-June-2004 11:53:42	7.263465499 s	1.915939907 s
10)	16-June-2004 11:53:43	7.566198649 s	302.733150 ms

10. Ten time stamps will fit on the display at one time. You can choose which ten to display by using the **Select Segment** control. You can also page through the segments one at a time by using the ADJUST knob on the front panel.

I²C and SPI Specifications

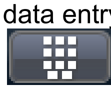
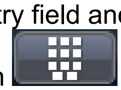
LeCroy I2C and SPI TD Specification, Vs. 1.1		
	I2C TD	**SPI TD**
Definition		
Protocol Setup	N.A.	Select CPOL, CPHA, DATA = MSB or LSB. Also, may select SIOP or SSPI defaults.
Decode Capability		
Format	Hexadecimal, Binary, ASCII	Hexadecimal, Binary, ASCII
Decode Setup	Threshold definition required. Default is to Percent amplitude. Choose to Decode address values including/not including the R/W bit in address value.	Threshold definition required. Default is to Percent amplitude. Select CPOL, CPHA, DATA = MSB or LSB,
# of Decoded Waveforms	**Up to 4 buses may be decoded at one time.** Sources can be Channels or Memory (Reference) Waveforms. In addition, zooms can be displayed (with decoded information).	**Up to 4 buses may be decoded at one time.** Sources can be Channels or Memory (Reference) Waveforms. In addition, zooms can be displayed (with decoded information).
Location	Overlayed over DATA waveform, on Grid	Overlayed over DATA waveform, on Grid
Visual Aid	Color Coding for FRAME, START/ReSTART bit, ADDR, R/W, DATA, ACK, and STOP bit	Color Coding for FRAME and DATA

LeCroy I2C and SPI TD Specification, Vs. 1.1		
	I2C TD	**SPI TD**
Trigger Capability		
Format	Hexadecimal or Binary ADDRESS and DATA can be set up with different formats.	Hexadecimal or Binary
Trigger Setup	Trigger on START, ReSTART, STOP, Missing ACK, ADDR, DATA, ADDR+DATA, ADDR+DATA Length, EEPROM	Trigger on DATA
ADDRESS Condition Setup	Specify One ADDRESS with condition of = 7 or 10 bit ADDRESS supported with full Read, Write, or R/W="Don't Care" selectability on both 7 and 10 bit ADDRESSes. Choose to Trigger on address values that include/don't include R/W bit in address value.	N.A.
DATA Condition Setup	<=, <, =, >, >=, <>, in range, out of range, don't care.	=
DATA Setup	Hexadecimal: # Data Bytes = 0 to 12. Data can be defined by nibble. Binary: Any combination of 0,1, or X for 1-96 bits Data pattern can be set to start on any byte in a 2048 byte window (EEPROM mode).	Hexadecimal: # Data Bytes = 0 to 8. Data can be defined by nibble. Triggers on that data pattern regardless of position. Binary: Any combination of 0,1, or X for 1-64 bits. Triggers on that data pattern regardless of position.
ACK Condition Setup	For any ADDR, ADDR+DATA, ADDR+DATA LENGTH, or EEPROM frame setup, select an ACK Condition of ACK, NO ACK, and DON'T CARE.	N.A.
Bit Rates	Full range over I2C specification for Standard, Fast, Fast-Mode Plus, and High-Speed modes. Auto-detected	Any. Auto-detected

Timebase Setup and Control

Set up the timebase by using the front panel **Horizontal** controls.

Setting up additional timebase setup and controls

1. Touch **Timebase** → **Horizontal Setup**. The **Timebase** dialog opens.
2. Touch inside the **Time/Division** data entry field and enter a value using the slider bar at the bottom of the window. Click the keypad button  on the slider bar to enter a value using the pop-up numeric keypad, or use the up/down arrows to adjust the value.
3. Touch inside the **Delay** data entry field and type in a value, or use the slider bar at the bottom of the window. Click the keypad button  on the slider bar to enter a value using the pop-up keypad. Touch the **Set To Zero** button to set the delay to zero.

Autosetup

When channels are turned on, you can run Autosetup on those channels. If no channels are turned on, all channels are affected. When more than one channel is turned on, the first channel in numerical order with a signal applied to it is automatically set up for edge triggering.

You can perform an autosetup of all these functions together by simply pressing **Auto Setup** 

on the front panel, or by touching **Auto Setup** 

in the **Vertical**, **Timebase**, or **Trigger** drop-down menu.

Real Time (SMART) Memory

LeCroy's Real Time (SMART) Memory feature ensures the highest time resolution for the time window displayed, without aliasing. Real Time Memory provides these advantages:

- Acquisition memory is automatically allocated as needed.
- Memory size optimization: Set Maximum Memory optimizes memory to obtain highest sampling rate, reducing the risk of aliasing. You can set a maximum memory up to 256 Mpts.
- Fixed Sample Rate allows setting of a specific sample rate, with the scope calculating the amount of memory needed for a timebase setting.
- The entire acquisition is displayed on the screen.
- High-speed compaction shows all significant features of your waveform.

You can set a maximum memory up to 256 Mpts.

Setting Up Real Time (SMART) Memory

1. Touch **Timebase** → **Horizontal Setup**.
2. Under **Real Time Memory**, touch the **Set Maximum Memory** button and touch inside the **Maximum Sample Point** field to enter a value using the scroll bar at the bottom of the window.

   OR

   Touch the **Fixed Sample Rate** button and touch inside the **Sampling Rate** field to enter a value using the scroll bar at the bottom of the window.
3. Touch inside the **Time/Division** data entry field and set a time per division.

Note: If you are currently acquiring waveforms, you will notice a change in sampling rate as you select different modes.

4. If you selected **Sequence** mode, touch inside the **Num Segments** data entry field and enter a value using the pop-up numeric keypad. If you want to use a timeout period, touch the **Enable Timeout** checkbox; then touch inside the **Timeout** data entry field and enter a value.

External Timebase vs. External Clock

An external timebase reference is used to synchronize the scope's internal timebase to an external frequency source. This allows multiple instruments to lock their timebases to a common source. The external timebase reference frequencies are model dependent. The WavePro 700Zi series uses an external timebase reference of 10 MHz ±0.01% with an amplitude of 650 $mV_{pk\text{-}pk}$ or 0.0 dBm ±30%.

An external sampling clock, applied via the auxiliary input, replaces the scope's internal timebase as the sampling clock. This means that the external sampling clock controls when the scope's digitizers sample the input waveforms. Since the external sampling clock uses the auxiliary input, an external trigger cannot be used when the external sampling clock is in use.

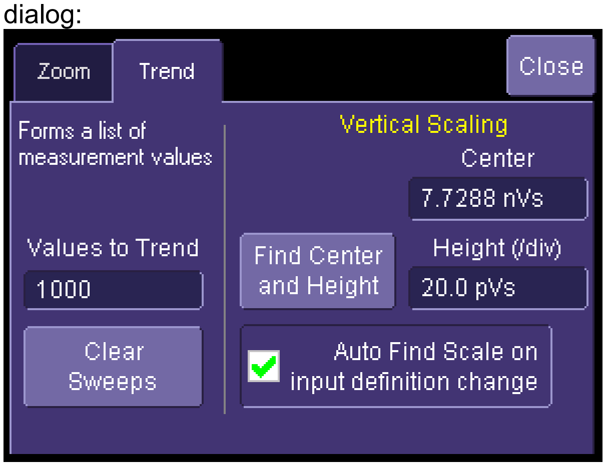
Note: The WavePro 700Zi series has a timebase of **10 MHz ±0.01** and a clock of **DC 100 MHz**.

Creating and Viewing a Trend

1. In the menu bar, touch **Measure** → **Measure Setup**.
2. Touch one of parameter tabs **P1** through **P12**.
3. Touch inside the **Source1** data entry field and select an input waveform from the pop-up menu.
4. Touch inside the **Measure** data entry field and select a parameter from the pop-up menu.
5. Touch the **Trend** button 

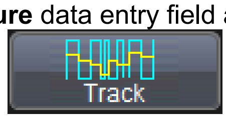
at the bottom of the dialog; then, from the **Math selection for Trend** menu, select a math function location (F1 to F8) to store the Trend display. The Trend will be displayed along with the trace label 

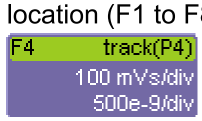
(example of a Trend Trace Label) for the math function you selected.
6. Touch the newly displayed Trend math function trace label if you want to change any settings in the Trend dialog:



Note: Turning off a trace for which trend data is being collected will reset the trend. If it is necessary to continue data collection for the trend, create a zoom trace of the channel trace before turning off the channel trace.

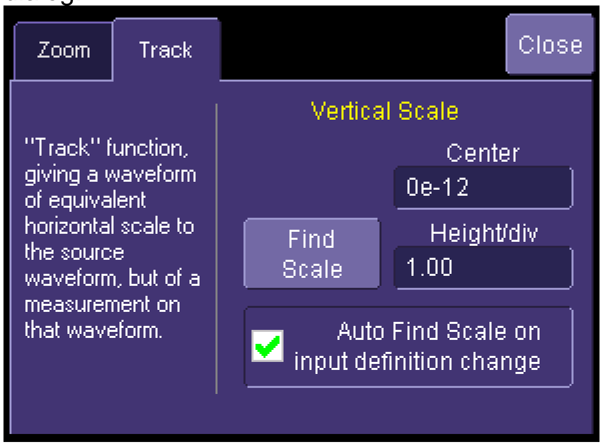
Creating a Track View

1. In the menu bar, touch **Measure** → **Measure Setup**.
2. Touch one of parameter tabs **P1** through **P12**.
3. Touch inside the **Source1** data entry field and select an input waveform from the pop-up menu.
4. Touch inside the **Measure** data entry field and select a parameter from the pop-up menu.
5. Touch the **Track** button 

at the bottom of the dialog; then, from the **Math selection for Track** menu, select a math function location (F1 to F8) to store the Track display. The Track will be displayed along with the trace label 

(example of a Track Trace Label) for the math function you selected.

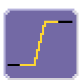
6. Touch the newly displayed Track math function trace label if you want to change any settings in the **Track** dialog:



Trigger Types

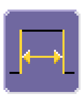
The triggers available to you are defined as follows:

Edge



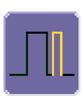
A simple trigger, Edge trigger is activated by basic waveform features or conditions such as positive or negative slope, and holdoff.

Width



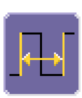
Width trigger allows you to define a positive- or negative-going pulse width bounded by a voltage level, above or below which a trigger will occur. Or you can specify a pulse width and voltage range, within or outside of which a trigger will occur.

Glitch



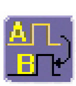
Glitch trigger is a simpler form of Width trigger. Use Glitch trigger when you want to define a fixed pulse-width time or time range only. Glitch trigger makes no provision for voltage levels or ranges.

Interval



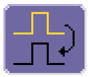
While Glitch trigger performs over the width of a pulse, **Interval** trigger performs over the width of an interval — the signal duration (the period) separating two consecutive edges of the same polarity: positive to positive or negative to negative. Use interval trigger to capture intervals that fall short of, or exceed, a given time limit. In addition, you can define a width range to capture any interval that is itself inside or outside the specified range - an Exclusion trigger by interval.

Qualified



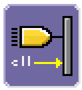
The Qualified (A-B) trigger allows arming of the trigger on Event A and triggering on Event B. If the arming event is a Pattern that occurs once (Pattern) or that occurs and stays satisfied (PatState), the triggering event can be an Edge, Width, Glitch, or Interval condition.

Qualified First

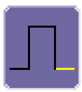

Not available on all scopes, **Qualified First** trigger is intended to be used exclusively in Sequence Mode to speed up the trigger rate. With Qualified First trigger, a single valid trigger is sufficient to acquire a full sequence. Other than in Sequence Mode, Qualified First is identical to the Qualified triggers.

In data storage applications, the index pulse can be defined as the qualifier signal and the servo gate signal as the trigger source.

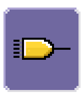
State


The **State** trigger is a level-qualified trigger which requires that the qualifying signal remain above or below a specified voltage level for a trigger to occur. For Sate trigger, you specify the time or number of events after the signal has gone above or below the voltage level when you want the trigger to occur.

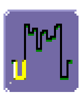
Dropout


Used primarily in single-shot applications, and usually with a pre-trigger delay, **Dropout** trigger can detect lost signals. The trigger is generated at the end of the timeout period following the last trigger source transition.

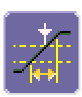
Logic


Logic trigger enables triggering on a logical combination (pattern) of five inputs: CH1, CH2, CH3, CH4, and EXT. You have a choice of four Boolean operators (AND, NAND, OR, NOR), and you can stipulate the high or low voltage logic level for each input independently.

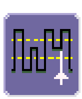
TV


Not available on all instruments, **TV** triggers provide stable triggering on standard or custom composite video signals. Use them on PAL, SECAM, or NTSC systems.

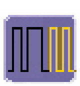
Slew Rate


SlewRate trigger activates a trigger when the rising or falling edge of a pulse crosses two threshold levels: an upper level and a lower level. The pulse edge must cross the thresholds faster or slower than a selected period of time. You can select both thresholds within a range of 2 ns to 20 s. Slew rate trigger is not available on all instruments.

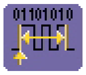
Runt


The **Runt** trigger occurs when a pulse crosses a first threshold line, but fails to cross a second threshold line before recrossing the first. You can select both voltage thresholds within a time range of 100 ps to 20 s. Other defining conditions for this trigger are the edge (triggers on the slope opposite to that selected) and runt width.

Serial Trigger and Decode

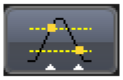

Serial trigger allows a serial trigger condition to be set from within the oscilloscope, using an easy-to-understand interface.

Serial



80-bit Serial Trigger without decode capability is available on SDA model scopes only.

Window



Trigger when signal or exits a window defined by adjustable thresholds.

Aux Input Trigger

Some instrument models give you the capability to trigger on an auxiliary input. When you select this option, the auxiliary trigger setup is routed to channel 3, and an information icon appears in the Channel 3 descriptor label.

CAUTION

If you select Aux Input trigger on a WavePro 700Zi Series oscilloscope, but do not input an external signal, the instrument will not operate.

Aux Input Trigger Setup

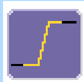
1. Touch **Trigger** → **Trigger Setup...** from the menu bar.
2. Touch inside the **View AUX IN on Channel 3** checkbox.  The **Coupling** field is disabled.
3. Press the Channel 3 front panel button to turn on Channel 3 and display the setup dialog.
4. Perform vertical setups for your auxiliary input in the Channel 3 dialog.
5. Touch **Vertical** → **Channels Status...**from the menu bar to view a summary of the Aux Input setup:

Software Assisted Trigger

For SDA 9000/11000/18000 analyzers, Software Assisted Trigger provides a quick way to create eye diagrams without the involvement of LeCroy's in-depth serial data analysis software. You can get a quick view of your signal and do the analysis whenever you wish.

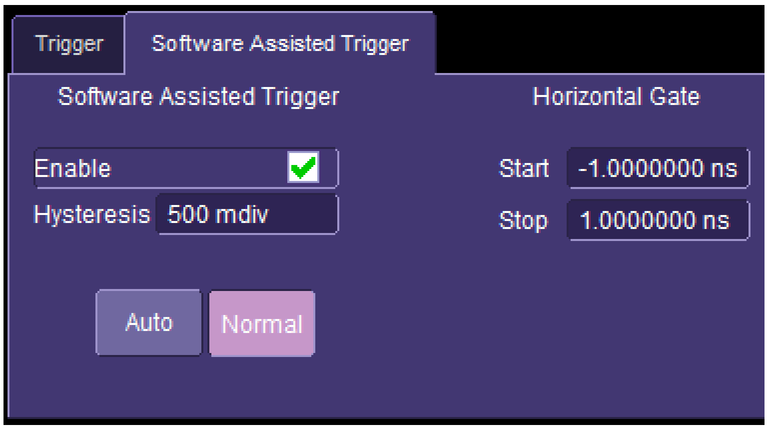
Software Assisted Trigger is used to find the trigger-level crossing point closest to the hardware trigger point. It then adjusts the time offset of the waveform so that it is aligned with the specified trigger level and slope.

Note: The following selections must be made to use the Software Assisted Trigger feature.

- **Edge** trigger type. 
- **Normal** trigger mode. 

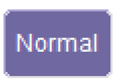
Software Assisted Trigger Setup

Make the following settings from the **Software Assisted Trigger** tab.



1. Touch the **Enable** checkbox to start software assisted triggering.
2. Create a trigger window by entering a **Hysteresis** value. This value sets a boundary above and below the trigger level set in the main Trigger dialog to exclude noise.
3. The **Auto** and **Normal** buttons determine the behavior when trigger crossings are not found in the trigger source waveform.

- **Auto** 

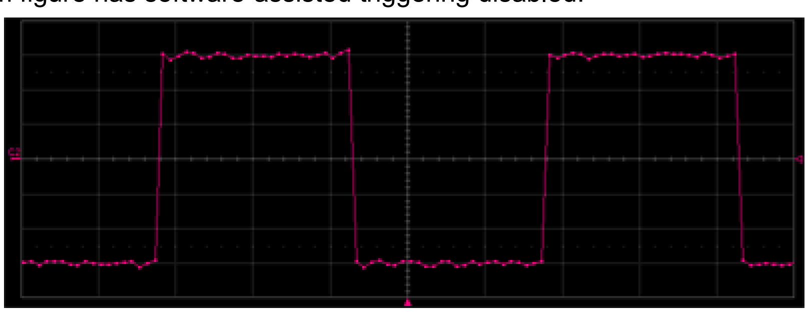
mode allows all waveforms through the channel.
- **Normal** 

mode allows waveforms only with a trigger crossing within the horizontal gate region through the channel.

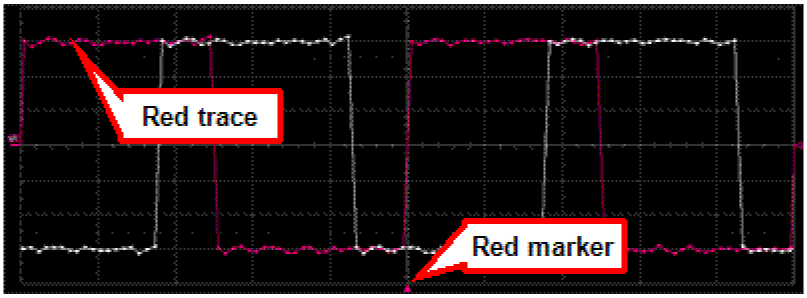
4. Set **Start** and **Stop** time values on the **Horizontal Gate** part of the Software Assisted Trigger tab. These values control where in the waveform the software-assisted trigger processing searches for trigger crossings.

Example

The following waveform figure has software-assisted triggering disabled:

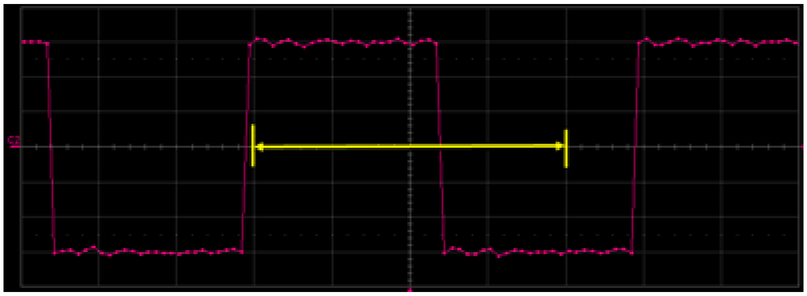


The next waveform figure has software-assisted triggering enabled. The red trace is aligned with the trigger point, indicated by the red marker below the grid. The white trace is the waveform at the input to software-assisted trigger processing. Notice how the white trace is the same as the previous figure.



The next figure shows a waveform outside the horizontal gate region (the closest positive edge is not within +/-1 ns). The **Auto** and **Normal** difference is as follows:

- In **Auto** software assisted trigger mode, the waveform would be displayed, but not aligned.
- In **Normal** mode, it would not be displayed.



Edge Trigger on Simple Signals

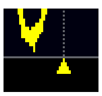
The instrument uses many waveform capture techniques that trigger on features and conditions that you define. These triggers fall into two major categories:

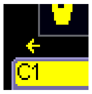
- **Simple Triggers** – activated by basic waveform features or conditions such as a positive or negative slope, and hold-off
- **SMART Triggers** – sophisticated triggers that enable you to use basic or complex conditions for triggering. Use SMART Triggers for signals with rare features, like glitches.

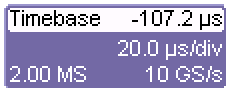
Trigger Settings

Horizontal and Vertical adjustments are typically made for all trigger types using either the **Delay** or **Level** knobs on the front panel of the instrument or their respective fields on the **Timebase** setup dialog.

- **Horizontal:** Turn the DELAY knob in the HORIZONTAL control group to adjust the trigger's horizontal position. Or, touch inside the **Delay** field in the timebase setup dialog and enter a value, using the pop-up keypad.

  The trigger location is shown by a marker  under the grid.

  Post-trigger delay is indicated by a left-pointing arrow 

at the lower-left of the grid. The time value is given in the title line of the **TimeBase** label 

at the lower-right of the grid.

- **Vertical:** Turn the LEVEL knob in the TRIGGER control group to adjust the trigger's vertical threshold.

  Turn this knob to adjust the level of the trigger source or the highlighted trace. Level defines the source voltage at which the trigger will generate an event: a change in the input signal that satisfies the trigger conditions.

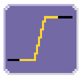
Alternatively, in the Trigger dialog, you can touch inside the **Level** field and type in a value using the pop-up numeric keypad. Quickly set a level of zero volts by touching the **Zero Level** button.

An arrow on the right side of the grid shows the threshold position. This arrow is only visible if the trigger source is displayed.

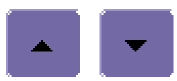
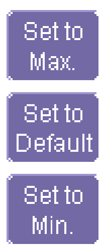
Edge Trigger Setup

Channel Setup

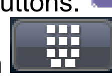
1. Touch the **Trigger** descriptor label.

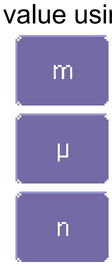
2. Touch the **Edge** trigger button. 
3. Touch inside the trigger **Source** field and select an input from the pop-up menu.

4. Select a coupling mode.
5. Touch inside the **Level** data entry field. 

Using the pop-up numeric keypad, enter a value in millivolts or use the up/down buttons  to increase or decrease the value in increments of 1 mV. Or, touch one of the preset value buttons. 

6. Select the holdoff by touching the **Time** or **Events** buttons. 

Use the slider bar (at the bottom of the window) to enter values via the keypad button  to enter a value using the pop-up keypad or the Default button 

to use preset values. Specify the unit of time, 

or use the **up/down** buttons (mentioned in the previous step) to increase or decrease the time value in increments of 200 ps. Or, touch one of the **preset value** buttons (also mentioned in the previous step).

7. Choose **Positive**, **Negative**, or **Either** from the slope choices. 

Note: Use the **TriggerScan** tab to run the **Trainer** (a wizard-like tool to configure triggers), **Save**, and **Load** trigger setups for reuse.

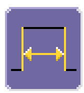
Width Trigger

How Width Trigger Works

Width trigger allows you to define a positive or negative-going pulse that is width bounded by a voltage level, above or below which a trigger will occur. You can specify a pulse width and voltage range, within or outside of which a trigger will occur.

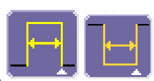
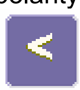
Width Trigger Setup

1. Touch the **Trigger** descriptor label. 

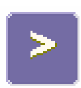
2. Touch the **Width** trigger button. 
3. Touch inside the trigger **Source** field and select a source on which to trigger. If you select an external trigger source, you will also need to set a coupling mode.

Touch inside the **Level** data entry field and enter a value using the slider bar at the bottom of the window. Click the keypad button  on the slider bar to enter a value using the pop-up keypad. Touch the Default button 

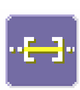
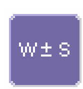
to set the level to zero mV.

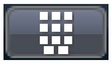
1. Select **Positive** or **Negative** polarity. 
2. Touch the **Less Than** button  and enter a pulse-width value in the **Upper Limit** field.

   OR

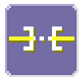
   Touch the **Greater Than** button  and enter a pulse-width value in the **Lower Limit** field.

   OR

   Touch the **In Range** button.  Then touch the **Delta** button  to set up a nominal range, plus or minus a delta value in seconds. Touch inside the **Nominal Width** and **Delta** fields and enter values, using the pop-up numeric keypads. Alternatively, touch the **Limits** button  to set up a precise pulse-width range by setting **Lower Limit** and **Upper Limit** values, using the slider bar at the bottom of the window.

Click the keypad button  on the slider bar to enter values using the pop-up keypads. Touch the Default button  to use preset values.

OR

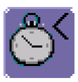
Touch the **Out Of Range** button  and perform the same range setups as for In Range triggering.

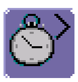
Note: Use the **TriggerScan** tab to run the **Trainer** (a wizard-like tool to configure triggers), **Save**, and **Load** trigger setups for reuse.

Qualified Trigger

How Qualified Triggers Work

Use a signals transition above or below a given level (its validation) as an enabling (qualifying) condition for a second signal that is the trigger source. These are Qualified triggers. For Edge Qualified triggers (the default), the transition is sufficient and no additional requirement is placed on the first signal. For State Qualified triggers the amplitude of the first signal must remain in the desired state until the trigger occurs. A qualified trigger can occur immediately after the validation, or following a predetermined time delay or number of potential trigger events. The time delay or trigger count is restarted with every validation.

 **Within Time** creates a time window within which a trigger can occur.

 **Wait Time** determines a delay from the start of the desired pattern. After the delay (timeout) and while the pattern is present, a trigger can occur. The timing for the delay is restarted when the selected pattern begins.

Events determines a minimum number of events of the trigger source. An event is generated when a trigger source meets its trigger conditions. On the selected event of the trigger source and while the pattern is present, a trigger can occur. The count is initialized and started whenever the selected pattern begins, and continues while the pattern remains. When the selected count is reached, the trigger occurs.

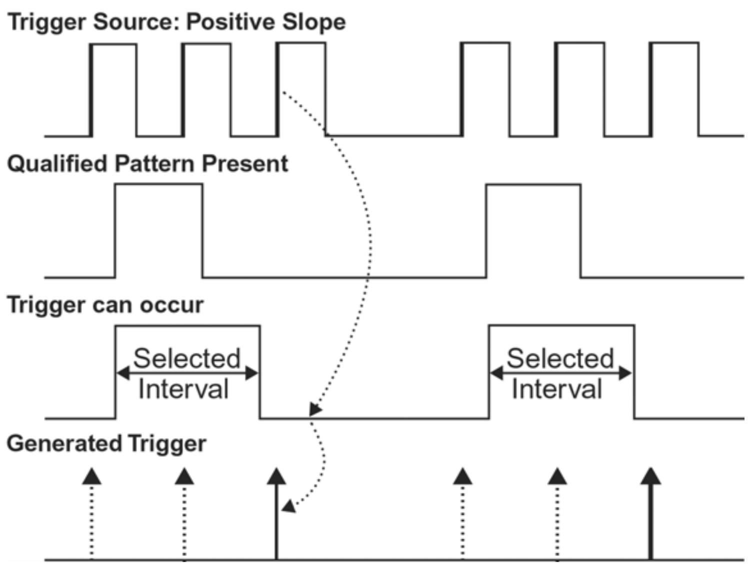


Figure 4-16. Edge Qualified and Wait: Trigger after timeout. The broken upward-pointing arrows indicate potential triggers, while the bold ones show where the actual triggers occur.

Qualified First Trigger

Not available on all scopes, Qualified First trigger is intended to be used exclusively in Sequence Mode to speed up the trigger rate. With Qualified First trigger, a single valid trigger is sufficient to acquire a full sequence. Other than in Sequence Mode, Qualified First is identical to the Qualified triggers.

In data storage applications, the index pulse can be defined as the qualifier signal and the servo gate signal as the trigger source.

Edge Qualified Trigger Setup

1. Touch the **Trigger** descriptor label. 

2. Touch the **Qualified** trigger button. 
3. Select a trigger type from the **Arm trigger on Event 'A'** section from **Edge**, **Pattern**, **State**, and **PatState** choices.
4. If a Pattern trigger type is selected in the previous step, choose from **Edge**, **Width**, **Glitch**, and **Interval** trigger types from the **Then trigger on Event 'B'** section. Otherwise, **Edge** is the only choice available.

State Triggers

Before we proceed to step 5 of Qualified Trigger, let's explain a few things about State Triggers. A State trigger is another Qualified trigger; however, instead of using the edges of the qualifying inputs, State trigger uses the logic state of the inputs to qualify the trigger. Therefore, the pattern must become true and remain true (for a period of time or number of events that you specify) to qualify the trigger.

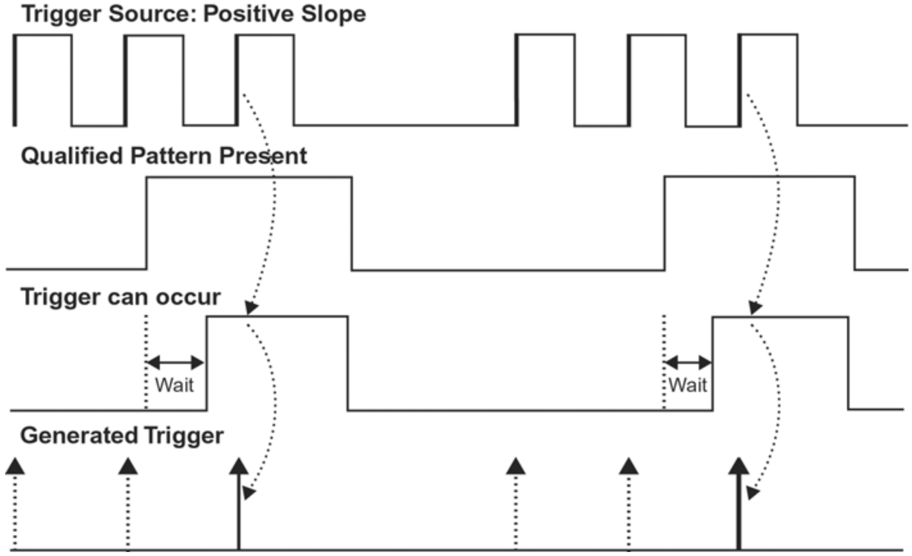
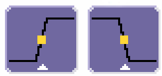


Figure 4-17. State Qualified and Wait: Trigger after timeout. The broken upward-pointing arrows indicate potential triggers, while the bold arrows show where the actual triggers occur.

5. Now, configure a setting to determine **When 'B' occurs** from **Any (Time)**, **Less Than**, **Greater Than**, and **Events**. If **Less Than**, **Greater Than**, or **Events** is selected, provide a corresponding **Before**, **After**, **N Events** value beneath using the **slider bar's value** slider, **pop-up keypad** button, or **Default** button.

6. Use the **Event 'A' Edge** tab to make Source, 

Coupling, 

Slope (positive or negative),  and **Level** selections. Add a level value using the **slider bar's value** slider, **pop-up keypad** button, or **Default** button.
7. Use the **Event 'B' Edge** tab to make **Source**, **Coupling**, **Slope**, and **Level** selections (done same as previous step). Use the **Find Level** button 

to automatically set the level.

Note: Use the **TriggerScan** tab to run the **Trainer** (a wizard-like tool to configure triggers), **Save**, and **Load** trigger setups for reuse.

Pattern (Logic) Trigger

How Pattern Trigger Works

Pattern Trigger enables triggering on a logical combination of up to five inputs: CH 1, CH 2, CH 3, CH 4, and EXT. The combination of inputs is referred to as a pattern. There are four logic gates available: AND, NAND, OR, NOR.

A trigger state is either high or low: high when a trigger source is greater than the trigger level (threshold) and low when less than it. For example, an AND pattern could be defined as true when the trigger state for CH 1 is high, CH 2 is low, and EXT is irrelevant (X or don't care). If any one of these conditions is not met, the pattern state is considered false.

Logic Applications

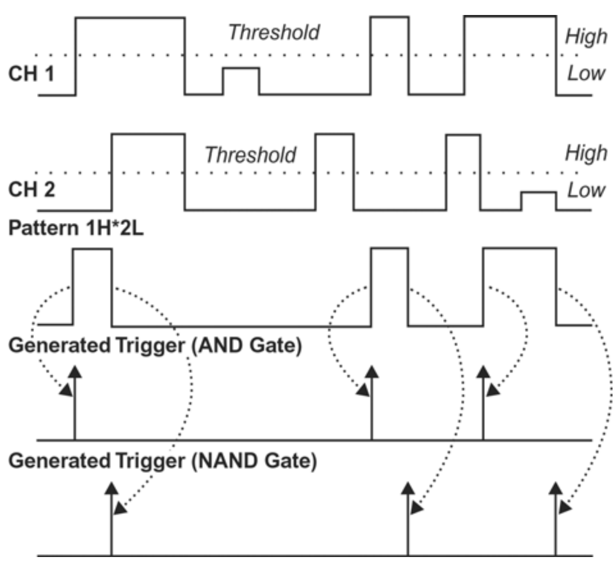
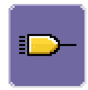
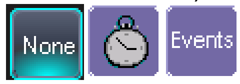
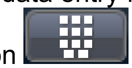


Figure 4-18. Pattern Trigger can be used in digital design for the testing of complex logic inputs or data transmission buses.

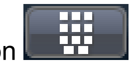
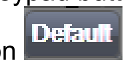
Pattern Trigger Setup

1. Touch the **Trigger** descriptor label. 

2. In the **Trigger Type** dialog, touch the **Pattern** trigger button. 
3. If you want to hold off the trigger (either in time or events) when the pattern becomes true, touch one of the **Holdoff By:** buttons on the **Holdoff** tab. 

Then touch inside a holdoff data entry field and enter a value using the slider bar at the bottom of the window. Click the keypad button  on the slider bar to enter a value using the pop-up keypad. Touch the Default button 

to use preset values.
4. If applicable, touch inside the **State** fields for channels 1-4 or EXT and select a logic state of either **Low** or **High**. Select **Don't Care** for all other inputs.

5. Touch inside the corresponding **Level** data entry field for each input included in the pattern, and enter a voltage level threshold, using the slider bar at the bottom of the window. Click the keypad button  on the slider bar to enter a value using the pop-up keypad. Touch the Default button 

to use preset values.

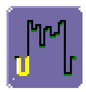
Note: Use the **TriggerScan** tab to run the **Trainer** (a wizard-like tool to configure triggers), **Save**, and **Load** trigger setups for reuse.

TV Trigger

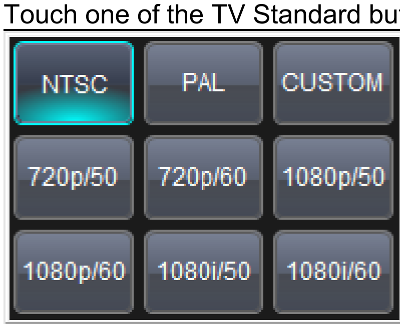
Not available on all instruments, **TV** triggers provide stable triggering on standard or custom composite video signals. Use them on PAL, SECAM, or NTSC systems.

TV Trigger Setup

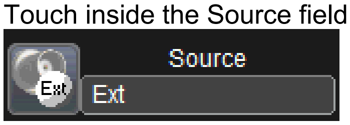
1. Touch the **Trigger** descriptor label. 

2. In the Trigger dialog, touch the **TV** trigger button. 

3. Touch one of the TV Standard buttons to select a television standard.

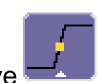
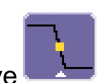


4. Touch inside the Source field and select the Ext (location of Trigger Out cable) from the pop-up menu.



Note: The channel that you have the VT75 Module connected to is automatically designated **TV Signal** in the menu.

5. Touch inside the **Polarity** field 

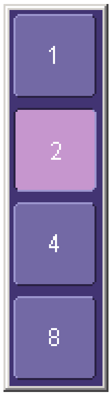
and select positive  or negative  polarity. This selection is independent of the polarity selected for Edge trigger (positive) in the **Trigger** setup dialog.

6. Check the **Trigger on Any Line** checkbox 

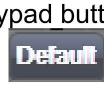
if you do not want to specify. When one is not specified, the scope triggers on any sync pulse.

7. Touch inside # of Fields 

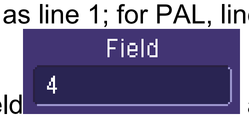
and make a selection (1 to 8) from the pop-up menu.



8. Touch inside the **Line** field 

and enter a line number using the slider bar at the bottom of the window. Click the keypad button  on the slider bar to enter a value using the pop-up keypad. Touch the Default button 

to use a preset value. The maximum line number that you can enter depends on the television standard (NTSC/PAL/SECAM) and the interlace factor selected. For NTSC, line 526 is the same as line 1; for PAL, line 626 is the same as line 1.

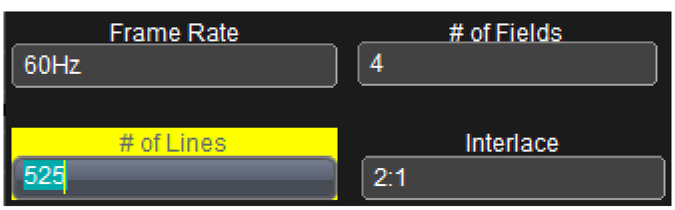
9. Touch inside Field 

and make a selection from the pop-up menu. The maximum field number that you can select is limited by your selection in **# of Fields**. If # of Fields above is set to **1**, this selection is disabled.

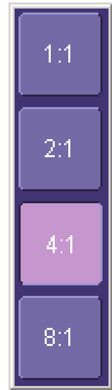
Custom TV Standard Trigger Setup

1. In the TV Standard field, touch the Custom button 

to select Custom mode.
2. Touch inside # of Lines, and enter a value up to 1500 using the **slider bar's value** slider, **pop-up keypad** button, or **Default** button.

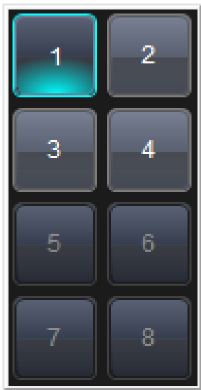


3. Touch inside Frame Rate and select 50, 60, 30, or 20Hz from the pop-up menu.

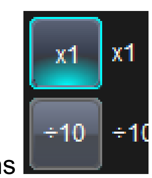


4. Touch inside Interlace and select an interlacing factor from the pop-up menu.

Note: A selection from this menu of 1:1 will reset the # of Fields selection to 1, since interlacing does not apply to a single field. The maximum allowable interlace factor is equal to the # of Fields set.

5. Use the Trigger On section of the dialog to configure the trigger for **ANY** line (using the checkbox) or a specific **Line**, Field, 

and **Level** value. Set the Line and Level values using the **slider bar's value** slider, **pop-up keypad** button, or **Default** button.

6. The **Ext** tab can be used to make other TV trigger input attenuations 

(choose either **X1 or ÷10**) and impedance 

(choose **50Ω**, **1MΩ**, or **Gnd**) selections.

Note: Use the **TriggerScan** tab to run the **Trainer** (a wizard-like tool to configure triggers), **Save**, and **Load** trigger setups for reuse.

SMART Triggers

Note: Not all SMART triggers are available on all instruments.

Glitch Trigger

How Glitch Trigger Works

Glitch trigger can be used to catch glitches. You can specify a pulse width or a pulse width range.

Pulse smaller than selected pulse width: Set a maximum pulse width. This glitch trigger is generated on the selected edge (positive or negative) when the pulse width is less than or equal to the set width.

The timing for the width is initialized and restarted on the opposite slope to that selected.

> **Note**: If the glitch's width is narrower than the signal's width, set the trigger to a narrower width than that of the signal. The signal's width, as determined by the instrument trigger comparator, depends on the DC trigger level. If that level were to be set at the middle of a sine wave, for example, the width could then be considered as the half period. But if the level were higher, the signal's width would be considered to be less than the half period.

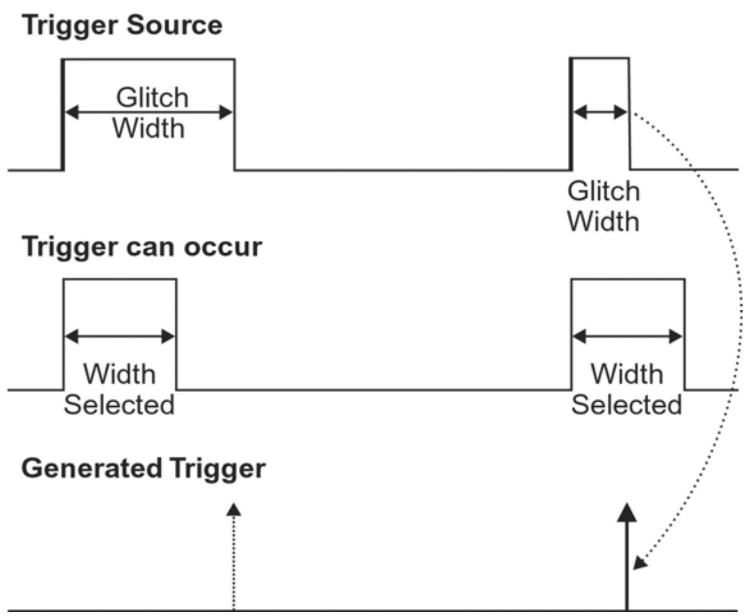
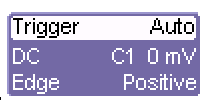
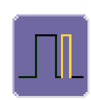


Figure 4-19. Glitch Trigger: In this example triggering on a pulse width less than or equal to the width selected. The broken upward-pointing arrow indicates a potential trigger, while the bold one shows where the actual trigger occurs.

Glitch Trigger Setup

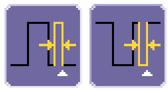
1. Touch the **Trigger** descriptor label. 

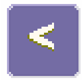
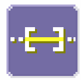
2. If applicable, touch the **Smart** trigger button,  then the **Glitch** trigger button. 

3. Touch inside the trigger **Source** field and select a source  on which to trigger.

4. Touch inside the **Level** data entry field and enter a value using the slider bar at the bottom of the window.

   Click the keypad button  on the slider bar to enter a value using the pop-up keypad. Touch the Default button  to use a preset value.

5. Select **Positive** or **Negative** polarity. 
6. Define the width of the glitch you are looking for. You can trigger on any glitch less than a chosen pulse-width (**Upper Limit**) or you can trigger on a specified range.

   - When you touch the **Less Than** button , the **Upper Limit** data entry field is made available.
   - When you touch the **In Range** button , the **Upper Limit** and **Lower Limit** fields are made available.
7. Touch inside the limit fields and enter a time value using the **slider bar's value** slider, **pop-up keypad** button, or **Default** button.

> **Note**: Use the **TriggerScan** tab to run the **Trainer** (a wizard-like tool to configure triggers), **Save**, and **Load** trigger setups for reuse.

Interval Trigger

How Interval Triggers Work

While Glitch trigger performs over the width of a pulse, Interval trigger performs over the width of an interval, with the signal duration (period) separating two consecutive edges of the same polarity: positive to positive or negative to negative. Use Interval trigger to capture intervals that fall short of, or exceed, a given time limit. In addition, you can define a width range to capture any interval that is itself inside or outside the specified range: an exclusion trigger by interval.

Interval Less Than: For this Interval Trigger, generated on a time interval smaller than the one that you set, choose a maximum interval between two like edges of the same slope (positive, for example).

The trigger is generated on the second (positive) edge if it occurs within the set interval. The instrument initializes and restarts the timing for the interval whenever the selected edge occurs.

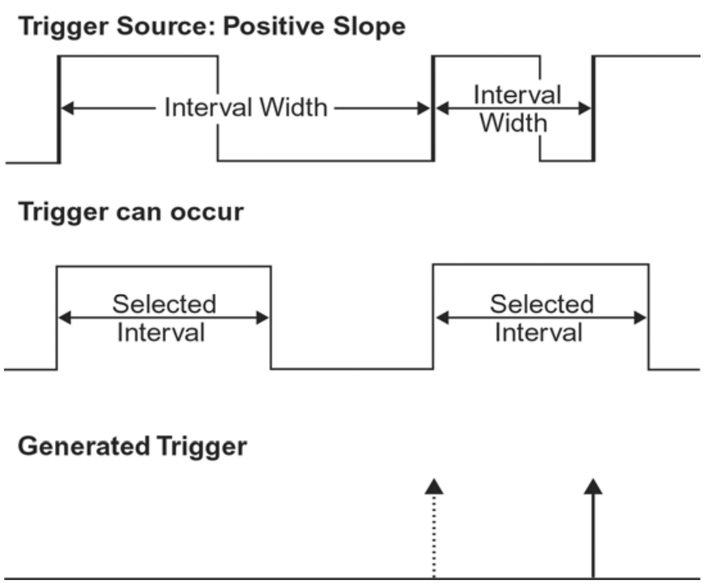


Figure 4-20. Interval Trigger that triggers when the interval width is smaller than the selected interval. The broken, upward-pointing arrow indicates a potential trigger, while the bold one shows where the actual trigger occurs — on the positive edge within the selected interval.

Interval Greater Than: For this Interval Trigger, generated on an interval larger than the one that you set, select a minimum interval between two edges of the same slope. The instrument generates the trigger on the second edge if it occurs after the set interval. The timing for the interval is initialized and restarted whenever the selected edge occurs.

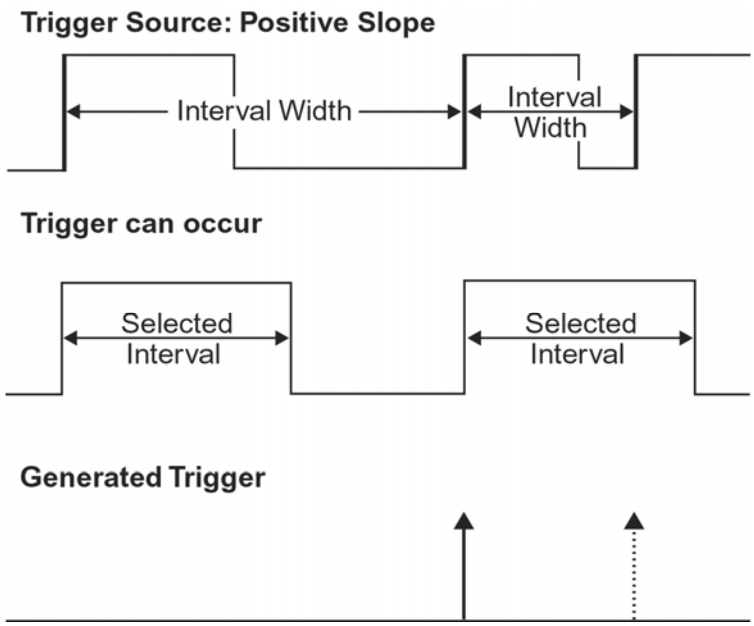


Figure 4-21. Interval Trigger that triggers when the interval width is larger than the set interval. The broken upward-pointing arrow indicates a potential trigger, while the bold one shows where the actual trigger occurs — on the positive edge after the selected interval.

Interval In Range: This Interval Trigger is generated whenever an interval between two edges of the same slope falls within a selected range. The instrument initializes and restarts the timing for the interval whenever the selected edge occurs.

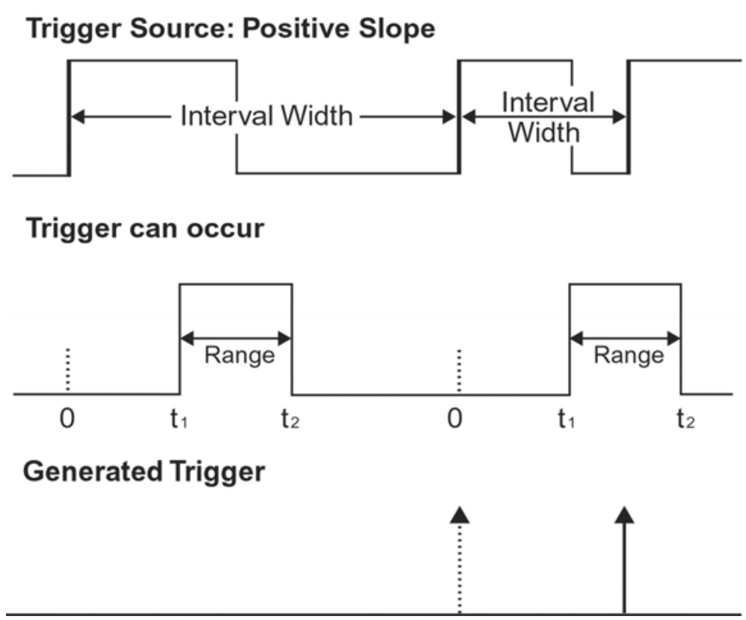
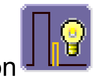
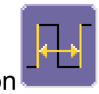


Figure 4-22. Interval Trigger that triggers when the interval falls within the selected range: t_1 = range's lower time limit; t_2 = range's upper limit. The broken upward-pointing arrow indicates a potential trigger, while the bold one indicates where the actual trigger occurs — on the positive edge within the selected range.

Interval Trigger Setup

1. Touch the Trigger descriptor label. 

2. If applicable, touch the **Smart** trigger button  , and then the **Interval** trigger button  .

3. Touch inside the trigger **Source** field and select a source 

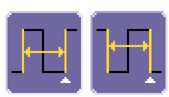
on which to trigger.

4. If available, touch inside the **Coupling** field and select a coupling mode. 

5. Touch inside the **Level** data entry field and enter a value using the slider bar at the bottom of the window.

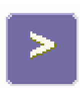
   Click the keypad button  on the slider bar to enter a value using the pop-up keypad. Touch the Default button 

to use a preset value.

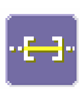
6. Select **Positive** or **Negative**polarity. 

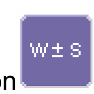
7. Touch the **Less Than** button  and enter a pulse-width value in the **Upper Limit** data entry field.

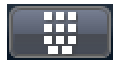
   OR

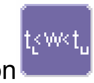
   Touch the **Greater Than** button  and enter a value in the **Lower Limit** data entry field.

   OR

   Touch the **In Range**button  .

   - Touch the **Delta** button 

to set up a nominal range, plus or minus a delta value in seconds. Touch inside the **Nominal Width** and **Delta** data entry fields and enter values using the slider bar at the bottom of the window. Click the keypad button  on the slider bar to enter values using the pop-up numeric keypads. Touch the Default button  to use preset values.

- Touch the **Limits** button  to set up a precise range. Touch inside the **Lower Limit** and **Upper Limit** data entry fields and enter values using the slider bar at the bottom of the window. Click the keypad button  on the slider bar to enter values using the pop-up numeric keypads. Touch the Default button  to use preset values.

Note: Use the **TriggerScan** tab to run the **Trainer** (a wizard-like tool to configure triggers), **Save**, and **Load** trigger setups for reuse.

Dropout Trigger

Used primarily in single-shot applications, and usually with a pre-trigger delay, Dropout trigger can detect lost signals. The trigger is generated at the end of the timeout period following the last trigger source transition.

How Dropout Trigger Works

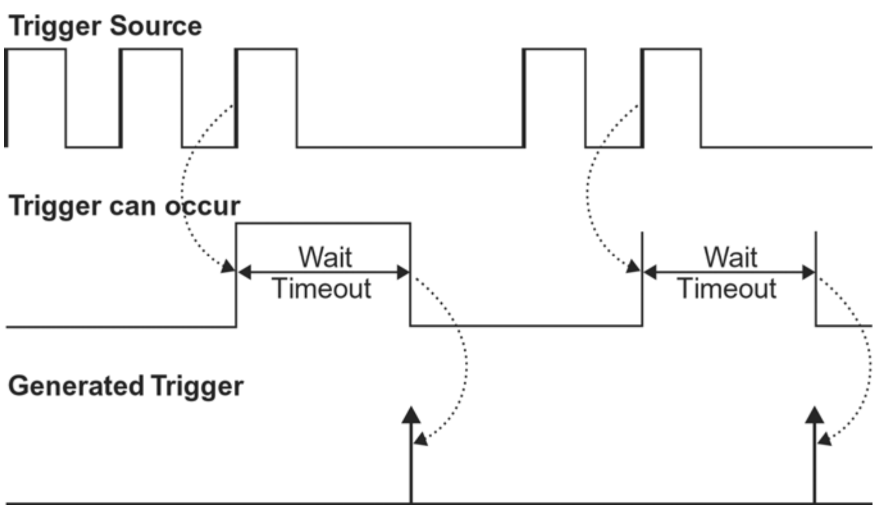
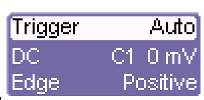
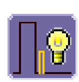
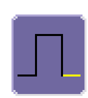


Figure 4-23. Dropout Trigger: occurs when the timeout has expired. The bold upward-pointing arrows show where the trigger occurs.

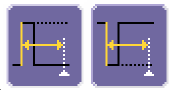
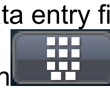
Dropout Trigger Setup

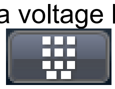
1. Touch the **Trigger** descriptor label. 

2. If applicable, touch the **Smart** trigger button,  and then the **Dropout** trigger button. 

3. Touch inside the trigger **Source** field and select a source  on which to trigger.

4. If available, touch inside the **Coupling** field and select a coupling mode.

5. Select **Positive** or **Negative**polarity. 
6. Touch inside the **Trigger after timeout if NO edge within** data entry field and enter a time value using the slider bar at the bottom of the window. Click the keypad button  on the slider bar to enter a value using the pop-up numeric keypad. Touch the Default button 

to use a preset value.
7. Enter a voltage level in the **Level** field using the slider bar at the bottom of the window. Click the keypad button  on the slider bar to enter a value using the pop-up numeric keypad. Touch the Default button 

to use a preset value. The Find Level button 

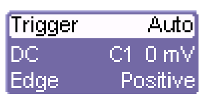
is available to automatically set the Level.

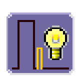
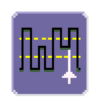
Note: Use the **TriggerScan** tab to run the **Trainer** (a wizard-like tool to configure triggers), **Save**, and **Load** trigger setups for reuse.

Runt Trigger

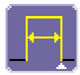
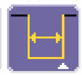
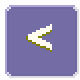
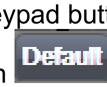
Not available on all instruments, the **Runt** trigger is programmed to occur when a pulse crosses a first threshold line and fails to cross a second threshold line before re-crossing the first. You can select both voltage thresholds within a time range of 100 ps to 20 s. Other defining conditions for this trigger are the edge (triggers on the slope opposite to that selected) and runt width.

Runt Trigger Setup

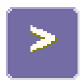
1. Touch the **Trigger** descriptor label. 

2. If applicable, touch the **Smart** trigger button,  and then touch the **Runt** trigger button. 
3. Touch inside the trigger **Source** field and select a source 

on which to trigger. If you select an external trigger source, you also need to set a coupling mode.

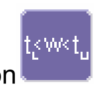
4. Touch inside the **Level** data entry field and enter a value, using the pop-up keypad.
5. Select **Positive** or **Negative**polarity.  
6. Touch the **Less Than** button  on some oscilloscopes and enter a pulse-width value in the **Upper Limit** field using the slider bar at the bottom of the window. Click the keypad button  on the slider bar to enter a value using the pop-up keypad. Touch the Default button 

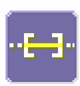
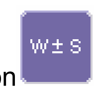
to use preset values.

   Touch the **Greater Than** button  on some oscilloscopes and enter a pulse-width value in the **Lower Limit** field using the **slider bar**, **pop-up keypad** button, or the **Default** button.

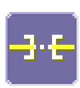
   OR

   Touch inside the **Nominal Width** and **Delta** fields and enter values, using the pop-up numeric keypads.

   Alternatively, touch the **Limits** button  to set up a precise pulse-width range by setting **Lower Limit** and **Upper Limit** values, using the **slider bar**, **pop-up keypad** button, or the **Default** button.

   Touch the **In Range** button  on some oscilloscopes . Then touch the **Delta** button  to set up a nominal range, plus or minus a delta value in seconds using the **slider bar**, **pop-up keypad** button, or the **Default** button.

   OR

   Touch the **Out Of Range** button  on some oscilloscopes and perform the previous part of this step for In Range triggering.

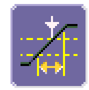
> **Note**: Use the **TriggerScan** tab to run the **Trainer** (a wizard-like tool to configure triggers), **Save**, and **Load** trigger setups for reuse.

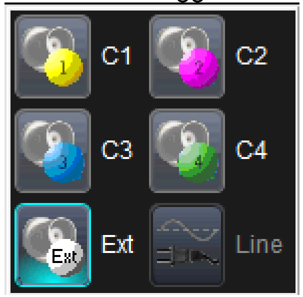
Slew Rate

Not available on all instruments, **Slew Rate** trigger activates a trigger when the rising or falling edge of a pulse crosses two threshold levels: an upper level and a lower level. The pulse edge must cross the thresholds faster or slower than a selected period of time. You can select both thresholds within a range of 100 ps to 20 s.

Slew Rate Trigger Setup

1. Touch the **Trigger** descriptor label. 

2. Touch the **Smart** trigger button,  and then touch the **Slew Rate** trigger button. 

3. Touch inside the trigger **Source** field and select a source 

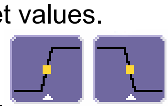
on which to trigger. If you select an external trigger source, you also need to set a coupling mode.

4. Touch inside the **Level** data entry field and enter a value using the slider bar at the bottom of the window.

   Click the keypad button  on the slider bar to enter a value using the pop-up keypad. Touch the

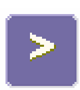
   Default button 

to use preset values.

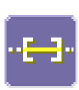
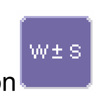
5. Select **Positive** or **Negative**polarity. 

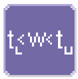
6. Touch the **Less Than** button  on some oscilloscopes and enter a pulse-width value in the **Upper Limit** field using the **slider bar**, **pop-up keypad** button, or the **Default** button.

   OR

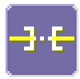
   Touch the **Greater Than** button  on some oscilloscopes and enter a pulse-width value in the **Lower Limit** field using the **slider bar**, **pop-up keypad** button, or the **Default** button.

   OR

   Touch the **In Range** button  on some oscilloscopes . Then touch the **Delta** button 

to set up a nominal range, plus or minus a delta value in seconds. Touch inside the **Nominal Width** and **Delta** fields and enter values using the **slider bar**, **pop-up keypad** button, or the **Default** button. Alternatively, touch the **Limits** button  to set up a precise pulse-width range by setting **Lower Limit** and **Upper Limit** values, using the **slider bar**, **pop-up keypad** button, or the **Default** button.

OR

Touch the **Out Of Range** button  on some oscilloscopes and perform the previous part of this step for In Range triggering.

> **Note**: Use the **TriggerScan** tab to run the **Trainer** (a wizard-like tool to configure triggers), **Save**, and **Load** trigger setups for reuse.

Trigger Setup Considerations

Trigger Modes

1. AUTO mode causes the scope to sweep even without a trigger. An internal timer triggers the sweep so that the display remains, even when the signal does not cause a trigger.
2. In NORMAL mode, the scope sweeps only if the input signal reaches the set trigger point. Otherwise it continues to display the last acquired waveform.
3. In SINGLE mode, only one sweep occurs each time you press the button.
4. STOP mode inhibits all sweeps until you select one of the other three modes.

Determining Trigger Level, Slope, Source, and Coupling

Level defines the source voltage at which the trigger circuit generates an event (a change in the input signal that satisfies the trigger conditions). The selected trigger level is associated with the chosen trigger source. Note that the trigger level is specified in volts and normally remains unchanged when the vertical gain or offset is modified.

The Amplitude and Range of the trigger level are limited as follows:

- ±5 screen divisions with a channel as the trigger source
- ±400 mV with EXT as the trigger source
- ±4 V with EXT/10 as the trigger source
- ±40 mV with EXT*10 as the trigger source
- none with LINE as the trigger source (zero crossing is used)

> **Note:** Once specified, Trigger Level and Coupling are the only parameters that pass unchanged from trigger mode to trigger mode for each trigger source.

Coupling refers to the type of signal coupling at the input of the trigger circuit. Choices of trigger coupling vary by scope model.

With DC coupling, all of the signal's frequency components are coupled to the trigger circuit for high-frequency bursts.

Slope determines the direction of the trigger voltage transition used for generating a particular trigger event.

Choose **Positive**, **Negative**, or **Either** from the slope choices. 

Like coupling, the selected slope is associated with the chosen trigger source.

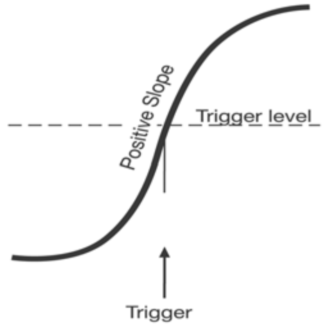
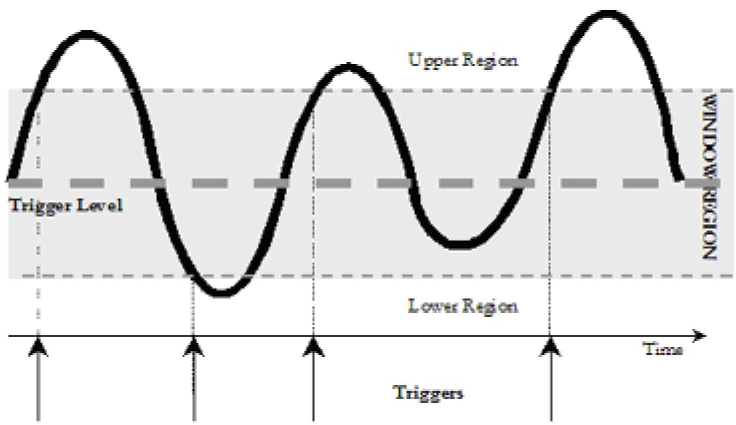


Figure 4-24. Edge trigger works on the selected edge at the chosen level. The slope (positive or negative) is specified in the Trigger label permanently displayed below-right of the grid.

Some instruments include the choice of **Window** slope, which sets a threshold above and below the trigger level beyond which the signal must pass to generate a trigger. The slope can be either positive or negative.



Trigger Source

The trigger **Source** may be one of the following:

- The acquisition channel signal (CH 1, CH 2, CH 3 or CH 4) conditioned for the overall voltage gain, coupling, and bandwidth.
- The line voltage that powers the oscilloscope (LINE). This can be used to provide a stable display of signals synchronous with the power line. Coupling and level are not relevant for this selection.
- The signal applied to the EXT BNC connector (EXT). This can be used to trigger the oscilloscope within a range of ±400 mV on EXT, ±4 V with EXT/10 as the trigger source, or ±40 mV with EXT*10 as the trigger source.
- A logic pattern.

Holdoff by Time or Events

Holdoff is an additional condition of Edge trigger. It can be expressed either as a period of time or an event count. Holdoff disables the trigger circuit for a given period of time or number of events after the last trigger occurred. Events are the number of occasions on which the trigger condition is met. The trigger will resume when the holdoff has elapsed and the trigger's other conditions are met.

Use holdoff to obtain a stable trigger for repetitive, composite waveforms. For example, if the number or duration of sub-signals is known you can disable them by choosing an appropriate holdoff value. Qualified triggers operate using conditions similar to holdoff.

Hold Off by Time

Sometimes you can achieve a stable display of complex, repetitive waveforms by placing a condition on the time between each successive trigger event. This time would otherwise be limited only by the input signal, the coupling, and the instrument's bandwidth. Select a positive or negative slope, and a minimum time between triggers. The trigger is generated when the condition is met after the selected holdoff time, counted from the last trigger. The delay is initialized and started on each trigger.

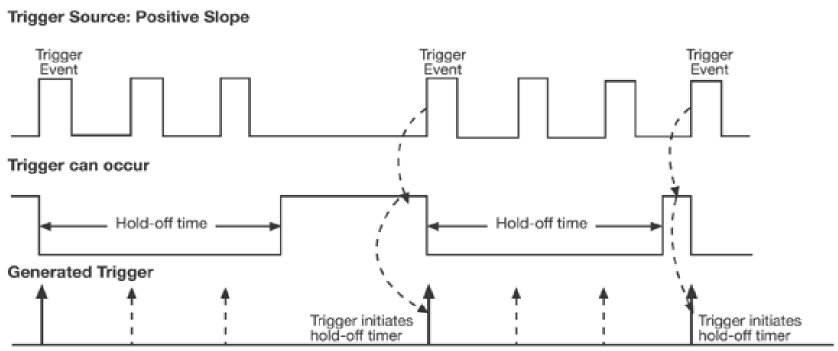


Figure 4-25. Edge Trigger with Holdoff by Time. The bold edges on the trigger source indicate that a positive slope has been selected. The broken upward-pointing arrows indicate potential triggers, which would occur if other conditions are met. The bold arrows indicate where the triggers actually occur when the holdoff time has been exceeded.

Hold Off by Events

Select a positive or negative slope and a number of events. An event is the number of times the trigger condition is met after the last trigger. A trigger is generated when the condition is met after this number, counted from the last trigger. The count is restarted on each trigger. For example, if the event number is two, the trigger will occur on the third event.

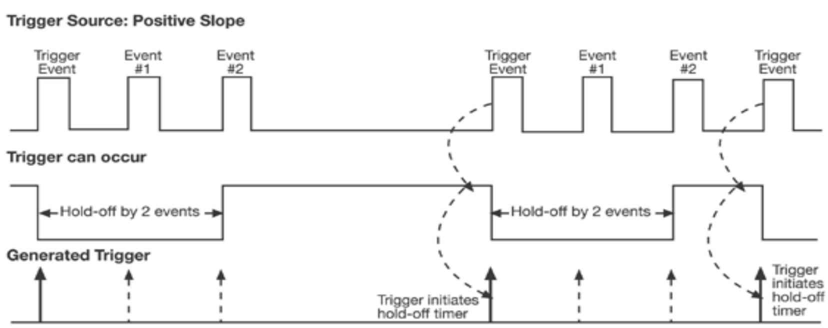
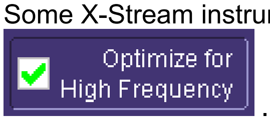


Figure 4-26. Edge Trigger with Holdoff by Events (in this example, two events). The bold edges on the trigger source indicate that a positive slope has been selected. The broken, upward-pointing arrows indicate potential triggers, while the bold ones show where triggers actually occur after the holdoff expires.

Optimizing for High Frequency

Some X-Stream instruments give you an option to reject high or low frequencies by means of a checkbox



. By default, the checkbox is selected, meaning, the instrument is optimized for high frequency waveforms. However, if you are measuring a waveform that is 10 MHz or slower, the checkbox should be unchecked to avoid triggering on an incorrect slope.

TriggerScan

TriggerScan is a debugging tool that helps you quickly find rare waveform glitches and anomalies. With TriggerScan, you can build a list of trigger setups to look for rare events and automatically sequence through each one. TriggerScan can use any type of trigger setup available including edge, width, and qualify as well as Smart Triggers (such as, glitch and runt triggers). TriggerScan automates two key processes in triggering rare events:

1. Trains the system by looking at normal acquired waveforms. During the training, the oscilloscope analyzes the waveforms to determine what waveforms normally look like. Using this information, it generates a list of smart trigger setups to trigger on abnormal situations.
2. Loads the smart trigger setups from the Trainer and cycles through these. As triggers occur, they are overlaid on the screen. All acquisition settings are preserved and you can use all the functions of the oscilloscope to find the root cause of these anomalies including, WaveScan, Histograms, and advanced analysis.

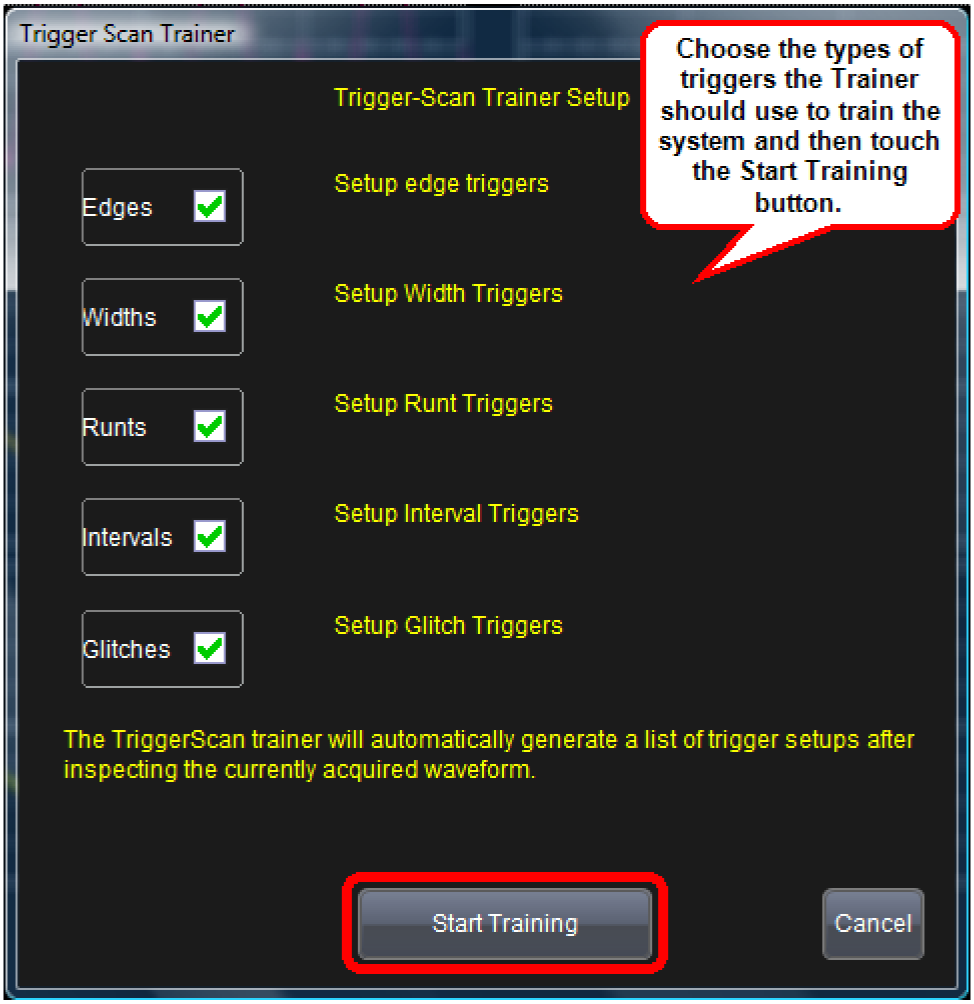
Training TriggerScan

The TriggerScan Trainer inspects a currently acquired waveform and automatically builds a list of common trigger setups used to find rare events.

PLEASE NOTE THE FOLLOWING:

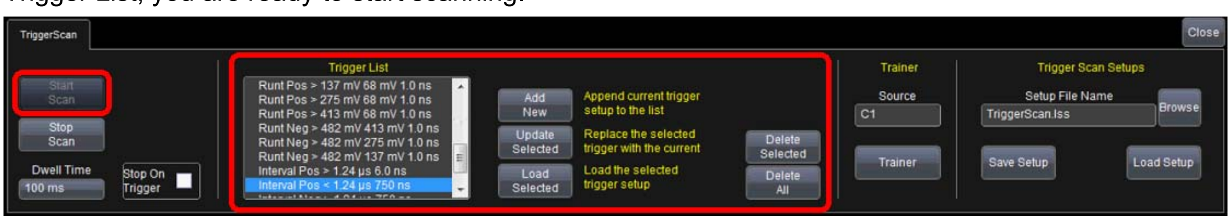
- You must acquire and display at least 3 cycles of a signal before running the Trainer.
- You should run the Trainer if you want to change the trigger types or if you change the channel or signal.

1. Touch **Trigger** → **Trigger Setup** from the menu bar.
2. On the **Trigger** dialog, click the **TriggerScan** tab.
3. Touch inside the **Source** data entry field and select a channel as the source for the training.
4. Touch the **Trainer** button.
5. On the **TriggerScan Trainer Setup** window, choose the types of triggers the Trainer should use to train the system and then touch the **Start Training** button. The training begins. When it is complete, a list of smart trigger setups is displayed in the Trigger List.



Starting TriggerScan

After you have run the Trainer, the Trigger List displays a list of smart trigger setups. You can add or remove trigger setups. You can also update the selected smart trigger setup. Once you have made any changes to the Trigger List, you are ready to start scanning.



1. Touch **Trigger** → **Trigger Setup** from the menu bar.
2. On the **Trigger** dialog, click the **TriggerScan** tab.
3. If you want to add a new trigger setup, touch the **Trigger** tab and set the new trigger. Then, touch the **Add New** button to add the new trigger to the Trigger List.
4. If you want to delete a trigger setup, highlight the setup in the Trigger List and touch the **Delete Selected** button.

Note: If you want to delete all trigger setups in the Trigger List, touch the **Delete All** button.

5. If you want to replace the selected trigger setup with the current trigger setup, highlight the setup in the Trigger List and touch the **Update Selected**button.
6. Once you have made any changes to the Trigger List, touch Start Scan. The oscilloscope automatically sequences through all the trigger setups.

PLEASE NOTE THE FOLLOWING:

- You can tune the dwell time that the scope will wait before loading the next trigger setup using the **Dwell Time** data entry field.
- If you have Persistence display mode enabled, all trigger events are recorded on the display. Refer to the **Persistence Setup** topic for instructions on enabling Persistence display mode.
- If you want TriggerScan to stop when the scope triggers next, check the **Stop On Trigger** checkbox. You can use this to isolate specific trigger setups.

Saving TriggerScan Setups

You should save TriggerScan setups once you have made any modifications to the Trigger List. The current Trigger List will not be preserved after exiting the application unless you save it.

1. On the **TriggerScan** dialog, touch inside the **Setup File Name** data entry field and enter a file name using the pop-up keyboard.

   OR

   Touch the **Browse** button and select a location and file name.
2. Touch the **Save Setup** button.

Note: You can load previously saved TriggerScan setups by touching the **Browse** button, locating the file and then clicking **Load Setup**.

Status

From **Status** tab on the **Utilities** dialog, you can view system status. The **System Status** read-only dialog displays system information including model number, serial number, firmware version, and installed software and hardware options.

Accessing the System Status Dialog

1. Touch **Utilities** → **Utilities Setup** on the menu bar.
2. Touch **System Status** on the **Utilities** dialog.

   OR

   Touch the **Status** tab.

Remote Communication

From the **Remote** tab on the **Utilities** dialog, you can set up remote communication. You can select a network communication protocol, establish network connections, and configure the Remote Control Assistant log from the **Remote** dialog. The choice of communication protocols is limited to TCPIP and GPIB.

PLEASE NOTE THE FOLLOWING:

- GPIB is an option and requires a GPIB card to be installed in a card slot at the rear of the scope.
- The instrument uses Dynamic Host Configuration Protocol (DHCP) as its addressing protocol. Therefore, it is not necessary to set up an IP address if your network supports DHCP. If it does not, you can assign a static address in the standard Windows 2000 network setup menu.

The Remote Control Assistant monitors communication between your PC and oscilloscope when you are operating the instrument remotely. You can log all events, or errors only. This log can be invaluable when you are creating and debugging remote control applications.

Setting Up Remote Communication.

If you are connecting the scope to a network, first contact your Information Systems administrator. If you are connecting the scope directly to your PC, connect a GPIB or Ethernet cable between them.

1. Touch **Utilities** → **Utilities Setup** on the menu bar.
2. Touch the **Remote** tab.
3. Make a **Port** selection: **TCPIP** (transmission control protocol/Internet protocol) or **GPIB** (general purpose interface bus). If you do not have a GPIB card installed, the GPIB selection will not be accessible.
4. If you are using GPIB, set a GPIB address by touching inside the **GPIB Address** data entry field and enter an address.

OR

If you are using TCPIP, press the **Net Connections** button; the Windows **Network Connections** window opens. You can reconfigure the oscilloscope's connection if it is already connected to the network or make a new connection.

Note: Your instrument allows you to restrict remote control access to certain clients. To restrict access, under **Security**, touch the **Yes** button and enter the IP addresses or domain name server names you want to restrict separated by a comma.

Configuring the Remote Control Assistant Event Log

1. Touch **Utilities** →**Utilities Setup** on the menu bar.
2. Touch the **Remote** tab.
3. Under **Remote Control Assistant**, touch inside the **Log Mode** data entry field. Select **Off**, **Errors Only**, or **Full Dialog** from the pop-up menu.
4. To export the contents of the event log to an ASCII text file, touch the **Show Remote Control Log** button. The **Event Logs** pop-up window opens. Touch inside the **DestFilename** data entry field and enter a file name, using the pop-up keyboard. Then, touch the **Export to Text File** button.

Hardcopy

From the **Hardcopy** tab on the **Utilities** dialog, you can print screen images. On the **Hardcopy** dialog, you can print the screen image to a printer, to the clipboard, or to a file. You can choose to print the waveform and grids only, the waveform and grids with the dialog, or the entire screen. You can also e-mail your screen images.

Printing

For instructions on printing and printer setup, refer to the **Printing** topic.

Clipboard

The **Clipboard** selection on the **Hardcopy** tab saves the screen image on the clipboard so you can paste a file into another application (like MS Word, for example).

Saving a Screen Image to the Clipboard

1. Touch **Utilities** → **Utilities Setup** on the menu bar.
2. Touch the **Hardcopy** tab, then the **Clipboard** icon.
3. Touch the **Colors** field and choose **Standard** to print the screen as it appears, **Print** to print the waveforms on a white background, or **Black & White** to print the waveforms in black & white.

Note: Choosing **Print** colors saves printer toner/ink.

4. Touch the **Hardcopy Area** field to choose which part of the screen you want to print. Choose **Grid Area Only** if you do not need to print the dialog area and you only want to show the waveforms and grids. Choose **DSO window** if you want to print the dialogs with the waveforms and grids. Choose **Full Screen** if you want to print the entire screen.
5. Touch the **Print Now** button .

Note: Once you have configured your **Clipboard** settings on the **Hardcopy** dialog, you can touch the PRINT front panel button and touch **Save Screen to Clipboard** on the **Print** flyout menu. The instrument uses the Clipboard settings to save the screen image to the clipboard.

File

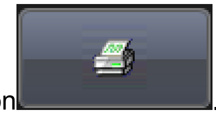
The **File** selection on the **Hardcopy** tab saves the screen image as a file to storage media such as a USB drive or hard drive.

Saving a Screen Image to a File

1. Touch **Utilities** → **Utilities Setup** on the menu bar.
2. Touch the **Hardcopy** tab, then the **File** icon.
3. Touch inside the **File Format** data entry field and select a graphic file format from the pop-up menu.

4. Touch the **Colors** field and choose **Standard** to print the screen as it appears, **Print** to print the waveforms on a white background, or **Black & White** to print the waveforms in black & white.

Note: Choosing **Print** colors saves printer toner/ink.

5. Touch inside the **File Name** data entry field and enter a name for the display image, using the pop-up keyboard.
6. Touch inside the **Directory** data entry field and type the path to the folder you want to print to, using the pop-up keyboard. Or touch the **Browse** button and navigate to the folder.
7. Touch the **Hardcopy Area** field to choose which part of the screen you want to print. Choose **Grid Area Only** if you do not need to print the dialog area and you only want to show the waveforms and grids. Choose **DSO window** if you want to print the dialogs with the waveforms and grids. Choose **Full Screen** if you want to print the entire screen.
8. Touch the **Print Now** button .

Note: Once you have configured your **File** settings on the **Hardcopy** dialog, you can touch the PRINT front panel button and touch **Save Screen to File** on the **Print** flyout menu. The instrument uses the File settings to save the screen image to a file (incrementing the filename by one each time you touch **Save Screen to File**).

E-Mail

The **E-mail** selection on the **Hardcopy** tab gives you the option to e-mail your screen images, using either the MAPI or SMTP protocols. Before you output to e-mail from the **Utilities** dialog, you first have to set up the e-mail server and recipient address in **Preference Setup**.

E-Mailing a Screen Image

1. Touch **Utilities** → **Utilities Setup** on the menu bar.
2. Touch the **Hardcopy** tab, then the **E-mail** icon.
3. Touch inside the **File Format** data entry field and select a graphic file format from the pop-up menu.
4. Touch the **Colors** field and choose **Standard** to print the screen as it appears, **Print** to print the waveforms on a white background, or **Black & White** to print the waveforms in black & white.

Note: Choosing **Print** colors saves printer toner/ink.

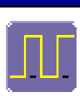
5. Touch the **Prompt for message to send with mail** checkbox if you want to include remarks with the image.
6. Touch the **Hardcopy Area** field to choose which part of the screen you want to print. Choose **Grid Area Only** if you do not need to print the dialog area and you only want to show the waveforms and grids. Choose **DSO window** if you want to print the dialogs with the waveforms and grids. Choose **Full Screen** if you want to print the entire screen.
7. Touch the **Print Now** button .

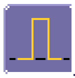
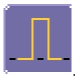
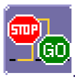
Note: Once you have configured your **Email** settings on the **Hardcopy** dialog, you can touch the PRINT front panel button and touch **Send Screen to Email** on the **Print** flyout menu. The instrument uses the Email settings to send the screen image.

Auxiliary Output Signals

From **Aux Output** tab on the **Utilities** dialog, you can output auxiliary signals. In addition to a calibration signal, the following signals can be output through the AUX OUTPUT connector:

Note: Not all of these options are available on some oscilloscope models.

Button:	Description:
	**Square Wave** You can set a Frequency value from 5.0 Hz to 5 MHz.

Button:	Description:
	**Trigger Out** — can be used to trigger another scope.
	**DC level —** a reference level
	**Trigger Enabled** — can be used as a gating function to trigger another instrument when the scope is ready
	**Pass/Fail** — allows you to set a pulse duration from 1 ms to 500 ms; generates a pulse when pass/fail testing is active and conditions are met.
	**Aux Output Off** — turns off the auxiliary output signal

Setting Up Auxiliary Output

1. Touch **Utilities** → **Utilities Setup** on the menu bar.
2. Touch the **Aux Output** tab.
3. If you want a 1 kHz, 1 V Square Wave, touch the **Set to 1 kHz, 1 V Square Wave** button.

   OR

   If you want a specialized output, touch one of the following buttons under **Use Auxiliary Output For**: **Square Wave**, **Trigger Enabled**, **Trigger Out**, **Pass/Fail**, **or Off**.

Note: If you choose **Pass/Fail**, touch inside the **Pulse Duration** field (if applicable) and enter a value from 1 ms to 500 ms, using the pop-up numeric keypad.

4. If applicable, touch inside the **Amplitude** data entry field and enter a value, using the slider bar at the bottom of the window. Click the keypad button  on the slider bar to enter a value using the pop-up numeric keypad.
5. Touch inside the **Frequency** data entry field and enter a value, using the slider bar at the bottom of the window. Click the keypad button  on the slider bar to enter a value using the pop-up numeric keypad.

Date and Time

From the **Date and Time** tab on the **Utilities** dialog, you can manually set the time and date or get it from the Internet. If you elect to get the date and time from the Internet, you need to have the oscilloscope connected to the Internet through the LAN connector on the side panel. You can also set time zones and daylight savings time.

Setting the Date and Time Manually

1. Touch **Utilities** → **Utilities Setup** on the menu bar.
2. Touch the **Date/Time** tab.
3. Touch inside each of the **Hour**, **Minute**, **Second**, **Day**, **Month**, and **Year** data entry fields and enter a value, using the pop-up numeric keypad.
4. Touch the **Validate Changes** button.

Setting the Date and Time from the Internet

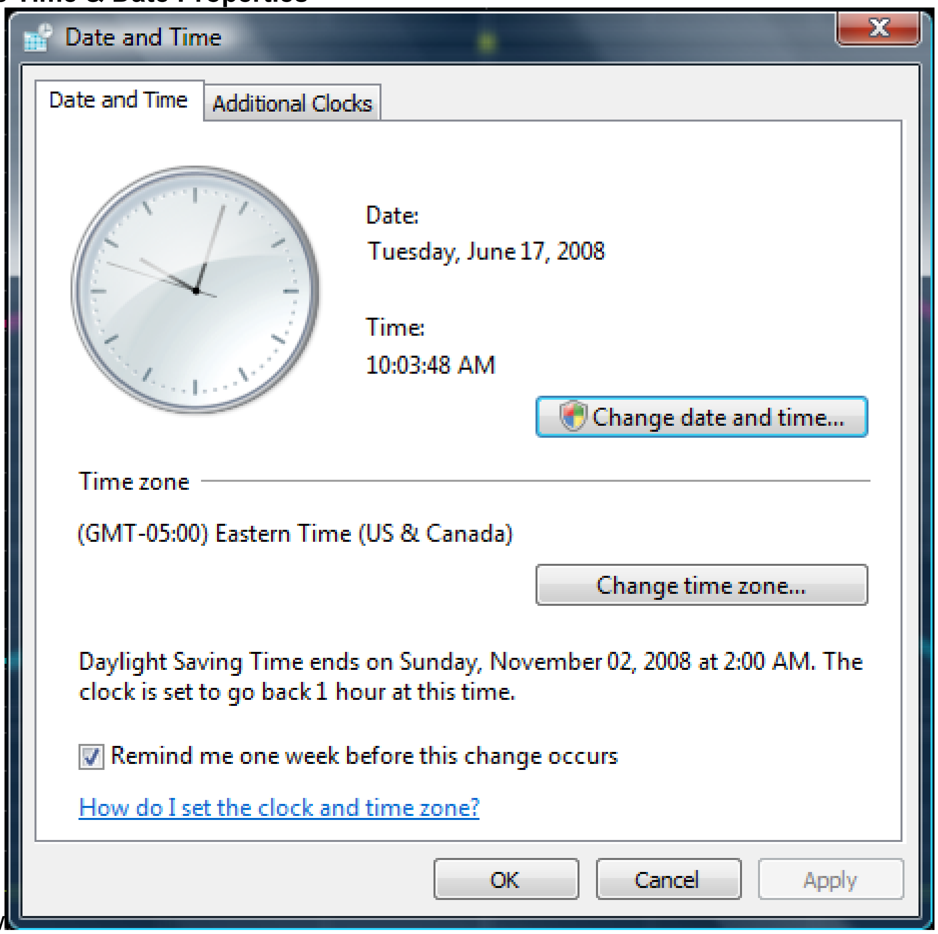
Note: The Simple Network Time Protocol (SNTP) is used.

1. Ensure that the scope is connected to the Internet through the LAN connector at the side of the scope.
2. Touch **Utilities** → **Utilities Setup** on the menu bar.

3. Touch the **Date/Time** tab.
4. Touch the **Set from Internet** button.

Setting the Date and Time from Windows

1. Touch **Utilities** → **Utilities Setup** on the menu bar.
2. Touch the **Date/Time** tab.
3. Touch the **Windows Date/Time** button .
4. Use the **Time & Date Properties** window



to configure the time, including time zone.

Options

From the **Options** tab on the **Utilities** dialog, you can add or remove software options. For information about software options, contact your local LeCroy Sales and Service office, or visit our Web site at http://www.lecroy.com/options.

Options that you purchase add performance to your instrument. This added performance is seen in the new math functions or parameters that you can choose from when doing Measure or Math setups.

Preferences

Enabling Audible Feedback

You can choose to have audible confirmation each time you touch a screen or front panel control. When Audible Feedback is enabled, the oscilloscope beeps each time you touch the screen or front panel control.

1. Touch **Utilities** → **Preferences** on the menu bar.
2. Touch the **Audible Feedback Enable** checkbox so that the oscilloscope beeps with each touch of the screen or front panel control.

Enabling Auto-calibration

You can choose to have your instrument automatically recalibrate itself whenever there is a significant change in ambient temperature. If you do not enable this option, the oscilloscope recalibrates only at startup and whenever you make a change to certain operating conditions.

1. Touch **Utilities** → **Preferences** on the menu bar.
2. Touch the **Automatic Calibration Enable** checkbox.

Optimizing Performance

If this option is available on your oscilloscope, you can set up the oscilloscope to optimize either calculating speed or display speed. If the display update rate is of primary concern to you, optimize for Display. If acquisition and analysis are more important, optimize for analysis. Optimizing for analysis can be useful when persistence or averaging is used, giving higher priority to waveform acquisition at the expense of display update rate.

The choices are presented as a spectrum with highest values at the extremes:

1. Touch **Utilities** → **Preferences** on the menu bar.
2. Touch one of the optimization icons.

Setting an Offset Control

As you change the gain, this control allows you to either keep the vertical offset level indicator stationary (when **Div** is selected) or to have it move with the actual voltage level (when **Volts** is selected). The advantage of selecting **Div** is that the waveform will remain on the grid as you increase the gain; whereas, if **Volts** is selected, the waveform could move off the grid.

> **Note:** Regardless of whether you select **Volts** or **Div**, the **Offset** shown in the channel setup dialog always indicates volts. However, when **Div** is selected for the Offset Control, the offset in volts is scaled proportional to the change in gain, thereby keeping the division on the grid constant.

1. Touch **Utilities** → **Preferences** on the menu bar.
2. Touch the **Acquisition** tab.
3. Under **Offset Setting constant in:**, touch either the **Div** or **Volts** button.

Setting a Delay Control

As you change the timebase, this control allows you to either keep the horizontal offset indicator stationary (when **Div** is selected) or to have it move with the trigger point (when **Time** is selected). The advantage of selecting **Div** is that the trigger point will remain on the grid as you increase the timebase; whereas, if **Time** is selected, the trigger point could move off the grid.

> **Note:** Regardless of whether you select **Time** or **Div**, the **Delay** shown in the timebase setup dialog always indicates time. However, when **Div** is selected for Delay In, the delay in time is scaled proportional to the change in timebase, thereby keeping the division on the grid constant.

1. Touch **Utilities** → **Preferences** on the menu bar.
2. Touch the **Acquisition** tab.
3. Under **Delay Setting constant in:**, touch either the **Div** or **Volts** button.

Configuring E-mail Settings

Before you can send e-mail from the oscilloscope, it must first be configured.

1. Touch **Utilities** → **Preferences** on the menu bar.
2. Touch the **E-mail** tab.
3. Choose an e-mail server protocol:
   MAPI (Messaging Application Programming Interface) is the Microsoft interface specification that allows different messaging and workgroup applications (including e-mail, voice mail, and fax) to work through a single client, such as the Exchange client. MAPI uses the default Windows e-mail application (usually Outlook Express).

SMTP (Simple Mail Transfer Protocol) is a TCP/IP protocol for sending messages from one computer to another through a network. This protocol is used on the Internet to route e-mail. In many cases no account is needed.

4. If you chose MAPI, touch inside the **Originator Address (From:)** data entry field and use the pop-up keyboard to type in the instrument's e-mail address. Then touch inside the **Default Recipient Address (To:)** data entry field and use the pop-up keyboard to enter the recipient's e-mail address.

   OR

   If you chose SMTP, touch inside the **SMTP Server** data entry field and use the pop-up keyboard to enter the name of your server. Touch inside the **Originator Address (From:)** data entry field and use the pop-up keyboard to type in the instrument's e-mail address. Then touch inside the **Default Recipient Address (To:)** data entry field and use the pop-up keyboard to enter the recipient's e-mail address.
5. To send a test e-mail text message, touch the **Send Test Mail** button. The test message reads "Test mail from [name of scope's email address]."

Acquisition Status

For each general category of oscilloscope operation, you can view a summary of your setups. To access these status dialogs, choose the following:

- **Vertical** → **Channels Status** on the menu bar
- **Timebase** →**Acquisition Status** on the menu bar
- **Trigger** → **Acquisition Status** on the menu bar
- **Math** → **Math Status**, **Zoom Status**, or **Memory Status** on the menu bar

In addition to these dialogs, summaries are also provided for XY setups, memory (M1-M4) setups, and time stamps for sequence mode sampling.

Service

 This button on the **Utilities Setup** dialog provides access to service dialogs, which are for the sole use of LeCroy service personnel. A security code is required to gain access.

Show Windows Desktop

Touch the **Show Windows Desktop** button  on the **Utilities Setup** dialog to minimize the instrument application. To maximize the application, touch the appropriate shortcut icon   .

Touch Screen Calibration

To start the calibration procedure, touch the **Touch-Screen Calibration** button .

During the calibration, you will be prompted to touch the center of a small cross in five key locations on the touch screen.

PLEASE NOTE THE FOLLOWING:

- Because sufficient accuracy cannot be achieved using your finger, use a stylus instead for this procedure.
- The calibration has a ten-second timeout in case no cross is touched.
- To avoid parallax errors, be sure to place your line of sight directly in front of each cross before touching it.

Adjusting Sensitivity and Position

Adjusting Sensitivity

1. Press the appropriate channel push button, for example 

to turn on channel 1.

   OR

   Touch **Vertical** → **Channel 1** from the menu bar.

2. Touch inside the **Trace On** checkbox 

to display the trace.
3. Turn the VERTICAL GAIN front panel knob for the selected channel.

   OR

   Touch inside the **Volts/Div** field and enter a value using slider bar. Click the **keypad** button to enter a value using the pop-up keypad, or use the up/down arrows.

   The voltage that you set is displayed in the trace descriptor label 

and in the **Volts/Div** field.

Adjusting the Waveform's Position

Turn the VERTICAL OFFSET adjust front panel knob directly above the channel button whose waveform you want to move vertically. Or you can touch inside the **Offset** field and type in a value on the pop-up keypad. To set the vertical offset to zero, touch the **Zero Offset** button directly below the **Offset** field.

Coupling

The choices of coupling are as follows:

- DC 50 Ω (all instruments)
- GROUND (all instruments)
- DC 1 MΩ (some instruments)
- AC 1 MΩ (some instruments)

Overload Protection

The maximum input voltage is 4 V peak. Whenever the voltage exceeds this limit, the coupling mode automatically switches from DC 50 Ω to GROUND. You will then have to manually reset the coupling to DC 50 Ω, as described next.

Setting Coupling

1. Touch **Vertical** → **Channel X Setup** from the menu bar.
2. Touch inside the **Coupling** field and select a coupling mode from the pop-up menu.

Probe Attenuation

Setting up Probe Attenuation

LeCroy's ProBus system automatically senses probes and sets their attenuation for you. If you want to set the attenuation manually,

1. In the menu bar, touch **Vertical**, then select a channel from the drop-down menu.

2. Touch inside the **Probe Atten.** data entry field 

. Touch a divide-by menu selection or touch **Var** (variable). If you choose **Var**, type in a value using the slider bar or the pop-up numeric keypad.

Bandwidth Limits

Reducing the bandwidth also reduces the signal and system noise, and prevents high frequency aliasing.

Setting Bandwidth Limits

Set bandwidth limits as follows:

1. In the menu bar, touch **Vertical**, then select a channel from the drop-down menu.
2. Touch inside the **Bandwidth** field and select a bandwidth limit value from the pop-up menu. The options are
   - Full (all X-Stream scopes)
   - 4 GHz
   - 3 GHz
   - 1 GHz
   - 200 MHz (all X-Stream scopes)
   - 20 MHz (all X-Stream scopes)

Linear and (SinX)/X Interpolation

Linear interpolation, which inserts a straight line between sample points, is best used to reconstruct straight-edged signals such as square waves. (Sinx)/x interpolation, on the other hand, is suitable for reconstructing curved or irregular wave shapes, especially when the sample rate is 3 to 5 times the system bandwidth.

Interpolation Setup

1. Touch the button for the channel you want to set up, 

 for example.
2. In the dialog area, touch inside the **Interpolation** data entry field under **Pre-Processing**. Pre-Processing is meant as before Math processing.
3. Touch inside the **Interpolation** data entry field. A pop-up menu appears offering **Linear** or **Sinx/x** interpolation.
4. Touch the button for the type of interpolation you want.

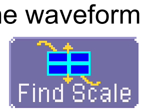
Inverting Waveforms

If available on your instrument, touch the **Invert** checkbox to invert the waveform for the selected channel.

Finding Scale

You can access the **Find Scale** button from the channel setup dialog. This feature automatically calculates peak-to-peak voltage, and chooses an appropriate Volts/Div scale to fully display the waveform.

Using Find Scale

1. Touch the trace label for the waveform you desire.
2. Touch the **Find Scale** icon 

.

Variable Gain

Variable Gain lets you change the granularity with which the gain is incremented. For example, when **Variable Gain** is disabled, the gain will increase or decrease in preset increments of 10 or 100 mV each time you touch the **Up/Down** buttons.

However, when **Variable Gain** is enabled, you can increase or decrease the gain in increments as small as 1 mV, depending on the scale of the waveform.

Enabling Variable Gain

1. Touch the descriptor label for the waveform whose gain you want to vary.
2. Touch the **Variable Gain** check box.

Channel Deskew

Unlike the Deskew math function, channel Deskew does no resampling, but instead adjusts the horizontal offset by the amount that you enter. The valid range is dependent on the current timebase +/- 9 divisions.

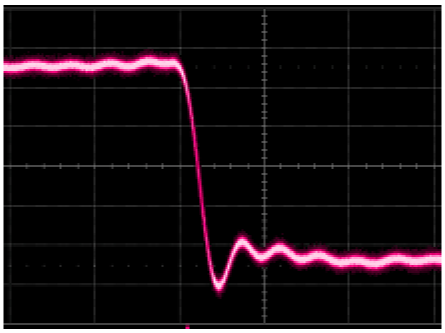
Channel Deskew Setup

1. In the menu bar, touch **Vertical**; from the drop-down menu, select a channel to set up.
2. Touch inside the **Deskew** data entry field and enter a value using the pop-up numeric keypad.

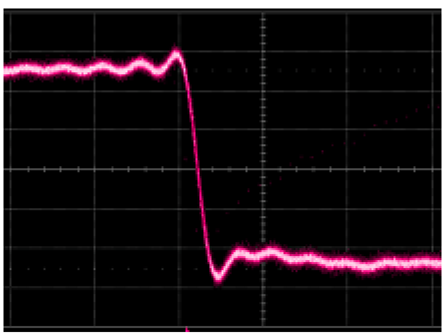
Group Delay Compensation

Group delay is defined as the rate of change of the total phase shift with respect to angular frequency through a device or transmission medium. An unavoidable consequence of correcting for group delay is that preshoot and overshoot will be added to the signal.

- **Pulse Response** - A delay compensation that minimizes preshoot. This selection is preferred in physics applications where there's no prediction of the future event.



- **Eye Diagram** - Delay compensation with less overshoot. This selection improves the accuracy of serial eye diagrams.



- **Flatness** - A delay compensation similar to eye diagram, this selection provides a flat frequency response.

Dark Calibration

Dark calibration is a compensation value added to a channel with an optical module. The **Dark Cal Level** is the residual power measured by the optical head with no input applied (dark input), and is used by the extinction ratio measurement. The dark calibration is independent of what type of module is connected to the selected channel, so an external optical-to-electrical converter can be compensated.

Performing Dark Calibration

1. In the menu bar, touch **Vertical**, then select a channel from the drop-down menu.
2. Touch the **Dark Cal** button 

.
3. A dialog box will appear instructing you to disconnect any optical signal from the channel. Touch the **Start Dark Calibration** button when this is done.
4. Dark calibration will be performed, and the dark level will appear in the **Dark Cal Level** field.

What Can AORM Do?

The Advanced Optical Recording Measurement (AORM) package for LeCroy digital oscilloscopes provides a set of waveform measurements and mathematical functions for the analysis of optical recording signals. Parameter measurements allow the categorizing and listing of measurement values in a variety of ways. The math functions (Histogramming and Trending) enable information to be graphically displayed.

The Advanced Optical Recording Measurement package provides parameter measurements for evaluating jitter due to intersymbol interface and emulation of DVD’s equalizer, slicer, and PLL. This functionality helps you to perform clock and jitter measurements, independent of a specific Integrated Circuit, allowing you to concentrate on optical head or media performance only. To support advanced optical recording drives that have constant angular velocity (CAV) or zone constant linear velocity (ZCLV), parameter measurements support automatic determination of the clock period.

Histogramming

Histograms can be created for any waveform parameter. They are displayed based on a set of user settings such as bin width or number of parameter events to be used. Histogram parameters are provided for measuring different histogram features such as standard deviation, number of peaks, and most populated bin. Histograms are selected by defining a trace as a math function, and selecting Histogram as the math function. As with other Zoom traces, histograms can be positioned and expanded by using the front panel **Position** and **Zoom** knobs.

Trending

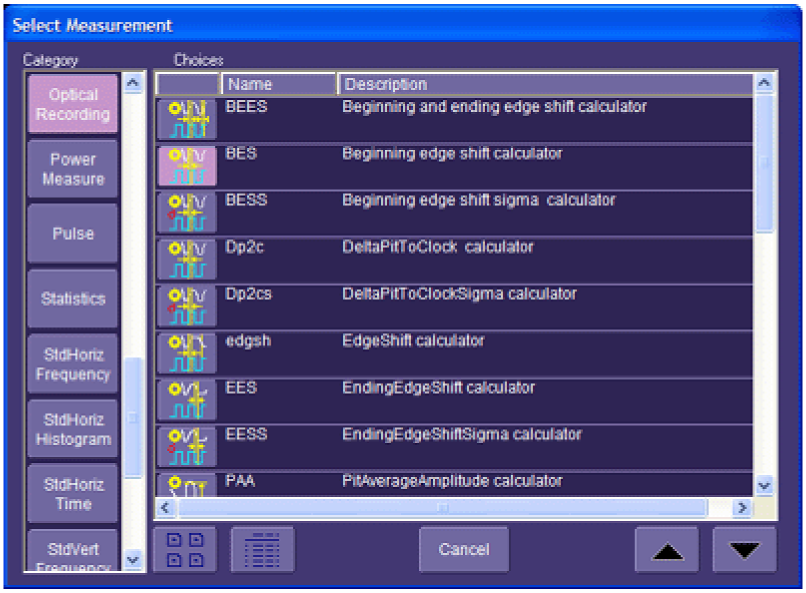
The Trend function allows you to create a graph containing successive waveform parameter measurement values. The trend function provides useful visual information on the variation of a waveform parameter within a sector, or even over multiple sectors. The Trend functionality, coupled with other scope features, enables you to graph certain parameters against one another.

Model of Optical Recording Processing

In many applications, it is important to make timing and jitter measurements directly from the RF signal, independent of a specific DVD chip. The optical recording processing function in AORM can perform this processing and can let you view the equalized data, sliced data, threshold, and/or the recovered clock. You can control the cutoff frequency and boost of the equalizing filter, the closed loop bandwidth of the 1st order integrating slicer, and the bandwidth of the phase-locked loop (PLL).

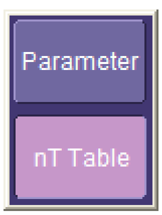
Selecting Parameters

1. Select **Measure** from the menu bar,
2. Touch the **Px** tab for the desired parameter position (P1 to P8).
3. Touch inside the **Measure** field, then select the **Optical Recording** group of parameters:



Parameters allow measurements of the section of waveform lying between the parameter cursors. The position of the cursors can be set by dragging, entering an exact value in the "Standard Cursors" dialog, or by means of the CURSORS front panel knobs. When you enable tracking by checking the **Track** checkbox, you can move the parameter cursors across the waveform so that measurement results can be taken on different sections of the waveform.

BES or EES Table



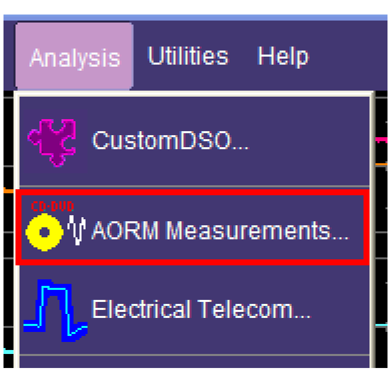
When the selected measurement is Begin Edge Shift (BES), End Edge Shift (EES), or their sigmas (BESS or EESS), the results can be shown in a table. Select **nT Table** from the **Show** menu to display a table of the average value of BES or EES for each subject nT vs. each “preceded” or “followed” nT:

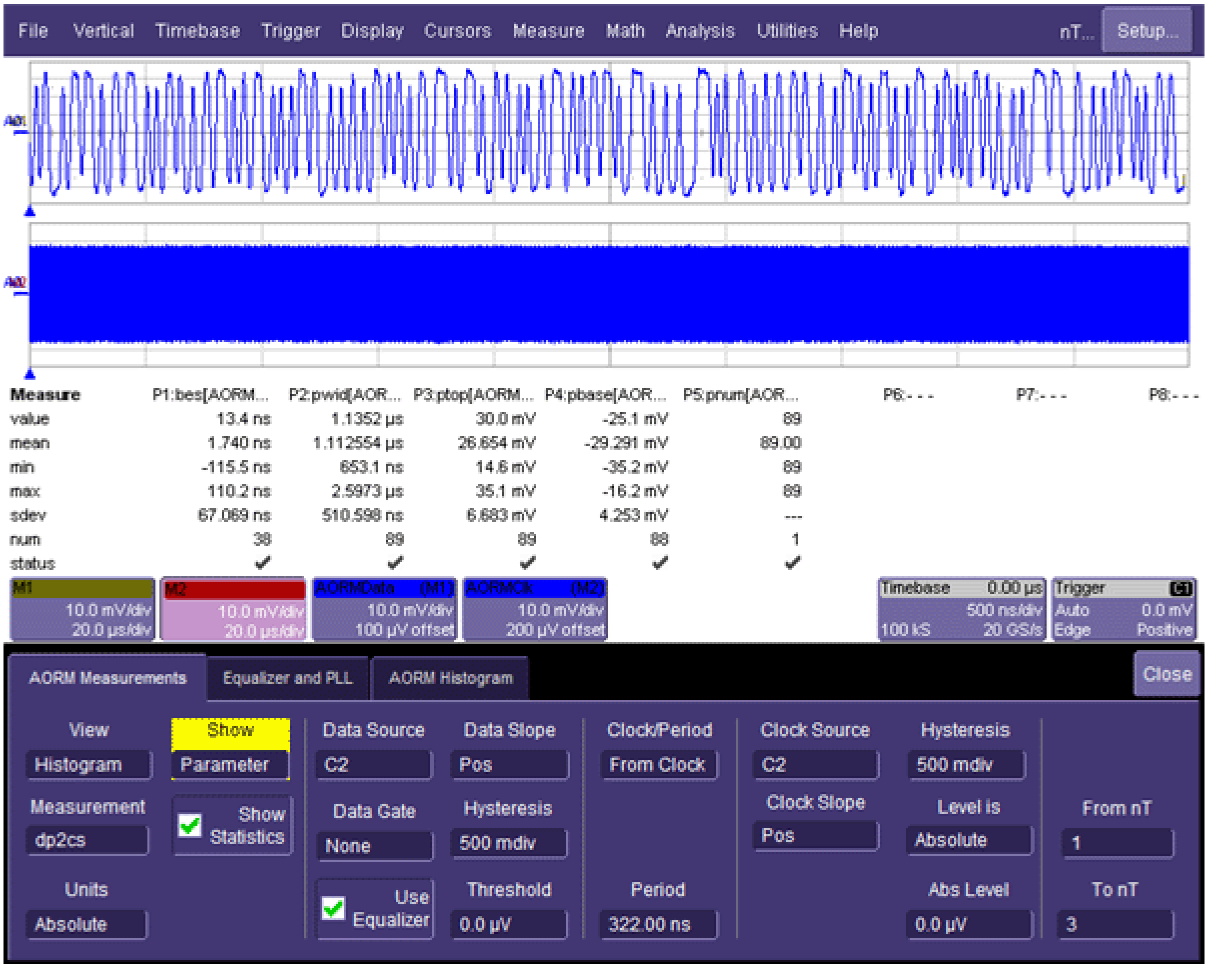
dp2cs								
1 T	---	---	68.0 ns	59.1 ns	48.9 ns	81.8 ns	36.7 ns	56.4 ns
9 T	54.1 ns	113.9 ns	21.4 ns	---	---	---	---	---
17 T	---	---	---	---	---	---	---	---
25 T	---							

Note: For EES, “Following nT” is displayed instead of “Preceding nT.” The nT is the range specified, starting from the low T. The subject T (i.e., s(n)) will also start from Low T to High T.

Setup and Measurement Dialog

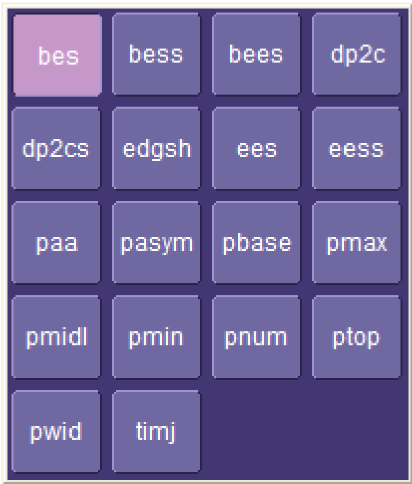
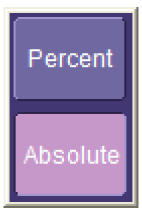
The “AORM Measurements” dialog is accessed from the menu bar’s **Analysis** menu. AORM is supplied with X-Stream software version 4.2 and later. This highly interactive dialog allows you to set up and configure the clock and data sources, select a measurement type, and analyze the waveform, including statistics on parametric measurements:

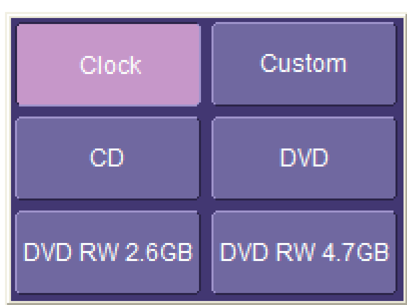




AORM Measurement Menus

Field	Description
View 	Allows you to quickly select the most common views, with graphing.

Measurement 	Selects the primary measurement to be made. In the Parameter view, the result for the selected measurement is displayed along with other parameters.	
Units 	The units used for the horizontal parameter results. **Absolute** refers to time or voltage units. **Percent** refers to results being calculated as a percent of the clock period.	
Show 	Determines which table of values is displayed below the grid. When **Parameters** is selected, a checkbox is displayed that enables you to turn on or off the display of statistics values. 	
Data	**Source**	This can be a channel, math function, or memory trace. A **Use Equalizer** checkbox is provided to apply a filter to the data source.
	Data Gate	This can be a channel, math function, memory trace, or other. It specifies the input that will be used to determine where to perform measurements on the input signal. The polarity is always high.
	Use Equalizer	This checkbox enables filtering of the data source.
	Data Slope	Polarity of the pits/spaces to use for the measurement, when appropriate. **Pos** polarity refers to pits, **Neg** polarity refers to spaces, **Both** can be selected to use both pits and spaces.
	Hysteresis	The Hysteresis selection imposes a limit above and below the

		Threshold, which precludes measurements of noise or other perturbations within this band. The width of the band is specified in divisions. **Guidelines for Use** 1. Hysteresis must be larger than the maximum noise spike you want to ignore. 2. The largest value of hysteresis usable is less than the distance from the threshold to the closest extreme value of the waveform. 3. Unless you know the largest noise and closest extreme level that will ever occur on any cycle, leave some margin on both sides of the threshold.
	Threshold	Sets the absolute crossing level.
Clock	The clock need only be specified if the parameter requires a clock for the calculation, or if it is used as the source of the period.	
	Clock/Period  	By selecting **From Data**, you can have the clock recovered from the data. In this case, all other clock setup fields except **Clock Slope** are unavailable. When **From Clock** is selected, the period is automatically measured from the clock provided. The clock must then be configured. When **Period Only** is selected,
	Period From  	Indicates the period of the clock. When a standard is selected, the period is set to the value defined by the standard. You can also set a **xN** multiplier (e.g., 10x).  **Custom** allows you to specify the clock period. When Custom is selected, another field is displayed in which to input the clock period.

	Clock Source	This can be a channel, math function, or memory trace.
	Clock Slope	Choose **Pos**, **Neg**, or **Near**. “Near” means the nearest clock edge to the data edge.
	Hysteresis	Size of the hysteresis band (in screen divisions) with the level at the center of the band. Any waveform being analyzed must pass beyond this band before the next threshold crossing is recognized. The **Level** may be set to **Absolute** or **Percent**.
Subject nT	For BES, EES, BEES, BESS, and EESS, this specifies the pit of interest. The results will be computed for each space/pit (pit/space) pair using the subject pit and all the spaces within the range specified.	
Preceded by low nT **Preceded by high nT**	Specifies the range of n indices that define the pits/spaces used in the calculation. The range of *n* coupled with T are used to categorize the pits/spaces based on their widths.	
From nT **To nT**	Specifies the range of *n* indices that define the pits/spaces used in the calculation. The range of *n* coupled with T is used to categorize the pits/spaces based on their widths.	

Measurements Table

Measurement	Parameters (setup for custom parameters)	XY (X-axis)
dp2c (s)	t@pit,pwid,pnum	t@pit
EDGESH	t@pit,pwid,pnum	t@pit
EES (s)	pwid,ptop,pbase,pnum	n/a
BES (s)	pwid,ptop,pbase,pnum	n/a
PAA	pwid,ptop,pbase,pnum	n/a
Pit width	t@pit, ptop,pbase,pnum	t@pit
timj	t@pit, ptop,pbase,pnum	t@pit
Pit base	pwid,ptop,pbase,pnum	pwid
Pit top	pwid,ptop,pbase,pnum	pwid
Pit minimum	pwid,ptop,pbase,pnum	pwid
Pit middle	pwid,ptop,pbase,pnum	pwid
Pit asym	pwid,ptop,pbase,pnum	n/a
Pit max	pwid,ptop,pbase,pnum	pwid
Pit num	pwid,ptop,pbase	n/a

View Menu Selections

View	Displays	Additional Functions
Parameter	The source traces will be displayed along with the custom parameters (see Measurements Table). If two traces are to be displayed, dual grids will be drawn.	**Statistics**: toggles the parameter statistics on/off.
Histogram	When **Histogram** is selected, a histogram of the selected parameter is displayed in a second grid, and a new tab, "AORM Histogram," appears.	**Find Center And Width**: determines the best scaling for the histogram based on up to the last 20,000 samples collected. This occurs automatically if the **Enable Auto Find** checkbox is checked.
Trend	When **Trend** is selected, a trend plot of the selected parameter is displayed in a second grid, and a new tab, "AORM Trend," appears.	**Find Scale**: determines the best scaling for the trend (center and height). This occurs automatically if the **Auto Find Scale** checkbox is checked.
XY Plot	Plots the trend of the selected measurement vs. either the trend t@pit or pwid, as appropriate; not available for all measurements (see Measurement Table, next, for details). When **XY Plot** is selected, a trend plot of the selected parameter is displayed in a second grid, and a new tab, "AORM Trend Y," appears.	**Find Scale**: determines the best scaling for trends (center and height). This occurs automatically if the **Auto Find Scale** checkbox is checked.

Equalizer and PLL Dialog

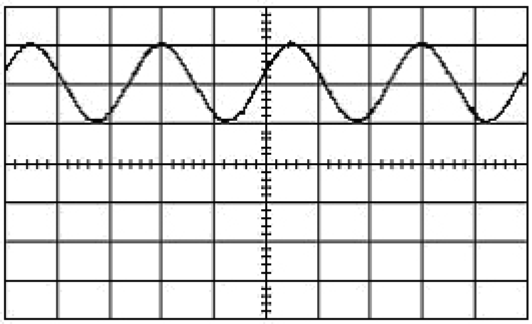
<table>
<tr><th>Field</th><th colspan="2">Description</th></tr>
<tr><td>Track Clock</td><td colspan="2">Check this checkbox to enable tracking.</td></tr>
<tr><td>Filter Cutoff and Boost</td><td colspan="2">An equalizer filter is applied prior to the measurements. You can adjust the cutoff frequency and boost of the filter.</td></tr>
<tr><td>Slicer Bandwidth</td><td colspan="2">The data passes through a slicer to level the data (removes the threshold due to low frequency effects). You can set the bandwidth of the slicer.</td></tr>
<tr><td>PLL Bandwidth</td><td colspan="2">If you checked the Clock From Data checkbox, a PLL is used to recover the clock from the data waveform. In that case, the bandwidth of the PLL can be adjusted.</td></tr>
<tr><td rowspan="2">PLL Startup Period From</td><td>Auto</td><td>When Auto is selected, the startup frequency will be determined by analyzing the peaks of the input signal.</td></tr>
<tr><td>User</td><td>When User is selected, you can set the startup frequency yourself. This is useful when the startup frequency cannot be reliably determined from the input signal.</td></tr>
</table>

Creating and Analyzing Histograms

The following guidelines provide a description of the oscilloscope's operational features for defining, using, and analyzing histograms. The sequence of steps is typical of this process.

Selecting the Histogram Function

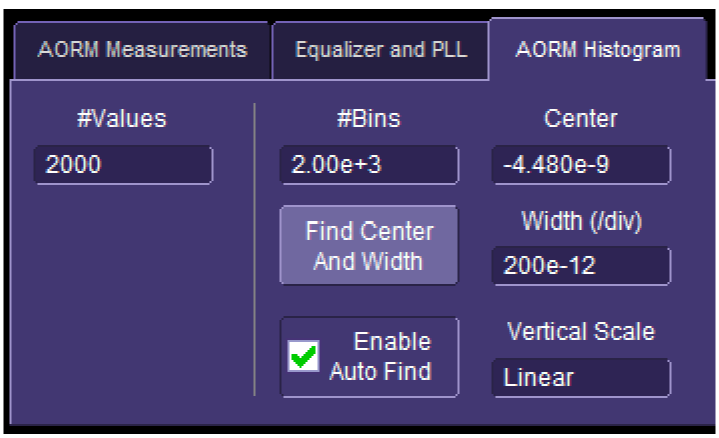
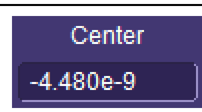
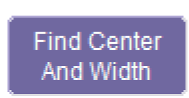
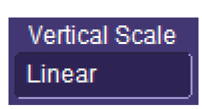
Histograms are created by graphing a series of waveform parameter measurements. The first step is to define the waveform parameter to be histogrammed. The next figure shows a screen display accompanying the selection of a frequency (freq) parameter measurement for a sine wave on Channel 1.

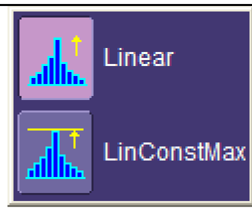


Four waveform cycles are shown, which will provide four freq parameter values for each histogram on each sweep. With a freq parameter selected, a histogram based on it can be specified.

Histogram Trace Setup Dialog

To the right of the main setup dialog, a Histogram setup dialog is displayed:

<table>
<tr><td>
</td><td>Each time a waveform parameter value is calculated it can be placed in a histogram bin. Enter the #Bins from 20 to 2000.
You can set from 20 to 2 billion parameter value calculations for histogram display in the #Values field.</td></tr>
<tr><td>
</td><td></td></tr>
<tr><td>
</td><td>The current horizontal per division setting for the histogram. The unit type used is determined by the waveform parameter type that the histogram is based on.</td></tr>
<tr><td>
</td><td></td></tr>
<tr><td>
</td><td>The vertical scale.
• Linear makes the vertical scale linear. The baseline of the histogram designates a bin value of 0. As the bin counts</td></tr>
</table>

	increase beyond that which can be displayed on screen using the current vertical scale, this scale is automatically increased in a 1-2-5 sequence. • **LinConstMax** sets the vertical scaling to a linear value that uses close to the full vertical display capability of the scope. The height of the histogram will remain almost constant.

Setting Binning and Histogram Scale

For either the **Linear** or **LinConstantMax** vertical scale option, the scope automatically increases the vertical scale setting as required, ensuring that the highest histogram bin does not exceed the vertical screen display limit.

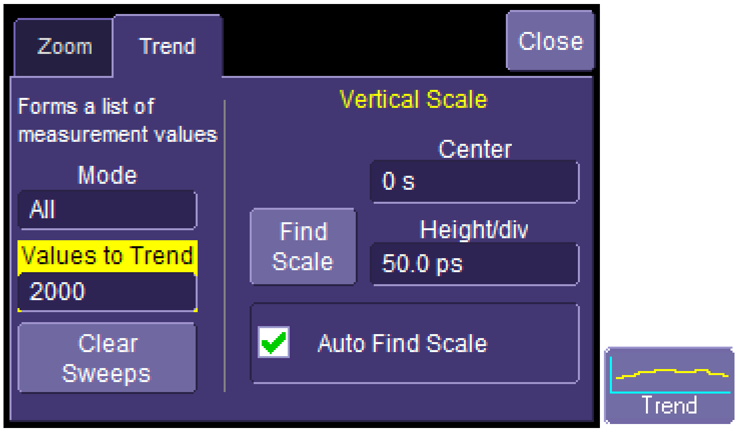
The **Center** and **Width** fields allow specification of the histogram center value and width per division. The width per division multiplied by the number of horizontal display divisions (10) determines the range of parameter values centered on the number in the **Center** field, used to create the histogram.

Displaying Trends

The Trend function for processing waveforms creates a graph of successive waveform parameter values. It provides useful visual information on waveform parameter variation. Used together with other scope features, it allows you to graph certain parameters compared to others.

To Configure a Trend:

1. From the menu bar select **Math**, then **Math Setup…** from the drop-down menu.
2. Touch an **Fx** tab that is not currently assigned a math function (i.e., Zoom function by default).
3. Touch inside the **Source1** field and select a source waveform from the pop-up menu.
4. Touch inside the **Operator1** field and select Trend from the **Select Math Operator** pop-up menu. The "Trend" setup dialog will appear at the right of the screen:



5. Decide whether all the parameter values, all per trace, or only the average of all parameter calculations for each waveform acquisition should be placed in the trend.

   All - every parameter calculation on each waveform will be placed in the trend.

   Average - trends only the average of all the values calculated on a given acquisition and yields one point in the trend per acquisition.

   All per Trace - for each acquisition, clears the buffer and places all parameter calculations from the new data in the trend. Unless this is specifically required, **All** should be selected.
6. Choose the number of values to be placed in the generated trend.
7. If desired, you can also configure the center and height of the trend in the base units of the parameter being trended. However, this is not a requirement, and **Find Scale** can be used to center the trend after it has been calculated.

   Center is for selecting the mantissa, exponent, or number of digits resolution. The configuration is the value at the horizontal center line on the grid, while units are those of the parameter trended.

   Height/div selects the vertical value of each vertical screen division. Units are those of the parameter trended.

Trend Calculation

Once the trend has been configured, parameter values will be calculated and trended on each subsequent acquisition. Immediately following an acquisition, its trend values will be calculated. The resulting trend is a waveform of data points that can be used the same way as any other waveform. Parameters can be calculated on it, and it can be zoomed, serve as the *x* or *y* trace in an XY plot, and can be used in cursor measurements.

The sequence for acquiring trend data is:

1. trigger
2. waveform acquisition
3. parameter calculations
4. trend update
5. trigger rearm

If the timebase is set in non-segmented mode, a single acquisition occurs prior to parameter calculations. However, in segment mode, an acquisition for each segment occurs prior to parameter calculations. If the source of trend data is a memory, storing new data to memory effectively acts as a trigger and acquisition. Because updating the screen can take significant processing time, it occurs only once a second, minimizing trigger dead time (under remote control the display can be turned off to maximize measurement speed).

Parameter Buffer

The parameter buffer allows you to include up to one million values in the trend calculation.

Capture of Parameter Events

The number of events captured per waveform acquisition or display sweep depends on the parameter type. Acquisitions are initiated by the occurrence of a trigger event. Sweeps are equivalent to the waveform captured and displayed on an input channel (1, 2, 3, or 4). For non-segmented waveforms, an acquisition is identical to a sweep. Whereas for segmented waveforms, an acquisition occurs for each segment, and a sweep is equivalent to acquisitions for all segments. Only the section of a waveform between the parameter cursors is used in the calculation of parameter values and corresponding trend events.

How to Read Trends

A trend is like any other waveform: its horizontal axis is in units of events, with earlier events in the leftmost part of the waveform and later events to the right. And its vertical axis is in the same units as the trended parameter. When the trend is displayed, trace labels appear in their customary place on the screen identifying the trace, the math function performed, and giving horizontal and vertical information:

- # number of events per horizontal division
- Units per vertical division, in units of the parameter being measured
- Vertical value at point in trend at cursor location when using cursors
- Number of events in trend that are within unzoomed horizontal display range.
- Percentage of values lying beyond the unzoomed vertical range when not in cursor measurement mode.

View Modes

The two modes available for AORM, “Parameter” and “List by nT,” both display measurements either as waveform parameters or as a list of values. The following table indicates which measurements can be made in each mode.

Measurement	Parameter	nT Table
BEES	x	x
BES	x	x
BESS	x	x
Dp2c	x	x
Dp2cs	x	x
EES	x	x
EESS	x	x
EDGSH	x	x
PAA	x	x
Pit asym	x	
Pit base	x	x
Pit max	x	x
Pit middle	x	x
Pit minimum	x	x

Pit moda	x	
Pit number	x	x
Pit res	x	
Pit top	x	x
Pit width	x	x
T@pit	x	
Timj	x	x

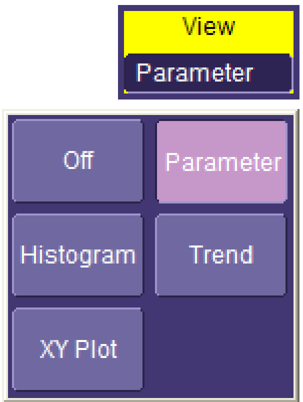
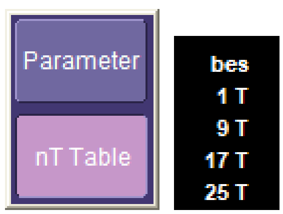
Configuration Options

All configuration options are available for each parameter, except as noted in this table:

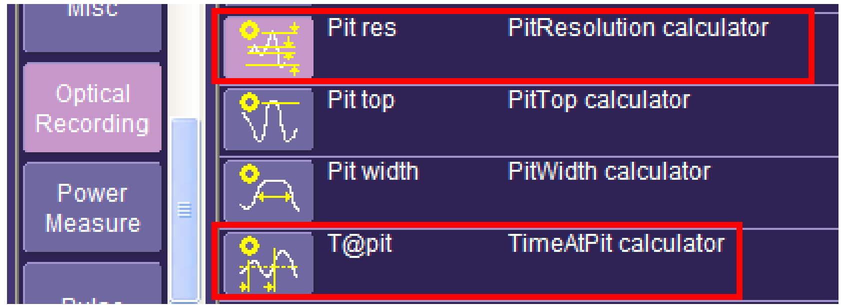
Parameter	Range of nT	Subject nT
Dp2c	x	
Dp2cs	x	
BEES		x
BES	x	x
BESS	x	x
EES	x	x
EESS	x	x
EDGSH	x	
PAA	x	
Pit asym	x	
Pit base	x	
Pit max	x	
Pit middle	x	
Pit minimum	x	
Pit moda	x	
Pit num	x	
Pit res*	x	
Pit top	x	
Pit width	x	
T@pit*	x	
Timj	x	

* Available from Measure dialog

Configuration Menus

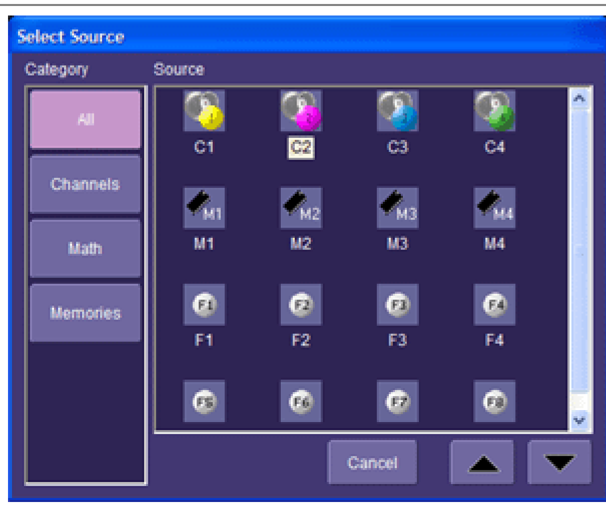
<table>
<tr>
<td>
</td>
<td>Display the AORM dialog by touching Analysis in the menu bar at the top of the screen and selecting AORM Measurements… from the drop-down menu.</td>
</tr>
<tr>
<td>
</td>
<td>Touch inside the View field and select a graph display from the pop-up menu.</td>
</tr>
<tr>
<td>
</td>
<td>Touch inside the Show field and select a display mode. A default group of parameters is automatically displayed when Parameter is selected. In addition, if Show Statistics is checked, standard statistics (mean, min, max, sdev, num) are also displayed.
When nT Table is selected, a default list of nT is displayed:

</td>
</tr>
</table>

Touch inside the **Measurement** field and select a measurement from the pop-up menu. Besides the parameters included in this menu, others are available from the **Select Measurement** menu, accessible from the "Measure" dialog.



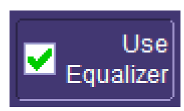
Setting Levels

To identify pits or spaces, thresholds and hysteresis are set.

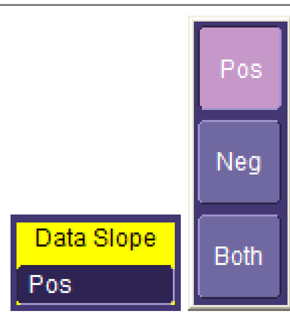


Touch inside the **Data Source** field and select a signal source.

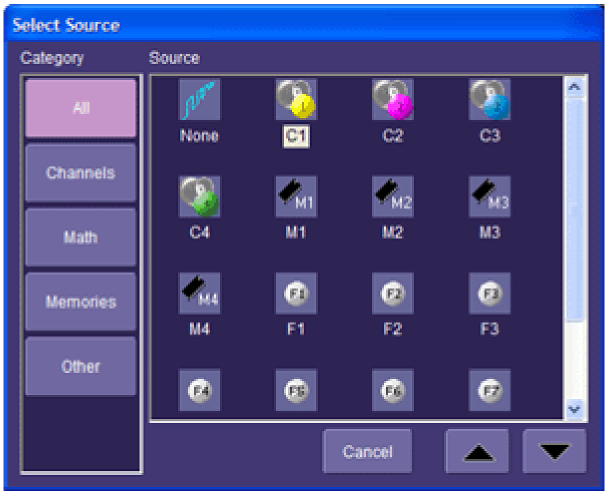
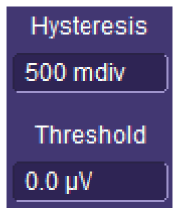
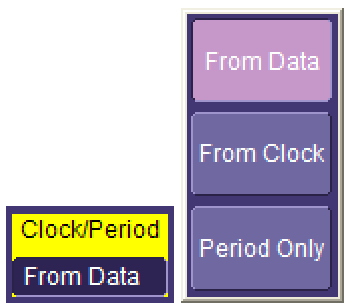
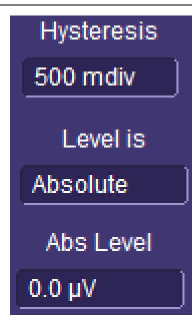
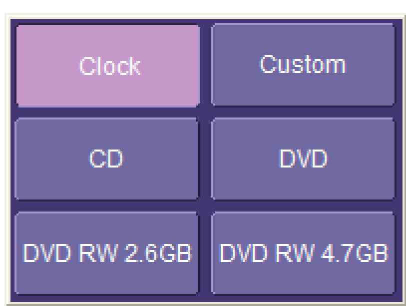
Check **Use Equalizer** to apply a filter to the Data Source.



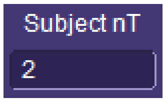
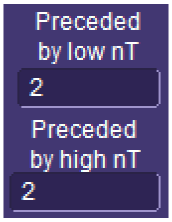
To set up the equalizer, touch the **Equalizer and PLL** tab.



Touch inside the **Data Slope** field and select an edge from the pop-up menu.

	Touch inside the **Data Gate** field and select a signal source. You can also select **None**. Data Gate specifies the input that will be used to determine where to perform measurements on the input signal. If a Data Gate is selected, the level is assumed to be high.
	The **Hysteresis** selection imposes a limit above and below the **Threshold**, which precludes measurements of noise or other perturbations within this band. The width of the band is specified in milli-divisions.
	Select **From Data** from the Clock/Period menu if you want to extract the clock from your input waveform. In this case, all other clock setup fields become unavailable except **Clock Slope**. Choose **Pos**, **Neg**, or **Near**. “Near” means the nearest clock edge to the data edge.
	To keep noise out of parametric measurements, set up a hysteresis band, in divisions, about a level. The level can be set to **Absolute** (volts) or **Percent**. In either case, touch inside the **Abs Level** or **Pct Level** field and enter a value, using the pop-up keypad.
	Select a clock period by having the period calculated from the clock source, by choosing a standard, or by manually setting a clock period. If you choose a clock period from a standard, you can also set a multiplier: 

SETTING nT

	For BES, EES, BEES, BESS, and EESS, this specifies the pit of interest. The results will be computed for each space/pit (pit/space) pair using the subject pit and all the spaces within the range specified.
	Specifies the range of n indices that define the pits/spaces used in the calculation. The range of n coupled with T are used to categorize the pits/spaces based on their widths.

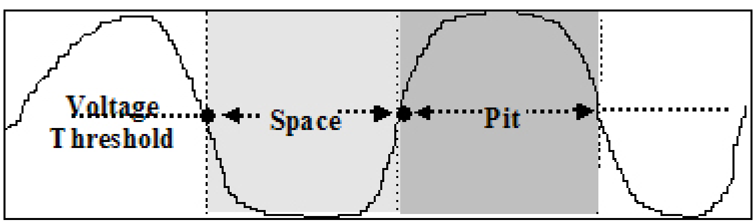
Maximizing Performance

A basic guideline that you should follow to maximize the performance of calculation in multiple parameter configurations is that precisely the same value for the clock period T, Threshold level, and Hysteresis value should be used.

Following this guideline ensures that parameters can make use of results obtained in previous parameter calculations. In most cases there is no need for different configurations of the above three items in different parameter setups.

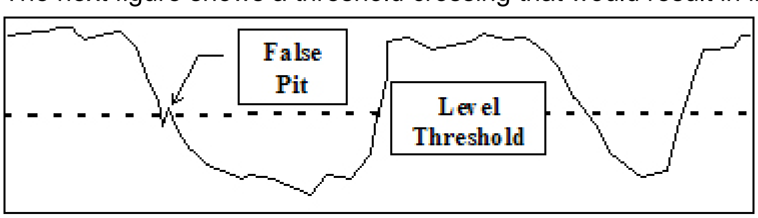
Pit or Space Identification

This is determined uniquely by the threshold, hysteresis, and edge polarity of threshold crossings. A positive threshold crossing indicates the start of a positive polarity pit and the end of a negative polarity space. A positive threshold crossing followed by a negative threshold crossing fully delineates a pit. A negative crossing followed by a positive crossing fully delineates a space, as illustrated in the following figure.

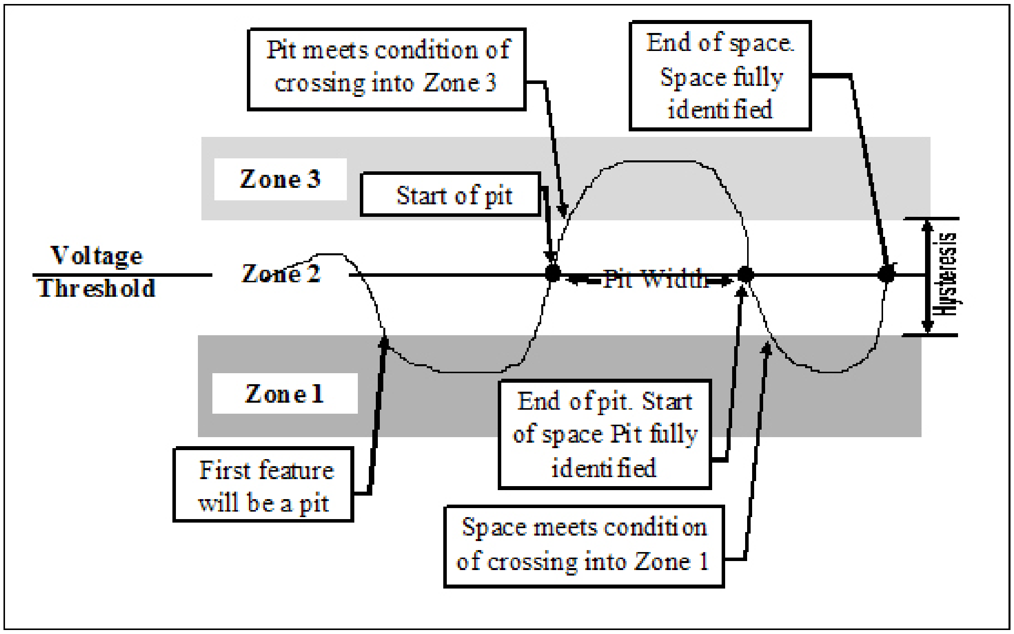


In order to prevent false pit and space identifications, hysteresis is provided. Hysteresis adds an additional condition that must be met before a threshold crossing is recognized as a pit/space edge. It requires that the waveform make an excursion of a certain distance from the threshold before the next threshold crossing is recognized.

The next figure shows a threshold crossing that would result in incorrect pit identification without hysteresis.



The hysteresis band shown in the next figure is centered on the user- selected voltage level threshold.



The hysteresis band divides the display into three zones. The ORM Package uses both the voltage threshold and hysteresis settings to identify pits and spaces.

Criteria for identifying a "feature" (pit or space):

- The first feature identified after the left parameter cursor can be either a pit or space. If the signal first enters Zone 1, the first feature identified (if additional constants are met) will be a pit. If the signal first enters Zone 3, it will be a space.
- After first crossing into Zone 1 or Zone 3, the next time the signal crosses the voltage threshold, it is recorded as the start time of a feature.
- If the first feature to be identified is a pit (signal entered Zone 1 first), after crossing the voltage threshold the signal must cross into Zone 3 and then pass the voltage threshold again to complete all conditions for identification as a pit. The first time that the signal crosses the voltage threshold after entering Zone 3 is recorded as the end time of the pit and the start time of the following space. The time between the start and end of the pit is recorded as the pit width. If the first feature to be identified is a space, the signal first entered Zone 3. The algorithm is used with directions reversed.
- For the entire signal, only a space can be identified after a pit, and only a pit can be identified after a space.
- All subsequent features are identified by crossing into the appropriate zone after the end of the previous feature. For a pit this is Zone 3, and for a space it is Zone 1. The end of the previous feature is the beginning of the current feature being identified. The subsequent first time the signal crosses the voltage threshold is recorded as the time of the feature being identified. At this point, the feature has been fully identified.

nT Pit-Space Categorization

Because optical recording data is encoded using a pulse-width modulation mechanism, it is often useful to perform signal analysis for selected pulse widths. Exploiting the fact that optical recording data widths are ideally integral multiples of the data clock period 'T', the AORM Package separates optical recording signal pits and spaces into groups whose widths fall into the same integral multiple of clock periods. As a result, ORMs can be configured to provide values for only pits or spaces, or both of these for a selected 'nT' value ('nT' denotes an integer multiple of the clock period) or for a range of 'nT's.

The ideal clock period (T) is configured on the parameter nT setup.

Categorization of pits and spaces by nT based on width is done using the following equation:

$$(n-0.5)\cdot T \leq w < (n+0.5)\cdot T$$

When this condition is met, the pit or space of width *w* is said to belong to the n^{th} index.

Beginning Edge Shift (BES)

Description

BES provides a measurement of the time between the beginning edge of the subject *n* in a specified space/pit pair and the nearest specified clock edge. The measurement is calculated between the points where the data and clock signals cross selected voltage thresholds. The clock period T can be entered by the user, or measured from a user supplied clock signal, as described below.

The value calculated depends on the clock and data edges selected, as shown in the table below. The data slope menu selects the polarity of the subject *n* pit/space. If Pos (positive) is selected, the measurement is performed from the beginning edges of positive polarity pits and categorized by the preceding space. If Neg (negative) is selected, the measurement is performed from the beginning edges of negative polarity spaces and categorized by the preceding pit. If Both is selected, the beginning edges of both pits and spaces are used in the calculation and categorized by the preceding inverse polarity space/pit. The sizes of pits or spaces used in the measurement are also determined by the range of 'nT' values chosen.

CLOCK EDGE	DATA SLOPE		
	Positive	Negative	Both
Positive	time between beginning edge of positive polarity subject pit and nearest positive clock edge	time between beginning edge of negative polarity subject space and nearest positive clock edge	time between beginning edge of subject pits and spaces to nearest positive clock edge
Negative	time between beginning edge of positive polarity subject pit and nearest negative clock edge	time between beginning edge of negative polarity subject space and nearest negative clock edge	time between beginning edge of subject pits and spaces to nearest negative clock edge
Near	time between beginning edge of positive polarity subject pit and nearest clock edge	time between beginning edge of negative polarity subject space and nearest clock edge	time between beginning edge of subject pits and spaces and nearest clock edge

The next figure shows the measurement of the beginning edge shift on a single subject 4T pit preceded by a 3T space. In this example, the clock is specified as the positive edge. For each space/pit combination, the beginning edge shift is calculated as the time difference between the beginning pit edge and the clock edge. Additionally, the measurements will be sorted by the space/pit pairs. For the positive polarity pit example shown in the figure after next, measurements t+ and t- are for a single beginning edge shift measurement configured for positive edge, or negative edge. If nearest is selected, the smaller of t- or t+ is used.

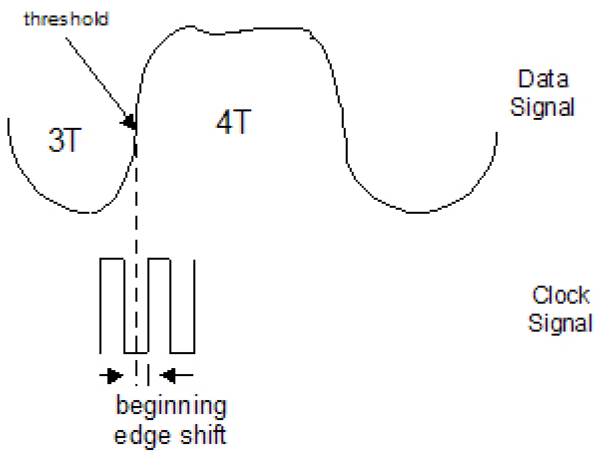


Figure 5-1. Beginning Edge Shift measurement of subject 4Tpit

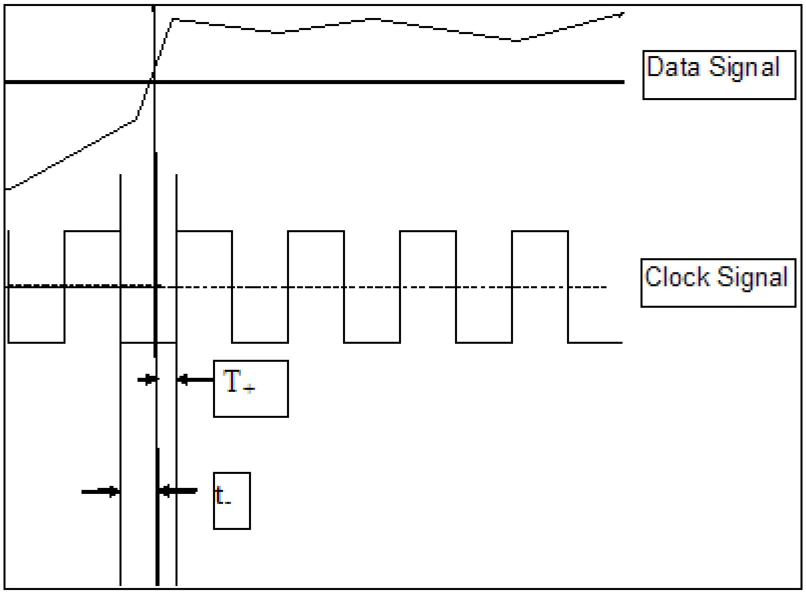


Figure 5-2. Zoom of Positive Polarity Pit Edge -- example measurement

$$bes = \Delta t_+ \cdot \frac{100\%}{T}$$

$$or \quad \Delta t_- \cdot \frac{100\%}{T}$$

Figure 5-4. The measurement has configurable units. If absolute time is specified, the value is simply the time indicated above. If percent is specified, the value of the measurement is the time normalized to the clock period:

For all pits, a valid measurement will be obtained only when both pit/space edges can be determined, (that is, there is a hysteresis-qualified threshold crossing beginning and ending the pit/space pair of interest between the parameter cursors), and there is a clock edge of both polarities surrounding the leading pit or space edge between the parameter cursors.

Display Options

ORM parameter calculations can be displayed, histogrammed, and trended in a variety of ways.

DISPLAY TYPE	VALUE DISPLAYED
Parameter Statistics Off	All values of the time between beginning edge of the subject *n* pit (space) and nearest clock edges for all subject pits (spaces) preceded by the spaces (pits) within the selected 'nT' range for the last acquisition.
Parameter Statistics On	Average, minimum, maximum, and sigma of the beginning edge shift calculated for all identified pit/space pairs within the selected 'nT' range for all acquisitions since the last CLEAR SWEEPS operation.
nT Table	List of values of the average beginning edge shift for each 'nT' space (pit) within the selected range preceding the subject pit (space) for the last acquisition.
Histogram Function	Histogram graph of the value of the beginning edge shift calculated for all pit/space pairs within the selected 'nT' range for all acquisitions since the last CLEAR SWEEPS operation.
Trend Function	Trend graph of the value of the beginning edge shift calculated for all pit/space pairs within the selected 'nT' range for all acquisitions since the last CLEAR SWEEPS operation.

Beginning Edge Shift Sigma (BESS)

Description

BESS provides a measurement of the mean, normalized standard deviatio*n* of the Beginning Edge Shift measurements (see BES). When a single *n* is specified, or when you are in 'nT Table' Show mode, the value calculated for the *n*th index is calculated using the following equation for standard deviation:

Beginning Edge Shift Sigma cannot be calculated for a given index *n* unless there are at least two Beginning Edge Shift values calculated or that *n* index.

$$BESS_n = \sigma(BES_n)$$

$$BESS_n = \sqrt{\frac{\sum BES^2{}_n - \frac{\left(\sum BES_n\right)^2}{N_n}}{N_n - 1}}$$

When Beginning Edge Shift is configured as a custom parameter with a range of *n*, the value calculated is the standard deviation of the distribution that results by normalizing each independent distribution categorized by the space (pit) nT preceding the subject pit (space). Distributions are normalized by subtracting the mean of the distribution from all of the elements in the distribution. This results in the following equation for overall Beginning Edge Shift Sigma resulting from the individually categorized Beginning Edge Shift Sigma values:

$$BESS_{overall} = \sqrt{\frac{\sum\left(BESS_n^2 \cdot (N_n - 1)\right)}{\sum N_n - 1}}$$

Note: The value calculated by BESS will generally not be the same as the sigma of the BES measurement displayed on the parameter line when a range of *n* is used and statistics is on. This is because the two measurements are not the same. The BESS measurement normalizes the results for each *n* by subtracting the mean BES from each BES in the nth distribution. This results in a superposition of mean-centered distributions, not a superposition of 0-centered distributions contributing to BES measurements. BESS will always be less than or equal to the standard deviation of BES measurements.

Display Options

ORM parameter calculations can be displayed, histogrammed and trended in a variety of ways.

DISPLAY TYPE	VALUE DISPLAYED
Parameter Statistics Off	Single value of the standard deviation of the mean normalized beginning edge shift values for pits/spaces of interest for last acquisition.
Parameter Statistics On	Average, minimum, maximum, and sigma of the beginning edge shift sigma value calculated per acquisition for all acquisitions since the last CLEAR SWEEPS operation.
nT Table	List of values of the standard deviation of the beginning edge shift values for each 'nT' spaces (pit) within the selected range preceding the subject pit (space) for the last acquisition.
Histogram Function	Histogram of beginning edge shift sigma values calculated for each acquisition for all acquisitions since the last CLEAR SWEEPS operation.
Trend Function	Trend of the beginning edge shift sigma values calculated for each acquisition for all acquisitions since the last CLEAR SWEEPS operation.

ENDING EDGE SHIFT (EES)

Description

EES provides a measurement of the time between the ending edge of the subject *n* in a specified space/pit pair and the nearest specified clock edge. The measurement is calculated between the points where the data and clock signals cross selected voltage thresholds. The clock period T can be entered by the user or measured from a user supplied clock signal, as described below.

The value calculated depends on the clock and data edges selected, as shown in the table below. The **Data Slope** menu selects the polarity of the **subject n** pit/space. If **Pos** (positive) is selected, the measurement is performed from the ending edges of positive polarity pits and categorized by the following space. If **Neg** (negative) is selected, the measurement is performed from the ending edges of negative polarity spaces and categorized by the following pit. If **Both** is selected, the ending edges of both pits and spaces are used in the calculation and categorized by the following inverse polarity space/pit. The sizes of pits or spaces used in the measurement are also determined by the range of 'nT' values chosen.

CLOCK EDGE	DATA SLOPE		
	Pos	**Neg**	**Both**
Positive	time between ending edge of positive polarity subject pit and nearest positive clock edge	time between ending edge of negative polarity subject space and nearest positive clock edge	time between ending edge of subject pits and spaces to nearest positive clock edge
Negative	time between ending edge of positive polarity subject pit and nearest negative clock edge	time between ending edge of negative polarity subject space and nearest negative clock edge	time between ending edge of subject pits and spaces to nearest negative clock edge
Near	time between ending edge of positive polarity subject pit and nearest clock edge	time between ending edge of negative polarity subject space and nearest clock edge	time between ending edge of subject pits and spaces and nearest clock edge

The next figure demonstrates the measurement of the ending edge shift on a single subject 4T pit followed by a 3T space. In this example, the clock is specified as the positive edge. For each pit/space combination, the ending edge shift is calculated as the time difference between the ending pit edge and the clock edge. Additionally, the measurements will be sorted by the pit/space pairs. For the positive polarity pit example shown in the figure after next, the measurements t+, and t- are for a single ending edge shift measurement configured for positive edge, or negative edge. If nearest is selected the smaller of t- or t+ is used.

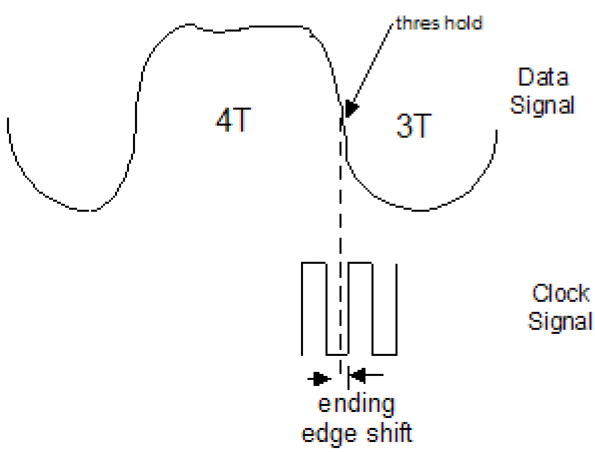


Figure 5-5. Ending Edge Shift measurement of subject 4T pit

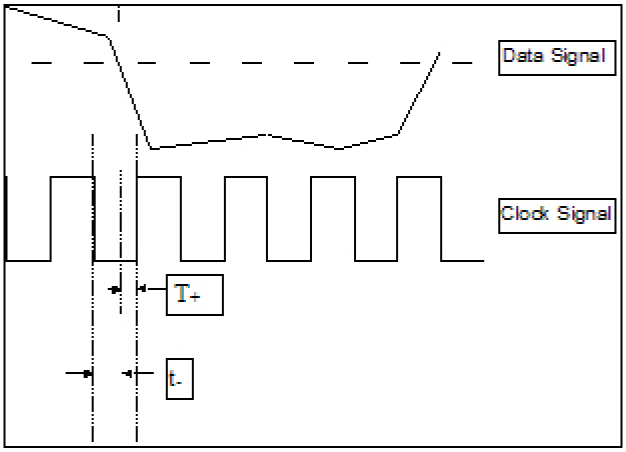


Figure 5-6. Zoom of Positive Polarity Pit Ending Edge -- example

The measurement has configurable units. If absolute time is specified, the value is simply the time as indicated above. If percent is specified, the value of the measurement is the time normalized to the clock period:

$$ees = \Delta t_{+} \cdot \frac{100\%}{T}$$

$$or \quad \Delta t_{-} \cdot \frac{100\%}{T}$$

For all pits, a valid measurement will be obtained only when both pit/space edges can be determined (that is, there is a hysteresis-qualified threshold crossing beginning and ending the pit/space pair of interest between the parameter cursors), and there is a clock edge of both polarities surrounding the ending pit or space edge between the parameter cursors.

Display Options

ORM parameter calculations can be displayed, histogrammed, and trended in a variety of ways.

DISPLAY TYPE	VALUE DISPLAYED
Parameter Statistics Off	All values of the average time between ending edge of the subject n pit (space) and nearest clock edges for all subject pits (spaces) followed by the spaces (pits) within the selected 'nT' range for the last acquisition.
Parameter Statistics On	Average, minimum, maximum, and sigma of the ending edge shift calculated for all identified pits/spaces pairs within the selected 'nT' range for all acquisitions since the last CLEAR SWEEPS operation.
nT Table	List of values of the average ending edge shift for each 'nT' space (pit) within the selected range following the subject pit (space) for the last acquisition.
Histogram Function	Histogram graph of the value of the ending edge shift calculated for all pit/space pairs within the selected 'nT' range for all acquisitions since the last CLEAR SWEEPS operation.
Trend Function	Trend graph of the value of the ending edge shift calculated for all pit/space pairs within the selected 'nT' range for all acquisitions since the last CLEAR SWEEPS operation.

Ending Edge Shift Sigma (EESS)

EESS provides a measurement of the mean, normalized standard deviation of the Ending Edge Shift measurements (see EES). When a single *n* is specified, or when you are in 'nT Table' Show mode, the value calculated for the nth index is calculated using the following equation for standard deviation:

$$EESS_n = \sigma\left(EES_n\right)$$

$$EESS_n = \sqrt{\frac{\sum EES^2{}_n - \frac{\left(\sum EES_n\right)^2}{N_n}}{N_n - 1}}$$

Ending Edge Shift Sigma cannot be calculated for a given index *n* unless there are at least two Ending Edge Shift values calculated for that n index.

When Ending Edge Shift is configured as a custom parameter with a range of *n*, the value calculated is the standard deviation of the distribution that results by normalizing each independent distribution categorized by the space (pit) nT following the subject pit (space). Distributions are normalized by subtracting the mean of the distribution from all of the elements in the distribution. This results in the following equation for overall Ending Edge Shift Sigma resulting from the individually categorized Ending Edge Shift Sigma values:

$$EESS_{overall} = \sqrt{\frac{\sum\left(EESS_n^2 \cdot \left(N_n - 1\right)\right)}{\sum N_n - 1}}$$

Note: The value calculated by EESS will generally not be the same as the sigma of EES measurement when a range of *n* is used and statistics are on. This is because the two measurements are not the same. The EESS measurement normalizes the results for each *n* by subtracting the mean EES from each EES in the nth distribution. This results in a superposition of mean-centered distributions, not a superposition of 0-centered distributions contributing to EES measurements. EESS will always be less than or equal to the standard deviation of EES measurements.

Display Options

ORM parameter calculations can be displayed, histogrammed and trended in a variety of ways.

DISPLAY TYPE	VALUE DISPLAYED
Parameter Statistics Off	Single value of the standard deviation of the mean normalized ending edge shift values for pits/spaces of interest for last acquisition.
Parameter Statistics On	average, minimum, maximum, and sigma of the ending edge shift sigma value calculated per acquisition for all acquisitions since the last CLEAR SWEEPS operation.
nT Table	List of values of the standard deviation of the ending edge shift values for each 'nT' spaces (pit) within the selected range following the subject pit (space) for the last acquisition.
Histogram Function	Histogram of ending edge shift sigma values calculated for each acquisition for all acquisitions since the last CLEAR SWEEPS operation.
Trend Function	Trend of the ending edge shift sigma values calculated for each acquisition for all acquisitions since the last CLEAR SWEEPS operation.

Beginning Ending Edge Shift (BEES)

BEES provides a measurement of both the beginning and ending edge shift for a subject *n* pit (space) preceded and followed by a specified space (pit). (See BES and EES.) The measurement is calculated between the points where the data and clock signals cross selected voltage thresholds. The clock period T can be entered by the user, or measured from a user supplied clock signal, as described below.

The value calculated depends on the clock and data edges selected, as shown in the table below. The **Data Slope** menu selects the polarity of the **subject n** pit/space. If **Pos** (positive) is selected, the measurement is performed from the beginning and ending edges of positive polarity pits and is preceded and followed by a space of the specified width. If **Neg** (negative) is selected, the measurement is performed from the edges of negative polarity spaces and is preceded and followed by a pit of the specified width.

CLOCK EDGE	DATA SLOPE	
	Positive	Negative
Positive	times between edges of positive polarity subject pit and nearest positive clock edge	times between edges of negative polarity subject space and nearest positive clock edge
Negative	times between edges of positive polarity subject pit and nearest negative clock edge	times between edges of negative polarity subject space and nearest negative clock edge
Near	times between edges of positive polarity subject pit and nearest clock edge	times between edges of negative polarity subject space and nearest clock edge

The next figure demonstrates the measurement of the beginning edge shift on a single subject 4T pit preceded and followed by a 3T space. In this example, the clock is specified as the positive edge. The beginning edge shift is calculated as the time difference between the beginning pit edge and the clock edge while the ending edge shift is calculated as the time difference between the ending pit edge and the clock edge.

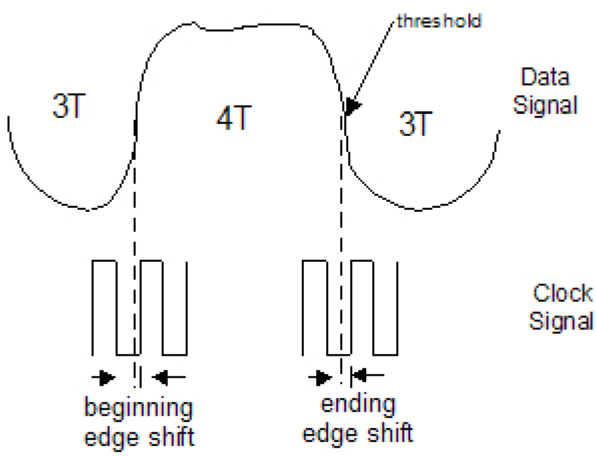


Figure 5-7. Beginning and Ending Edge Shift measurement of subject 4T pit

The measurement has configurable units. If absolute time is specified, the value is simply the time, as indicated above. If percent is specified, the value of the measurement is the time normalized to the clock period:

$$bees = \Delta t_{+} \cdot \frac{100\%}{T}$$

$$or \quad \Delta t_{-} \cdot \frac{100\%}{T}$$

For all pits, a valid measurement will be obtained only when both edges of the leading and trailing pits/spaces can be determined (that is, there is a hysteresis-qualified threshold crossing beginning the start pit/space and ending the end pit/space of interest between the parameter cursors), and there is a clock edge of both polarities surrounding the leading pit or space edge between the parameter cursors.

Display Options

ORM parameter calculations can be displayed, histogrammed, and trended in a variety of ways.

DISPLAY TYPE	VALUE DISPLAYED
Parameter Statistics Off	Single value of the average time between the edges of the subject n pit (space) and nearest clock edges for all subject pits (spaces) that are preceded and followed by the specified space (pits) for the last acquisition.
Parameter Statistics On	Average, minimum, maximum, and sigma of the beginning and ending edge shift calculated for all subject pits (spaces) that are preceded and followed by the specified space (pits) for all acquisitions since the last CLEAR SWEEPS operation.
nT Table	List of values of the beginning edge shift and the ending edge shift for all subject pits

DISPLAY TYPE	VALUE DISPLAYED
	(spaces) that are preceded and followed by the specified space (pits) for the last acquisition.
Histogram Function	Histogram graph of the values of the beginning and ending edge shift calculated for all subject pits (spaces) that are preceded and followed by the specified space (pits) for all acquisitions since the last CLEAR SWEEPS operation.
Trend Function	Trend graph of the value of the beginning and ending edge shift calculated for all subject pits (spaces) that are preceded and followed by the specified space (pits) for all acquisitions since the last CLEAR SWEEPS operation.
XY Plot	XY Plot displays the trend of one parameter vs. another.

Dp2c - DELTA PIT TO CLOCK

Description

Dp2c provides a measurement of the time between the leading edge of the pit (or spaces of interest) and the nearest specified clock edge. The measurement is calculated between the points where the data and clock signals cross selected voltage thresholds.

The value calculated depends on the clock and data edges selected, as shown in the table below. If in the **Data Slope** menu **Pos** (positive) is selected, the measurement is performed from the leading edges of positive polarity pits. If **Neg** (negative) is selected, the measurement is performed from the leading edges of negative polarity spaces. And if **Both** is selected, the leading edges of both pits and spaces are used in the calculation. The sizes of pits or spaces used in the measurement are also determined by the range of 'nT' values chosen.

CLOCK EDGE	DATA SLOPE		
	Pos	**Neg**	**Both**
positive	time between leading edge of positive polarity pit and nearest positive clock edge	time between leading edge of negative polarity space and nearest positive clock edge	time between leading edge of pits and spaces to nearest positive clock edge
negative	time between leading edge of positive polarity pit and nearest negative clock edge	time between leading edge of negative polarity space and nearest negative clock edge	time between leading edge of pits and spaces to nearest negative clock edge
near	time between leading edge of positive polarity pit and nearest clock edge	time between leading edge of negative polarity space and nearest clock edge	time between leading edge of pits and spaces and nearest clock edge

For the positive polarity pit example shown as the zoom of the measurement (next two figures), the measurements t□, t-, tn are for a single Delta Pit-to-Clock measurement configured for positive edge, negative edge, or nearest edge, respectively.

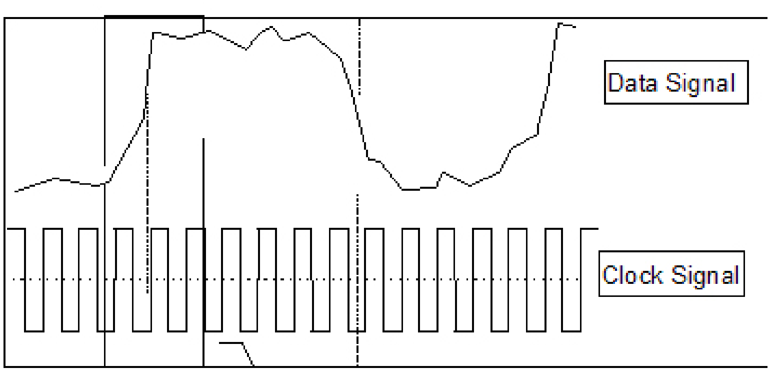


Figure 5-8. Delta Pit-to-Clock measurement

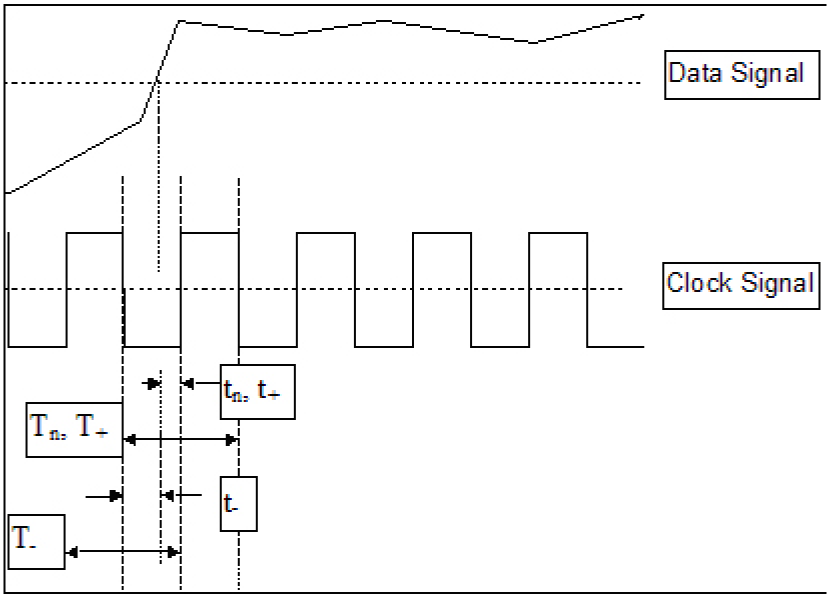


Figure 5-9. Zoom of Positive Polarity Pit Edge - example measurement

The measurement has configurable units. If absolute time is specified, the value is simply the time as indicated above. If percent is specified, the value of the measurement is the time normalized to the local clock period. The local clock period is calculated as the time between the two clock edges bracketing the clock edge used for the delta time measurement:

$$\Delta p2c = \Delta t_+ \cdot \frac{100\%}{T_+}$$

$$or \quad \Delta t_- \cdot \frac{100\%}{T_-}$$

$$or \quad \Delta t_n \cdot \frac{100\%}{T_n}$$

For all pits, a valid measurement will be obtained only when both pit/space edges can be determined (that is, there is a hysteresis qualified threshold crossing that begins and ends the pit/space of interest between the parameter cursors), and when there is a clock edge of both polarities surrounding the leading pit or space edge between the parameter cursors.

Display Options

ORM parameter calculations can be displayed, histogrammed and trended in a variety of ways.

DISPLAY TYPE	VALUE DISPLAYED
Parameter Statistics Off	All values of the average time between leading pit/space edges and nearest clock edges for all pits/spaces within the selected ‘nT’ range for the last acquisition.
Parameter Statistics On	Average, minimum, maximum, and sigma of the Delta Pit-to-Clock calculated for all identified pits/spaces within the selected ‘nT’ range for all acquisitions since the last CLEAR SWEEPS operation.
nT Table	List of values of the average Delta Pit-to-Clock for each group of pits/spaces of common ‘nT’ width for the last acquisition.
Histogram Function	Histogram graph of the value of the Delta Pit-to-Clock calculated for all pits/spaces within the selected ‘nT’ range for all acquisitions since the last CLEAR SWEEPS operation.
Trend Function	Trend graph of the value of the Delta Pit-to-Clock calculated for all pit/space within the selected ‘nT’ range for all acquisitions since the last CLEAR SWEEPS operation.

Dp2cs - DELTA PIT TO CLOCK SIGMA

Description

Dp2cs provides a measurement of the mean, normalized standard deviation of the Delta Pit-to-Clock measurements (see Dp2c). When a single *n* is specified, or in 'nT Table' Show mode, the value calculated for the nth index is calculated using the following equation for standard deviation:

$$\Delta P2CS_n = \sigma(\Delta P2C_n)$$

$$\Delta P2CS_n = \sqrt{\frac{\sum \Delta P2C^2{}_n - \frac{\left(\sum \Delta P2C_n\right)^2}{N_n}}{N_n - 1}}$$

Delta Pit-to-Clock Sigma cannot be calculated for a given index *n* unless there are at least two Delta Pit-to-Clock values calculated for that *n* index.

When Delta Pit-to-Clock is configured as a custom parameter with a range of *n*, the value calculated is the standard deviation of the distribution that results by normalizing each independent distribution categorized by nT. Distributions are normalized by subtracting the mean of the distribution from all of the elements in the distribution. This results in the following equation for overall Delta Pit-to-Clock Sigma resulting from the individually categorized Delta Pit-to-Clock Sigma values:

$$\Delta P2CS_{overall} = \sqrt{\frac{\sum\left(\Delta P2CS_n^{\ 2} \cdot (N_n - 1)\right)}{\sum N_n - 1}}$$

Note: The value calculated by DP2CS will generally not be the same as the sigma of DP2C measurement displayed on the parameter line when a range of *n* is used and statistics is on. This is because the two measurements are not the same. DP2CS measurement normalizes the results for each *n* by subtracting the mean DP2C from each DP2C in the nth distribution. This results in a superposition of mean centered distributions, not a superposition of 0 centered distributions contributing to DP2C measurements. DP2CS will always be less than or equal to the standard deviation of DP2C measurements.

Display Options

ORM parameter calculations can be displayed, histogrammed, and trended in a variety of ways.

DISPLAY TYPE	VALUE DISPLAYED
Parameter Statistics Off	Single value of the standard deviation of the mean normalized Delta Pit-to-Clock values for pits/spaces of interest for last acquisition.
Parameter Statistics On	Average, minimum, maximum and sigma of the Delta Pit-to-Clock sigma value calculated per acquisition for all acquisitions since the last CLEAR SWEEPS operation.
nT Table	List of values of the standard deviation of the Delta Pit-to-Clock values for each individual 'nT' in the selected range of 'nT' for the last acquisition.
Histogram Function	Histogram of Delta Pit-to-Clock sigma values calculated for each acquisition for all acquisitions since the last CLEAR SWEEPS operation.
Trend Function	Trend of the Delta Pit-to-Clock sigma values calculated for each acquisition for all acquisitions since the last CLEAR SWEEPS operation.

EDGSH - EDGE SHIFT

Description

Edge Shift provides a measurement of the difference between the width of pits, spaces, or both, and their ideal widths. These ideal widths are integer multiples of the clock period 'T'. The width of the pit or space is determined by the time between crossings of the selected voltage threshold (see pwid).

When a single *n* is specified for the Edge Shift custom parameter, for each pit-width value calculated, the Edge Shift is calculated as:

$edgsh_i = (w_i - n_i \cdot T)$ when absolute time units are specified

or

$edgsh_i = (w_i - n_i \cdot T) \cdot \frac{100.0\%}{T}$ when percent is specified,

where n_i is the n that makes the width closest to nT (i.e., *n* is the n category to which the width belongs). Thus:

$$(n_i - 0.5) \cdot T \leq w_i < (n_i + 0.5) \cdot T$$

where T is the configured period. It is very important for this parameter calculation that you enter exactly the ideal T.

For 'nT Table' Show mode, or custom mode with one *n* specified, the value displayed for the nth index is the average of all of the edge shift values calculated that belong to that index:

$$edgsh_n = \left(\frac{\sum w_i}{N_n} - n \cdot T \right) \cdot \frac{100.0\%}{T}$$

Where N_n is the number of pits belonging to the nth index. When edge shift is configured as a custom parameter with a range of *n*, the overall edge shift is calculated and displayed as the weighted average of the edge shift values calculated above:

$$edgsh_{overall} = \frac{\sum (edsh_n \cdot N_n)}{\sum N_n}$$

The measurement calculation is compliant with the definition of Edge Shift as defined by ISO/IEC JTC1.23.14517 Section 22.4.

Display Options

ORM parameter calculations can be displayed, histogrammed, and trended in a variety of ways.

DISPLAY TYPE	VALUE DISPLAYED
Parameter Statistics Off	All values of the overall edge shift for all pits/spaces within the selected 'nT' range for last acquisition.
Parameter Statistics On	Average, minimum, maximum, and sigma of all edge shift values calculated per acquisition for all acquisitions since the last CLEAR SWEEPS operation.
nT Table	List of values of the overall edge shift for each group of pits/spaces of common 'nT' width for the last acquisition.
Histogram Function	Histogram of the single overall edge shift value calculated for each acquisition for all acquisitions since the last CLEAR SWEEPS operation.
Trend Function	Trend of the single overall Edge Shift value calculated for each acquisition for all acquisitions since the last CLEAR SWEEPS operation.

Example

The example shows the CD data signal measured at the selected voltage threshold containing, in sequence, a 5T pit, 3T space, 3T pit and 4T space. If the clock period 'T' is 231.5 ns, then the 5T and 4T edge shift value is simply the difference between the width calculated and the ideal width (since there is only one pit/space of that 'nT' width), thus:

$$edgsh(4T) = (920 - 4 \cdot 231.5) \cdot \frac{100\%}{231.5} = -2.59\%$$

$$edgsh(5T) = (1160 - 5 \cdot 231.5) \cdot \frac{100\%}{231.5} = +1.08\%$$

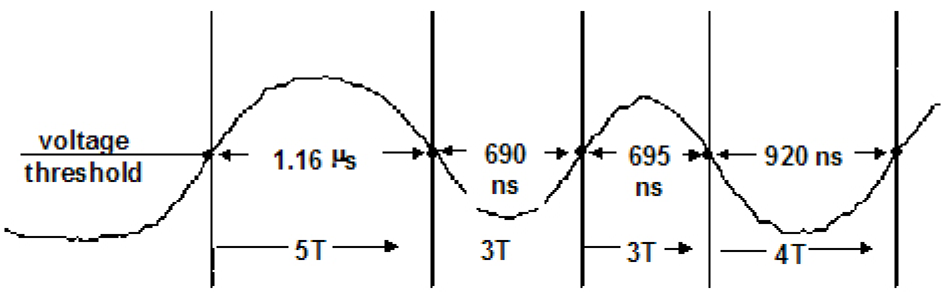
The 3T edge shift value is the average difference:

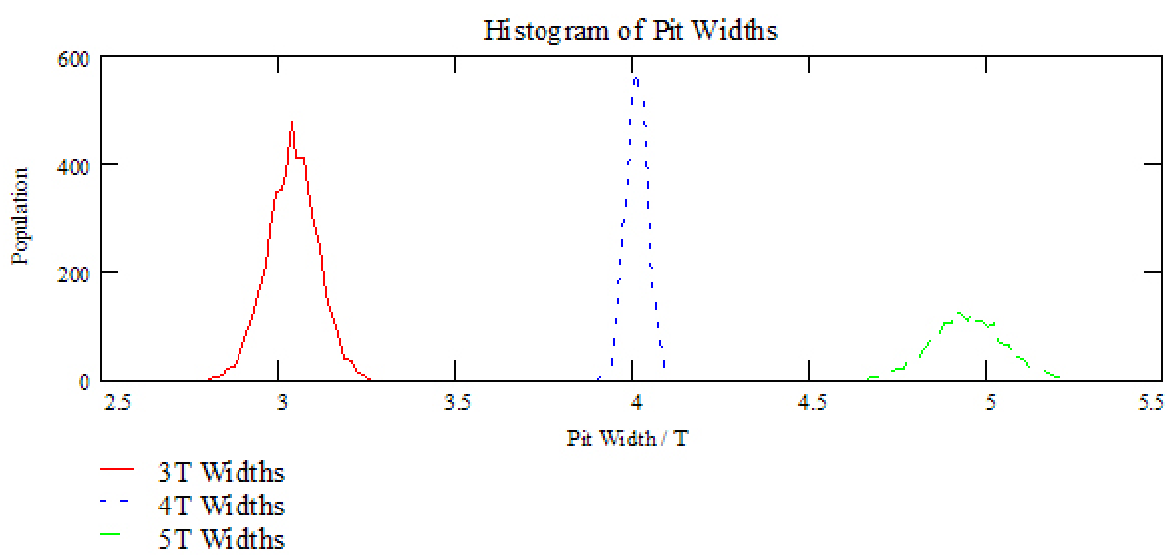
$$edgsh(3T) = \frac{\left((690 - 3 \cdot 231.5) \cdot \frac{100\%}{231.5}\right) + \left((695 - 3 \cdot 231.5) \cdot \frac{100\%}{231.5}\right)}{2} = +0.86\%$$

In an nT Table display, these three values would be shown in the appropriate nT location.

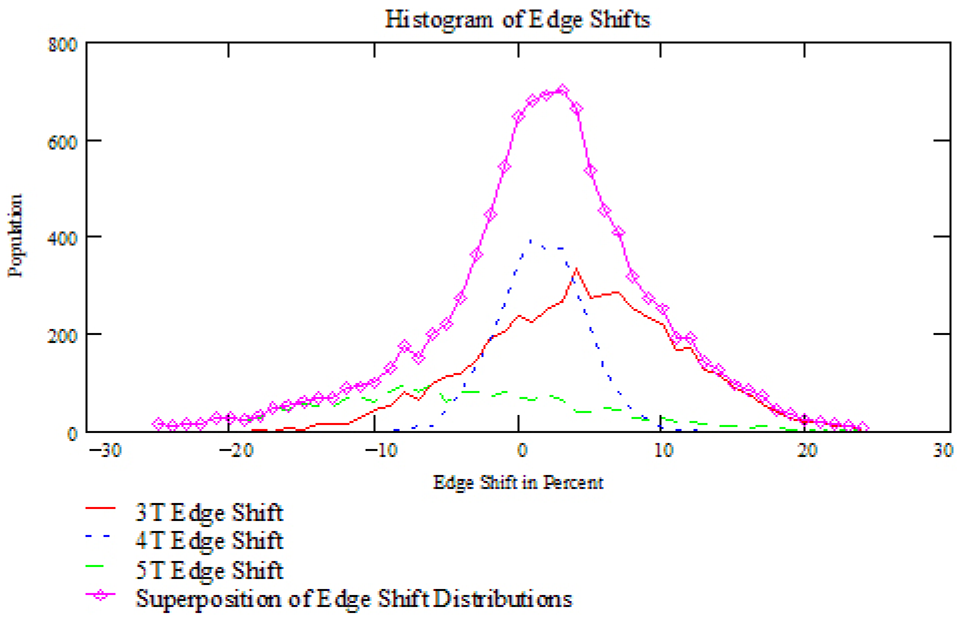
More On Edge Shift

A good approach to understanding the operation of the edge shift parameter with different modes of operation starts by considering the next figure, a histogram of 3T to 5T pit widths.





The Edge Shift parameter takes on each of these distributions separately. For each distribution, the ideal width (nT) is subtracted from the pit widths and the difference is calculated in percent. As a result, the Edge Shift distributions are calculated, as shown in the next figure.



The 3T, 4T, and 5T distributions are obtained when the Edge Shift custom parameter is configured for single *n* values and histogrammed. The final superposition distribution is obtained when the Edge Shift custom parameter is configured for ranges of *n* values (in this case 3T to 5T) and histogrammed.

The value displayed on the custom parameter line (with statistics off) is the mean of any of the resulting distributions for the last acquisition only. This average edge shift value is calculated internally without actually histogramming the values. The values displayed in 'nT Table' mode are the mean of the Edge Shift distributions resulting from each nT distribution for the last acquisition.

Note: The standard deviation of superimposed Edge Shift distributions is not the same as Timing Jitter.

LPER - Local Period

Description

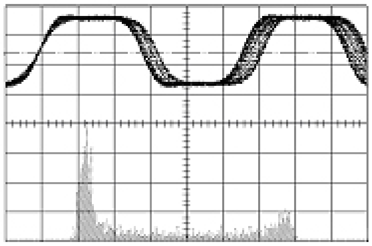
Local Period provides a measurement of the clock period of each clock cycle (up to the maximum number of cycles governed by memory limitations). Histogramming and statistics can be used to provide a clock jitter measurement. The starting edge (the edge that begins each cycle) is configurable.

Display Options

ORM parameter calculations can be displayed, histogrammed, and trended in a variety of ways.

Display Type	Value Displayed
Parameter Statistics Off	Single value of average period for all clock cycles for last acquisition.
Parameter Statistics On	Average, minimum, maximum, and sigma of the clock period for all clock cycles for all acquisitions since last CLEAR SWEEPS operation.
Histogram Function	Histogram graph of the value of the period for all clock cycles for all acquisitions since the last CLEAR SWEEPS operation.
Trend Function	Trend graph of the value of the period for all clock cycles for all acquisitions since the last CLEAR SWEEPS operation.

Example

	Histogramming the local period jittery clock signal shows that there are two frequency modes, one at a period of about 292 ns and the other at around 308 ns.

PAA - PIT AVERAGE AMPLITUDE

Description

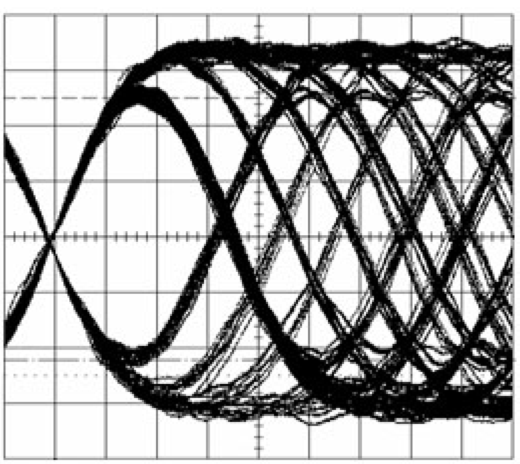
Pit Average Amplitude provides a measurement of the average amplitude of pits and spaces. The calculation is performed by calculating the difference between the average value of the base (pbase) for spaces of a particular ‘nT’ width and the average value of the top (ptop) of pits of the same ‘nT’ width. For example, the average value of the base for all 3T spaces is subtracted from the average value of the top for all 3T pits to obtain the 3T pit average amplitude. If a range of ‘nT’ values is selected and is displayed as a parameter, the measurement provides the weighted average amplitude based on the number of occurrences of each ‘nT’ pit/space width.

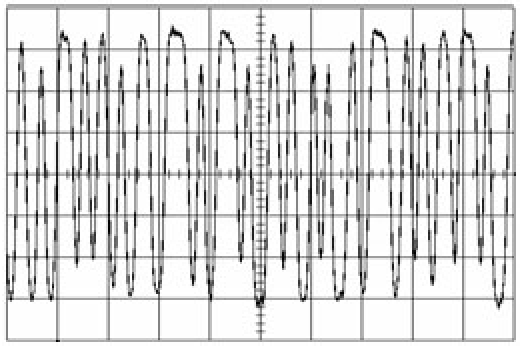
Display Options

ORM parameter calculations can be displayed, histogrammed, and trended in a variety of ways.

DISPLAY TYPE	VALUE DISPLAYED
Parameter Statistics Off	Single value of the average amplitude for all pits/spaces of interest for last acquisition.
Parameter Statistics On	Overall average, minimum, maximum, and sigma of the single average amplitude value calculated per acquisition for all acquisitions since the last CLEAR SWEEPS operation.
nT Table	List of the average amplitude values for each group of pits/spaces of common ‘nT’ width for the last acquisition.
Histogram Function	Histogram of the single average amplitude value calculated for each acquisition for all acquisitions since the last CLEAR SWEEPS operation.
Trend Function	Trend of the single average amplitude value calculated for each acquisition for all acquisitions since the last CLEAR SWEEPS operation.

Example

	Consider this persistence plot of an optical data waveform. Using cursors, the average amplitude of the 3T pits/spaces can be estimated. In this case, the value obtained is 47.2 mV.

	When the parameter paa is configured for 3T widths, the measurement result is also 47.2 mV. This value is calculated automatically.

PASYM - PIT ASYMMETRY

Description

Pit Asymmetry provides a measurement of the asymmetry of the middle voltage level for the high nT index pits/spaces compared to the middle voltage level of the low 'nT' index pits/spaces. The measurement calculation is compliant with the definition of Pit Asymmetry as defined by IEC 908:1987 Section 3.1. The negative value of the measurement is referred to as Pit Symmetry as defined by ISO/IEC 10149:1995 (E) Section 12.2. Pit Asymmetry is calculated by the formula:

$$PASYM = \frac{pmidl_{high\_n} - pmidl_{low\_n}}{paa_{high\_n}} \cdot 100\%$$

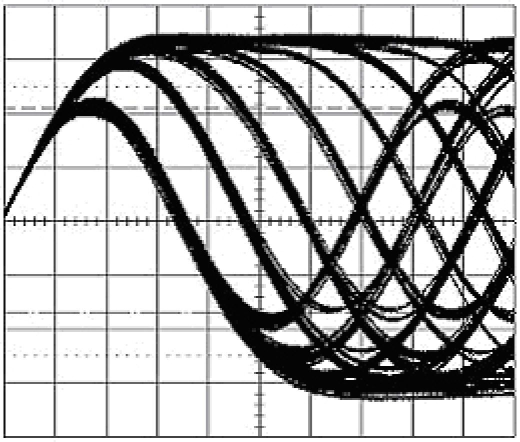
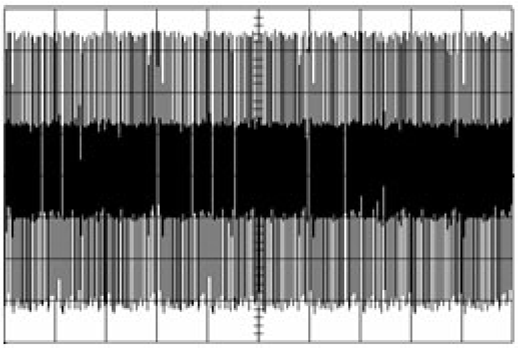
where paa is the average peak-peak amplitude. The low (smallest) and high (largest) 'nT' values to use in performing the calculation are provided by the user through the associated measurement configuration options. Midpoint designates the midpoint value between the average top and base for a specified 'nT.' The value shown is in units of percent.

Display Options

ORM parameter calculations can be displayed, histogrammed, and trended in a variety of ways.

DISPLAY TYPE	VALUE DISPLAYED
Parameter Statistics Off	Single value of the asymmetry for the last acquisition.
Parameter Statistics On	Average, minimum, maximum and sigma of the single asymmetry value calculated per acquisition for all acquisitions since the last CLEAR SWEEPS operation.
Histogram Function	Histogram of asymmetry value calculated per acquisition for all acquisitions since last CLEAR SWEEPS operation.
Trend Function	Trend of single asymmetry value calculated per acquisition for all acquisitions since last CLEAR SWEEPS operation.

Example

	This persistence plot of a bandwidth limited, smooth waveform illustrates asymmetry. Notice that the mid-level of the 3T waveform is offset from 0 V, and that the mid-level of the 11T waveform is approximately 0 V.
	Since the 3T middle level is offset, the expected asymmetry value is negative. This is the asymmetry calculated from a waveform with several thousand widths. The values are the asymmetry, the 3T middle level, the 11T middle level, and the 11T average amplitude.

PBASE - PIT BASE

Pit Base provides a best estimate of the bottom amplitude of a space. The concept of the base calculation is to automatically provide the same measurement that would be obtained from a persistence plot. The base of each space is determined through histogramming techniques described under Base and Top Calculation Details.

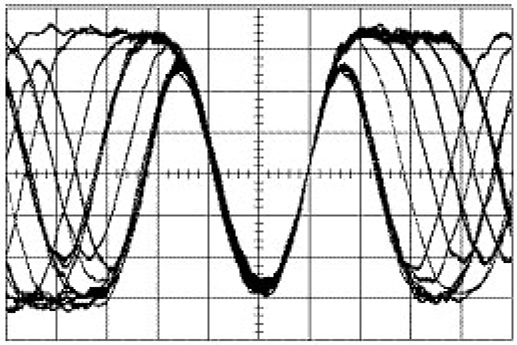
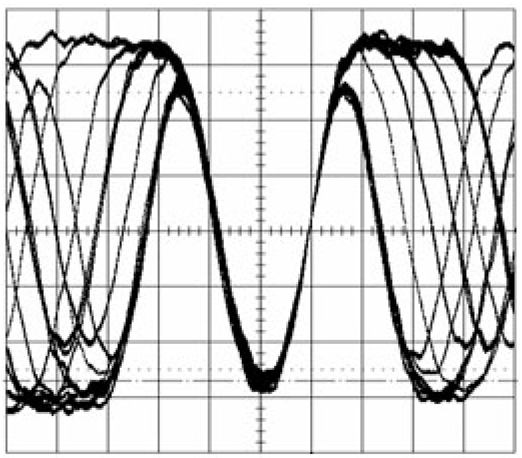
When pbase is configured as a custom parameter, all bases within the single nT or range of nT are calculated. Histogramming or trending such a configuration would result in one value per space in the nT range contributing a value to the histogram or trend. The value displayed on the custom parameter display line is the average of all such base calculations. 'nT Table' mode provides an average base measurement for each *n* index.

Display Options

ORM parameter calculations can be displayed, histogrammed, and trended in a variety of ways. The following table provides a concise description of the value or values displayed using each approach.

DISPLAY TYPE	VALUE DISPLAYED
Parameter Statistics Off	All values of the base for all spaces within the selected 'nT' range for the last acquisition.
Parameter Statistics On	Average, minimum, maximum, and sigma of the base for all spaces that are within the selected 'nT' range for all acquisitions since the last CLEAR SWEEPS operation.
nT Table	List of values of the average base for each group of spaces of common 'nT' width for the last acquisition.
Histogram Function	Histogram graph of the value of the base for all spaces within the selected 'nT' range for all acquisitions since the last CLEAR SWEEPS operation.
Trend Function	Trend graph of the value of the base calculated for space that is within the selected 'nT' range for all acquisitions since the last CLEAR SWEEPS operation.

Example

	This persistence waveform is created by setting a SMART Trigger® to capture only 4T spaces. The 4T base computed is -27.0 mV.
	When the same measurement is taken with the parameter cursors, it confirms that -27.0 mV is a reasonable value for the base.

PMAX - PIT MAXIMUM

Description

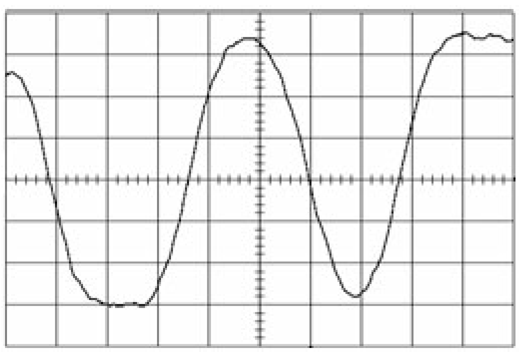
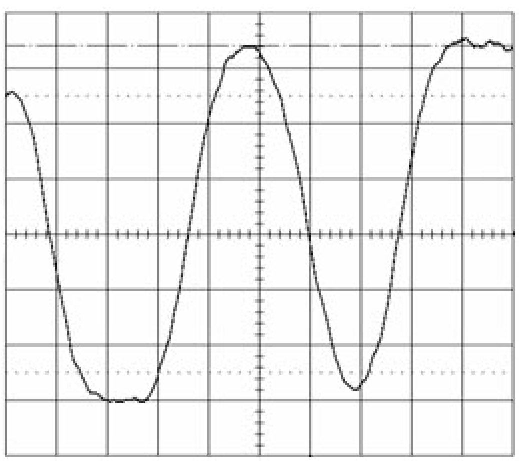
Pit Maximum provides a measurement of the maximum voltage value of pits of interest. It provides a comparison of how the maximum point in the waveform corresponds to the ptop value When pmax is configured as a custom parameter, all maximums within the single nT or range of nT are calculated. Histogramming or trending such a configuration would result in one value per pit in the nT range contributing a value to the histogram or trend. The value displayed on the custom parameter display line is the average of all such maximum calculations. ‘nT Table’ mode provides an average maximum value for the pits in each *n* index.

Display Options

ORM parameter calculations can be displayed, histogrammed and trended in a variety of ways. The table provides a concise description of the value or values displayed using each approach.

DISPLAY TYPE	VALUE DISPLAYED
Parameter Statistics Off	All values of the maximum for all pits within the selected 'nT' range for the last acquisition.
Parameter Statistics On	Average, minimum, maximum, and sigma of the maximum for all pits that are within the selected 'nT' range for all acquisitions since the last CLEAR SWEEPS operation.
nT Table	List of values of the average maximum for each group of pits of common 'nT' width for the last acquisition.
Histogram Function	Histogram graph of the value of the maximum for all pits within the selected 'nT' range for all acquisitions since the last CLEAR SWEEPS operation.
Trend Function	Trend graph of the value of the maximum calculated for each pit that is within the selected 'nT' range for all acquisitions since the last CLEAR SWEEPS operation.

Example

	This waveform contains a single pit, whose max is computed as 26.1 mV.
	The same measurement is verified using the measurement cursors.

PMIDL - PIT MIDDLE LEVEL

Description

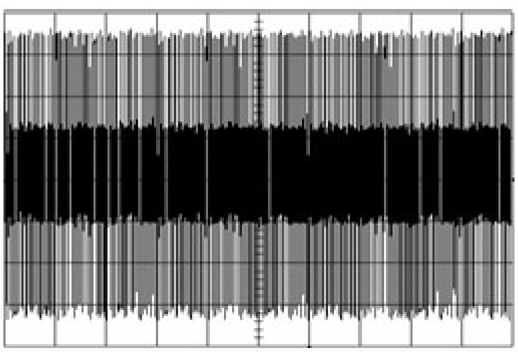
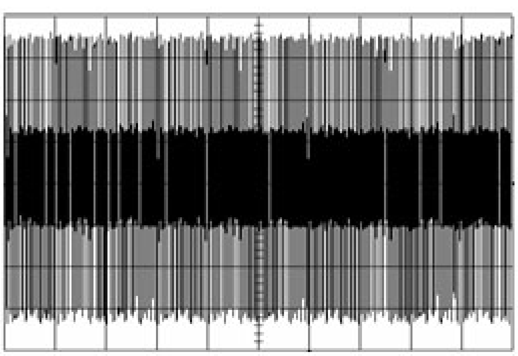
Pit Middle Level provides a measurement of the middle voltage level of pits or spaces. It is performed by first calculating the midpoint of the average value of the base (pbase) for spaces and the average value of the top of pits (ptop). If only 3T pits are specified, the resulting measurement is the 'decision level' (see ISO/IEC 10149:1995 (E) Section 12.1). If a range of 'nT' values is selected and is displayed as a parameter, the measurement provides the weighted average midpoint based on the number of occurrences of each 'nT' pit/space width. The measurement value can be used to determine not only the differences of the midpoint of different 'nT' width pits, but also the overall best data waveform voltage threshold setting to use for all ORMs.

Display Options

ORM parameter calculations can be displayed, histogrammed, and trended in a variety of ways.

DISPLAY TYPE	VALUE DISPLAYED
Parameter Statistics Off	Single value of the middle level for all pits/spaces of interest for last acquisition.
Parameter Statistics On	Overall average, minimum, maximum, and sigma of the single middle level value calculated per acquisition, for all acquisitions since the last CLEAR SWEEPS operation.
nT Table	List of the middle level values for each group of pits/spaces of common 'nT' width for the last acquisition.
Histogram Function	Histogram of the single middle level value calculated for each acquisition for all acquisitions since the last CLEAR SWEEPS operation.
Trend Function	Trend of the single middle level value calculated for each acquisition for all acquisitions since the last CLEAR SWEEPS operation.

Example

	This waveform contains thousands of pits. In 'nT Table' mode, the middle levels are displayed for each nT index. These values are the midlevels of the tops and bases for pits/spaces within the nT indices.
	The overall middle level is calculated based on a weighted average of the middle level for each nT. This value is the overall best threshold value for all pits/spaces within the 3T to 11T range.

PMIN - PIT MINIMUM

Description

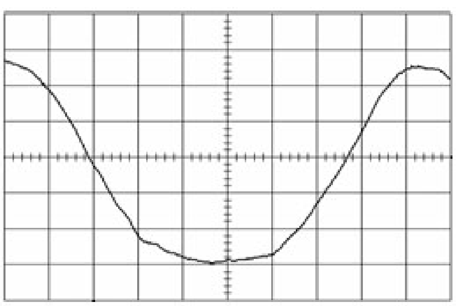
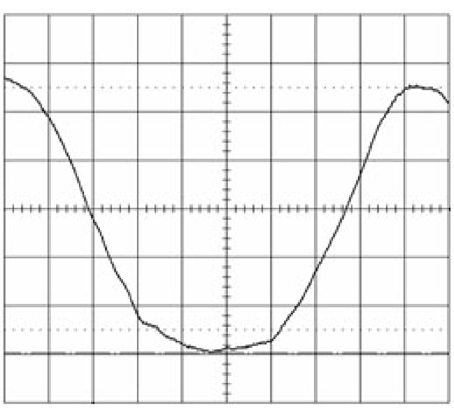
Pit Minimum provides a measurement of the minimum voltage value of pits of interest, and a comparison of how the minimum point in the waveform corresponds to the ptop value. When pmin is configured as a custom parameter, all minimums within the single nT or range of nT are calculated. Histogramming or trending such a configuration would result in one value per pit in the nT range contributing a value to the histogram or trend. The value displayed on the custom parameter display line is the average of all such minimum calculations. 'nT Table' mode provides an average minimum value for the pits in each n index.

Display Options

ORM parameter calculations can be displayed, histogrammed, and trended in a variety of ways.

DISPLAY TYPE	VALUE DISPLAYED
Parameter Statistics Off	All values of the minimum for all pits within the selected 'nT' range for the last acquisition.
Parameter Statistics On	Average, minimum, maximum, and sigma of the minimum for all pits that are within the selected 'nT' range for all acquisitions since the last CLEAR SWEEPS operation.
nT Table	List of values of the average minimum for each group of pits of common 'nT' width for the last acquisition.
Histogram Function	Histogram graph of the value of the minimum for all pits within the selected 'nT' range for all acquisitions since the last CLEAR SWEEPS operation.
Trend Function	Trend graph of the value of the minimum calculated for each pit that is within the selected 'nT' range for all acquisitions since the last CLEAR SWEEPS operation.

Example

	This waveform contains a single space. The pit min is computed as -9.7 mV.
	The measurement can be verified with the measurement cursors.

PMODA - PIT MODULATION AMPLITUDE

Description

Pit Modulation Amplitude provides a measurement of the ratio of the Pit Average Amplitude (paa) for the low 'nT' pits/spaces in the data signal to the Pit Top (ptop) of the high 'nT' pits in the data signal:

$$PMODA = \frac{paa_{low\_n}}{avg(top)_{high\_n}}$$

The low and high 'nT' values to be used for performing the calculation are provided by the user through the associated measurement configuration options. Some measurements of modulation amplitude require the low and high *n* index to be identical. The value shown is decimal. The measurement calculation is compliant with the definition of Modulation Amplitude as defined by IEC 908:1987 Section 9.2 and ISO/IEC 10149:1995 (E) Section 12.2.

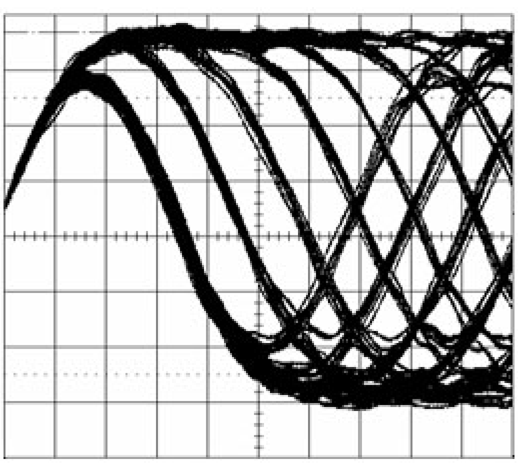
Note: This measurement must be performed on the DC-coupled optical data waveform, otherwise incorrect values will result.

Display Options

ORM parameter calculations can be displayed, histogrammed, and trended in a variety of ways.

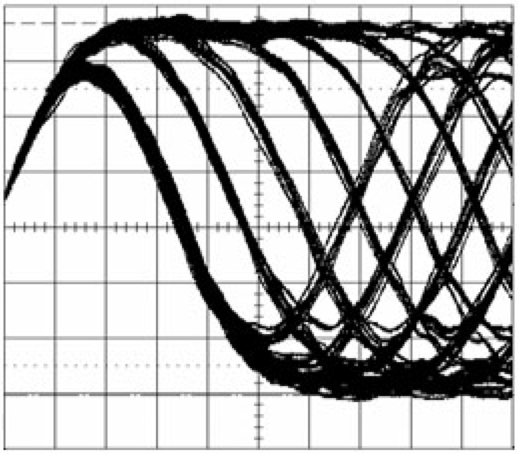
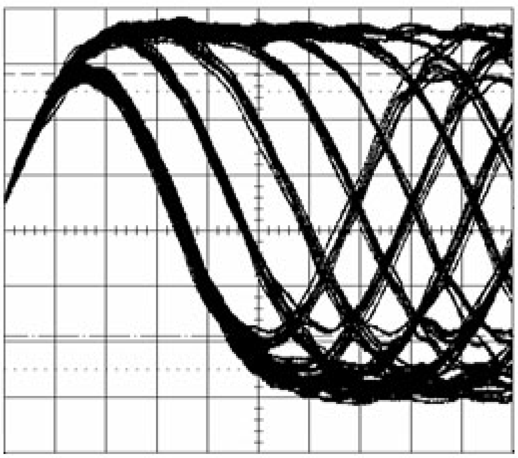
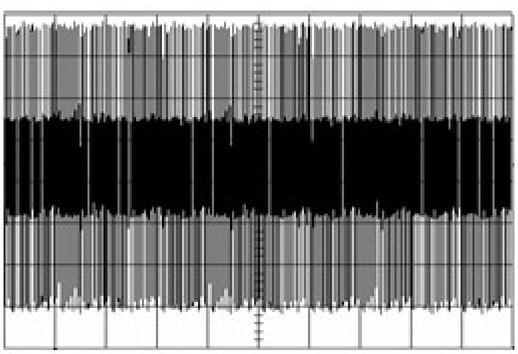
DISPLAY TYPE	VALUE DISPLAYED
Parameter Statistics Off	Single value of the modulation amplitude for the last acquisition.
Parameter Statistics On	Average, minimum, maximum, and sigma of the single modulation amplitude value calculated per acquisition for all acquisitions since the last CLEAR SWEEPS operation.
Histogram Function	Histogram of the modulation amplitude value calculated per acquisition for all acquisitions since the last CLEAR SWEEPS operation.
Trend Function	Trend of the single modulation amplitude value calculated per acquisition for all acquisitions since the last CLEAR SWEEPS operation

Example



The following persistence plots were generated using the DC-coupled signal.

In the first plot, the amplitude measurement cursor is reading the 11T top voltage of 76.7 mV.

	In the second, the cursor reads the difference between the 11T top and base of 67.2 mV.
	In the third plot, the cursor reads the difference between the 3T top and base of 47.2 mV.
	The last plot shows the waveform with the parameters calculated automatically. 1. pmoda 3T paa/11T top 2. pmoda 11T paa/11T top 3. paa 3T 4. paa 11T 5. top 11T P1 contains the ratio of P3 to P5. P2 contains the ratio of P4 to P5.

PNUM - PIT NUMBER

Description

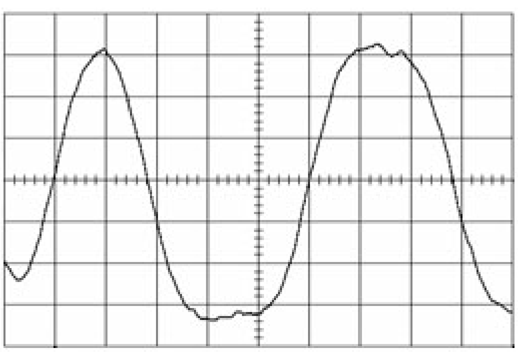
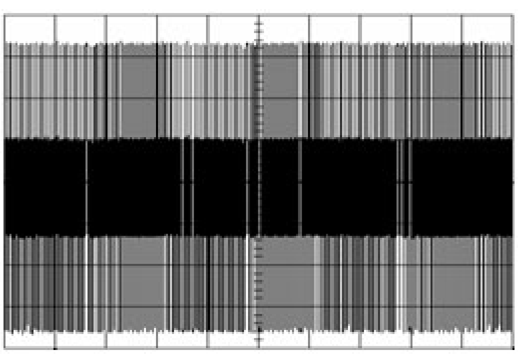
Pit Number provides a measurement of the number of pits or spaces of interest or both. When pnum is selected as a parameter measurement the total number of pits and/or spaces for the selected 'nT' range is displayed. In the nT Table mode the number for each 'nT' value is displayed.

Display Options

ORM parameter calculations can be displayed, histogrammed, and trended in a variety of ways. The table provides a concise description of the value or values displayed using each approach.

DISPLAY TYPE	VALUE DISPLAYED
Parameter Statistics Off	Single value of the total number of pits/spaces within the selected 'nT' range for the last acquisition.
Parameter Statistics On	Average, minimum, maximum, and sigma of the single value of the total number of pits/spaces within the selected 'nT' range, calculated per acquisition for all acquisitions since the last CLEAR SWEEPS operation.
nT Table	List of values of the number pits/spaces for each individual 'nT' in the selected range of 'nT' for the last acquisition.
Histogram Function	Histogram graph of the single value of the number of pits/spaces within the selected 'nT' range calculated each acquisition for all acquisitions since the last CLEAR SWEEPS operation.
Trend Function	Trend graph of the single value of the number of pits/spaces within the selected 'nT' range calculated each acquisition for all acquisitions since the last CLEAR SWEEPS operation.

Example

	In this waveform, each of the 3 pits/spaces is easily identified. There is a 4T pit, a 6T space, and a 5T pit. Each is counted and displayed in 'nT Table' mode.
	This is the long waveform showing the number of pits/spaces obtained: approximately 9,000.

PRES - PIT RESOLUTION

Description

Pit Resolution measures the ratio of the Pit Average Amplitude (see paa measurement description) of the smallest of the 'nT' pits or spaces in the data signal to that of the largest:

$$PRES = \frac{paa_{low\_n}}{paa_{high\_n}} \cdot 100\%$$

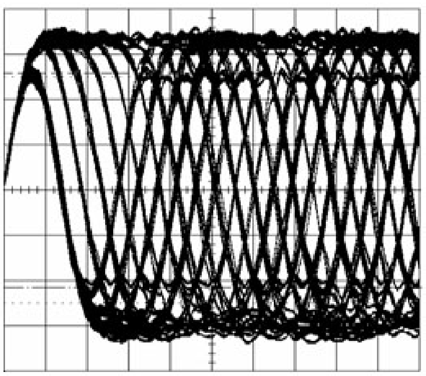
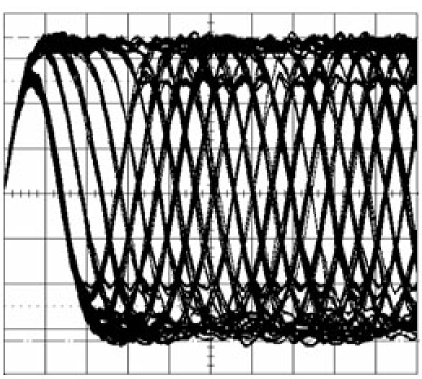
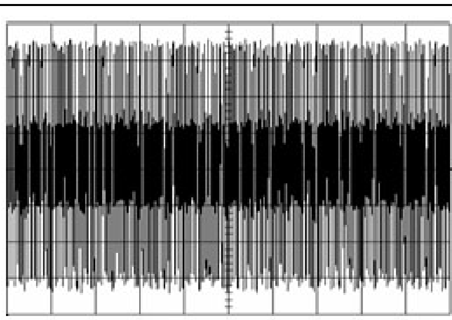
The low and high 'nT' values for performing the calculation must be provided by the user through the associated measurement configuration options. The value shown is in units of percent. The measurement calculation is compliant with the definition of Pit Resolution as defined by IEC 13549:1993 Section 15.3.1.

Display Options

ORM parameter calculations can be displayed, histogrammed, and trended in a variety of ways. The table provides a concise description of the value or values displayed using each approach.

DISPLAY TYPE	VALUE DISPLAYED
Parameter Statistics Off	Single value of the pit resolution for the last acquisition
Parameter Statistics On	Overall average, minimum, maximum, and sigma of the single pit resolution value calculated per acquisition for all acquisitions since the last CLEAR SWEEPS operation.
Histogram Function	Histogram of the single pit resolution value calculated per acquisition for all acquisitions since the last CLEAR SWEEPS operation.
Trend Function	Trend of the single pit resolution value calculated per acquisition for all acquisitions since the last CLEAR SWEEPS operation.

Example

	Consider the following persistence plots. In the first, the amplitude measurement cursor reads the difference between the 3T top and base: 47.3 mV.
	In the second, the cursor reads the difference between the 11T top and base: 67.3 mV. Therefore, the resolution is: $\frac{47.3}{67.3}\cdot 100\% = 70.3\%$
	This is the same waveform with the parameters calculated automatically. 1. pres 3T paa / 11T paa 2. paa 3T 3. paa 11T P1 contains the ratio of P2 to P3 in percent.

PTOP - PIT TOP

Description

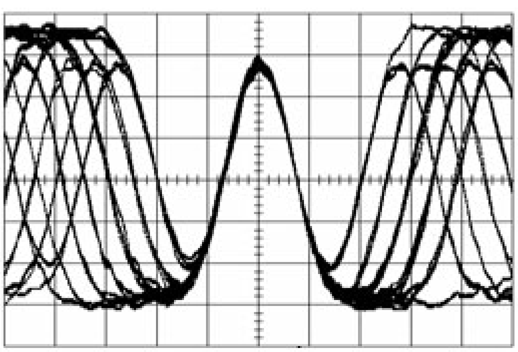
Pit Top provides a measurement of the best estimate of the top amplitude of a pit. The concept of the top calculation is to automatically provide the same measurement which would be obtained from a persistence plot. The top of each pit is determined through histogramming techniques described in detail under Base and Top Calculation Details. When ptop is configured as a custom parameter, all tops within the single nT or range of nT are calculated. Histogramming or trending such a configuration would result in one value per pit in the nT range contributing a value to the histogram or trend. The value displayed on the custom parameter display line is the average of all such top calculations. ‘nT Table’ mode provides an average top measurement for each *n* index.

Display Options

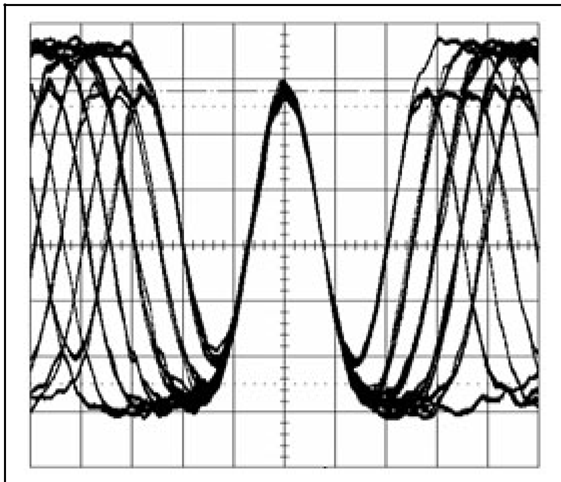
ORM parameter calculations can be displayed, histogrammed, and trended in a variety of ways.

DISPLAY TYPE	VALUE DISPLAYED
Parameter Statistics Off	All values of the top for all pits within the selected ‘nT’ range for the last acquisition.
Parameter Statistics On	Average, minimum, maximum, and sigma of the top for all pits within the selected ‘nT’ range for all acquisitions since last CLEAR SWEEPS operation.
nT Table	List of values of the average top for each group of pits of common ‘nT’ width for the last acquisition.
Histogram Function	Histogram graph of the value of the top for all pits within the selected ‘nT’ range for all acquisitions since the last CLEAR SWEEPS operation.
Trend Function	Trend graph of the value of the top calculated for pit that is within the selected ‘nT’ range for all acquisitions since last CLEAR SWEEPS operation.

Example



This persistence waveform was created by setting a SMART Trigger to capture only 3T pits. The computed 3T top is 25.8 mV.



When the same measurement is taken with the parameter cursors, it confirms that 25.8 mV is a reasonable value for the top.

PWID - PIT WIDTH

Description

Pit Width provides a measurement of the width of pits or spaces or both. The width of the pit or space is determined by the crossing of the selected voltage threshold. When pwid is selected as a parameter measurement it is generally useful to display the measurement calculation for a single 'nT' value. Otherwise the measurement will calculate the average width of 3T pits, 4T pits, and so on, which is meaningless. However, it is also often desirable to histogram the width of all pits and/or spaces. In this case the range of 'nT' values should be set to include all pit/space widths of interest.

Display Options

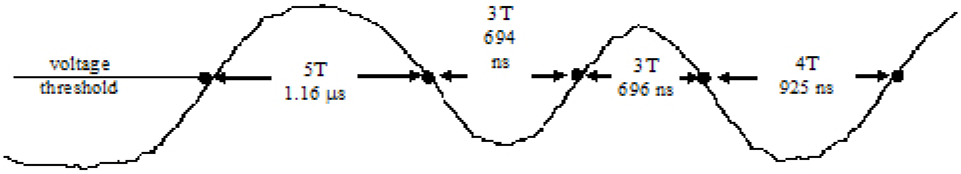
ORM parameter calculations can be displayed, histogrammed, and trended in a variety of ways. The table provides a concise description of the value or values displayed using each approach.

DISPLAY TYPE	VALUE DISPLAYED
Parameter Statistics Off	All values of the pit width for all pits/spaces within the selected 'nT' range for the last acquisition.
Parameter Statistics On	Average, minimum, maximum, and sigma of the pit width for all pits/spaces that are within the selected 'nT' range for all acquisitions since the last CLEAR SWEEPS operation.
nT Table	List of values of the average pit width for each group of pits/spaces of common 'nT' width for the last acquisition.
Histogram Function	Histogram graph of the value of the pit width for all pits/spaces within the selected 'nT' range for all acquisitions since the last CLEAR SWEEPS operation.
Trend Function	Trend graph of the value of the pit width for all pits/spaces within the selected 'nT' range for all acquisitions since the last CLEAR SWEEPS operation.

Example 1

The example shows that, measured at the selected voltage threshold, the CD data signal contains sequentially a 5T pit, 3T space, 3T pit, and 4T space. If the measurement is configured to select only 3T pits or spaces then the value displayed will be:

pwid = (694 ns + 696 ns) / 2 = 695 ns



Example 2: Histogramming

Consider the problem of determining the error margin in an optical recording system. Because the data is encoded in the widths of the pits/spaces, it would be ideal for the widths to be exact integer multiples of the period of the clock used to sample the data signal. In practice this is not the case, but in order to ensure error-free data recovery, it is important for the widths to be grouped and separated.

Histogramming can be used to analyze the grouping of pit widths and to determine whether the separation is acceptable.

The scope is set up to acquire the optical data waveform by assigning Channel 1 to the data signal at a time/div of 0.2 ms, so that many pits/spaces can be gathered quickly. The signal is AC-coupled, so the threshold is set to 0 mV.

The **pwid** custom parameter is assigned to P1 and configured in the following manner:

hysteresis = 0.5 divisions

threshold = 0 mV
polarity = Both
range of *n*
low *n* = 3
high *n* = 11
period = 231.5 ns

F1 is defined as a zoom of channel 1 so that the waveform can be viewed expanded and the pits and spaces can be identified.

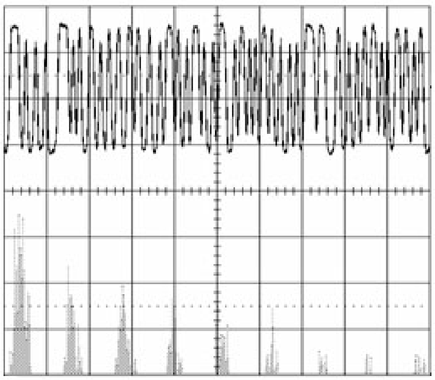
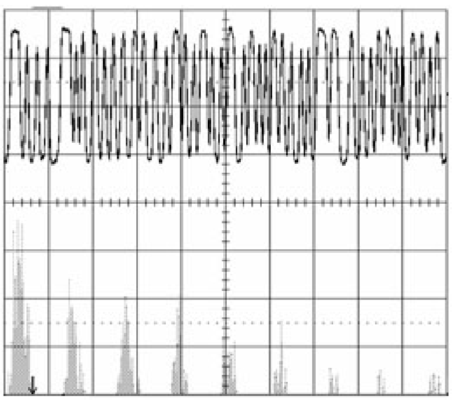
Using the "Math Setup" dialog, F2 is defined as the histogram of the **pwid** parameter on P1 and is set up as follows:

1. Use the maximum number of values (2,000,000,000)
2. Classify into 2000 bins
3. Linear vertical scale.

The trigger is set up to trigger on a pit edge and operated in normal trigger mode.

Note: Prior to acquisition, select each trace and press the RESET button to ensure that all the traces are reset.

In normal trigger mode, multiple waveforms are acquired and processed. The histogram will typically have data that is not well centered or is off screen. Touch the **FIND CENTER AND WIDTH** button to see the pit width distributions as they accumulate. After enough measurements have been taken, stop the triggering. After the histogram has been centered, the screen will look as follows:

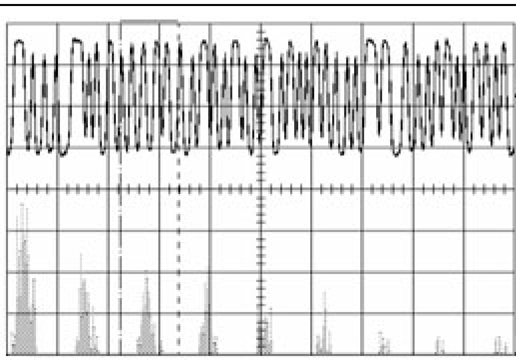
	The optical data waveform is on the top and the histogram is on the bottom. Notice the clustering of width distributions.
	By enabling cursor tracking, the difference cursors are swept across the histogram. As expected, the space between the 3T and 4T distributions is the shortest, because of inter-symbol interference and the many 3T widths. The spacing is 138 ns.

To measure the spread of widths for the distributions, set the measure mode to **My Measure** and configure the parameters:

1. P2: average of F2
2. P3: high of F2
3. P4: low of F2
4. P5: range of F2

Because parameter measurements are performed only on those portions of the waveform between the parameter cursors, activate tracking so that they can be swept across the histogram. Set the difference between the cursors so that they encompass one clock period. In this case, the histogram is shown at 0.2 µs per division. Set the difference between the parameter cursors to:

$$\frac{231.5\cdot10^{-9}}{.2\cdot10^{-6}} = 1.16 \quad divisions$$

	This screen shows the histogram statistics taken on the 5T distribution. The distribution has the largest spread of values: 102.5 ns. The mean is 1.1659 µs, which is 3.6% higher than the ideal of 1.1575 µs.

T@Pit - TIME AT PIT

Description

The Time-at-Pit parameter provides the time of each leading edge of every pit or space within the nT range specified from the trigger point (time = 0). The value displayed is the time of the first pit only.

The usefulness of this parameter is not in the displayed value, but in its trending. The intent is that two parameters (t@pit and another ORM parameter) can be set up with identical configurations: precisely the same number of pits or spaces is found in the waveform, and precisely the same number of parameter measurements is made. When both of these parameters are trended, the two trends will have the same number of events, and there will be a one-to-one correspondence between each event in each trend. If both trends are displayed, and time cursors are swept over each, values will be displayed for the ORM parameter value and the time within the acquisition where the parameter measurement was made. These times are useful when searching for abnormal events within a waveform.

Not only can the trend of t@pit provide the actual event time, it can be used as the X-axis in an XY plot to examine modulation characteristics of particular parameter measurements.

Example

This example typifies the usage of the t@pit parameter. Step-by-step instructions are given.

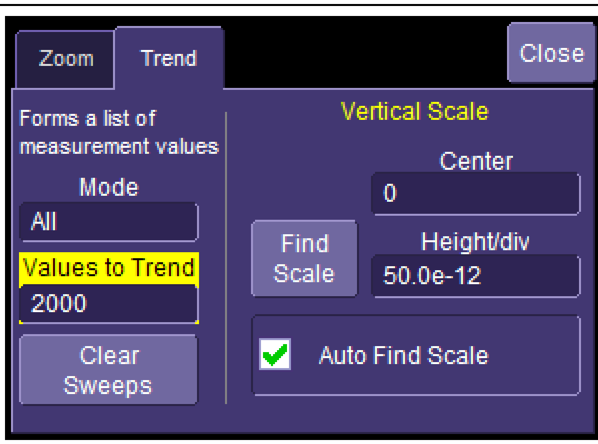
A large optical recording waveform is to be acquired, and the ordinary pit/space widths that can cause errors in the system need to be found. The waveform contains pits/spaces that have widths that are ideal integer multiples of the clock period 231.5 ns in a range from 3 to 11 times this clock period.

The scope is set up to acquire this waveform by assigning Channel 1 to the data signal at a time/div of 0.2 ms. This signal will contain approximately 1800 pits/spaces. The ideal threshold (determined by the **pmidl** parameter) is 1.9 mV.

The **pwid** custom parameter is assigned to P1 and the **t@pit** parameter to P2. Both parameters are configured in the following manner:

hysteresis = 0.5 divisions
threshold = 1.9 mV
polarity = Both
range of *n*
low *n* = 0
high *n* = 25
period = 231.5 ns

Using the "Math Setup" dialog, define F1 as the trend of the pwid parameter, and F2 as the trend of the t@pit parameter. For later use, define F3 as a zoom of Channel 1.



We are expecting 1800 pits/spaces, so make sure that the trends are set to use up to 2000 values for each math setup.

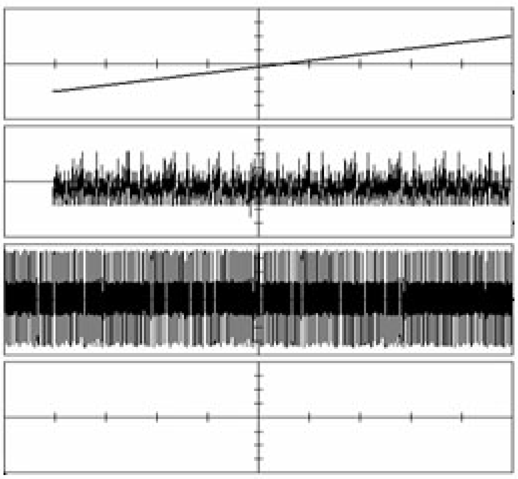
The trigger is set up to trigger on a pit edge and is operated in single-shot mode. For convenience, the waveforms are ordered on the screen in a particular manner:

1. F2: Trend of **t@pit**
2. F1: Trend of **pwid**
3. F3: Zoom of optical recording waveform

The reason for this order will become apparent.

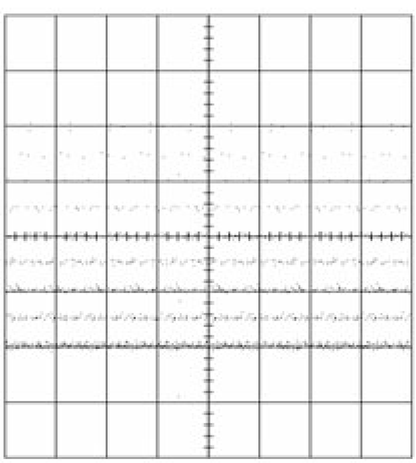
Press the single-shot trigger button to acquire the waveform. The waveform should be centered on the screen. Typically the trends will have data that is not well centered or is off screen. Centering is done by touching the **Find Scale** button in each trend setup dialog.

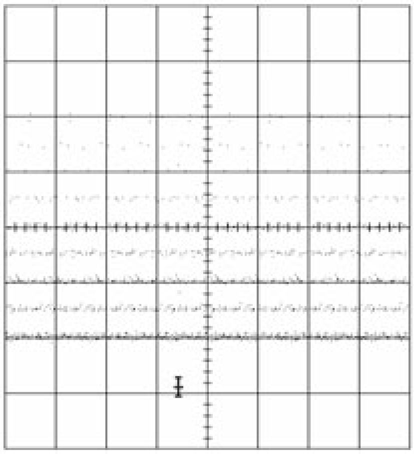
The next screen shows what each trace looks like after the waveform has been acquired and the trends centered.

	The waveforms are displayed in Quad grid mode. The trend of t@pit is basically linear, as expected because the time at each pit from the trigger is ascending. The trend of the pit widths looks basically as expected. Notice that there are exactly as many events inside both trends, a necessary condition.

From the menu bar, select **Display Setup…** and set the grid mode to **XY**. Bands of pit widths corresponding to widths that are ideal integer multiples of the clock period will be evident. Select **F2** (the trend of t@pit) and zoom to expand the time scale. Then select **F1** (the trend of pwid) and use the vertical ZOOM knob to adjust the band spacing. The vertical POSITION knob can be used to position the display vertically.

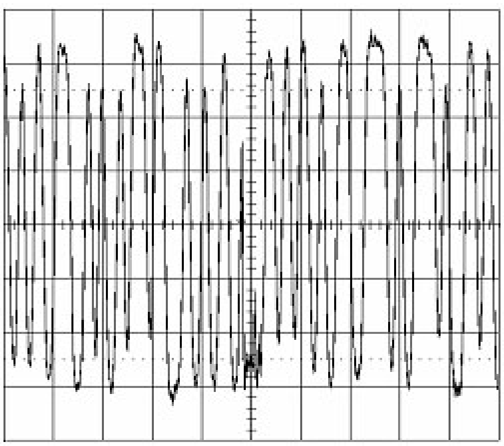
The next screens show the XY plot.

	The XY plot has been adjusted so that all the pit widths are displayed vs. time. Notice that all form bands. This is because all but two are ideal integer multiples of the clock period. The two pit widths dissimilar to the others are sitting just below the 3T pit width band, and between the 4T and the 5T band. These strange pit widths occurred around the middle of the waveform.

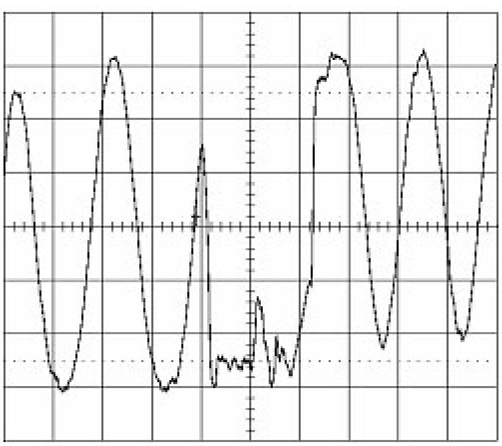


Using time cursors, it can be seen that the beginning of this problematic pit width occurred at pit # 973, 859.61 µs from the trigger. This smaller-than-normal pit is 245.40 ns.

Now that a problem has been identified, we would like to view the portion of the waveform in which the problem occurred. Change the display mode Single grid. Turn off the two trend traces, leaving only F3, which is the expanded trace of Channel 1. Move the absolute time cursor to the position in the trace at 859 µs; and, using the WAVEPILOT position controls, position the waveform so that the cursor on the trace is at the center of the screen. Expand the waveform using the horizontal zoom control.



Here is the waveform zoomed in a bit with the measurement cursor placed at 859.600 µs. As can be seen, there is some kind of aberration at the center of the trace.



Further zooming clearly identifies the problem: a burst error that prevented the positive polarity width that starts at 859.61 µs from reaching its peak value. This defect caused the reflectivity to drop and to erratically fluctuate throughout the duration of the burst error. The defect affected the width of the next pit as well, which created the pit width centered between the 4T and 5T band of the XY plot.

TIMJ - TIMING JITTER

Description

Timing Jitter provides a measurement of the standard deviation of the difference of the width of pits and/or spaces from the mean width. The width of the pit/space is determined by the crossing of the selected voltage threshold.

The measurement calculation is compliant with the definition of Timing Jitter as defined by ISO/IEC JTC1.23.14517 Section 22.4.

Display Options

ORM parameter calculations can be displayed, histogrammed, and trended in a variety of ways. This table provides a concise description of the value or values displayed using each approach.

DISPLAY TYPE	VALUE DISPLAYED
Parameter Statistics Off	Single value of the overall timing jitter for pits/spaces of interest for last acquisition.
Parameter Statistics On	Average, minimum, maximum, and sigma of the overall timing jitter value calculated per acquisition for all acquisitions since the last CLEAR SWEEPS operation.
nT Table	List of values of the timing jitter for each individual 'nT' in the selected range of 'nT' for the last acquisition.
Histogram Function	Histogram of the overall timing jitter values calculated for each acquisition for all acquisitions since the last CLEAR SWEEPS operation.
Trend Function	Trend of the overall timing jitter values calculated for each acquisition for all acquisitions since the last CLEAR SWEEPS operation.

Example

A waveform is acquired with 3T, 4T, and 5T pit widths as follows:

3T	4T	5T
695 ns	925 ns	1.16 µs
690 ns		1.18 µs
696 ns		

T is 231.5 ns, and the timing jitter parameter has been configured for a range of 3T through 5T.

The 3T mean is 693.66 ns. The 4T mean is 925 ns. The 5T mean is 1.17 µs.

The 3T timing jitter is calculated by taking the standard deviation of the difference between each width and the 3T mean. This is 3.214 ns, normalized by:

$$Timj_{3T} = 3.214 \cdot \frac{100\%}{231.5} = 1.389\%$$

The 4T timing jitter cannot be calculated because there is only one value (at least two values are required).

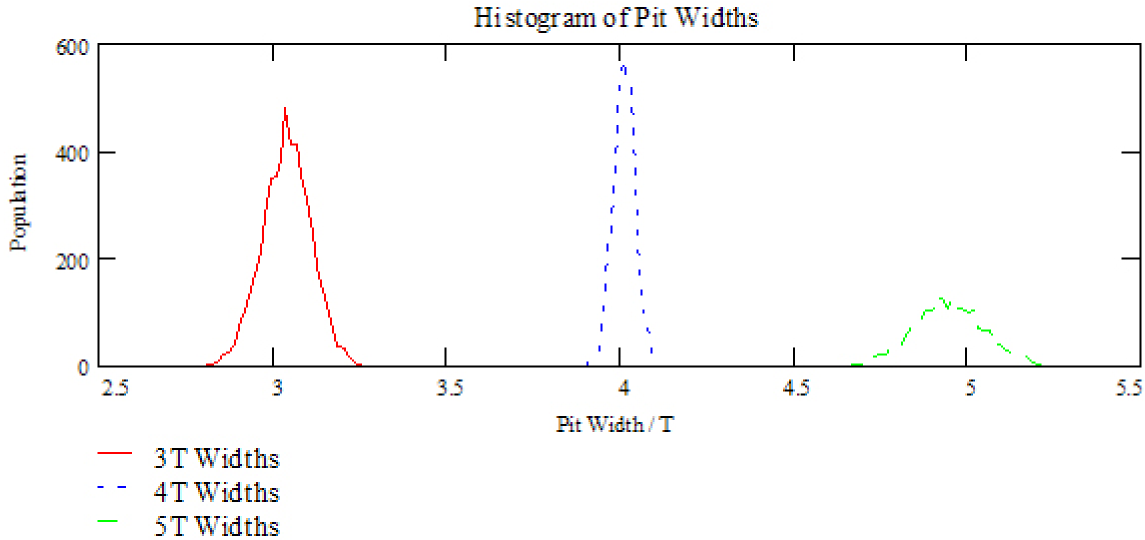
The 5T timing jitter is +6.109%.

The overall timing jitter is calculated using a weighting formula, which results in the standard deviation of the mean centered distributions. In this example, it is calculated as:

$$Timj_{overall} = \sqrt{\frac{1.389^2 \cdot (3-1) + 6.109^2 \cdot (2-1)}{(3+2-1)}} = 3.208\%$$

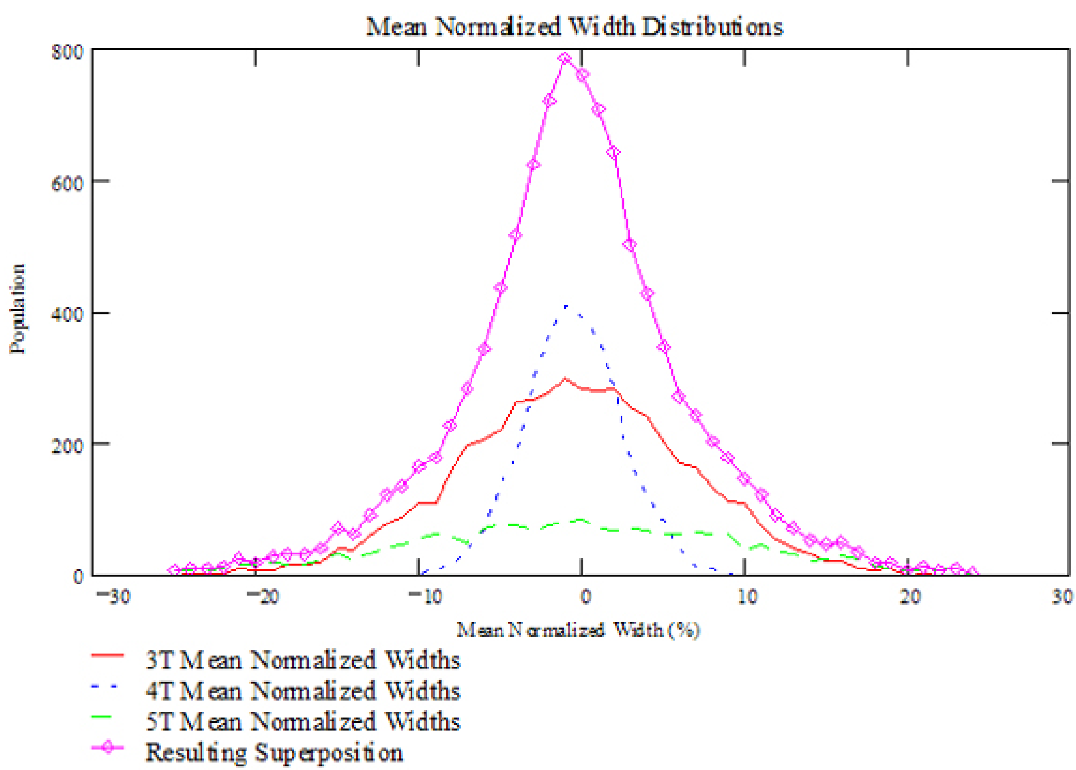
More About Timing Jitter

In order to understand the operation of the timing jitter parameter with different modes of operation, consider the histogram of 3T to 5T pit widths in the next figure.



The Timing Jitter parameter considers each of these distributions separately. For each distribution the standard deviation is calculated. This is the timing jitter displayed for each nT distribution. Overall timing jitter is calculated by subtracting the mean width of each distribution from the widths in those distributions and considering the resulting superposition.

The sigma of the 3T, 4T, and 5T distributions are what is obtained when the Timing Jitter custom parameter is configured for single n values (the sigmas are the same as the sigma of the edge shift calculation). The sigma of the resulting superposition is what is obtained when the Timing Jitter custom parameter is configured for ranges of n values (in this case 3T to 5T).



The value displayed on the custom parameter line (with statistics off) is the sigma of any of the resulting distributions for the last acquisition only. This timing jitter value is calculated internally without having to actually histogram the values. The values displayed in nT Table mode are the sigma of the width distributions resulting from each nT distribution for the last acquisition.

Note: Timing Jitter is always less than or equal to the standard deviation of superimposed Edge Shift distributions.

Signals, Coupling, and Threshold Settings

Which optical recording signal, or combination of signals should be used in a calculation? How should the signal be coupled, or the threshold set? The answers to these questions are sometimes uncertain. This section offers tips on how to answer them.

Choice of Signals

Generally, the choice of signals depends on the aim of the measurement. For example, if the quality of the signal direct from the media is being examined, generally the signal at the output of the photodetector should be used. Alternatively, a conditioned signal could serve the purpose.

A “sliced” or logic conditioned signal should normally be chosen when precise timing measurements are desired and propagation delay through the logic device (comparator) is not an issue. Timing measurement accuracy is improved when a fast signal is used, as opposed to the slower signals at the photodetector, for the following reasons:

- A fast edge usually results in more accurate timing measurements because of interpolation algorithms, as long as points are sampled along the edge.
- A fast edge provides a threshold crossing time and, therefore, measurement accuracy more immune to noise.
- The use of the signal at the output of a logic device or comparator decreases the sensitivity of the measured threshold-crossing time to the exact value of the threshold level selected.
- The use of the signal at the output of a logic device or comparator typically solves other threshold problems as well, in systems that dynamically adjust the threshold based on the optical recording data signal. Sliced or logic signals facilitate the use of a fixed threshold.

Coupling

DC coupling is required only for measurements of absolute DC values. Measurements requiring it include **ptop**, **pbase**, **pmin**, **pmax**, and **pmoda**. Otherwise, AC coupling is best used on signals that are not outputs of logic devices or comparators: those that might have varying thresholds.

Threshold Selection

If DC coupling must be used, there are some further considerations for threshold selection. While all of the optical recognition measurements specify thresholds used to extract the pits/spaces (by recording threshold crossings), there is a variance in the sensitivity of parameters to the exact threshold value selected. The sensitive parameters are those that are time related or whose values depend on the exact time of the threshold crossing. Those insensitive to the exact threshold value are parameters that use the threshold crossing time only to categorize the parameter result according to width (that is, they use the crossing time only to find the width for determining the nT index to which the pit/space belongs).

In the case of threshold-insensitive parameters, it usually suffices to use a fixed threshold somewhere in the middle of the optical recognition waveform. Even if the signal’s middle shifts, the fixed threshold is usually adequate.

Additionally, if the signal is AC coupled, it will tend not to shift much, and the fixed threshold will be perfectly adequate.

The problem arises when what is required is a DC-coupled signal with a threshold that changes dynamically throughout the waveform. There remains a possible solution, but the scope setup is slightly more complicated.

Consider the fact that AC coupling can be regarded as rejection of the DC component of a signal, or subtracting it from the signal. In many systems, the threshold is determined in precisely this manner by applying a low-pass filter to the signal, and then applying this value, with the signal itself, to the input of a comparator. If a threshold value determined in this manner is available in the circuit, the threshold signal, along with the optical recognition data signal itself, should be acquired. Waveform math can then be used to subtract the threshold signal. This is done by defining a trace as the Arithmetic Difference of the raw data signal and the threshold signal. The new trace is then used as the optical recognition data signal in the parameter calculations.

Regardless of how the signal is coupled, there are other considerations involved in determining the appropriate threshold. If waveform math is used, the threshold is always 0 V. Otherwise, the optimum threshold is best determined using the pmidl parameter.

Some optical recognition standards define the middle level of the 3T signal as the “decision level.” Pmidl configured for the single 3T pits/spaces is an ideal candidate for the best threshold value. Another candidate is the pmidl value calculated using the entire range of n indices possible. In this way, pmidl calculates the best overall threshold level as a weighted average of middle levels calculated for each n index.

In AORM, the ODATA function can be used to remove these effects. Its “leveled” output subtracts the “threshold” (low frequency content of the signal) from the input data.

Using Parameters with Trends and XY Plots

X-axis	Y-axis
t@pit	Dp2c edgsh pbase pmax pmin ptop pwid
pwid	pbase pmax pmin ptop
ptop	pmax
pbase	pmin

We saw in the t@pit parameter description how the ORM parameters have certain unique characteristics that make particular measurements useful when trended together with XY plots. And how the t@pit parameter is essential to those measurements. Plots that can be generated on single acquisitions include those listed in the table at left.

The reason that these plots are considered useful on single acquisitions is because the parameters are guaranteed to be configurable in a manner that meets the following criteria:

- Each parameter is capable of providing multiple values per acquisition.
- Each parameter pair is configurable in a manner that guarantees the same number of events per waveform.
- Each parameter pair must be configurable in a manner that guarantees a one-to-one correspondence between parameter calculation values.

Example and Step-by-Step Instructions

Here is an example typifying the use of XY plots without the t@pit parameter. A complete example using t@pit has been provided in the section dedicated to this parameter description.

Consider a situation in which it is desirable to find the relationship of the pit top value to the pit width in an optical recognition data waveform:

The scope is set up to acquire this waveform by assigning Channel 1 to the data signal at a time/div of 0.2 ms. This signal will contain approximately 1800 pits/spaces. The ideal threshold has been determined by the pmidl parameter as 1.9 mV.

The **ptop** custom parameter is assigned to **P1**, and the **pwid** parameter is assigned to **P2**. Use configuration tracking to configure both parameters in the following manner:

hysteresis = 0.5 divisions
threshold = 1.9 mV
polarity = Pos
range of n
low *n* = 3
high *n* = 11
period = 231.5 ns

In math setup, **F1** is defined as the trend of the **ptop** parameter and **F2** as the trend of the **pwid** parameter. Because we are expecting 1800 pits/spaces, make sure that for each math setup the trends are set to use up to 20,000 values.

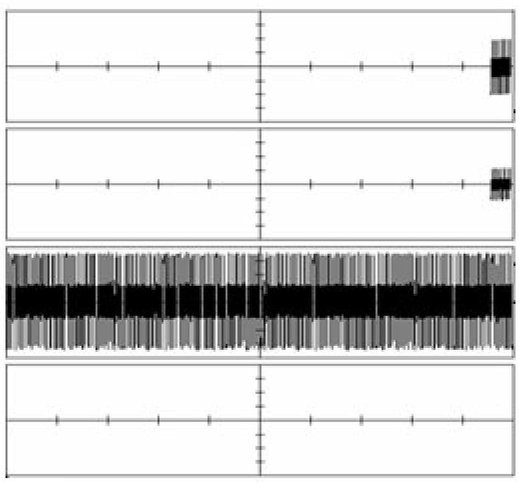
> **Note:** If configuration tracking is used on the ptop parameter, the pwid parameter must be visited in order to set the polarity to positive because ptop inherently implies positive polarity pits.

The trigger is set to trigger on a pit edge and is operated initially in single-shot mode. For convenience, the waveforms are ordered on the screen in a particular manner so that they will automatically work correctly with XY display mode:

1. F2: Trend of **t@pit**
2. F1: Trend of **pwid**
3. Channel 1: optical recognition data signal

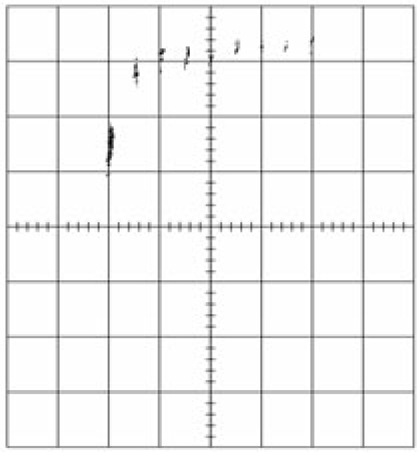
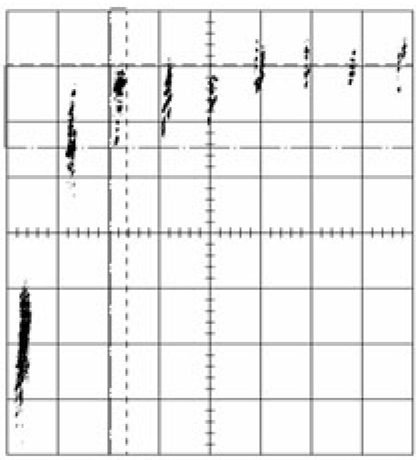
> **Note:** Prior to acquisition, select each trace and press the RESET button to ensure that all the traces are reset.

The single-shot trigger button is pressed and the waveform acquired. The waveform should be centered on the screen. The trends will typically have data not well centered or off-screen. These traces can be positioned on the grid by touching the FIND SCALE button in each trend setup dialog. The screen shown here is what each trace looks like after the waveform has been acquired and the trends centered.

	The waveforms are displayed in Quad grid mode. Notice that there are exactly as many events inside both trends, a necessary condition. Although the trends are very short (containing only 902 out of the total 20,000 pits allowed) repeated triggering will eventually fill in both trends sufficiently.

Set the display mode to **XY**. Clusters of pit top values will be apparent: clustered because the tops tend to be approximately the same amplitude and the pit widths approximate multiples of the clock period. Select **F2** (the trend of pwid) and use the vertical ZOOM control to expand the X-axis scale. Select **F1** (the trend of ptop) and use the ZOOM knob to adjust the vertical scale. The vertical POSITION knob can be used to position the display vertically.

This is what the XY plot looks like:

	The XY plot has been adjusted so that all of the pit tops are displayed vs. Pit width. Notice that all of the pit widths form clusters. Press NORMAL trigger, and the clusters will become even more dense. You can have up to 20,000 points in the XY plot.
	Using the XY cursors, a variety of measurements can be performed simultaneously. For example, here it can be seen that the 5T pit width varies by approximately 82 ns and the top varies by approximately 2.58 mV. Of course these could be seen through automatic parameter calculations without using XY plots, but the XY plot can sometimes provide information that would not otherwise be observable.

Improving Horizontal Measurement Accuracy

Horizontal measurement accuracy pertains to timing-related measurements. In the AORM package, these are Dp2c, Dp2cs, edgsh, Iper, pwid, t@pit, and timj. In many cases, measurement accuracy can be improved by considering certain items pertaining to how a DSO operates and how parameters are measured.

DSOs sample the signal, building a waveform that consists of points at intervals determined by the sample rate. One obvious consideration for maximizing horizontal measurement accuracy is to ensure that the highest sample rate possible is used. On low time/divs, waveforms become long. Thus it is important to set the Max Sample Points value in the SMART Memory dialog to the largest possible value. This ensures the highest sample rate based on the time/div setting.

Times are calculated for ORM parameters by interpolating between points that straddle the threshold specified. Measurement accuracy is improved when the edge is:

- fast enough to enable points straddling the threshold that are far from the threshold, and
- slow enough, and the sample rate high enough, to enable points to be sampled on the edge.

In most cases, these considerations are taken into account by sampling at the highest rate possible and by ensuring that the volts/div setting is as low as practically possible.

Note on RIS (Random Interleaved Sampling)

RIS is a mechanism used by the oscilloscope to increase the effective sample rate by interleaving samples taken over multiple waveform acquisitions. The scope enters RIS mode automatically when the time/div setting is set extremely low.

Because multiple acquisitions are interleaved in RIS, a highly stable trigger signal must be maintained, and precisely the same waveform acquired on each acquisition.

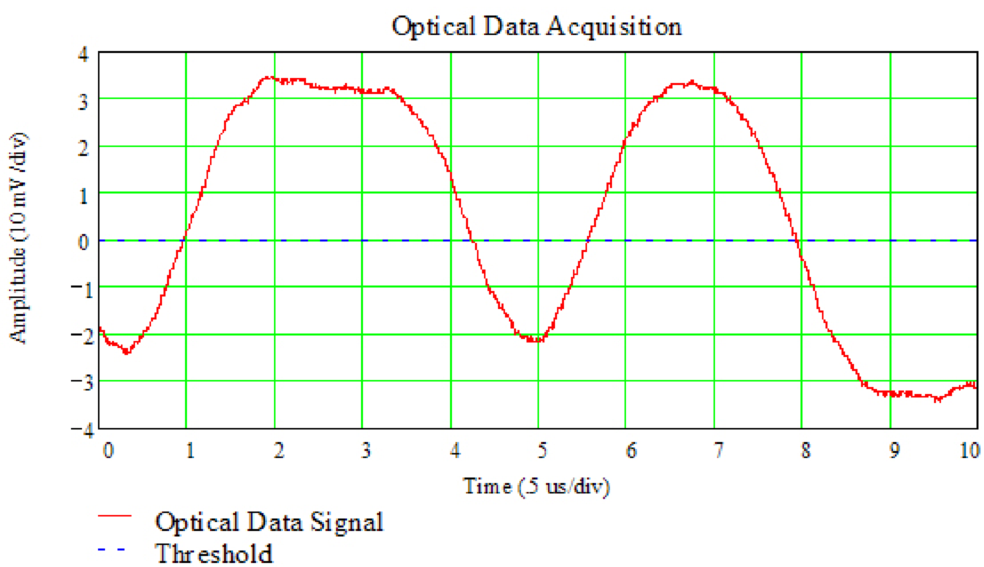
For most ORMs, RIS is neither appropriate nor recommended. If not used properly, it will result in erroneous measurements.

(For more on RIS, see your scope's *Operator's Manual*.)

Base and Top Calculation

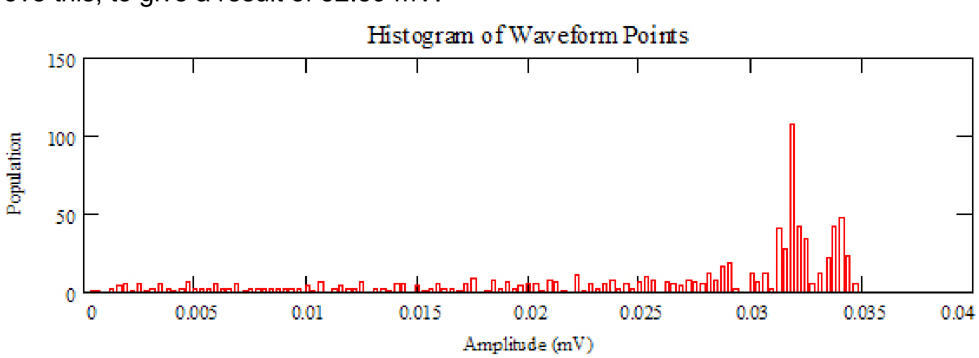
The base and top are designed to emulate results in the past obtained from persistence plots. In general, the top was calculated by examining the most intense region near the top of a waveform in an eye-pattern persistence map. The AORM package improves on this in that the tops of all pits are calculated independently: rogue amplitude variations in the waveform can be identified.

The waveform in the next figure contains two pits. We need only consider the first of these.

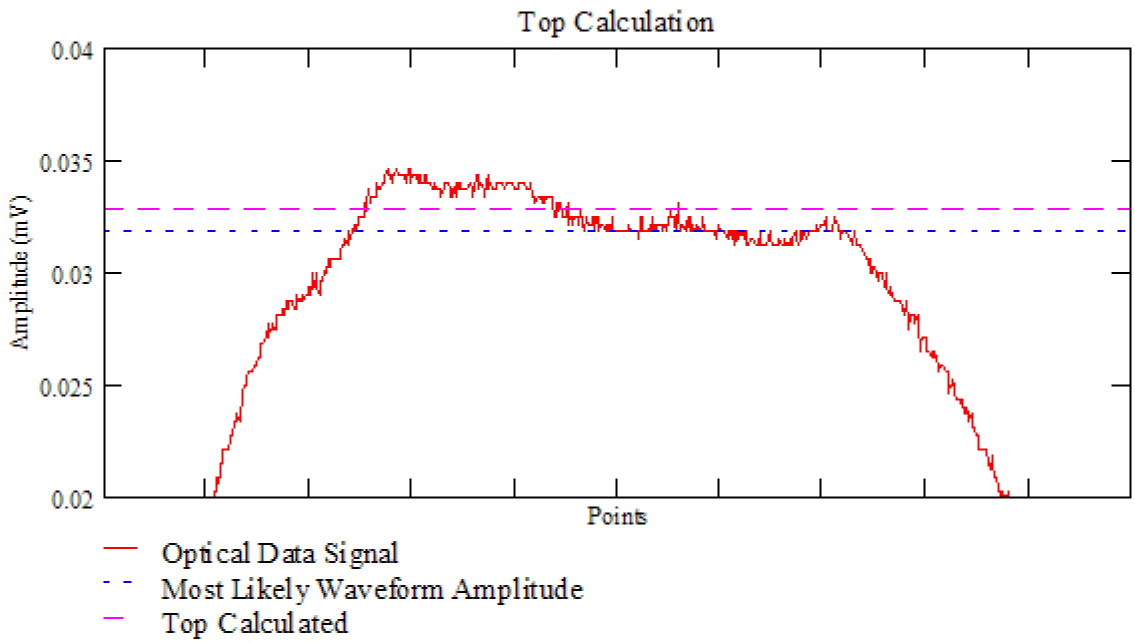


The AORM package histograms the values inside each pit to determine the most likely amplitude: the most densely populated region.

The next figure shows the histogram of the pits' amplitudes. It is easily seen that the most likely amplitude is approximately 32 mV (exactly 31.9 mV). The top is calculated by averaging all of the waveform data points at or above this, to give a result of 32.89 mV.



In the next figure, the top bisects the two flattest regions at the top of the waveform and, in effect, calculates the value that would be estimated from examination of an eye-pattern persistence map.



Introduction to AORM Theory

An understanding of statistical variations in parameter values is needed for many waveform parameter measurements. Knowledge of the average, minimum, maximum, and standard deviation of the parameter may often be enough, but in many other instances a more detailed understanding of the distribution of a parameter's values is desired.

Histograms provide the ability to see how a parameter's values are distributed over many measurements. They divide a range of parameter values into sub-ranges called bins. A count of the number of parameter values calculated (events) that fall within its sub-range is maintained for each bin.

While the range can be infinite, for practical purposes it need only be defined large enough to include any realistically possible parameter value. For example, in measuring TTL high-voltage values a range of ±50 V is unnecessarily large, whereas one of 4 V ±2.5 V is more reasonable. It is this 5 V range that is subdivided into bins. And if the number of bins used were 50, each would have a sub-range of 5 V per 50 bins or 0.1 V/bin. Events falling into the first bin would then be between 1.5 V and 1.6 V. The next bin would capture all events between 1.6 V and 1.7 V. And so on.

After several thousand events, the graph of the count for each bin (its histogram) provides a good understanding of the distribution of values. Histograms generally use the X-axis to show a bin's sub-range value, and the Y-axis for the count of parameter values within each bin. The leftmost bin with a non-zero count shows the lowest parameter value measurements. The vertically highest bin shows the greatest number of events falling within its sub-range.

The number of events in a bin, peak, or histogram is referred to as its population. The next figure shows a histogram's highest population bin as the one with a sub-range of 4.3 to 4.4 V, which is to be expected of a TTL signal. The lowest value bin with events is that with a sub-range of 3.0 to 3.1 V. Because TTL high voltages need to be greater than 2.5 V, the lowest bin is within the allowable tolerance. However, because of its proximity to this tolerance and the degree of the bin's separation from all other values, additional investigation may be desirable.

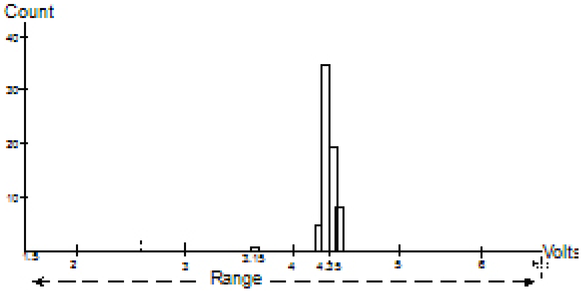
LeCroy DSO Process

LeCroy digital oscilloscopes generate histograms of the parameter values of input waveforms. But first, the following must be defined:

- The parameter to be histogrammed
- The trace on which the histogram will be displayed
- The maximum number of parameter measurement values to be used in creating the histogram

- The measurement range of the histogram
- The number of bins to be used

Once these are defined, the oscilloscope is ready to make the histogram.



The sequence for acquiring histogram data is:

1. trigger
2. waveform acquisition
3. parameter calculations
4. histogram update
5. trigger re-arm.

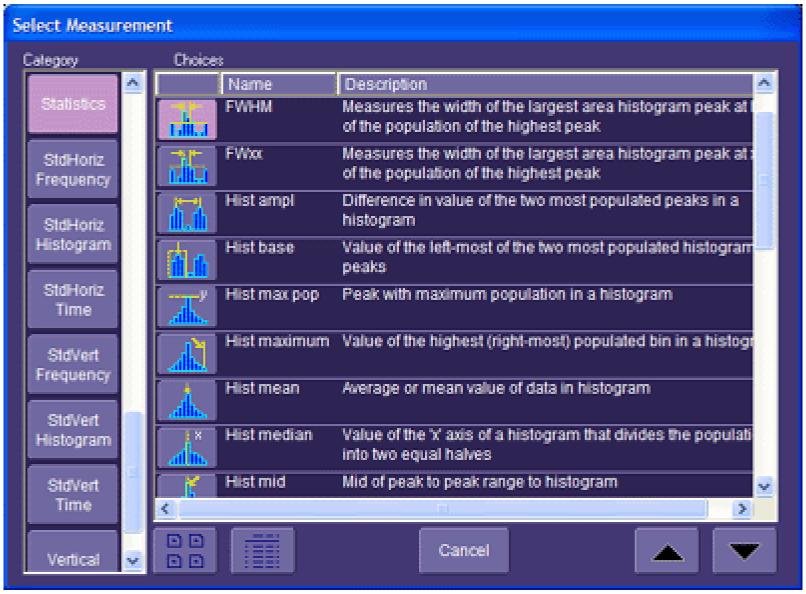
If the timebase is set in non-segmented mode, a single acquisition occurs prior to parameter calculations. However, in Sequence mode an acquisition for each segment occurs prior to parameter calculations. If the source of histogram data is a memory, storing new data to memory effectively acts as a trigger and acquisition. Because updating the screen can take significant processing time, it occurs only once a second, minimizing trigger dead-time (under remote control the display can be turned off to maximize measurement speed).

Histogram Parameters

Once a histogram is defined and generated, measurements can be performed on the histogram itself. Typical of these are the histogram's:

- Average value, standard deviation
- Most common value (parameter value of highest count bin)
- Leftmost bin position (representing the lowest measured waveform parameter value)
- Rightmost bin (representing the highest measured waveform parameter value).

Histogram parameters are provided to enable these measurements. Accessible by selecting **Statistics** from the **Select Measurement** menu, they are calculated for the selected waveform section between the parameter cursors.



avg average of data values in histogram

fwhm full width (of largest peak) at half the maximum bin

fwxx full width (of largest peak) at xx% the maximum bin

hampl histogram amplitude between two largest peaks

hbase histogram base or leftmost of two largest peaks

high highest data value in histogram

hmedian median data value of histogram

hrms rms value of data in histogram

htop histogram top or rightmost of two largest peaks

low lowest data value in histogram

maxp population of most populated bin in histogram

mode data value of most populated bin in histogram

pctl data value in histogram for which specified 'x'% of population is smaller

pks number of peaks in histogram

range difference between highest and lowest data values

sigma standard deviation of the data values in histogram

totp total population in histogram

xapk x-axis position of specified largest peak.

Zoom Traces and Segmented Waveforms

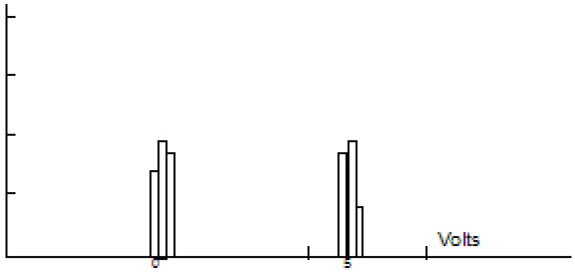
Histograms of zoom traces display all events for the displayed portion of a waveform between the parameter cursors. When dealing with segmented waveforms, and when a single segment is selected, the histogram will be recalculated for all events in the displayed portion of this segment between the parameter cursors. But if **All Segments** is selected, the histogram for all segments will be displayed.

Histogram Peaks

Because the shape of histogram distributions is particularly interesting, additional parameter measurements are available for analyzing these distributions. They are generally centered around one of several peak value bins, known (with its associated bins) as HistX@peak.

Example

In the next figure, a histogram of the voltage value of a five-volt amplitude square wave is centered around two peak value bins: 0 V and 5 V. The adjacent bins signify variation due to noise. The graph of the centered bins shows both as peaks.



Determining such peaks is very useful because they indicate the dominant values in a signal.

However, signal noise and the use of a high number of bins relative to the number of parameter values acquired can give a jagged and spiky histogram, making meaningful peaks hard to distinguish. The scope analyzes histogram data to distinguish peaks from background noise and from histogram definition artifacts such as small gaps, which are due to very narrow bins.

Binning and Measurement Accuracy

Histogram bins represent a sub-range of waveform parameter values, or events. The events represented by a bin may have a value anywhere within its sub-range. However, parameter measurements of the histogram itself, such as average, assume that all events in a bin have a single value. The scope uses the center value of each bin's sub-range in all its calculations. The greater the number of bins used to subdivide a histogram's range, the less the potential deviation between actual event values and those values assumed in histogram parameter calculations.

Nevertheless, using more bins may require performance of a greater number of waveform parameter measurements in order to populate the bins sufficiently for the identification of a characteristic histogram distribution.

In addition, very fine-grained binning will result in the creation of gaps between populated bins that may make determination of peaks difficult.

The oscilloscope's parameter buffer is very effective for determining the optimal number of bins to be used. An optimal bin number is one where the change in parameter values is insignificant, and the histogram distribution does not have a jagged appearance. With this buffer, a histogram can be dynamically redisplayed as the number of bins is modified by the user. In addition, depending on the number of bins selected, the change in waveform parameter values can be seen.

DVD Processing Model

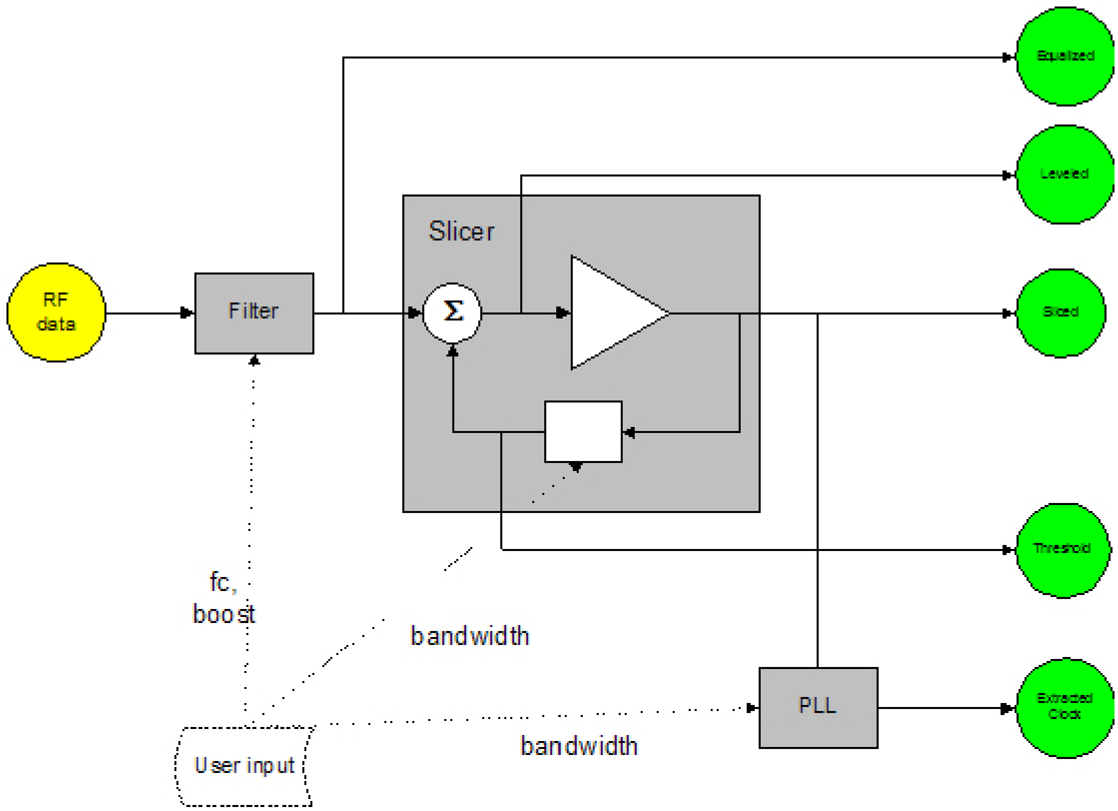
In many applications, it is important to make measurements directly from the RF signal, independent of a specific DVD chip. The OData processing function provided in the Advanced ORM package can emulate the filter, slicer, and/or phase-locked loop (PLL) of a typical optical recording drive. A schematic of this function is shown in the following diagram. You can view the equalized, leveled data, threshold, sliced data, or the extracted clock.

You can control the cutoff frequency and boost of the equalizing filter, the closed loop bandwidth of the slicer, and the bandwidth of the PLL. Alternatively, you can input the equalized signal and still look at the slicer or PLL output of the function.

Additionally, some of the advanced optical drives have a header and data section. A Gate signal differentiates the header and data sections. The OR Data function lets you input a gate signal, and allows you to choose when to analyze the data (gate high or low), so that either the header or the data area can be analyzed.

DVD RAM

For noisy signals if less than three width peaks are found, the PLL start frequency is set so that T is the nearest value to the expected bit rate for which the first width peak is an integer multiple of T. Also, the PLL start phase is derived from the first two edges instead of the first one.



Where:

- Equalized – applies a low pass filter with boost to the input data. This should not be used if the input data is already equalized (filtered).
- Leveled – Applies the filter (if the input data is raw RF) and then subtracts the sliced threshold from it.
- Sliced – This is the output of the slicer. It is similar to Leveled except the amplitude of each pulse is normalized to "1" and "0."
- Threshold – this waveform comprises the low frequency components of the original signal.
- Extracted CLK – the sliced data is passed through a PLL and the recovered clock signal is produced.

Filtering

A low-pass filter that removes high-frequency noise and provides equalization is needed for the newer optical recording systems (e.g., DVD). In the DVD read-only and recordable specifications are given the frequency characteristics of the low pass filter (LPF) and equalizer (EQ) as a graph. The combination of these must meet within 1 dB below 7 MHz, and it is recommended to meet it up to 10 MHz. Also, group delay variation for frequencies </= 6.5 MHz must be >/= ±3 ns, and gain at 5.0 MHz minus gain at 0 Hz must be 3.2 ±0.3 dB. For the LPF, it gives an example implementation to achieve these characteristics as a 6th order Bessel filter with a cutoff frequency fc (-3dB) = 8.2 MHz, and an example for the EQ is a three-tap transversal filter.

The OData function implements the 6th order Bessel filter as a FIR filter to provide the low-pass filter capability. The number of coefficients of the FIR depends on the ratio between the cutoff frequency fc and the sample rate fs. For a 1x DVD with an fc of 8.2 MHz, sampled at 500 MS/s, approximately 220 taps are required. Sampling at 1 GS/s is about twice that. Ideally, the sampling rate should be 10 to 20 times the clock rate. For a 1x DVD with a clock period of 37 ns, the sample rate should be 500 MS/s.

The three-tap equalization filter (EQ) is applied to the data after it has been low-pass filtered. The three samples input to the EQ are not adjacent; they are at 0 and ±2T, where T is a 1/channel bit rate.

Because the spacing in DSO samples depends on data rate and sample rate, T is likely to be a non-integer number of samples. In this case, interpolation is used to find the values at -2T and +2T.

Slicer

The Slicer is a 1st order integrating slicer with a programmable closed loop bandwidth (e.g., 5 kHz for 1x DVD as specified in DVD-R Annex G and DVD Annex H). Besides producing the sliced data, the slicer can output the difference of the input signal and the slicer threshold level. The slicer threshold will be determined by an exponential average of data samples computed as:

New thresh = (n - 1)/n * old thresh + 1/n * new data

Where *n* is chosen to meet the bandwidth requirement at the current sample rate.

Parameter	Description	Possible Values
Cutoff frequency	Cutoff frequency for the Equalizer (low-pass filter)	10 kHz to 800 MHz
Boost	Boost for the Equalizer	0 to 20 dB
Slicer BW	Controls the bandwidth of the Slicer	1 to 200 kHz
PLL BW	Controls the bandwidth of the PLL (used for clock recovery only)	1 to 200 kHz
Gate	Optionally, you can specify a channel that will be used to gate the input signal. When specified, you must also specify the polarity of the gate (i.e., process when low or high).	None, C1, … C4, M1, M4, other traces High or Low

Notes on ODATA Math Function

The ORDATA math function can accept as input either unequalized or already equalized data, and produce:

- Equalized: If the input data is not already equalized, the instrument applies equalization using the Filter Cutoff and Boost settings from the "Equalizer and PLL" dialog. The result is low-pass filtered such that -3dB frequency, without boost, is the filter cutoff frequency, and has high frequency boost applied such that the specified boost is reached at about 61% of the filter cutoff frequency. The default values for cutoff frequency (8.2 MHz) and boost (3.2 dB) are the correct settings for 1x DVD and DVD-R.
- Leveled: The data is fed to a 1st order integrating low-pass filter whose bandwidth is set by Slicer BW in the "Equalizer and PLL" dialog. The default slicer bandwidth (5.0 kHz) is correct for 1x DVD and DVD-R. The output of this filter is subtracted from the waveform to move the correct slice level to zero. Leveled output may be used for pit width measurements, etc. The correct level for the parameter threshold is always zero volts when it is used on leveled data.
- Extract Clk: The sliced data is sent through a PLL emulation, and the output is the PLL's VCO output. This uses the PLL BW setting in the "Equalizer and PLL" dialog. The default PLL Bandwidth (9 kHz) is the correct setting for 1x DVD and DVD-R. The VCO's starting frequency and phase are preset to attempt to start the PLL in a "locked" condition on each sweep.

This appendix contains more information about each of these operations, including known limits on their operations. Extracting the clock from the data has the most dependencies.

Equalized

Equalization can be applied if three conditions are met:

- We can make the low-pass filter.
- We can apply boost.
- The waveform is large enough to still have valid points after the filtering.

A warning message is displayed if any of these conditions is not met. If one of the following warning messages appears, the waveform is NOT equalized:

- "LP fc low & sample rate too high, can't LP filter": The number of coefficients needed for the finite impulse response (FIR) low-pass filter exceeded the maximum number supported. The maximum is adequate for 1x DVD at 16 GS/s, which means the maximum ratio of sample rate to cutoff frequency is 16e9/8.2e6 =

1951.22. This is far above the maximum it is reasonable to use. See the note on computation time under "Operational Notes" below.

- "Acquisition too small to apply EQ filters": The valid region of the waveform is reduced by EQ spacing on each side. This error message means that the result would then have no valid points.
- "LP fc low & sample rate too high, can't EQ filter": This message is shown if current EQ spacing is greater than 8191 samples, an implementation restriction. The EQ spacing is set to correspond to 2T, assuming that the cutoff frequency is correctly set; it is calculated from the cutoff frequency as follows:

EQ spacing in samples = 2.0/(fc * 26.16/8.2) * sample interval

Operational Notes

1. Even if the input data is already equalized, it is often helpful to tell the ODATA function that it is not, but set the boost to zero. This greatly reduces noise. White noise has power per Hz of bandwidth, and reducing the scope's bandwidth to around 8.2 MHz gets rid of 99% of white noise.
2. Applying high-frequency boost makes short pulses larger and has less effect on longer pulses. The correct boost should not greatly increase the signal's overall amplitude.
3. The output of the equalization is not delayed, as it would be by an analog filter. We compensate for the known delay through the digital filter and replace each input point with the corresponding equalized point.
4. The FIR LP filter plus 3.2 dB boost from the three-tap EQ filter produces the transfer function shown in the next figure when the FIR fc is set to 8.2 MHz. The highest peak is 20 log (dB) magnitude. The bowed trace below it is the real component of the TF. The flat line at zero is the imaginary component of the TF. It is zero indicating that there is no delay at all from input to output.
5. The computation time for the low-pass filter is generally longer than the time required for the sum of the rest of the computations done by the ODATA math function. This is because the low-pass filter is a finite impulse response filter (emulating the shape of a 6th order Bessel filter). It can require hundreds of multiplies-and-adds per sample in the waveform. The higher the sample rate relative to the bit time, T, the longer the FIR is. It is adequate to sample at 10 to 20 times the channel bit time, T. For 1x DVD, T is 26.16 MHz. Twenty times that is 523 MHz, so 500 MS/s is a good sample rate.

6. The three-tap EQ filter uses as input the point to be replaced and the points 2T away on each side. Since 2T may not correspond to an integer number of scope samples, linear interpolation between scope samples is used to get the values at exactly 2T away on each side.

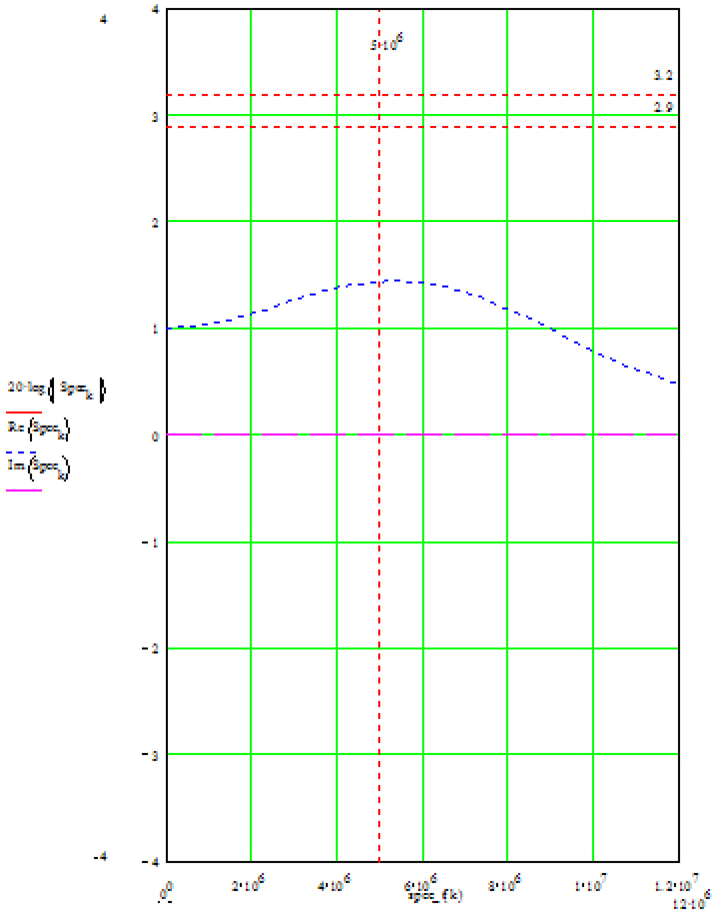


Figure 5-10. Simulation result showing transfer function of the digital low-pass filter and 3-tap EQ (boost) filter, set to 8.2 MHz cutoff frequency and 3.2 dB boost.

Leveled

There are no additional conditions to produce leveled data. The threshold is calculated and subtracted even if the equalization could not be applied for the reasons previously explained.

Sliced

The slicer uses a fixed hysteresis around zero, which corresponds to half a division (+ and - 1/4 division) when not vertically zoomed. (Remember that the slicer works on Leveled data, so zero is the correct threshold; the dynamically computed threshold has already been subtracted.) This means a peak must cross zero and exceed it by a quarter of a division or it will be ignored, as if it were noise after the previous crossing. A signal that is four divisions high (half of full scale) should have no problem meeting this requirement.

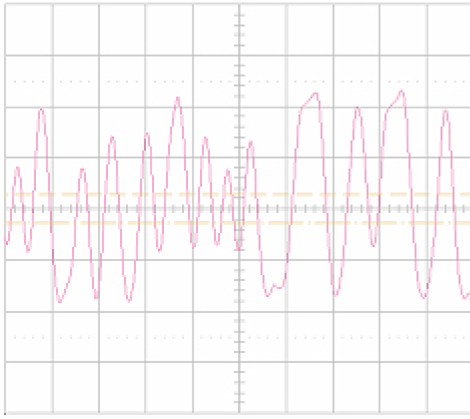


Figure 5-11. Leveled DVD data, with cursors showing the approximate hysteresis requirements for Sliced data.

The slicer produces waveforms that are exactly five divisions high. Each edge has two samples that are between the high and low levels and are positioned 2.5 divisions apart such that the zero cross time of the sliced output edge is the same as the zero cross time of the leveled data.

Extract CLK

It is usually not possible to get data and clock signals correctly aligned from an optical drive to visualize how the data edges align with the clock; in some cases, the clock may not be available at all. This function produces a clock waveform from the data by passing it through a software PLL. This output may be overlapped on the display with Leveled or Sliced output on another trace; and it can be used for measurements of the clock frequency. If the JTA option is present, a JitterTrack of Frequency of the extracted clock may give interesting insight into timing variation in the input signal.

The only user-set parameter for clock extraction is the "PLL BW" setting on the "Equalizer and PLL” dialog. The PLL Bandwidth is the unity gain intercept of the open-loop transfer function of the PLL. The closed loop -3 dB frequency is approximately 1.274 time that. The loop filter meets the specification shown in Annex H of the DVD Physical specifications (or Annex G of the DVD-R Physical specifications). For 1x DVD the PLL BW should be set to 9 kHz. In that case the software PLL has this closed-loop response:

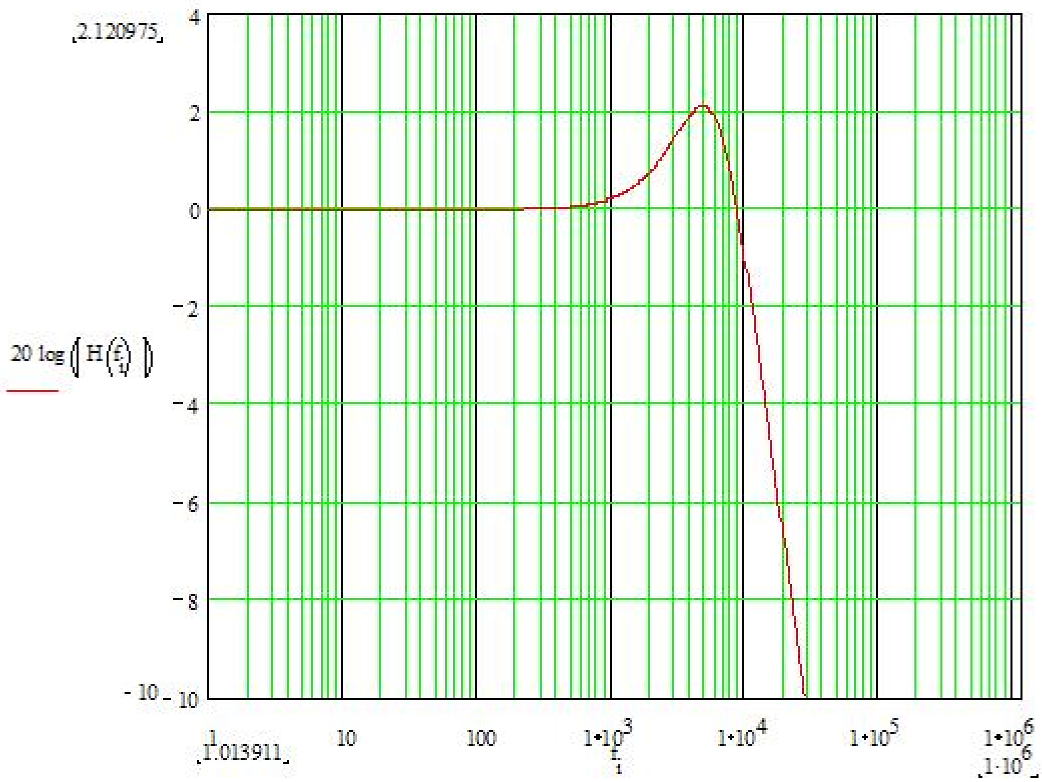


Figure 5-12. PLL closed-loop transfer function when "PLL BW" is set to 9 kHz.

The bandwidth of any PLL is a trade-off between jitter (phase noise) and desirable properties like a wide locking range and fast tracking. The "lock range" is the maximum frequency step for which the PLL can acquire lock without slipping a cycle. If we set up the VCO to start at other than the correct frequency (which corresponds to a frequency step), the PLL must change frequency to match the data. With PLL BW set to 9 kHz, the lock range is only about 25 kHz, slightly less than 0.1% of the expected clock frequency. The pull-in range is much broader but the pull-in time can be quite long. If we start the VCO just 0.4% away from the correct frequency, it would take hundreds of microseconds for the PLL to lock.

Since the acquired data may be a millisecond or less in duration, extracting the clock depends critically on the scope's ability to determine T (1/clock frequency) from the data and on starting the PLL's VCO at that frequency and at about the right phase. When it can do that, the VCO starts up locked and does not have to settle noticeably. If it cannot find the frequency, the warning message: "ORDATA VCO start freq is 3.19*LP fc, didn't find it" will be displayed. As the message says, if the scope cannot find the frequency, it starts the VCO at 3.19024 * LP fc. That ratio is 26.16/8.2 (to six significant digits). That is correct for DVD according to the specification; however, it may not be within 0.1% for a real drive. Experience shows that drives read a couple of percent fast.

To make the clock extraction successful, the scope must be successful in finding the starting frequency from the data. Here are some things you should do to make this successful:

1. Capture as clean a signal as possible. Remember that a passive probe is 10 MΩ resistive only at low frequencies and, therefore, may significantly load a high-speed signal. A passive probe's response will roll off well below the scope's DC 50 Ω bandwidth. Consider using a differential probe such as the AP033 or AP034, or an FET probe such as the AP020. Remember to attach the ground lead.

2. Equalize properly. If the signal you are probing is already equalized but not very clean, you can tell ODATA that it is RF anyway and set the boost to zero. That way the data will be low-pass filtered, which greatly reduces noise. If you don't equalize when you need to, or if you apply boost to an already equalized signal, the scope will probably not be able to determine the starting VCO frequency from the data, you will see the warning described above, and the extracted clock may not be good.

3. Sample at about 20 times the expected clock frequency. If you sample closer to 10 times the clock or below that, the extraction algorithm may not be able to correctly separate the peaks in the width distribution to determine the frequency at which to start the PLL. If you sample much more than 20 times the clock, the widths (in samples)

may be too spread out from detectable peaks in the distribution. (See ***How the Starting VCO Frequency is Determined*** for more details.) Example: CD data has T = 231 ns, about 4.33 MHz. We can extract the clock from CD data at 100 MS/s (23x) and 200 MS/s (46x) or 250 MS/s (58x). At 50 MS/s (11.5x) and at 500 MS/s (115x), it sometimes does not find the right starting frequency. Another example: DVD has T = 1/26.16 MHz, about 38.2 ns. We can extract the clock from DVD data at 500 MS/s (19x), 1 GS/s (38x), and 2 GS/s (76x). At 250 MS/s (9.5x) and at 4 GS/s (153x), it sometimes does not find the right starting frequency.

Following are some interesting pictures to show what can be handled:

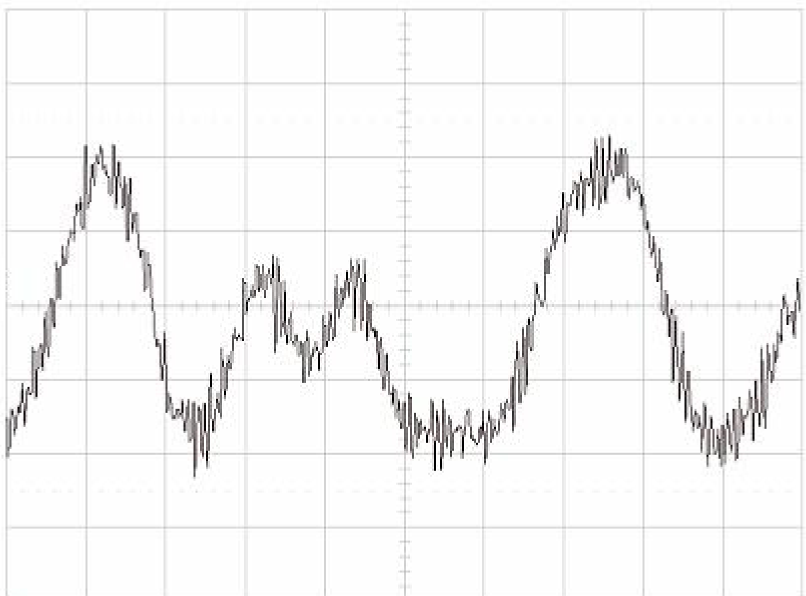


Figure 5-13. A small section of a 1 ms long noisy DVD waveform. Acquired with an ungrounded AP020 probe at 500 MS/s.

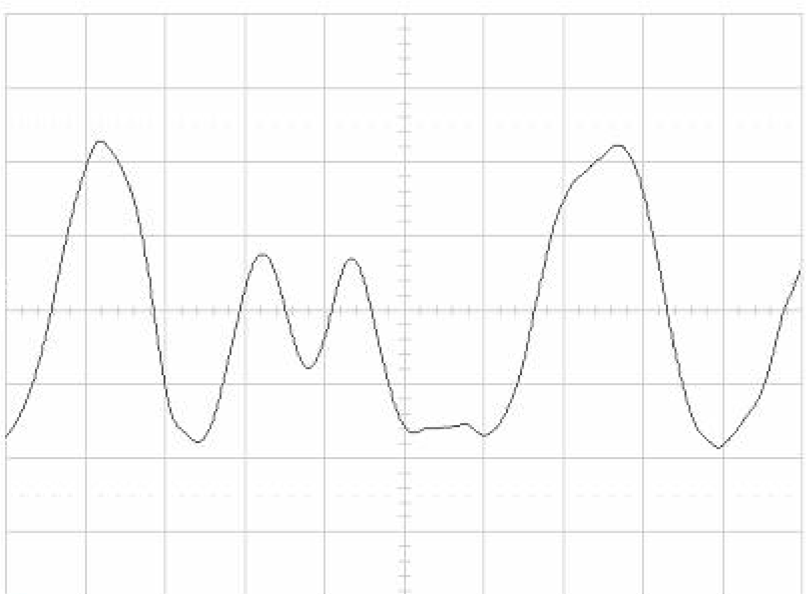
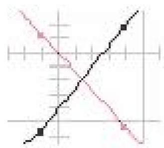
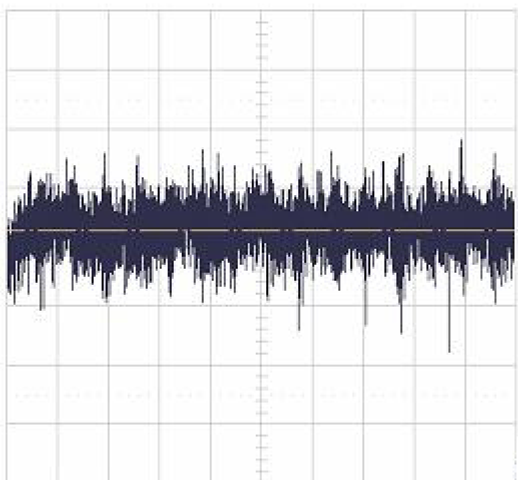


Figure 5-14. Same piece of the same signal, equalized and leveled.

It should be mentioned that the extracted clock output is also exactly five divisions high (without vertical zoom), and its edges are linear from -0.1 to +0.1 * T and from +0.4 to +0.6 * T. Therefore, if there are 20 samples per T, each edge of the extracted clock signal has four samples between the top and base. These samples are placed proportional to phase, so that the edge crosses 0 at exactly 0 and 180 degrees VCO phase.

The phase steering target for the VCO is, roughly speaking, that the data transitions should happen on the falling edge of the VCO output. To be more precise, we steer such that the VCO phase will be 180 degrees at the sample where a zero crossing in the data is detected. Because the software VCO works on a sample-by-sample basis, there is, on average, a half sample delay from the VCO falling edge's zero cross to the data zero cross. At 20 samples per T, this half sample error is 2.5% of T, not noticeable without zooming in. At 10 samples per T, it is 5% of T. The following small figure shows part of a zoom on a rising data edge and a falling clock edge sampled at 500 MS/s (2 ns per sample). The horizontal scale is 2 ns per division. The samples are bold. Note that the data crosses zero at 1 ns after the falling edge of the VCO output crosses zero. This is the expected result.

	The signal used is a 4.36 MHz square wave, which has a transition every 3T when 1/T = 26.16 MHz. During the first 50 µs or so, the phase settles in from initial startup. After that all the zero crossings are half a sample apart, as shown in this picture.
	JitterTrack of Frequency (requires JTA option) of the extracted clock. The startup frequency was correct to within a few kHz, and the PLL did not slip. It is possible that the starting frequency was precise but the starting phase was not; the effect would be the same. JitterTrack shows frequency as a function of time. The vertical scale is 20 kHz per division; the cursor is positioned at 27.107 MHz. The horizontal scale is 0.1 ms per division.

How the Starting VCO Frequency & Phase Are Determined

The PLL's VCO is started at a frequency of 1/T. Due to the accuracy required, T is determined in two steps. The first step produces an estimate of T starting with very few assumptions. The second step starts with the estimate of T and refines it. The information used in both steps is the sample at which a transition (through zero) occurred in the sliced data, for up to the first 2000 edges. If the source waveform has less than 2000 edges, the accuracy of this procedure may be reduced.

If the source waveform has less than 50 edges, the instrument does not even attempt to estimate T. The PLL will start at 3.19024 * LP fc. Because of the low bandwidth of the PLL, it does not make much sense to try to extract the clock from a very short waveform; the PLL will not have time to react.

The first step calculates the width of the first (up to) 2000 pulses, sorts the widths, and finds the first three peaks in the distribution of widths. The distribution is "smoothed" by a five-bin wide boxcar filter to prevent small local events from misleading the peak detection. This is the primary reason why the signal must be over-sampled by greater than 10x. The distribution of widths is similar to a histogram of pwid (pit width) on "leveled" output of the ODATA function, using a threshold of 0.0 mV and measuring All widths. The spacing between the peaks is approximately T, close enough to determine the lowest nT. The instrument calculates the estimate of T from the means of the first three peaks, which are assumed to be lowest n, lowest n + 1, and lowest n + 2 (i.e., 3T, 4T, and 5T). This estimate is generally good to better than 1%.

The second step uses the location of the first (up to 2000) transitions, in order. It uses the estimate of T to calculate n between each pair of same-polarity edges. If the estimate is within 1%, we have at least 50% margin. A 50% margin occurs if a pair of same-polarity edges is 25T apart. On a good waveform, the count is likely to be exact. On a noisy or distorted waveform, it may be that some peaks are miscounted, but as long as some are long and some are short, the final total will be nearly correct. Finally, T is computed as:

(time at the last transition - time at the first transition)/(total n between them)

If there are 2000 edges, an average of 4T apart, the separation between first and last edge is 8000T. If our count of n is off by 1, that is a 0.0125% error. We can tolerate up to 7 counts error (0.0875%) before the PLL will not start locked. When the waveform is correctly equalized, this does not happen.

A highly asymmetric waveform will not have clean peaks in the distribution of its pulse widths, which also means that many of the pulses will be nearly (n + 0.5)T. On such a waveform, we may not be able to determine T. The possible reasons for failing to determine T (and therefore the VCO start frequency) are:

- Less than 50 edges in the waveform.
- Could not distinguish the first three peaks in the distribution of widths.

As mentioned above, you should sample at about 20x to 50x the clock frequency to make clock extraction work reliably.

An attempt is made to start the VCO not only at the correct frequency but also at the correct phase. The phase is pre-set such that the first edge in the waveform will occur on a falling edge of the VCO output. The first edge is just as likely to be out of place as any other edge in the waveform, of course. If the VCO starts significantly at the wrong phase it will either slow down or speed up for a short while until it gets to the right phase. A JitterTrack shows this clearly. On a 4x DVD waveform we captured, which just happens to have a significantly out-of-place first edge, the frequency is disturbed slightly for the first 15 µs or so; the frequency shift during this time is very small, on the order of 0.1%, as the phase adjusts.

Introduction to 8B/10B

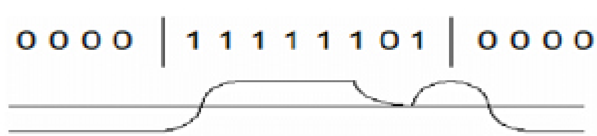
The new LeCroy Serial Bus Analyzer ("SBA" option key) features:

- Powerful search capability allows captured waveforms to be searched for a user-defined sequence of symbols.
- Multi-lane analysis decoding of up to four simultaneously captured lanes.
- Waveform can be analyzed by other protocol analysis software packages (future capability)
- When combined with the high bandwidth capabilities of the SDA11000, this software option permits symbolic decoding of data streams up to 10 Gb/s.

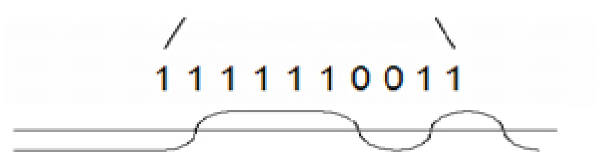
Description of Encoding and Decoding

In real-world transmission systems, there are several techniques to prevent long strings of zeros or ones from occurring in the data stream. 8B/10B encoding and decoding is one of those techniques.

Here is a very simplified picture of what might happen when a long string of ones has a single embedded zero in it. The waveform that results from this data stream may not make it to the threshold crossing in time to be sampled.

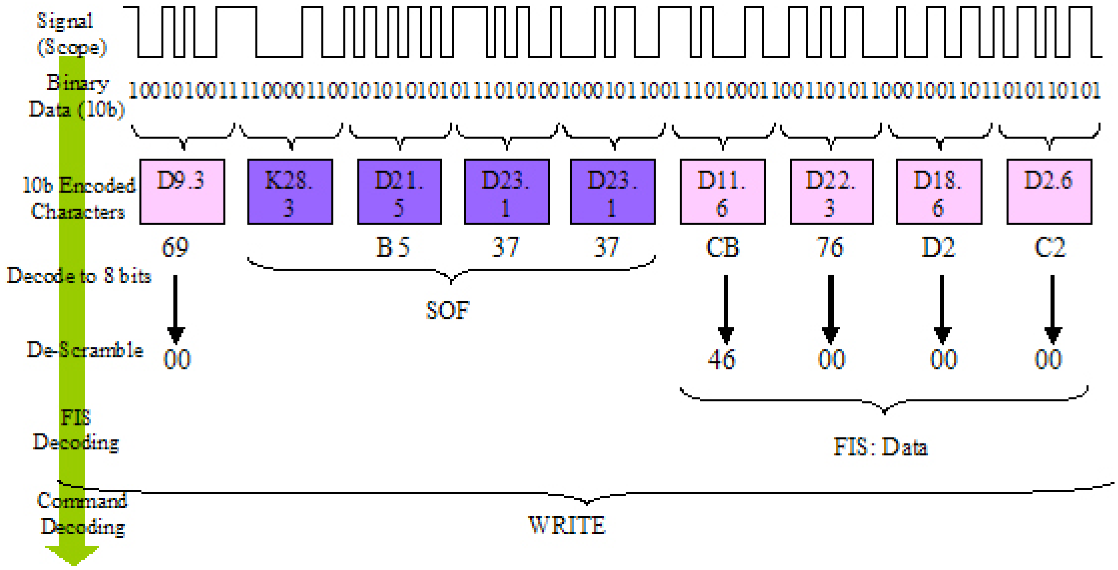


When 8B/10B encoding is applied, every 8 bits are mapped into 10 bits so that long and short runs of zeros and/or ones are avoided. In this case the new waveform might look something like this:



Running Disparity

The Running Disparity (RD) monitors the percentage of "1s" present in the message relative to the percentage of "0s". In order to “compensate” for an increasing number of 1s relative to 0s, the system keeps track of the Running Disparity, such that when the running disparity is “positive” (high percentage of 1s at the transmitter), the following character being transmitted will be selected from the symbol table with a “0” or negative Running Disparity.



As the above diagram shows, decoding a digital waveform requires the capture of the waveform in a real-time or sampling oscilloscope, followed by parsing of the digital data in 10-bit increments until certain characters (symbols) are detected.

For example, SAS utilizes K28.3 and K28.5 as primitive delimiters

- K28.3: 1st character of all primitives except ALIGN
- K28.5: 1st character of ALIGN primitive

Name	abcdei fghj output		Description
	Current rd-	Current rd+	
K28.3	001111 0011	110000 1100	Occurs only at Byte 0 of all primitives except for the ALIGNₚ primitive
K28.5	001111 1010	110000 0101	Occurs only at Byte 0 of the ALIGNₚ primitive

Here are the rest of the primitives defined in the specification:

Primitive name	Byte 3 contents	Byte 2 contents	Byte 1 contents	Byte 0 contents
ALIGNₚ	D27.3	D10.2	D10.2.	K28.5
CONTₚ	D25.4	D25.4	D10.5	K28.3
DMATₚ	D22.1	D22.1	D21.5	K28.3
EOFₚ	D21.6	D21.6	D21.5	K28.3
HOLDₚ	D21.6	D21.6	D10.5	K28.3
HOLDAₚ	D21.4	D21.4	D10.5	K28.3
PMACKₚ	D21.4	D21.4	D21.4	K28.3
PMNAKₚ	D21.7	D21.7	D21.4	K28.3
PMREQ_Pₚ	D23.0	D23.0	D21.5	K28.3
PMREQ_Sₚ	D21.3	D21.3	D21.4	K28.3
R_ERRₚ	D22.2	D22.2	D21.5	K28.3
R_IPₚ	D21.2	D21.2	D21.5	K28.3
R_OKₚ	D21.1	D21.1	D21.5	K28.3
R_RDYₚ	D10.2	D10.2	D21.4	K28.3
SOFₚ	D23.1	D23.1	D21.5	K28.3
SYNCₚ	D21.5	D21.5	D21.4	K28.3
WTRMₚ	D24.2	D24.2	D21.5	K28.3
X_RDYₚ	D23.2	D23.2	D21.5	K28.3

For PCI Express, on the other hand, a compliant device issues a “Polling” or “Compliance” pattern in each of its lanes to indicate that the device is operational. The polling pattern is composed of the following sequence of symbols:

Symbol	K28.5	D21.5	K28.5	D10.2
Current Disparity	0	1	1	0
Pattern	0011111010	1010101010	1100000101	0101010101

Significance of symbols:

K28.5-	Comma when disparity is negative, specifically: "0011111010"
K28.5+	Comma when disparity is positive, specifically: "1100000101"
D21.5	Out of phase data character, specifically: "1010101010"
D10.2	Out of phase data character, specifically: "0101010101"
D	Delay Character: K28.5

Recommended System Configuration

- SDA, DDA, or WaveMaster 4000A, 4020, 6000A, 6020, or SDA11000 real-time digital storage oscilloscope.
- X-Stream firmware version 4.7 or newer
- At least one Serial Data Standards package (for example, SDA-SATA or SDA-PCIE-G2)

Option Key Installation

When ordered as an option for a new instrument, no installation is necessary. Installation is required, however, when the option is ordered after the oscilloscope is purchased. An option key will be issued at the time the option is purchased.

To enter the option key code,

1. Touch **Utilities** in the menu bar, then Utilities **Setup…** in the drop-down menu.
2. Select the **Options** tab from the utilities dialog.
3. In the Options dialog, touch the **Add Key** button and enter the option key in the dialog box, using the on-screen keyboard.

Figure 5-15. Entering the option key code for the SBA software option

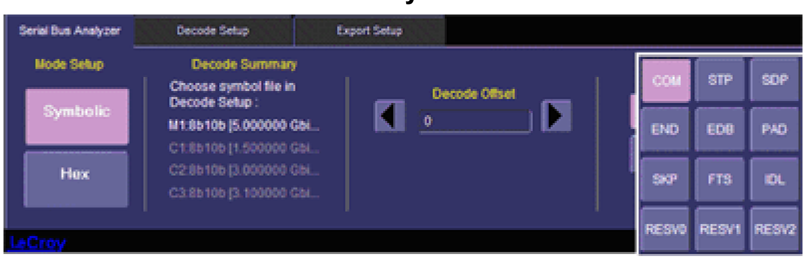
Loading a Waveform into the Serial Bus Analyzer – Decode Setup

1. Select **Analysis** from the SDA menu, then **Serial Bus (SBA)…** from the drop-down menu.
2. Touch the **Decode Setup** tab.

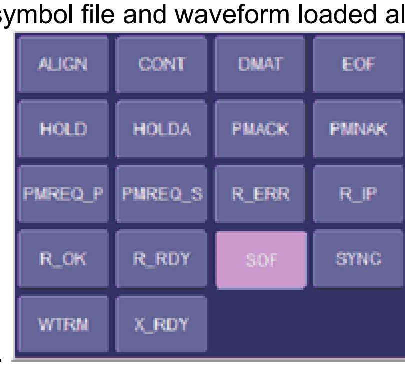
3. Select a signal **Source**: a Channel, Memory location, or Math trace. Up to four individual traces can be loaded.
4. Enter the bit rate for the signal under test. Most common serial data standards have between 1.5 and 3 Gb/s bit rates.
   - SATA Gen1: 1.5 Gb/s
   - SATA Gen2: 30 Gb/s
   - PCI Express Gen 1: 2.5 Gb/s
   - PCI Express Gen 2: 5 Gb/s
5. Select a **Symbol File**. This text (*.txt) file contains the symbol sequence for each primitive (command) applicable to the waveform under study. There is a primitive file for each serial data standard. The **Protocol** field will be updated with the symbol file loaded. When no Symbol File has been selected, the generic “8b10b” name will be displayed.

Searching for a Symbol or Hex Equivalent

1. Select **Analysis** from the SDA menu, then **Serial Bus (SBA)…**
2. Touch the **Serial Bus Analyzer** tab



3. Select **Symbolic** or **Hex** for search mode setup.
4. The "Decode Summary" displays the symbol file and waveform loaded along with the Primitive type, depending on the serial data standard.



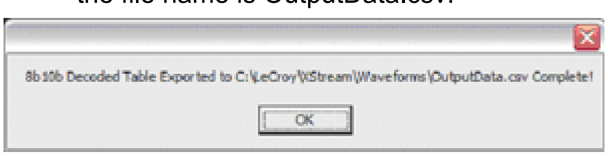
5. Choosing a primitive (for example, ALIGN), yields the following zoomed (Z1) display:

6. The Z1 display is zoomable independent of the source file. Whenever Z1 is chosen, the symbol search applies to the source and symbol files where Z1 is defined.
7. In order to switch to a different source file, touch the **SBA Setup…** button in the Z1 dialog and configure the source and symbol files.

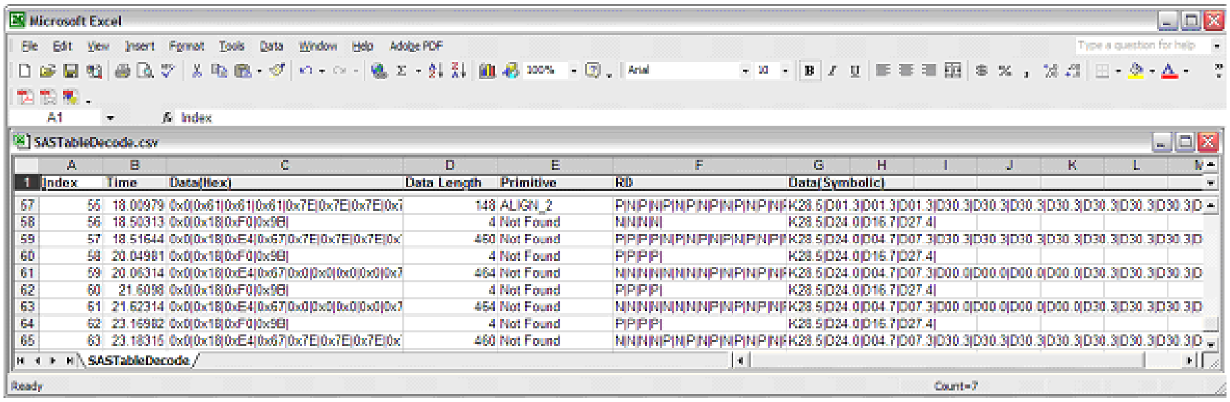
Creating a Symbol-decoded Output File from a Waveform -- Export Setup

1. From the menu bar, select **Analysis**, then **Serial Bus (SBA)…** from the drop-down menu.
2. Touch the **Export Setup** tab. The following menu is displayed:

3. Select an **Export File**. By default the output file will be created in D:\Applications\SBA directory. Or, Browse to create a *csv (comma-separated values) file by entering a name using the pop-up keyboard.
4. Select a **Script File**. The default script file is called Exportdata.lss, and it is located in the D:\Applications\SBA directory.
5. Touch the **Export** Button. The desired output file is created in the same default directory. In this example, the file name is OutputData.csv:



6. The output file can be opened in Microsoft Excel and will have the following format:



7. The fields contain Index, Time, Hex Data, Length (number of bytes), Name of Primitive, Running Disparity count (P=positive, N=Negative, Z= Neutral) and the Symbolic (encoded) data.

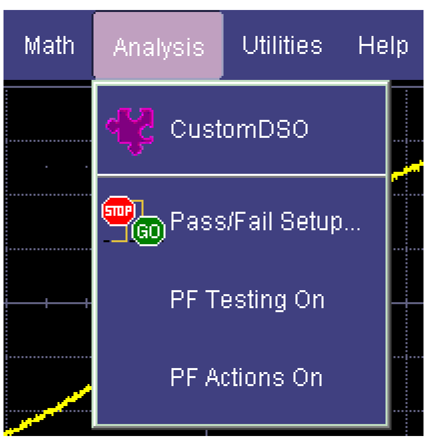
CustomDSO Overview

CustomDSO, in its Basic mode, allows you to create DSO setups that can be called by the touch of a single button. The recalled setups can themselves include calls to other setups. A very simple example would be a toggle between two setups. Rings of three or more setups are possible, as are trees, or any other topology that you need. Basic mode also allows you to recall VBScripts that can set up all or part of the scope and do many other things.

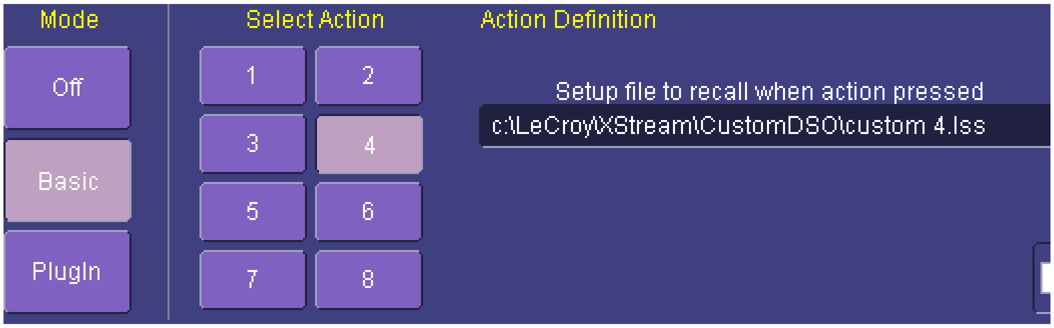
Another more powerful feature is the PlugIn, which allows you to add your own ActiveX controls to a setup. These controls are powered by routines written in Visual Basic. With ActiveX controls you can create your own user interfaces to suit your own preferences. A large number of interactive devices are available: button, checkbox, radio button, list box, picture box, and common dialogue box.

Invoking CustomDSO

CustomDSO can be invoked from the **Analysis** drop-down menu:



If CustomDSO is already in Basic mode, the following dialog will be displayed:

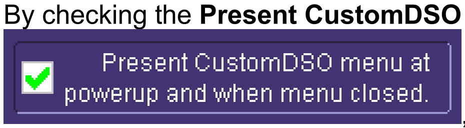


CustomDSO Basic Mode

The **Basic** CustomDSO mode offers eight Action buttons, each of which can call a different setup when touched. The "Action Definition" field is used to enter a CustomDSO setup file name by means of the pop-up keyboard. The **Test Basic Mode** dialog is used to execute the setups loaded under Action Definition.

Checking the **Execute scripts asynchronously** checkbox 

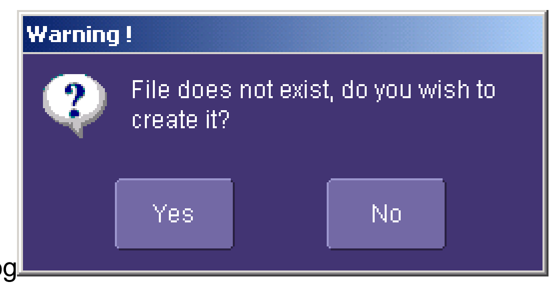
allows scripts to run in a separate process. Thus, the scope application can perform another function while the script is being executed.

By checking the **Present CustomDSO menu at powerup and when menu closed** checkbox 

, the eight CustomDSO buttons will continue to be available at the bottom of the screen after you close the CustomDSO dialog. Furthermore, they will appear automatically each time the DSO is powered up.

Editing a CustomDSO Setup File

If the file does not exist, touch the **Edit** button 

; a dialog 

will appear for you to create the file. If the file does already exist, the **Edit** button enables you to modify it. The **Edit** button allows you to edit the file that is named in the **Setup file to recall** field, and not the file of the setup that the instrument is currently in, unless these happen to be the same.

In the example used here, three setup files were made, called CustomA.lss, CustomB.lss and CustomC.lss. Fragments from all three are shown below.

```
Set CustomDSO = XStreamDSO.CustomDSO
‘ CustomDSO Setup A.lss
CustomDSO.ActionScript1 = “c:\LeCroy\XStream\CustomDSO\A.lss”
CustomDSO.ActionEnable1 = False
CustomDSO.ActionScript1 = “c:\LeCroy\XStream\CustomDSO\B.lss”
CustomDSO.ActionEnable1 = True
CustomDSO.ActionScript1 = “c:\LeCroy\XStream\CustomDSO\C.lss”
CustomDSO.ActionEnable1 = True
CustomDSO.ActionScript1 = “c:\LeCroy\XStream\CustomDSO\A.lss”
```

```
1169 CustomDSO.ActionEnable1 = False

1160 Set CustomDSO = XStreamDSO.CustomDSO
1161 ' CustomDSO Setup B.lss
1162 CustomDSO.ActionScript1 = "c:\LeCroy\XStream\CustomDSO\A.lss"
1163 CustomDSO.ActionEnable1 = True
1164 CustomDSO.ActionScript1 = "c:\LeCroy\XStream\CustomDSO\B.lss"
1165 CustomDSO.ActionEnable1 = False
1166 CustomDSO.ActionScript1 = "c:\LeCroy\XStream\CustomDSO\C.lss"
1167 CustomDSO.ActionEnable1 = True
1168 CustomDSO.ActionScript1 = "c:\LeCroy\XStream\CustomDSO\A.lss"
1169 CustomDSO.ActionEnable1 = False

1160 Set CustomDSO = XStreamDSO.CustomDSO
1161 ' CustomDSO Setup C.lss
1162 CustomDSO.ActionScript1 = "c:\LeCroy\XStream\CustomDSO\A.lss"
1163 CustomDSO.ActionEnable1 = True
1164 CustomDSO.ActionScript1 = "c:\LeCroy\XStream\CustomDSO\B.lss"
1165 CustomDSO.ActionEnable1 = True
1166 CustomDSO.ActionScript1 = "c:\LeCroy\XStream\CustomDSO\C.lss"
1167 CustomDSO.ActionEnable1 = False
1168 CustomDSO.ActionScript1 = "c:\LeCroy\XStream\CustomDSO\A.lss"
1169 CustomDSO.ActionEnable1 = False
```

The text in green following a single quotation mark is a VBS comment and causes no action.

The text in red contains the path and name of the setup file associated with the numbered button. This setup will be called when the button is pressed.

The boolean (in blue) decides whether the action button will invoke the setup or remain inactive.

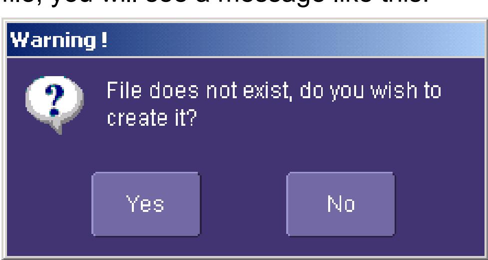
For example, in setup B, **A.lss** and **C.lss** can be invoked, but not B, which is already in place.

As you see from the line numbers in the program fragments, the setup files are rather long because they include all the information needed to set the DSO to the required state. But if you want to make a very short file that changes only a few variables (for example, the action button settings) you can make a file that includes only the relevant instructions. This usage assumes that the remainder of the DSO is already in the required state. This is an example of the complete compatibility of the instrument's software. The same commands can be used in setups, in scripts, or in remote control commands in external programs, whether resident in the instrument or in an external computer.

Creating a CustomDSO Setup File

If you touch the **Edit** button 

when the **Setup file to recall** field contains the name of a non-existent file, you will see a message like this:



If you then touch **Yes**, the DSO will display a file like this:

```
' XStreamDSO ConfigurationVBScript ...
' Created by CustomDSO ...
On Error Resume Next
set dso = CreateObject("LeCroy.XStreamDSO.1")
' dso.Display.GridMode = "Dual"
' dso.Acquisition.C1.VerScale = 0.1
' dso.Acquisition.Horizontal.HorScale = 1e-6
' dso.Acquisition.TriggerMode = "Auto"
```

You can add to this fragment any commands you need.

CustomDSO PlugIn Mode

This is the mode in which CustomDSO really shows its power. You can insert any ActiveX control or graph.

Creating a CustomDSO PlugIn

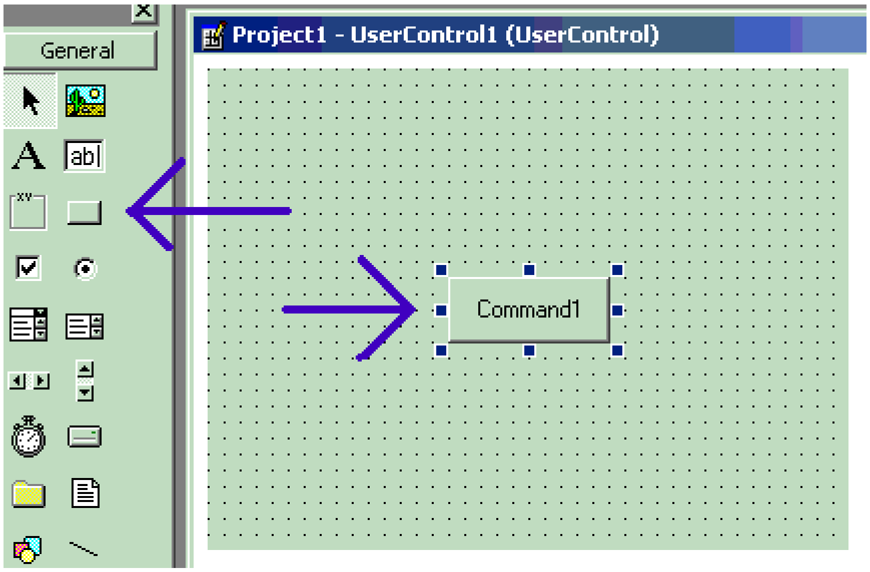
Follow these steps to create an example Visual Basic PlugIn:

1. Start a new VB project.

   Select **ActiveX Control** from the **New** tab.
2. Resize the control.

   A. In the Properties window set Width 11940.
   B. In the Properties window set Height 2475.
3. Place two buttons on the control.

   A. Double click on the command button at left of screen (left arrow below).
   B. Move and resize the resulting button as required, using the handles (right arrow below).



   C. Repeat for the second button.
   D. In the Properties window set the Name properties to **SingleButton** and **AutoButton**, respectively.
   E. Set the button Caption properties to **Single** and **Auto**, respectively
4. Create code for the buttons.

A. Double click on the **Single** button.
B. In the resulting code window, insert code to make the following subroutine:

```
Private Sub SingleButton_Click()
Dim app as Object
Set app = CreateObject("LeCroy.XStreamApplication")
app.Acquistion.TriggerMode = "Stopped"
End Sub
```

Double click on the **Auto** button.

In the resulting code window, insert code to make the following subroutine:

```
Private Sub AutoButton_Click()
Dim app as Object
Set app = CreateObject("LeCroy.XStreamApplication")
app.Acquistion.TriggerMode = "Auto"
End Sub
```

5. Test the Component in Internet Explorer. (This is an optional, but very useful step, because you can test your work without installing anything in the instrument.)

   A. Start the instrument.
   B. Click the **Run** button In Visual Basic.
   C. Click the **Stop** button in Visual Basic when you have finished.
6. Make the Project in Visual Basic.

   A. Click the **Stop** button in Visual Basic.
   B. Select Make Project1.ocx from the File menu.
7. Install the PlugIn in the instrument.

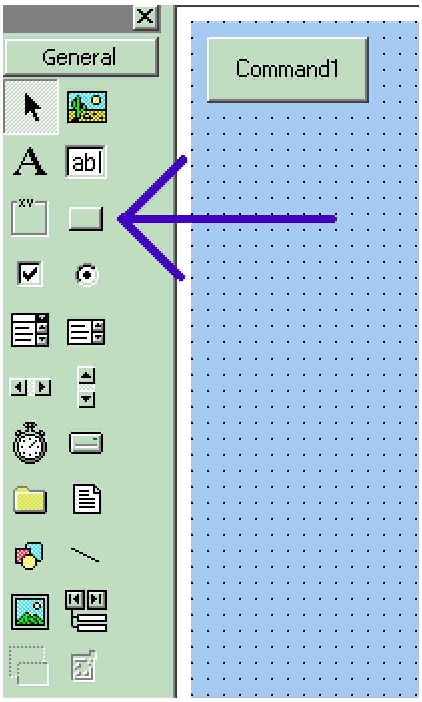
   A. Start the instrument.
   B. Select **ActiveDSO** in the Analysis Menu.
   C. Select **PlugIns** mode.
   D. Type "Project1.UserControl1" in the "COM ProgID of Plug-In" text box.
   E. Click the Install button under the text box.
8. Now Click the new **Auto** and **Single** buttons to see their effects.

Properties of the Control and Its Objects

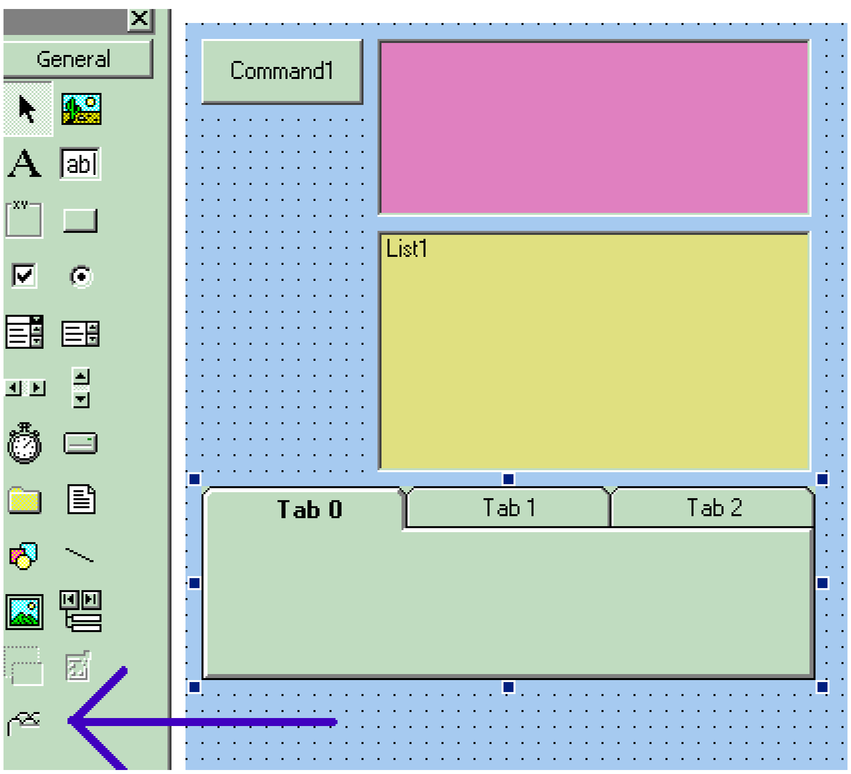
Using the **View Properties** button in Visual Basic, you can customize your PlugIn to your exact requirements. Among the most useful properties are the following: Height, Width, BackColor, Name, Caption.

You can gain access to the properties of your objects by Clicking View – Properties. Positions and sizes of objects can be changed from View – Object, by dragging the object or one of its handles.

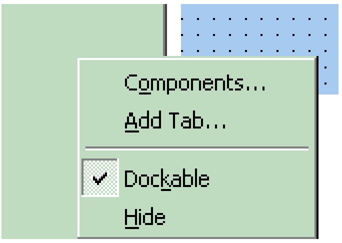
You can insert any available control into your plug-in. The basic control set is shown in a toolbar at the left of the screen in the picture below. Double click on any control to insert it into the plug-in. In the following example, a command button has just been inserted.



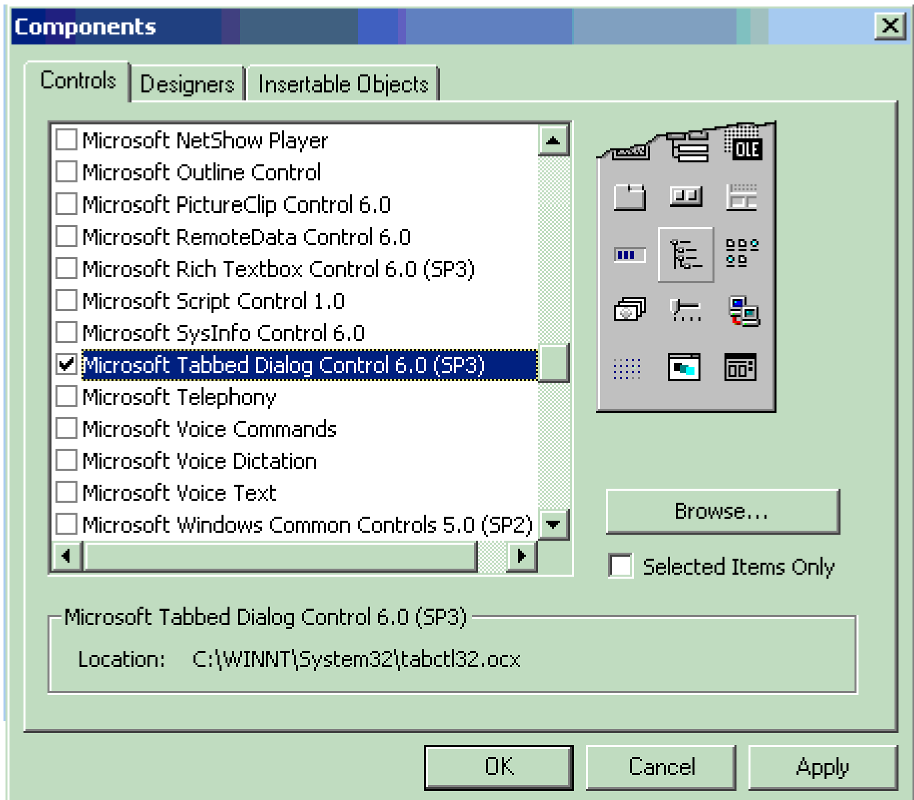
In the next example you can see a command button, a picture box, a list box and a Tabbed Dialog Control.



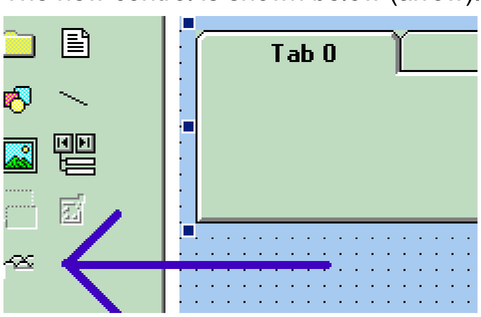
The Tabbed Control (arrow) is not in the basic tool box. To gain access to it, right click in the tool box at left (but not on an icon.) You will see this menu:



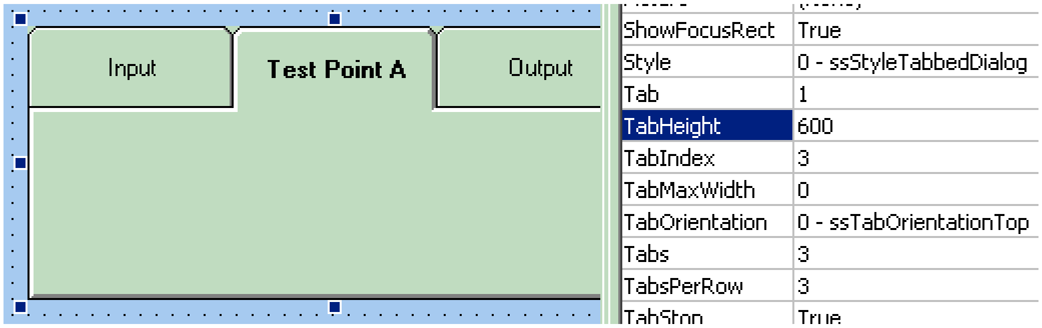
Now select the Microsoft Tabbed Control as shown below, and click on Apply. The control will be added into the toolbox at the left of the screen, where you can double click on it as usual.



The new control is shown below (arrow).

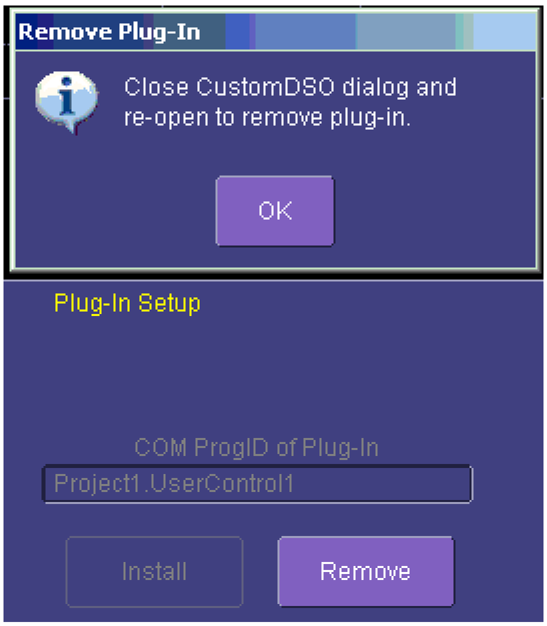


The system is very versatile, and you can place controls on the tabs of the Tabbed Control. Look in the properties window to see how you can customize your tabs, as illustrated below.



Removing a PlugIn

Remove a plug-in by clicking Remove in the PlugIn dialog, as follows:



Close the CustomDSO dialog, reopen, and the plug-in is gone.

Example 1

First Example PlugIn – Exchanging Two Traces on the Grids

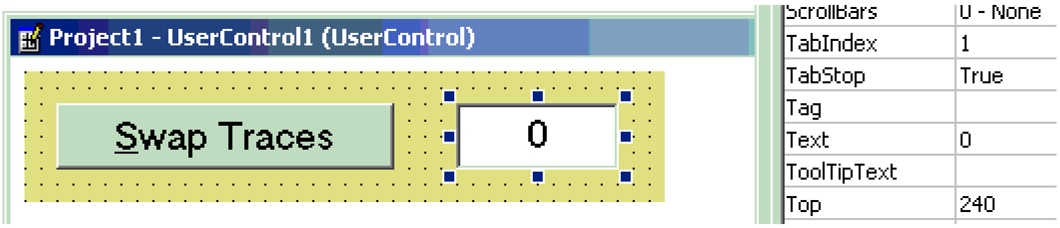
The example assumes that the instrument is in dual-grid mode, and that there are at least two visible traces. The routine looks for the visible traces, in the order C1 . . . C4, F1 . . . . FxThe number of math traces available depends on the software options loaded on your scope. See Specifications. , and it exchanges the first two it finds whenever the button is pressed. Note that arrays of objects can be constructed, allowing numerous objects to be accessed in simple loops.

```
Private Sub Command1_Click()

Dim wm As Object
Set wm = CreateObject("LeCroy.XStreamApplication")

Set acq = wm.Acquisition ' To save typing
Set mat = wm.Math        ' To save typing
```

```
Dim t(16) As Object
' Create an array of objects to allow looping.
Set t(1) = acq.C1 : Set t(2) = acq.C2
Set t(3) = acq.C3 : Set t(4) = acq.C4
Set t(5) = mat.F1 : Set t(6) = mat.F2
Set t(7) = mat.F3 : Set t(8) = mat.F4
Set t(9) = mat.F5 : Set t(10) = mat.F6
Set t(11) = mat.F7 : Set t(12) = mat.F8

Dim trace As Integer
trace = 0: views = 0
' Exchange the traces on the grids.
    Do
    trace = trace + 1
        If t(trace).View = "True" Then
        views = views + 1
            If t(trace).UseGrid = "YT1" Then
            t(trace).UseGrid = "YT2"
            Else
            t(trace).UseGrid = "YT1"
            End If
        End If
    Loop Until ((trace = 12) Or (views = 2))
' Show the parity of the last swap.
    If Command1.Caption = "Swap A" Then
    Command1.Caption = "Swap B"
    Else
    Command1.Caption = "Swap A"
    End If

Dim TextString As String
TextString = Text1.Text
Dim TextValue As Integer
TextValue = Val(TextString) + 1
TextString = Str(TextValue)
TextString = Trim(TextString)
Text1.Text = TextString

End Sub
```

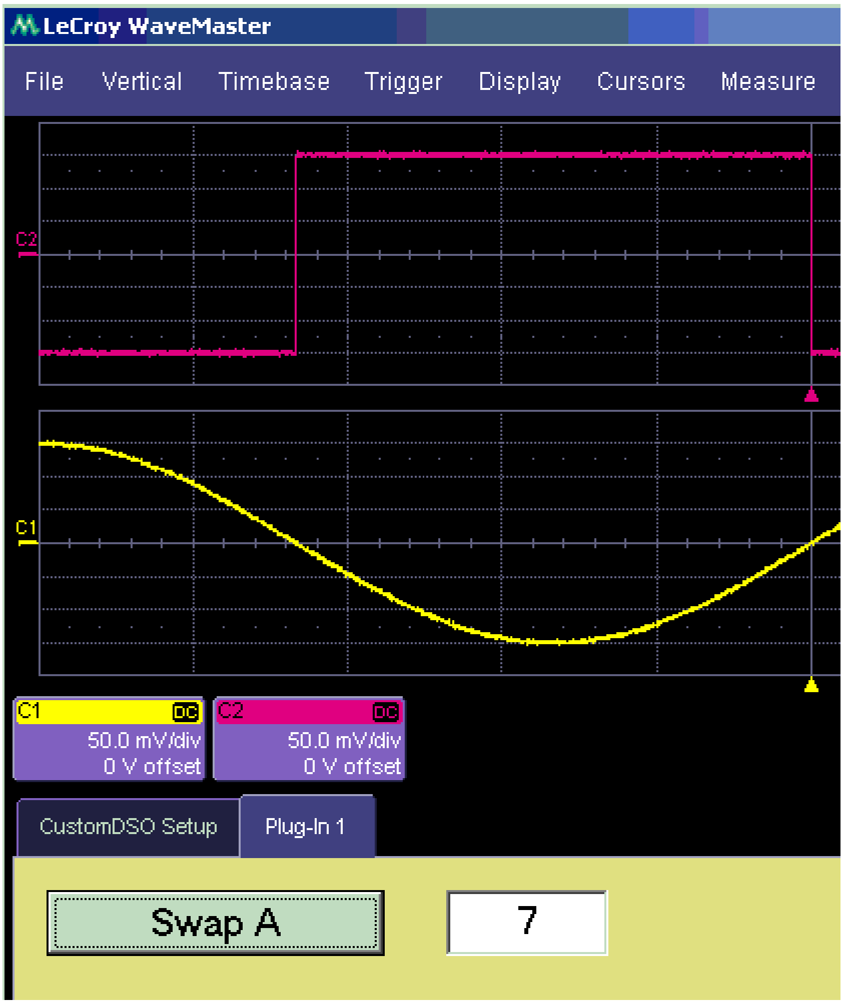
This routine exchanges the first two traces that it finds. You can make it exchange <u>all</u> the traces on a dual grid by changing the penultimate line to this - Loop Until trace = 12

The next figure shows the Visual Basic Screen just after the Text Box text has been set to “0” in the Properties Window, thus defining the initial value.



Here is the result after seven swaps. The counting method could be useful in any routine where numerous operations, such as triggers, have to be performed. In fact, the caption of the button could have been used to show the number of operations.

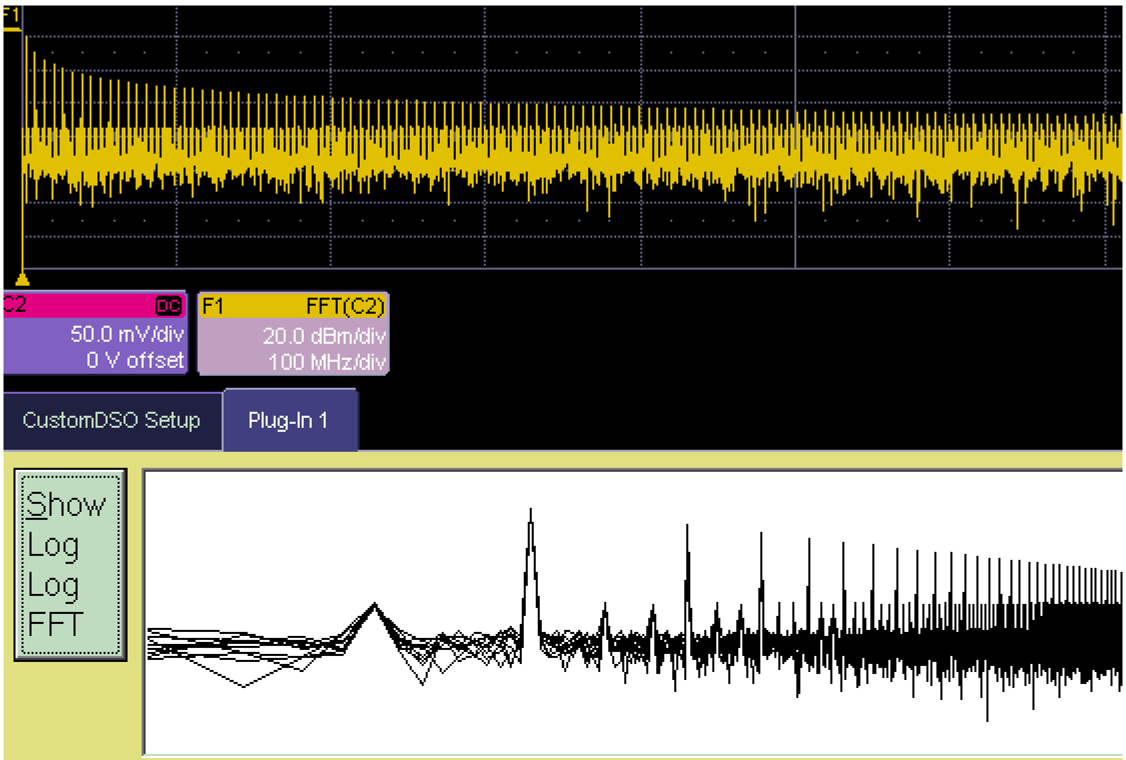
ActiveX offers a large range of standard controls, including list boxes for selection from a list, and picture boxes for drawing graphs and charts.

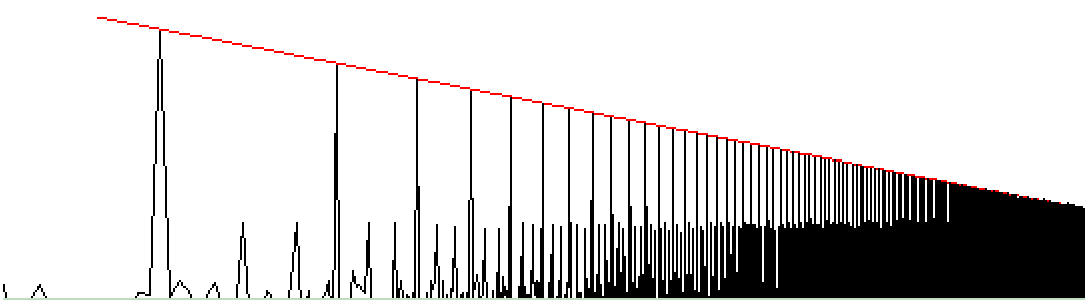


Example 2

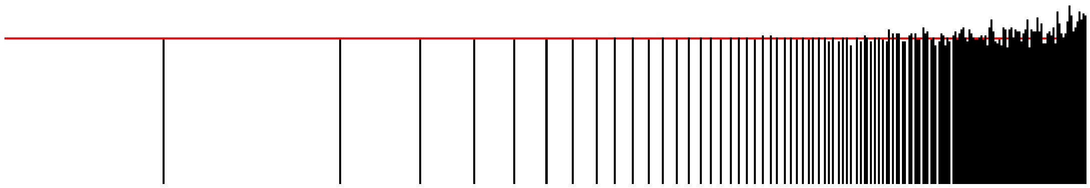
Second Example PlugIn – Log-Log FFT Plot

A frequent requirement is to plot a frequency spectrum on two logarithmic scales. The instrument provides a vertical scale, so CustomDSO has only to change the horizontal one. Here is an example. The first figure has been truncated on the right side.





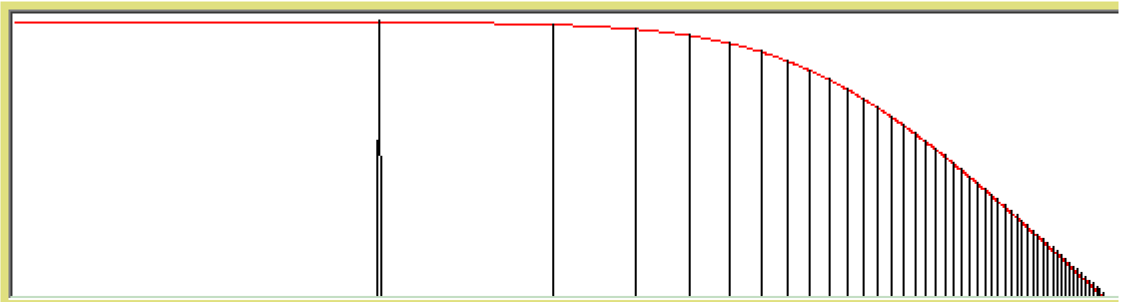
These examples were made with two different instrument setups: in the second, the FFT was zoomed vertically. The graph has a red line to represent the theoretical envelope for the peaks. This has great potential for testing the transmission characteristics of amplifiers and filters, since the output can be compared with a theoretical curve. Furthermore, if the output is divided by the curve, the result for a perfect DUT would be a horizontal line, which is easy to inspect. The example below has been magnified vertically by a factor of ten. The rise at the right side occurs because the signal is descending into the noise.



```
Private Sub Command1_Click()
'   Draw a DSO trace on a logarithmic horizontal scale.
Dim WM As Object
Set WM = CreateObject("LeCroy.XStreamApplication")

Dim Samples As Long
Samples = WM.Math.F1.Out.Result.Samples
Samples = Samples - 1 ' Make it a round number.
'   Calculate the horizontal scale.
LogSamples = Log(Samples)
XScale = Samples / LogSamples

'   Set the scale using DSO variables
Dim Top, Bot As Single
Top = WM.Math.F1.Out.Result.VerticalFrameStop
Bot = WM.Math.F1.Out.Result.VerticalFrameStart
Picture1.Scale (0, Top)-(Samples, Bot)

Dim Wave
Wave = WM.Math.F1.Out.Result.DataArray

Dim Black, White, Blue, Red As Long
Black = 0: White = &HFFFFFF
Blue = &HFF4444: Red = &HFF

'   Draw a theoretical curve for the peaks.
StartPoint = Top + 20#: EndPoint = -54.5
Picture1.Line (0, StartPoint)-(Samples, EndPoint), Red

'  Draw the plot with linear interpolation between points.
    For X = 1 To Samples
    LogX = XScale * Log(X): Y = Wave(X)
        If X > 1 Then
        Picture1.Line (LogX, Y)-(OldLogX, OldWave), Black
        End If
    OldLogX = LogX: OldWave = Y
```

```
    Next X

End Sub
```

Here is an example showing a simple one-pole roll-off compared to a curve.



Control Variables in CustomDSO

The simplest way to select variables for use in CustomDSO is to use LeCroy's X-Stream Browser.

DFP Filters Overview

The Need

In today's complex environment, data is frequently composed of a mixture of analog and digital components spread over a broad range of frequencies. In many applications, the relevant data is encoded or obscured. Capturing the right signals becomes a challenge. Engineers find it increasingly difficult to examine only those parts of the data they are interested in. Traditional (or even smart) oscilloscope triggering cannot always provide a satisfactory answer.

For example, servo motors from disk drives add a low frequency component to the high frequency data output. It is hard to achieve an accurate analysis of data unless the low component is removed.

Another common example is switched power supply units, which inject the switching frequency component into many system parts. Viewing digital signals mixed with this switching frequency component could be very difficult. Filtering is definitely required.

Yet another example is in ADSL residential connectivity, where data is transmitted over 256 narrow bands. Each band is only 4.7 kHz wide, and the gap between two adjacent bands is also 4.7 kHz. Examining such complex waveforms with regular DSOs is almost impossible; filtering out unwanted frequency components is necessary.

The Solution

At present, these needs are addressed in two ways. One way is building analog filters and placing them in front of the oscilloscope, providing an already filtered signal to the DSO. The disadvantages of this approach are many. Analog filters depend heavily on the accuracy and stability of analog components. Although in some cases analog filters are easily implemented, they are quite impractical for low (< 100 Hz) or high (> 100 MHz) frequency ranges. In comparison, digital filters can provide the desired results in those cases.

The second approach, practiced by many engineers, is using the DSO as a digitizer. The digitized data output is then transferred to a PC for processing. This solution frequently provides the required results, but it might be too slow or too limited in flexibility for some applications.

With Digital Filter Package 2 (DFP2), LeCroy provides a solution that combines the best of both worlds. This package includes seven of the most useful finite impulse response filters (FIR), in addition to a custom design feature. It also includes four infinite impulse response (IIR) filter types (Butterworth, Chebyshev, Inverse Chebyshev, Bessel). You can easily set the Cutoff Frequency in addition to the Stop Band Attenuation and Pass Band Ripple for each filter.

It is even possible to use single filters or multiple filters cascaded for even more complex filtering. Once filtered, waveforms include mostly relevant frequency components, undesired parts being greatly attenuated.

If you want filters with special characteristics, the custom design feature allows you to design unique filters tailored to your specific needs. The required filter can be designed with a digital filter design or with a math package such as MATLAB or Mathcad. Filter coefficients can be directly downloaded from the program into the scope, using the DSOFilter utility. It is also possible to specify the filter coefficients on an Excel spreadsheet and to use DSOFilter to download them from the spreadsheet to the scope.

DFP2's flexibility is shown by the following example:

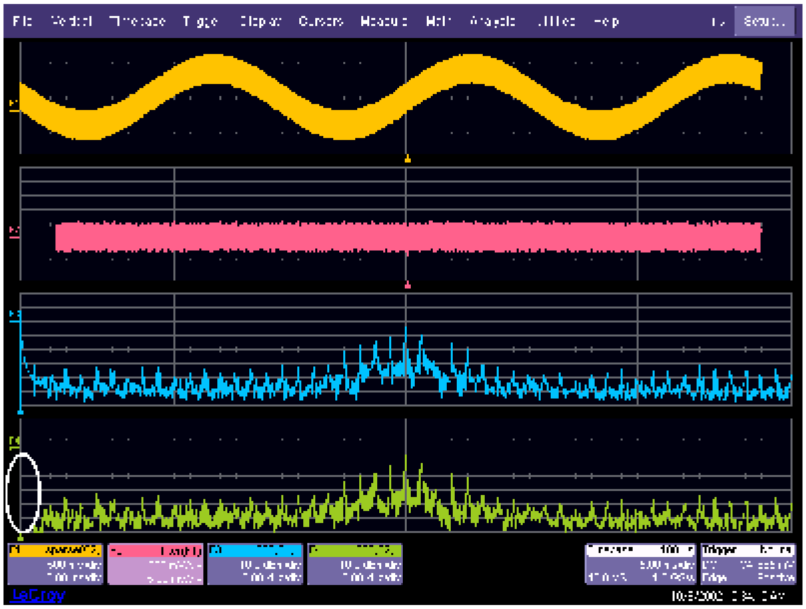


Figure 5-16. 1. A 25 kHz square wave combined with an unwanted 60 Hz sinusoidal component.

Figure 5-17. 2. A high-pass filter set to attenuate signals lower than 1 kHz is applied to remove the unwanted 60 Hz component.

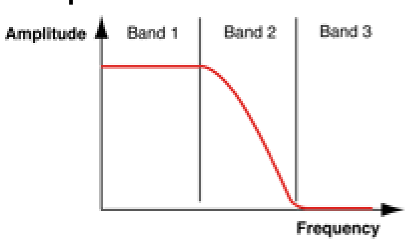
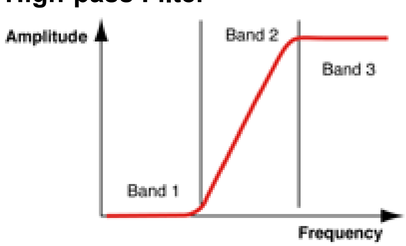
Figure 5-18. 3. FFT of the unfiltered trace.

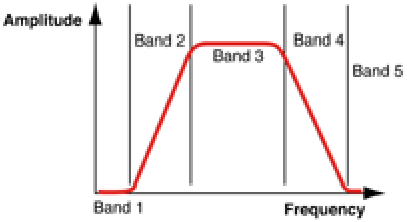
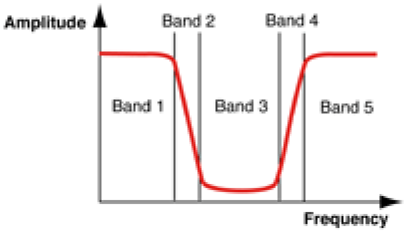
Figure 5-19. 4. FFT of the filtered trace. Note the absence of the 60 Hz component.

Enhanced Solutions

DFP2 can be coupled with other LeCroy software products such as JTA2 or DDM2 to enhance the capabilities of these products and to provide improved solutions. For Jitter Measurement, for example, the DFP2 Band-pass Filter can be coupled with the JTA2 package to measure jitter over a narrow frequency range.

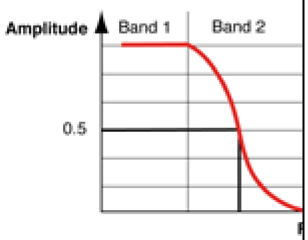
KINDS OF FILTERS[1]1. Filters are optimal FIR filters of less than 2001 taps, according to the Parks-MacLellan algorithm described in Digital Filter Design and Implementation by Parks and Burrus, John Wiley & Sons, Inc., 1987, and then adjusted by windowing the start and end 20% with a raised cosine for improved time domain characteristics and better ultimate rejection in the frequency domain, slightly increasing 1st stop-band ripple height.

Low-pass Filter  	Low-pass filters are useful for eliminating accumulated high-frequency noise and interference, and for canceling high-frequency background noise. Sample applications are in datacom, telecommunications, and disk drive and optical recording analysis for accurate RF signal detection. Band 1: Pass Band — DC to top of the transition region; signal passes unattenuated. Band 2: Transition Region — edge frequency to edge frequency plus width; increasing attenuation. Band 3: Stop Band — above end of transition region; signal is highly attenuated.
High-pass Filter  	High-pass filters are useful for eliminating DC and low-frequency components. Sample applications include Disk Drive and Optical Recording analysis (emulation of the SLICING function). Band 1: Stop Band — DC to bottom of the transition region; highly attenuated. Band 2: Transition Region — edge frequency minus width to edge frequency; decreasing attenuation. Band 3: Pass Band — above edge frequency; signal passes unattenuated.

Band-pass Filter 	Band-pass filters are useful for emphasizing a selected frequency band. Sample applications include radio channel identification, broadband transmission, ADSL, clock generators (i.e., eliminating the central frequency and displaying harmonics only), and telecommunications (Jitter measurement over a selected frequency range). Band 1: First Stop Band — DC to bottom of first transition region; highly attenuated. Band 2: First Transition Region — lower corner minus width to lower corner; decreasing attenuation. Band 3: Pass Band — signal passes unattenuated. Band 4: Second Transition Region — upper corner to upper corner plus width; increasing attenuation. Band 5: Second Stop Band — signal highly attenuated.
Band-stop Filter 	Band-stop filters are useful for eliminating a narrow band of frequencies. Sample applications include medical equipment, such as ECG monitors where the dominant ripple at 50/60 Hz is rejected, leaving the low energy biological signals intact. Digital troubleshooting: the inherent frequency of the switched power supply is blocked, revealing power line voltage drops and glitches caused by the system clock generator. Band 1: First Pass Band — DC to bottom of first transition region; signal passes unattenuated. Band 2: First Transition Region — lower corner minus width to lower corner; increasing attenuation. Band 3: Stop Band — signal is highly attenuated. Band 4: Second Transition Region — upper corner to upper corner plus width; decreasing attenuation. Band 5: Second Pass Band — signal passes unattenuated.

Communications Channel Filters

Raised Cosine (a low-pass filter)



These filters belong to the low-pass filter category (with a variety of shapes). Raised cosine is one of a class of filters used to minimize intersymbol interference: the time domain impulse response crosses zero at all bit time intervals except the one with the impulse.

Applying raised root cosine twice (or at the sending and receiving end of a signal, for example) results in a raised cosine filter effect. Sample applications include wireless cellular communications such as WCDMA, datacom, telecommunications, disk drive and optical drive analysis.

Band 1: Pass Band — DC to corner frequency minus half width; signal passes unattenuated.

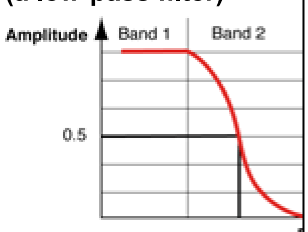
Band 2: Transition Region — corner minus half width to corner plus half width; attenuation increases with frequency with a rolloff shape of 0.5cos(a) + 0.5, where a ranges from 0 to p over the transition region. This region is determined by b, which is specified as a percentage of the corner frequency.

Band 3: Stop Band — above corner frequency plus half width; highly attenuated.

The impulse function for the raised cosine filter is:

$$h(t)=\frac{\left[\frac{\sin\left(\pi \frac{t}{T_s}\right)}{\pi \frac{t}{T_s}}\right]\cos\left(\pi\beta \frac{t}{T_s}\right)}{1-\left(2\beta \frac{t}{T_s}\right)^2}$$

Raised Root Cosine (a low-pass filter)



Band 1: Pass Band — DC to corner frequency minus half width; signal passes unattenuated.

Band 2: Transition Region — corner minus half width to corner plus half width; attenuation increases with frequency with a rolloff shape of 0.5[cos(a) + 0.5]½, where a ranges from 0 to p over the transition region. This region is determined by b, which is specified as a percentage of the corner frequency.

Band 3: Stop Band: — above corner frequency plus half width; signal is highly attenuated.

The impulse function for the square-root raised cosine filter is:

$$h(t)=\frac{4\beta}{\pi\sqrt{T_s}}\,\frac{\cos\left((1+\beta)\pi \frac{t}{T_s}\right)+\frac{\sin\left[(1-\beta)\pi \frac{t}{T_s}\right]}{4\beta \frac{t}{T_s}}}{1-\left(4\beta \frac{t}{T_s}\right)^2}$$

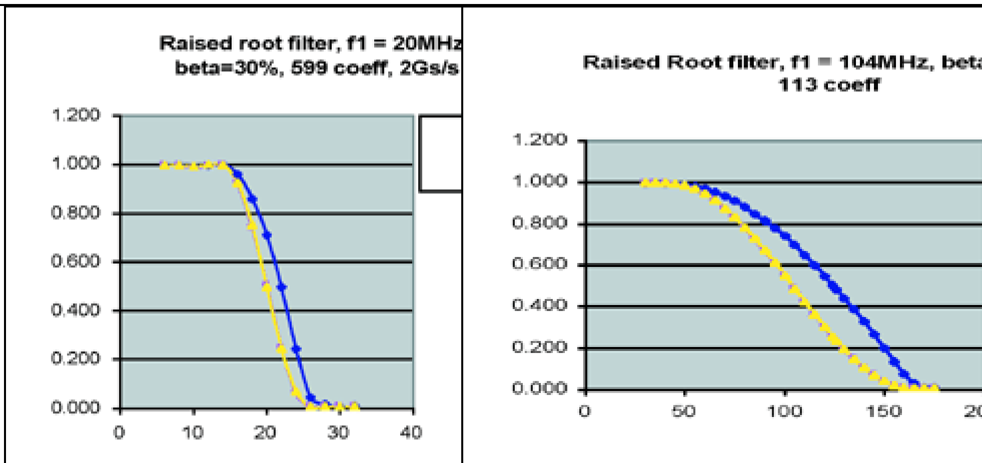
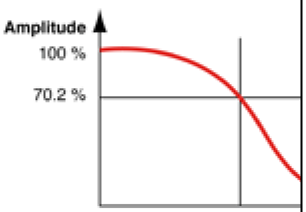


Figure 5-20. Illustrated above are two raised root cosine filters with different beta. The diagrams show that applying this filter twice results in a raised cosine response.

Gaussian



Band 1: Pass Band — DC to half power bandwidth% times modulation frequency, pass; 3 dB down at half power bandwidth.

The shape of a Gaussian filter's frequency response is a Gaussian distribution centered at DC. The signal becomes more attenuated with increasing frequency. It is not possible to specify a transition region or a stop band for Gaussian filters. However, the BT value, a fraction of the symbol frequency, determines the filter's width, where:

B = half power bandwidth

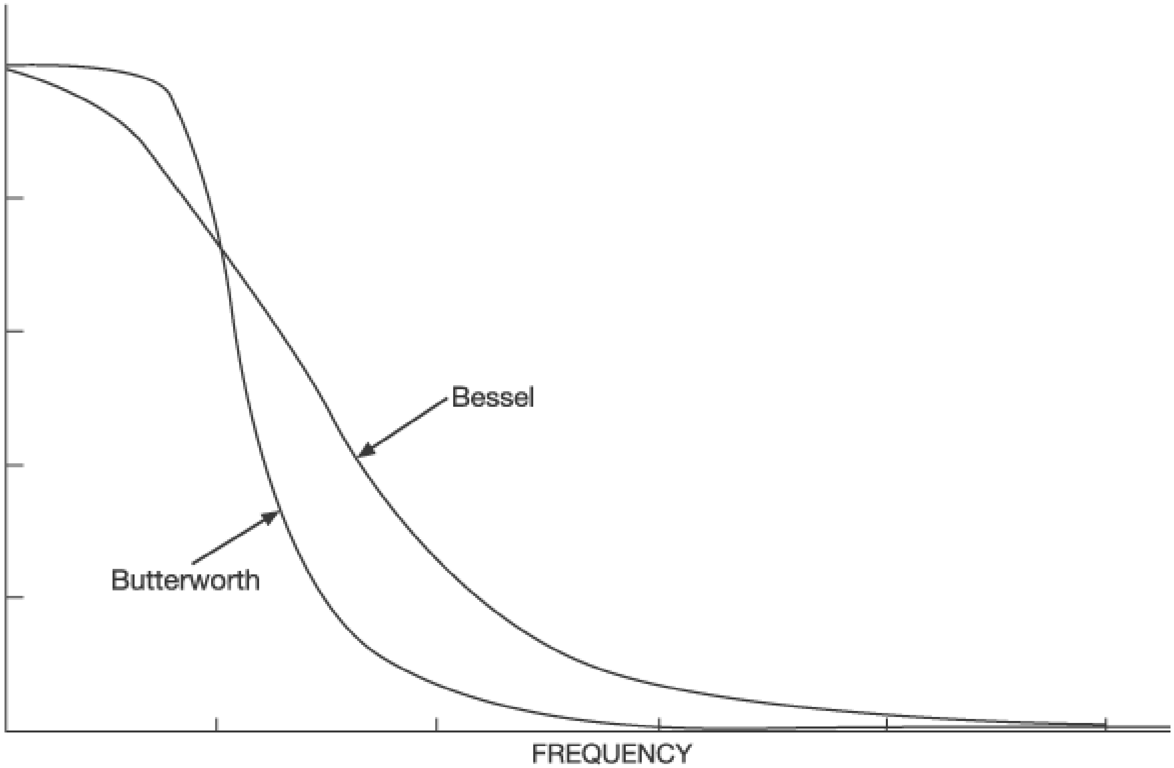
T = bit (or modulation period)

IIR Filters

Infinite Impulse Response (IIR) filters are digital filters that emulate analog filters. The four types offered by the DFP2 option are as follows:

- Butterworth
- Chebyshev
- Inverse Chebyshev
- Bessel

The Butterworth or "maximally flat" filter has the flattest amplitude response of all the available filters. The Bessel filter is noted for its uniform phase response as a function of frequency. The following figure shows a comparison between Butterworth and Bessel filters. Note that the Bessel filter has a wider transition bandwidth, but is linear phase within the passband.



If you need the fastest rolloff, the Chebyshev filters have the narrowest transition region for a given number of stages. However, the Chebyshev filter has ripple in the passband, while the Inverse Chebyshev filter exhibits a flat passband response, but has ripple in the stop band.

In the setup of these filters, you have control of cutoff frequencies, transition region width, and stop band attenuation.

Introduction

Filter Setup

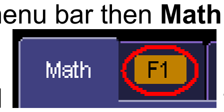
Multirate Filters

Custom Filters

Filter Specifications

Filter Setup

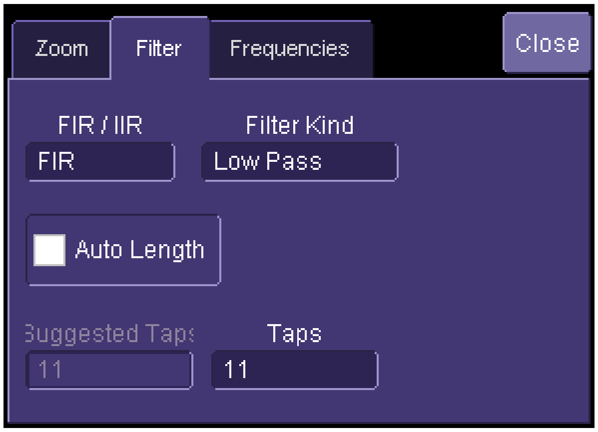
DFP Filter Setup

1. Touch **Math** in the menu bar then **Math Setup...** in the drop-down menu.
2. Touch the **Fx** tab (F1  for example) for the math trace you want to display your filtered waveform.

3. Touch the **Single** function button  if you want to perform just one filtering function on the trace, or touch the **Dual** function button  to perform math on, or apply another filter to, the filter output.
4. Touch inside the **Source1** field and select a source waveform from the pop-up menu.
5. Touch inside the **Operaor1** field and select **Filter**  from the pop-up menu. A mini-dialog to set up the filter will open at right.

Note: Other math choices in the **Operator1** menu include **Boxcar**, ERES, and interpolation. The boxcar "filter" is a simple average taken over a user-specified number of points (the "length").

6. Touch inside the **FIR/IIR** field and select finite or infinite response filterFIR (non-recursive) filters require a limited number of multiplications, additions, and memory locations. On the other hand, IIR (recursive) filters, which are dependent on previous input or output values, in theory require an infinite number of each.
.
7. Whether you selected FIR or IIR, touch inside the **Filter Kind** field and select a filtering operation. Some choices are not available for IIR.



8. If you selected FIR, touch inside the **Type** field and choose an FIR filter type. Then touch inside the **Taps**[The number of coefficients. The suggested number of taps is a minimal suggestion: using even more taps can give a more desirable response. Using less than the suggested number of taps will not meet the requested specifications.] data entry field and enter a value, using the pop-up numeric keypad. Alternatively, you can touch the **Auto Length** checkbox; the **Taps** field is grayed out and the scope calculates the optimum number of coefficients.

   If you selected IIR, touch inside the **Type** field and choose an IIR filter type. Then touch inside the **Stages** data entry field and enter a value, using the pop-up numeric keypad. Alternatively, you can touch the **Auto Length** checkbox; the **Stages** field is grayed out and the scope calculates the optimum number of stages.
9. Touch the **Frequencies** tab.
10. Depending on the class (FIR/IIR) and kind of filter you selected, and whether or not Auto Length is enabled, you can change the cutoff frequencies, transition width (edge width), stop band attenuation, and pass band ripple.

Custom Filters

Custom Filter Setup

If the standard filters provided with DFP2 are not sufficient for your needs, you can create filters with virtually any characteristic, up to 2000 taps.

The required custom filter can be designed with a digital filter design or math package such as MATLAB or Mathcad. The filter coefficients can then be loaded into the scope from an ASCII file. The file consists of numbers separated by spaces, tabs, or carriage returns.

Note: Do not use commas as separators.

For a custom IIR filter there needs to be a multiple of 6. Each stage consists of 3 numbers for the numerator polynomial followed by 3 numbers for the denominator polynomial. They are in the order a b c where the polynomial is of the form: $a + b * z^{-1} + c * z^{-2}$.

Example 1: Creating an FIR Filter Coefficient File Using Mathcad

```
N := 200   i := 0..N
sinx(x) := sin(x)/x
```

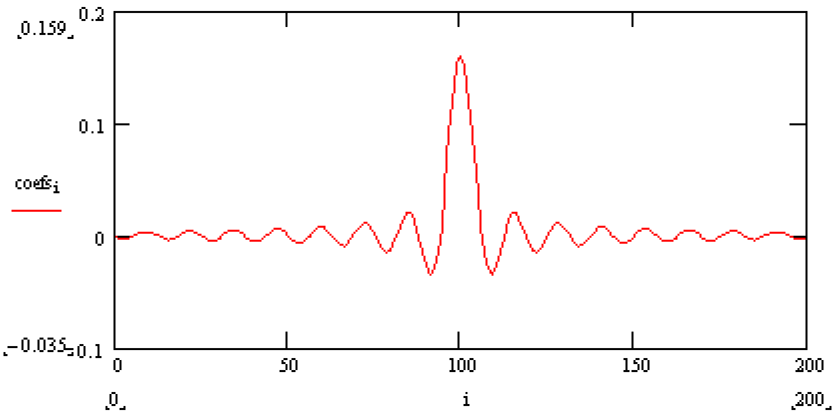
$$\mathit{coefs}_i := \frac{1}{2\pi}\,\sin x\left[\frac{i-100.0001}{2}\right]$$

200 point sin(x)/x, a low-pass filter.

Note: Real world filters would either be windowed or made by the Remez exchange algorithm. The point of this example is to show how to transfer a filter to the scope.

$$\mathit{check} := \sum \mathit{coefs}$$

check = 0.987   This is the DC gain of the filter

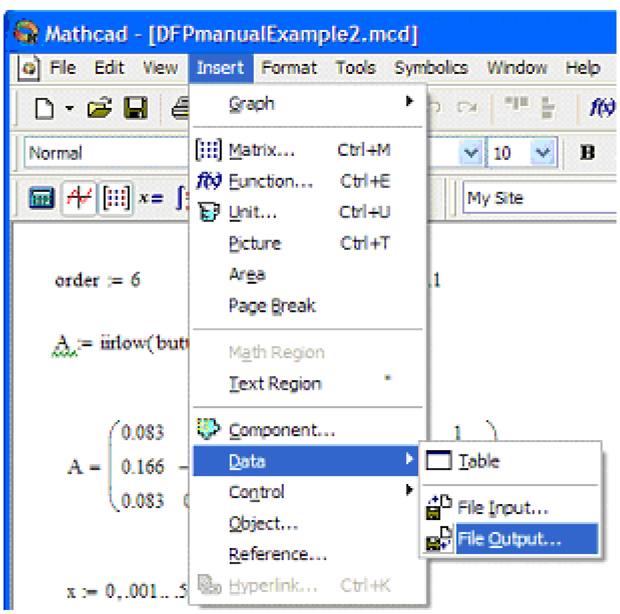


Now create an ASCII file containing the coefficients: FirFilter.txt

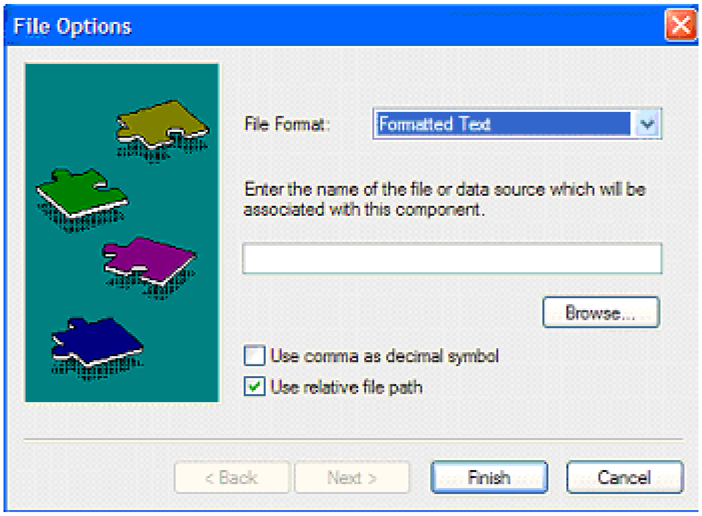
Writing Data to a Data File

To write values from Mathcad version 11 to a data file, you can use the File Read/Write component, as follows:

1. Click in the blank spot in your worksheet.
2. Choose Insert, Data, File Output from the menu.



3. This opens the File Options wizard:



4. From the File Format drop-down list choose "Tab Delimited Text".
5. Press Finish. The File Output component icon will appear in your spreadsheet, with the path to the data file under it. Underneath that, the cursor is on a placeholder for the name of the Mathcad variable containing the data to be written to the data file. Enter the name of the variable.

When you click outside the component, the values in the Mathcad variable will be written to the filename you specified. Each time you calculate the worksheet, the data file is rewritten.

On the instrument, when the Custom filter kind is selected, a file selection box will appear. Select the file saved from Mathcad; the coefficients will be used in the filter.

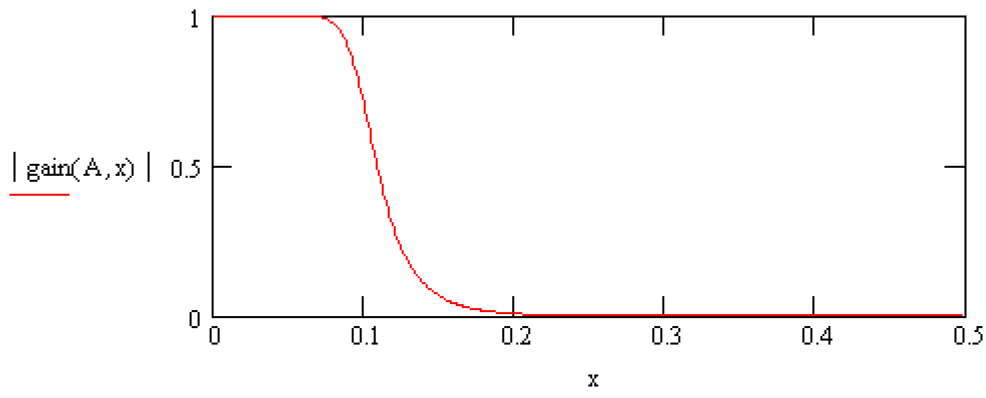
Example 2: Creating an IIR Filter Coefficient File Using Mathcad

Note: This example uses the Mathcad Signal Processing Extension Pack.

```
order := 6   fcutoff := .1
A := iirlow(butter(order),fcutoff)
```

$$A = \begin{bmatrix} 0.083 & 1 & 0.067 & 1 & 0.061 & 1 \\ 0.166 & -1.404 & 0.135 & -1.143 & 0.122 & -1.032 \\ 0.083 & 0.736 & 0.067 & 0.413 & 0.061 & 0.276 \end{bmatrix}$$

```
x := 0,.001 .. .5
```



Now create an ASCII file containing the coefficients: IirFilter.txt

IirFilter.txt

A

> **Note:** The diskette icon, the file name, and the "A" below it are the representation of a Mathcad "File Output" component. It is inserted by selecting **Insert**, **Data**, **File Output**. You must specify the file name ("IirFilter.txt" in the example) and fill in the variable name that is the source of the data ("A" in the example). Be sure to specify a complete path for the file.
>
> **Note:** In the example above, because “A” has a predefined meaning (as a unit) in Mathcad 11, it appears with a green underline. However, earlier versions of Mathcad give no warning about using "A" as a variable name, and it may still be used for this purpose.

What gets written to Iirfilter.txt is as follows:

```
0.0828825751812225 1 0.0674552738890719 1 0.0609096342883086 1

0.165765150362445 -1.40438489047158 0.134910547778144 -1.1429805025399
0.121819268576617 -1.03206940531971

0.0828825751812225 0.735915191196472 0.0674552738890719 0.412801598096189
0.0609096342883086 0.275707942472944
```

Writing Data to a Data File

To write values from Mathcad to a data file, you can use the File Read/Write component, as follows:

1. Click in the blank spot in your worksheet.
2. Choose **Component** from the **Insert** menu.
3. Select **File Read** or **Write** from the list and click **Next**. This launches the first part of the File Read or Write Setup Wizard.
4. Choose **Write to a data source** and press **Next** to go to the second page of the Wizard:

   From the **File Format** drop-down list in this Wizard, choose **Tab Delimited Text**.
5. Type the path to the data file you want to write, or click the **Browse** button to locate it.
6. Press **Finish**. You'll see the File Read or Write component icon and the path to the data file.

   In the place holder that appears at the bottom of the component, enter the transposed name of the Mathcad variable containing the data that will be written to the data file. It is important to transpose the variable (Ctl + 1) so that the variables appear in the correct order.

When you click outside the component, the values in the Mathcad variable will be written to the filename you specified. Each time you calculate the worksheet, the data file is rewritten.

On the instrument, when the **Custom** filter kind is selected, a file selection box will appear. Select the file saved from Mathcad; the coefficients will be used in the filter.

Multirate Filters

Description

In many of today's development environments, digital filter design has become most challenging. Specifications typically require higher order filters, implying increased storage capacity for filter coefficients and higher processing power. Moreover, high-order filters can be difficult, if not impossible, to design. In applications such as 3G wireless systems, for example, at the receiver end data must be filtered very tightly in large magnitude in order to be processed.

Although the LeCroy DFP option provides many filter types, the correlation between edge frequencies and sample rate may be a limiting factor: edge frequencies are limited from 1% to 49.5% of the sample rate, while the minimum transition width region is 1% of the sample rate.

Multirate, multistage filters are a practical solution for the design and implementation of FIR filters with narrow spectral constraints. Multirate filters change the input data rate at one or more intermediate points within the filter itself, while maintaining an output rate that is identical to the input rate. This approach provides a solution with greatly reduced filter lengths, as compared to standard single-rate filters.

This can be achieved in two or more simple steps. First, a filter (with a relatively limited edge frequency) is applied and the results are decimated. Then, a second filter is applied to the decimated waveform, substantially reducing the lower edge frequency limit.

Example

A sine wave with a frequency of 3 MHz has a higher frequency noise component. A low-pass filter is required to remove the noise component. The sample rate of the scope is 2 GS/s. The minimum edge frequency of the low-pass filter for this sample rate is 20 MHz. While this filter is sufficient for removing part of the noise, it cannot remove the high frequency component completely. In such a case, the problem can be solved in two stages.

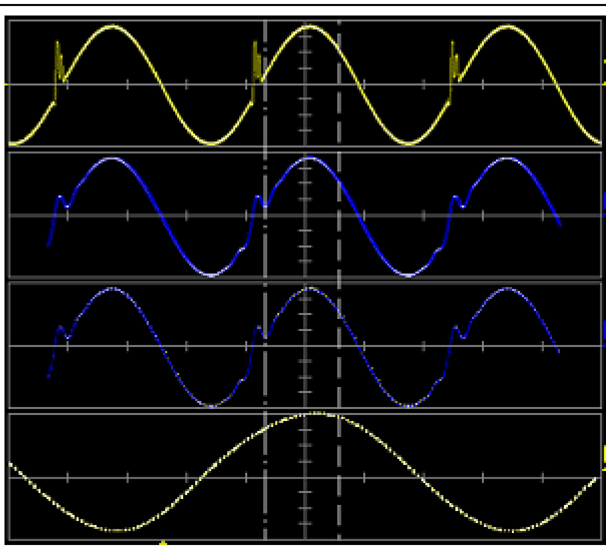


Figure 5-21. 1. A noisy sine wave with a frequency of 3 MHz.

Figure 5-22. 2. The first low-pass filter with 20 MHz edge frequency and 30 MHz transition region is applied.

Figure 5-23. 3. A sparsed version of trace A.

Figure 5-24. 4. A second low-pass filter with an edge frequency of 5 MHz and a transition region width of 6 MHz is applied to the sparsed trace in 3.

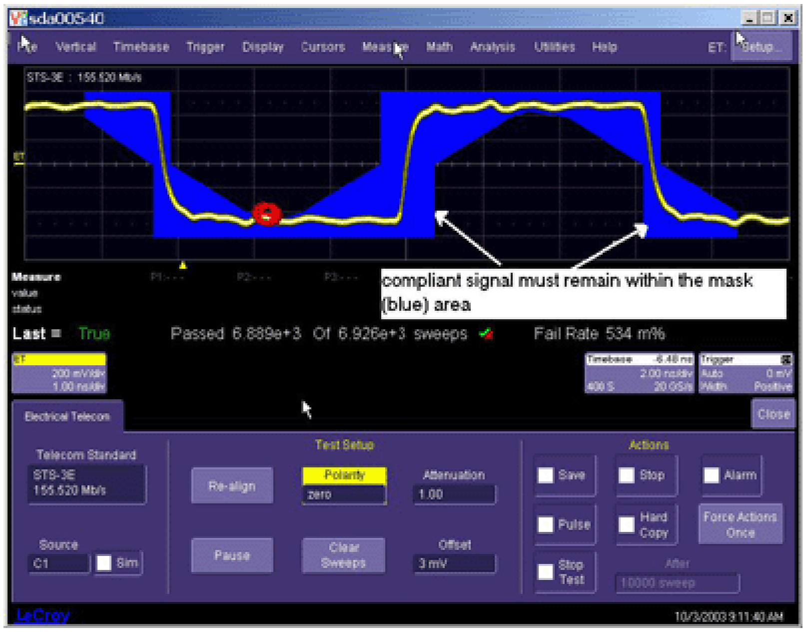
The last trace shows the zoomed signal, which was filtered by a multistage filtering method. Notice that all high frequency noise components were removed.

Specifications

- The pass-band gain of all filters (except custom) is normalized to 1.
- FIR Coefficients: 2001 max.
- IIR Stages: 29 max.
- Filter Kinds: high pass, low pass, band pass, band stop, raised cosine, raised-root cosine, Gaussian, custom
- IIR Filter Types: Butterworth, Chebyshev, Inverse Chebyshev, Bessel

ET-PMT Overview

ET-PMT is a software package that measures pulse mask compliance of electrical telecommunications signals. Pulse mask testing consists of acquiring the given signal in an oscilloscope and comparing the voltage vs. time waveform to a standard mask. The mask defines regions in V,t space where a compliant signal must remain. An example of a pulse mask is shown below.



In addition, there are often alignment criteria for each test. For example, a one level may consist of a positive-going or negative-going pulse (as in the DS1 standard). The test would then have a selection for which pulse to measure and the ET-PMT software will automatically find only those pulses.

The ET-PMT package supports the following ANSI and ITU standards.

ANSI T1	ITU-T
DS1	E1 (twisted pair)
DS3	E1 (coax)
STS-1	E2
STS-3E	E3
-	E4
-	STM-1E

In addition to these standards, It is also possible to define modified or custom pulse mask tests.

Compatibility

The functions of ET-PMT described here are compatible with the following oscilloscopes from LeCroy:

- WavePro 7100, 7200, and 7300
- WaveMaster 8620, 8600A, 8500A, and 8300A
- WaveRunner 6050, 6100, 6200
- SDA 6020, 6000A, 5000A, and 3000A (included as standard on all SDA models)

Note: ET-PMT is standard on all SDA model serial data analyzers. The Serial Data Mask package (SDM) also includes the functions of ET-PMT.

Probing

Telecommunications signals require specific load impedance for compliance testing to be accurate. The twisted pair standards require 100 ohm (DS-1) and 120 ohm (E-1) terminations, and the other standards require 75 ohm terminations. A set of adapters (test fixture) for this purpose is available from LeCroy (part number TF-ET). These adapters require the use of an LPA-BNC ProLink adapter if they are being used with a WaveMaster or SDA.

Operation

Selecting a Standard

1. To invoke Pulse Mask Testing, touch **Analysis** in the menu bar, then **Electrical Telecom** in the drop-down menu. The Electrical Telecom dialog will appear at the bottom of the oscilloscope display:

2. Touch inside the **Telecom Standard** field to select the desired standard. Selecting a standard from this menu sets the bit rate, mask, and pulse isolation criteria for the measurement.
3. Touch inside the **Source** control to select the channel to which the signal is connected. Specific terminations are required for each standard according to the table below. The Electrical Telecom adapters listed in the table are available from LeCroy in the TF-ET test fixture kit.

Telecom Standard	Required Termination	LeCroy Adapter
E1 TP	120 ohm	AP120
E1 coax	75 ohm	PP090
E2	75 ohm	PP090
E3	75 ohm	PP090
E4	75 ohm	PP090
STM-1E	75 ohm	PP090
DS-1	100 ohm	AP100
DS-3	75 ohm	PP090
STS -1	75 ohm	PP090
STS-3E	75 ohm	PP090

Test Setup

Setup

The **Setup** button applies the appropriate settings to the oscilloscope for testing the selected standard. Different standards require a particular termination, and an error message will appear at the bottom of the oscilloscope screen if the wrong (or no) adapter is present. However, this error will not prevent the instrument from making the measurement; that is, measurements can be made without the specific adapters. But if the signal is out of range for the standard, the setup operation will generate an error message and the **Run** button will be grayed out. The signal will appear on the screen, but no testing will be possible.

Before the **Setup** button is pressed, the **Run** and **Clear Sweeps** buttons appear grayed out. These buttons become available (not grayed out) upon successful completion of a setup. At that time, a **Re-Align** button will replace the **Setup** button.

Polarity

In many electrical standards, such as DS-1, alternate “ones” are inverted. Each time a one is transmitted it is either a positive or negative going pulse depending upon the polarity of the previous one. This type of coding is referred to as AMI (alternate mark inversion). The **Polarity** control allows you to select which polarity (positive or negative) pulse to test.

The STS-3E and STM-1E standards use CMI (code mark inversion) pulse coding. In CMI coding, a one remains high for the full bit period while a zero has a transition to the low state in the middle of the bit period. The Polarity control allows you to select whether a 1 or 0 is to be tested.

Attenuation

The **Attenuation** control allows you to enter an attenuation value to be applied to the test signal. This control allows for amplitude correction for cable or other systematic losses. The attenuation value can be set from 0.5 to 1 in steps of 0.01, with 1 meaning “no external attenuation” and 0.5 corresponding to an amplitude reduction of 50%. If a cable with 3 dB of loss were being used to couple the signal under test to the instrument, for example, the attenuation value should be set to 0.708.

Offset

The offset control allows for the correction of DC offsets in the signal under test. This control can be varied from -50 mV to 50 mV.

Clear Sweeps and Pause

The Clear Sweeps button allows you to reset the sweep count and start testing over again. The Pause button, which suspends testing without resetting the sweep count, becomes a Run button once it is pressed. Pressing the Run button, in this case will start the test from where it was last suspended.

Actions

The Actions section contains a set of check boxes that determine what actions are taken when a test fails. Actions are selected by checking the appropriate box; multiple actions are possible for each failure. The available actions are:

Save - Stores the failed waveform in a file. When this option is selected, the Save Waveform tab appears in the main dialog. Touch this tab to set the file name and storage location.

Stop - Stops testing on the first failure.

Alarm - Generates an audible alarm on each failure.

Pulse - Generates a pulse at the auxiliary output BNC connector on the front panel of the instrument. When this option is selected, the Aux Output tab appears. Touch this tab to set the type of pulse.

Hardcopy - Prints a hard copy image of the failed mask test. Hard copies can be sent to any valid printer (either networked or directly connected to the instrument). It is also possible to print to file or email the image. When this option is selected, the Hardcopy tab appears. Touch this tab to set the format and devices.

Stop Test - Gives you the ability to stop the test after a predetermined number of sweeps have been completed. When this option is selected, the After field alongside the checkbox becomes active. Touch inside the After field to set the number of sweeps, by means of the pop-up keypad.

The **Force Actions** Once button immediately executes all of the selected actions when it is pressed.

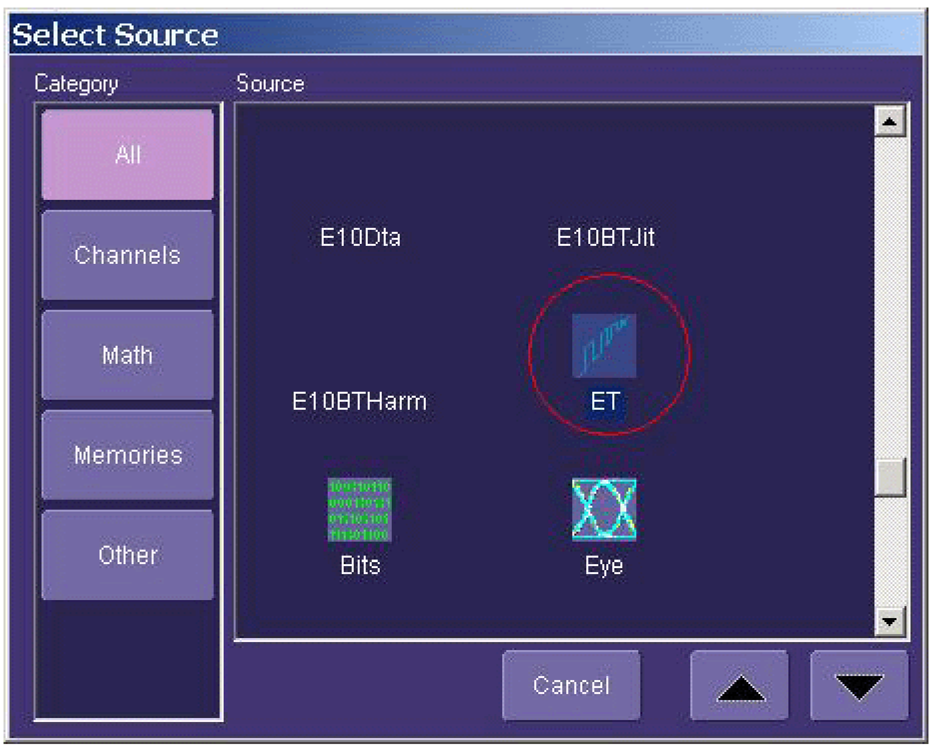
Adding Measurements to a Mask Test

In addition to measuring pulse compliance relative to a given mask, parametric measurements are also available. A full set of measurements including amplitude, overshoot and rise time are available from the oscilloscope’s Measure menu. Measurements are made on the displayed waveform as it appears on the mask test screen.

Setting up Parametric Measurements

1. In the menu bar, touch Measure; then Measure Setup in the drop-down menu. The menu shown below is displayed:

2. Touch inside the Source1 field and select “ET” as the source.

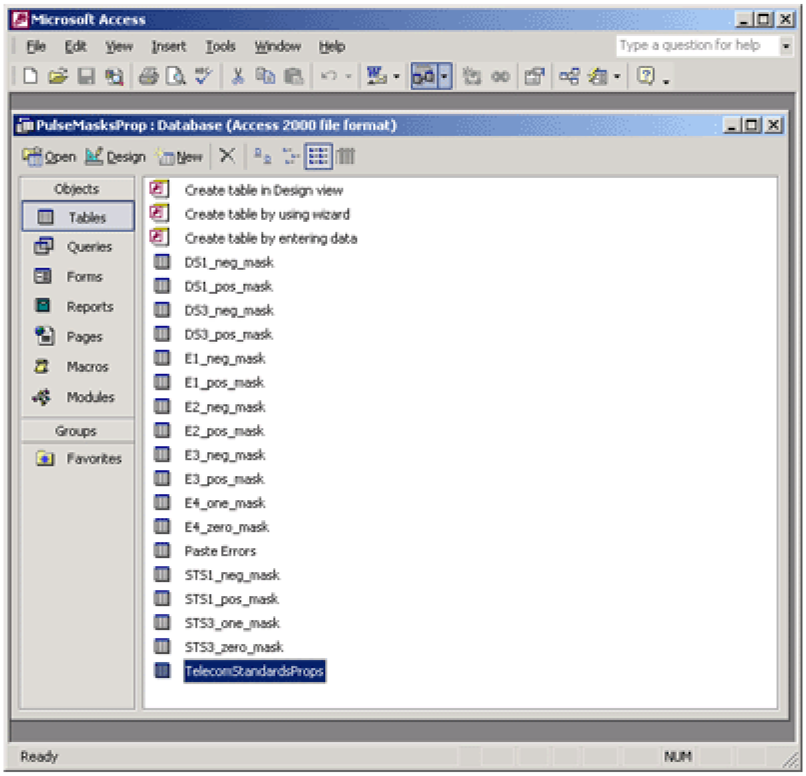


3. Touch inside the Measure field and select the desired measurement from the pop-up menu.
4. Touch the Always On checkbox in the “Help Markers” section of the dialog to display cursors on the waveform, showing exactly where the measurements are being made.

Custom Pulse Mask Test Files

The pulse mask test file included with the ET-PMT software includes the standards listed in the table. In addition to these standard pulse masks, custom masks can also be created. Custom masks include a mask, along with pulse alignment criteria and acquisition settings such as waveform averaging and persistence. You must use Microsoft Access 2000 to edit the mask definition file.

The mask properties are defined by the file "PulseMasksProp.mdb" in the "D:\Masks" directory. This Microsoft Access 2000 file is shown here:



This file contains mask tables that give the coordinates of each mask, and a table named "TelecomStandardsProps" that contains all of the information defining the test and how each mask is to be used.

The fields in this table are defined as follows:

ID - An index that is automatically generated. Do not enter a value in this field.

Old - For internal use only; leave blank.

Standard - The name of the standard to which the selected mask is associated

Legacy Standard - Optional; not used for user defined standards

Symbol - Defined mode within standard (pos, neg, one, zero, transmit, receive)

Bit rate - Bit rate of signal; defines period.

Required Bandwidth - Minimum bandwidth needed to accurately test signal.

Minimum Sample Rate - Minimum sample rate needed to accurately test signal.

Probe - Required probe or adapter; left blank if none.

Type - Defines the type of signal and how it will be aligned and tested. The choices are:

- Absolute
- Absolute + Offset
- Relative
- Relative Peak

Coding - Type of coding (CMI code mark inversion, AMI alternate mark inversion, etc.); used for information only.

Pattern Isolation - Bit pattern used to isolate the pulse under test. Syntax: bbbb/pp/aaa where bbbb is the symbol values before the desired pattern and aaa is the value of the bits after the desired pattern. For example, an isolated "one" would look like: 00/1/00. Bipolar pulses are defined as 1 and -1; and CMI is handled as two bits per unit interval. A CMI 0, for example, would look like: 1/01/0.

V div - The vertical scale required for nominal amplitude in the mask. This voltage is adjusted for relative masks.

Nom Ampl - The nominal amplitude of the pulse to be tested.

T div - Horizontal scale to have signal in mask.

ET Delay - The time in seconds between the center of the pulse (1/2 bit) and the edge where the trigger is set

ET Center - The center of the mask in DIV (usually 5.0)

Base Point - Optional (currently not used)

Offset Tol (for "Absolute + Offset" type only) - The permitted tolerance to adjust the offset for a better mask fit

Gate Start - The limit in DIV in the waveform data at which the mask test is started

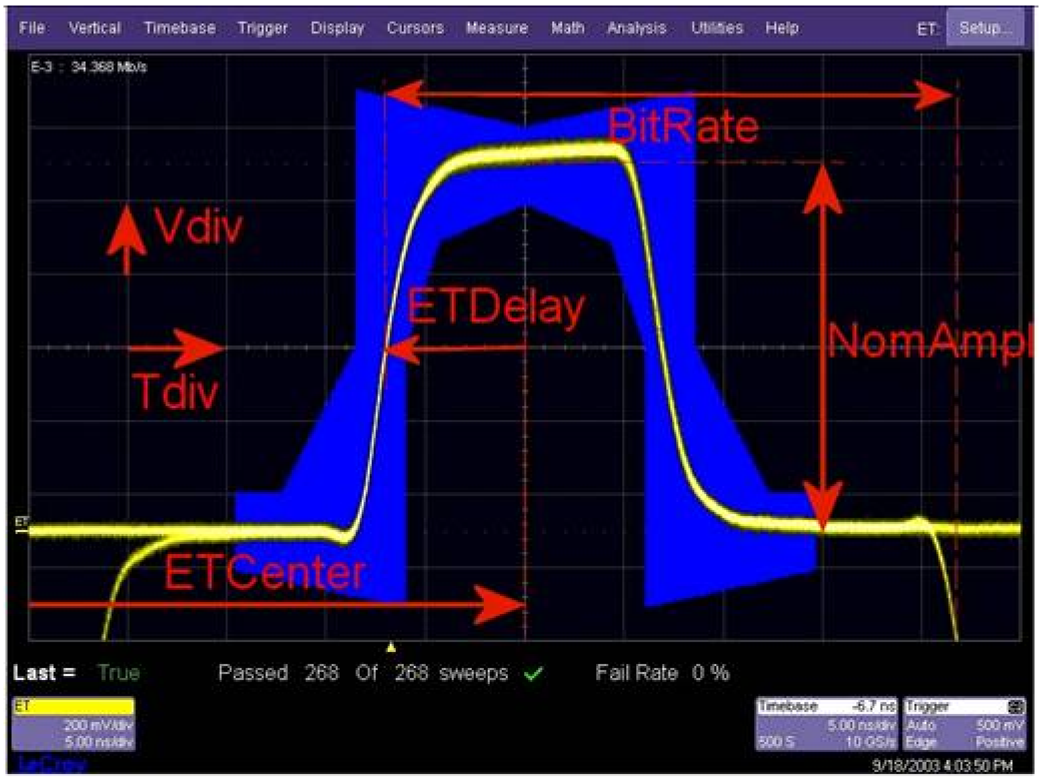
Gate Stop - The limit in DIV in the waveform data at which the mask test is stopped. This parameter and the one above it allow you to perform mask testing on specific pulses within a longer, more complex waveform.

Mask Data (optional) - Hyperlink to the table that contains the mask data

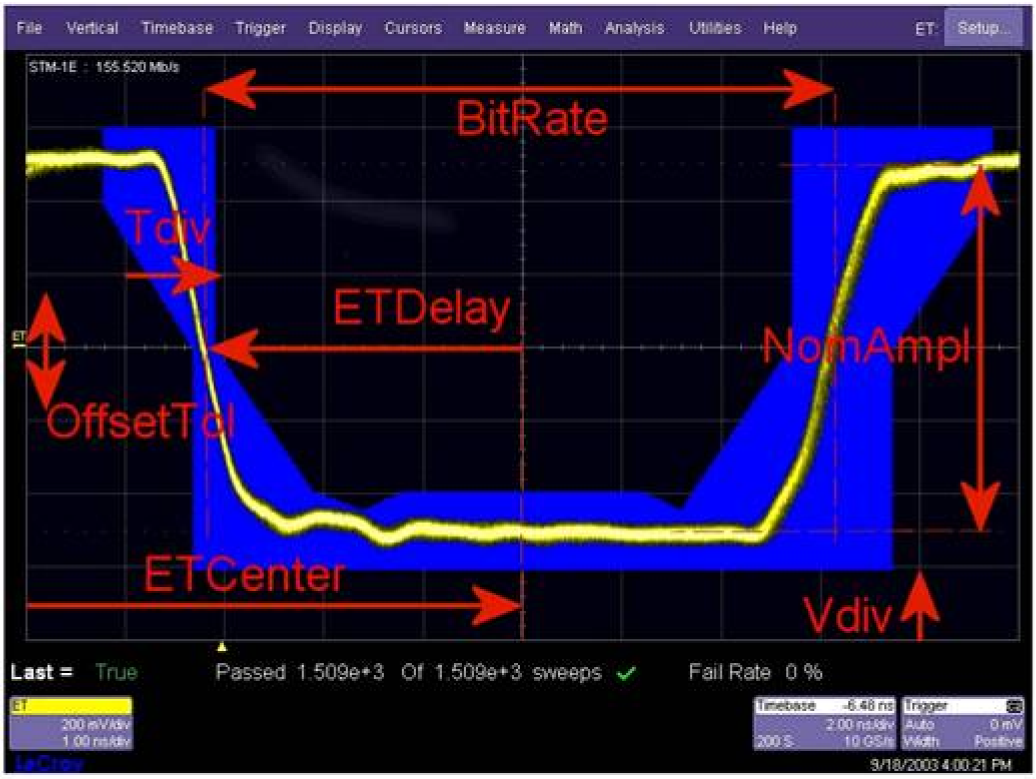
Mask File Name (optional) - Used if MaskData is not specified; file name of the custom mask (*.msk file). This file type is created by LeCroy's MaskMaker software.

The following images show graphically how the above parameters are used for the four different types of masks set in the Type field. They show how the various parameters are used to align and scale the waveforms to the mask.

Absolute

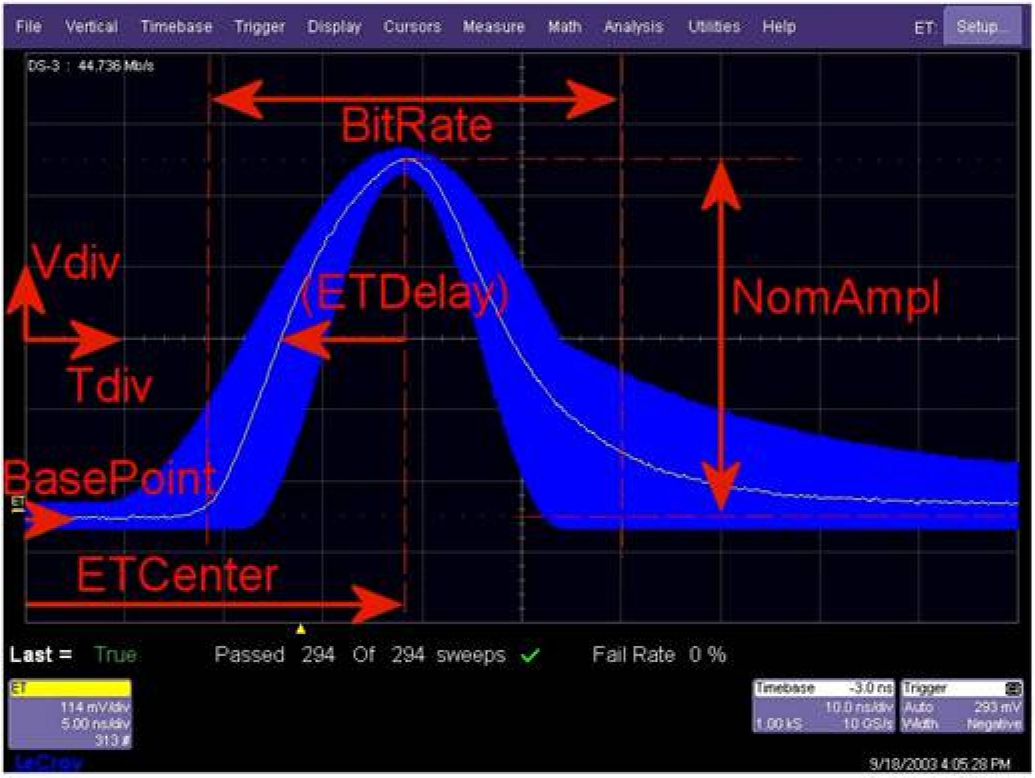


Absolute + Offset



Relative

Relative Peak



Accessing JTA2

To access JTA2's special features, you must first purchase and install the option. Once installed, JTA2's math and parameter selections will appear in the Math and Measure menus.

Timing Functions

JitterTrack,PersistenceHistogramandPersistenceTrace are timing functions in LeCroy's JitterPro and JTA jitter and timing analysis packages. The JitterTrack feature is key to identifying the source of excessive jitter or non-normal jitter characteristics. A timeline of signal jitter that is synchronous with the signal under test allows you to view patterns that would remain invisible using other systems, zoom to areas containing maximum jitter, and troubleshoot the problem. PersistenceHistogram is the ideal quantitative "companion" to persistence display. It histograms a horizontal or vertical slice of the persistence waveform. Utilizing averagerange and sigma, settings, PersistenceTrace computes a vector trace from a bit map to give insight into edge details down to a few picoseconds.

- **JitterTrack** graphically plots as a function of time the amplitude of the waveform attributes **Cycle-to-Cycle** variation, **Duty Cycle**, **Interval Error**, **Period**, **Width**, and **Frequency**.

  Interval Error, for example, calculates the timing error of a signal compared with an ideal, expected interval defined by a user-specified reference frequency, the most common estimator of jitter. "The short-term variations of a digital signal's significant instants, from their ideal positions in time," are plotted. This is the perfect tool for characterizing clocks in synchronized telecom networks such as SONET and SDH.

  A special **data** function, available for most of these attributes, enables work on random data streams.
- **Persistence Histogram** analyzes a vertical or horizontal slice of a persistence map of multiple waveforms. The resultant bar chart shows a numerical measurement of the timing variations of a signal, which are observed qualitatively in the persistence display of the signal. A typical application is characterizing the jitter in a communications signal eye diagram.
- Persistence Trace is a method for displaying the data acquired from multiple sweeps of a waveform. A vector trace is computed, based on the bit map of the underlying multiple signal acquisitions. Detail is then represented in a choice of three graphic

forms, each representing a different characteristic of the waveform. Insight into edge details is given down to a few picoseconds — valuable in applications such as the examination of fast signal transitions.

Timing Parameters

Timing parameters can also be used to measure cycle-to-cycle jitter, the width of positive and negative pulses, the duty cycle of either polarity, and an infinite number of cycles on long records. Pulses or cycles can be counted using one of these parameters.

As interpolation filtering is applied to signal edges in the vicinity of measurement points, timing parameters operate on acquired waveform levels that may be selected in either volts or percentage of signal amplitude. Each parameter calculation is performed over all cycles or edges present in the input signal, without limitations.

Statistical Tools

The information obtained from applying timing parameters can then be analyzed using the statistical tools, histograms and trends:

- **Histograms** characterize and present as a bar chart the statistical distribution of a timing parameter's set of values. In addition, there are 18 statistical histogram parameters, which operate directly on the histogram.
- **Trends** represent the evolution of timing parameters in line graphs whose vertical axes are the value of the parameter, and horizontal axes the order in which the values were acquired.

How JitterTrack Works

Using "Clock" or "Data"

Use this function to plot as a bar chart the evolution over time of this and five other waveform attributes in simple steps.

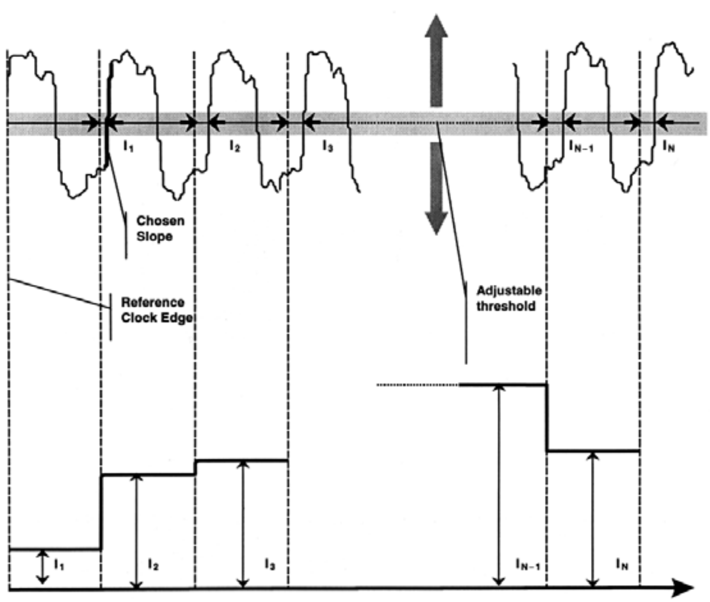


Figure 5-25. How JitterTrack's Interval Error works when "Clock" Mode is selected

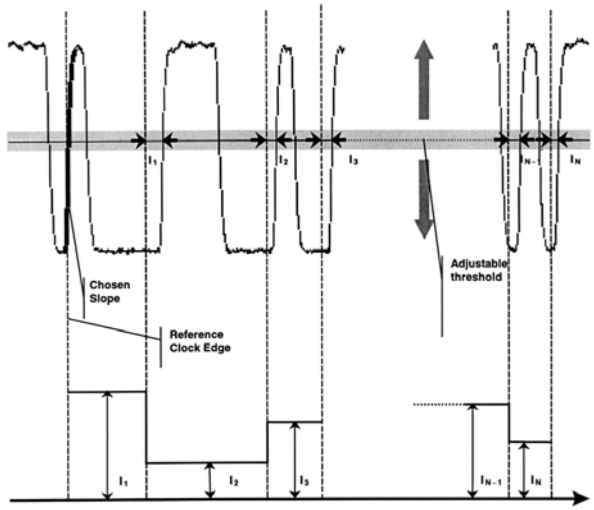


Figure 5-26. When "Data" Mode is selected.

1. Set the desired reference clock frequency for an ideal position against which the signal is to be compared, or use "Find Frequency."
2. Specify the level at which the jitter measurement is to be made, as well as the rising or falling edge on which the measurement is to start.
3. Timing errors are graphically revealed.

When to Use JitterTrack

The JitterTrack Function charts the evolution in time of these waveform attributes:

- Cycle-to-Cycle deviation
- Duty Cycle
- Interval Error
- Period
- Pulse Width
- Frequency

Each is time-correlated to its source trace and contains the same number of points as the waveform.

JitterTrack or Trend?

Whether it is more appropriate to use JitterTrack or the statistical tool, Trend will largely depend on the application, as well as the other factors set out in the tables below. While JitterTrack sample points are evenly spaced in time, those of Trend are not. Trend plots any parameter available in the instrument against its event count, as in a scatter or an XY diagram.

Characteristic	Trend	JitterTrack
Representation	parameter Value vs. Events	attribute value vs. time
Attributes or Parameters Supported	all parameters	Cycle-Cycle Period Duty Cycle Width Interval Error Frequency
Behavior	cumulative over several acquisitions up to 1 million events	non-cumulative (resets after every acquisition) unlimited number of events

When you need to	Use…
monitor the evolution of a waveform parameter or attribute over several acquisitions...	*Trend*— Jitter works only on one acquisition at a time
time-correlate an event and a parameter value...	*JitterTrack*
monitor an evolution in the frequency domain...	*JitterTrack*— Trend points are not evenly spaced in time and therefore cannot be used for FFT (Fast Fourier Transform).
monitor JTA parameters...	*Trend*

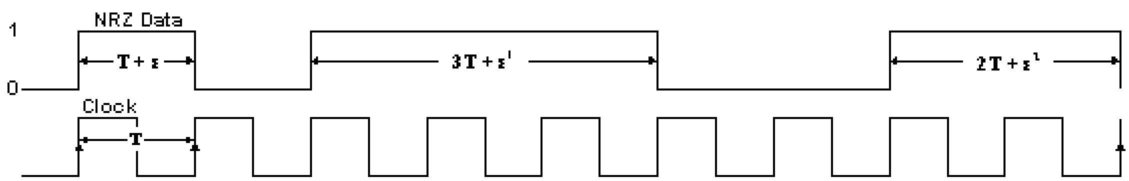


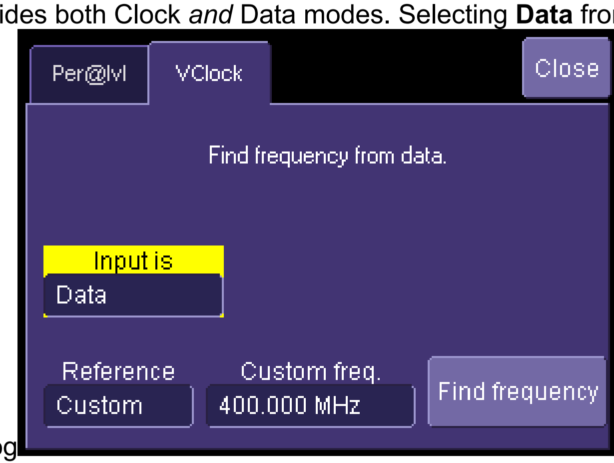
Figure 5-27. Random NRZ (Non-Return to Zero) data stream and its corresponding clock signal.

Clock or Data?

For most waveform attributes, JitterTrack offers the choice of Clock or Data modes for measuring clock signals or data streams. "Data" should be used (where available) when the pulse widths, intervals, periods or other significant instants being measured are randomly distributed and contain multiples of the clock period.

On the one hand, apart from jitter, clock signals ought to be regular. On the other hand, data streams by their very nature have irregular pulse widths.

A clock signal is normally required to characterize jitter. But such a signal will not be available if the waveform being measured is a data stream, whose very randomness hides the clock signal. To overcome this, JitterTrack provides both Clock *and* Data modes. Selecting **Data** from the VClock



dialog gives the superior timing resolution through normalization (see table) required for correctly measuring jitter in data signals.

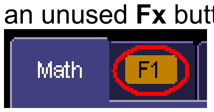
The diagram on the previous page shows a data stream in relation to its clock signal. It illustrates how data pulses contain, within themselves, multiples of their clock-signal pulse widths. Analyzing the positive pulses in the data stream, we observe a great variance between each sample in, for instance, the range T to 3T . In fact, it is the small variations (the jitter) that are important. And they could be normalized if clock frequency, and clock frequency over pulse width, were known. This normalization, provided by JitterTrack, reduces pulse variations and increases timing resolution so that errors (ε) can be clearly observed. It does this by reducing the jitter range, dividing each measurement equal to n × T by n.

Modes	CLOCK	DATA
Jitter Range	3T + ε	ε << 3T
Resolution	coarse	fine

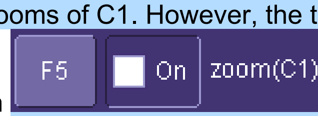
Table 5-1.Comparing a Random Data Stream Analyzed Using Clock and Data Modes.

Setting Up Jitter Measurements

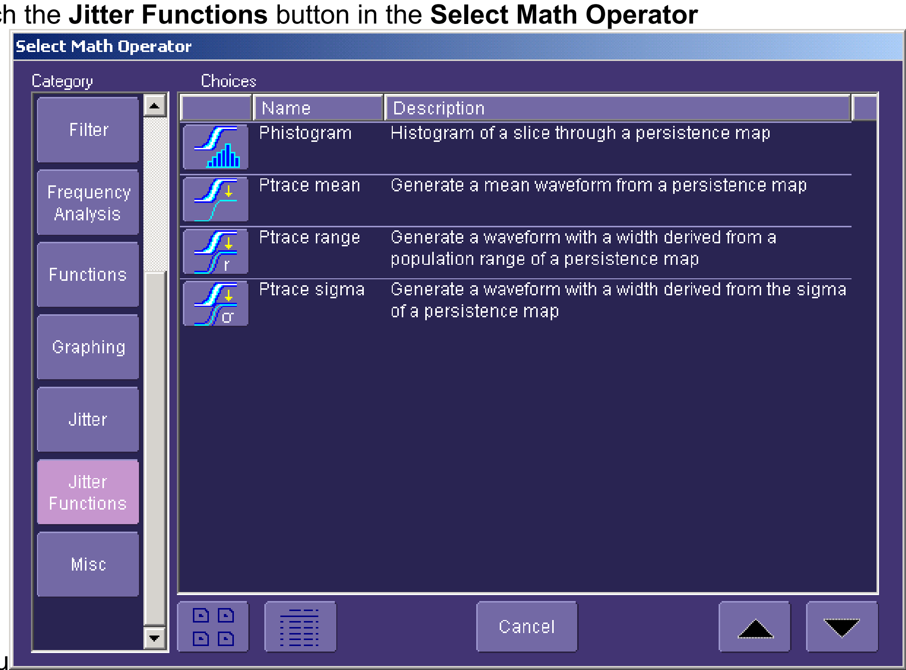
Jitter Math Setup

1. Touch **Math** in the menu bar, then **Math Setup...** in the drop-down menu.
2. In the "Math" dialog, touch an unused **Fx** button to simply make a selection from the **Select Math Operator** menu. Or, touch an **Fx** tab 

for more setup options.

> **Note:** By default, unused **Fx** positions are designated as zooms of C1. However, the traces are disabled, as indicated by an unchecked **On** box alongside the **Fx** button 

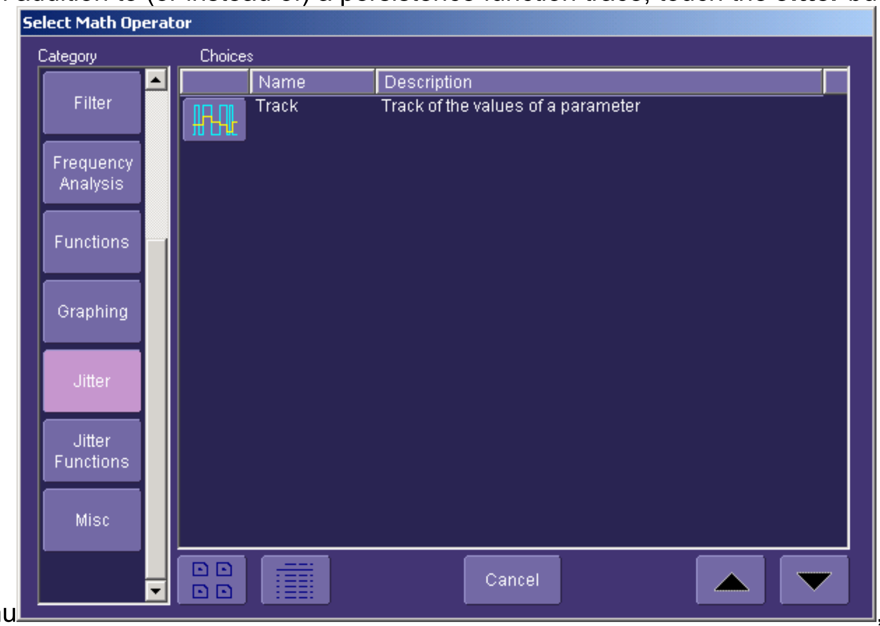
.

3. Touch the **Jitter Functions** button in the **Select Math Operator** menu 

for a list of persistence functions.
4. Touch a persistence function. The **Select Math Operator** menu closes, and the trace is automatically enabled.

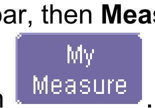
JitterTrack

If you want to enable JitterTrack in addition to (or instead of) a persistence function trace, touch the **Jitter** button in the **Select Math Operator** menu

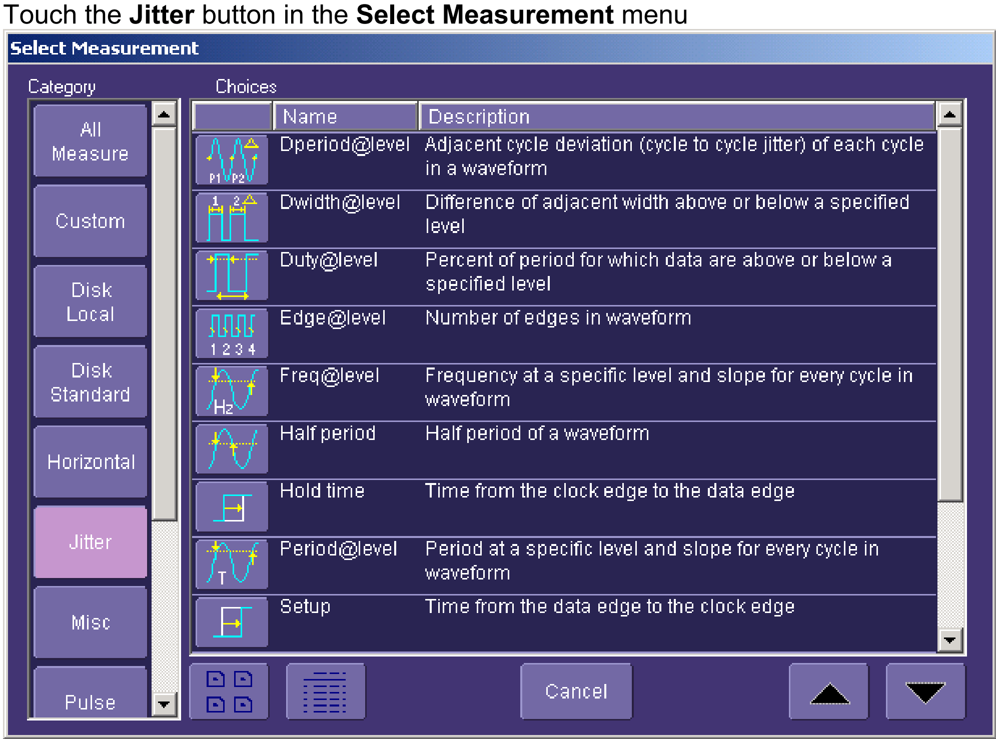


, then the **Track** button. The **Select Math Operator** menu closes, and the JitterTrack is automatically enabled.

Jitter Parameters Setup

1. Touch **Measure** in the menu bar, then **Measure Setup...** in the drop-down menu.
2. Touch the **My Measure** button 

.
3. In the "Measure" dialog, touch an unused **Px** button to simply select a jitter parameter from the **Select Measurement** menu. Or, touch a **Px** tab for more setup options.
4. Touch the **Jitter** button in the **Select Measurement** menu



; a list of persistence functions appears.

5. Touch a parameter. The setup dialog for the Px position you selected opens automatically. A mini-dialog also opens to the right of the main dialog, giving you more setup options for the selected parameter.

When to Use Persistence Histograms

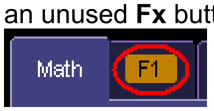
The Persistence Histogram function builds a histogram from a persistence map to reveal the features that are only visible when several acquisitions have been superimposed on one another.the histogram as statistical tool simply graphs waveform parameters such as amplitude, frequency, or pulse width on an acquisition or series of acquisitions. In contrast to this,

Both Histogram and Persistence Histogram bar charts are divided into intervals, or bins. While each bin in the histogram bar chart contains a class of similar parameter values, the Persistence Histogram analyzes both vertical and horizontal "slices" of the persistence map. Vertically, each bin contains a class of similar amplitude levels; horizontally, each bin contains a class of similar time values.

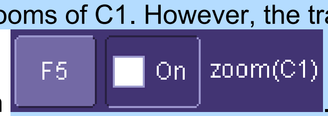
For a Histogram of	Use
a crossover point in time or in amplitude on an eye diagram	Persistence Histogram (Vert. and Horiz. Slices)
cumulative jitter on an eye diagram	Persistence Histogram (Horiz. Slice)
signal-to-noise ratio on an eye diagram	Persistence Histogram (Vert. Slice)
the different interval widths present in a long data stream	Histogram)*p@lv* (of Timing Parameter
cumulative jitter on a long record of a clock signal	Histogram)*tie@lv* (of Timing Parameter
cycle-to-cycle jitter	Histogram)Δ*p@lv* (of

Setting Up Persistence Histograms

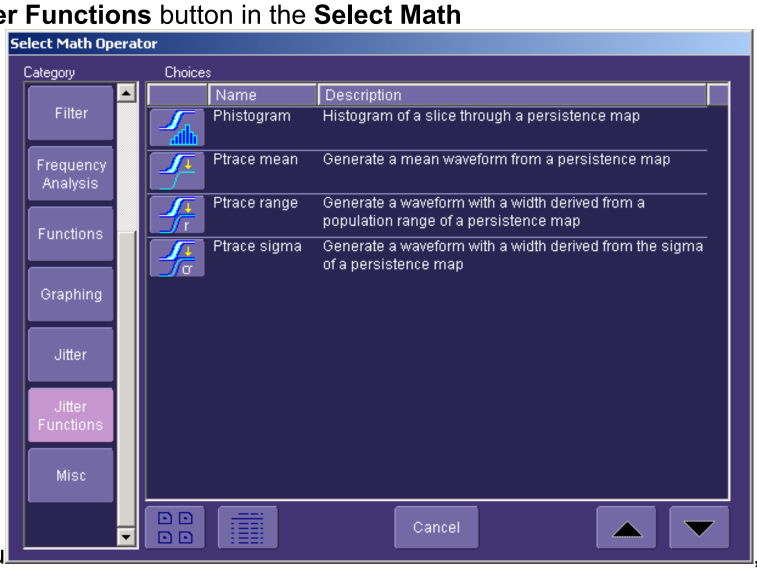
Selecting the Math Function

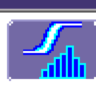
1. Touch **Math** in the menu bar, then **Math Setup...** in the drop-down menu.
2. In the "Math" dialog, touch an unused **Fx** button to simply make a selection from the **Select Math Operator** menu. Or, touch an **Fx** tab 

for more setup options.

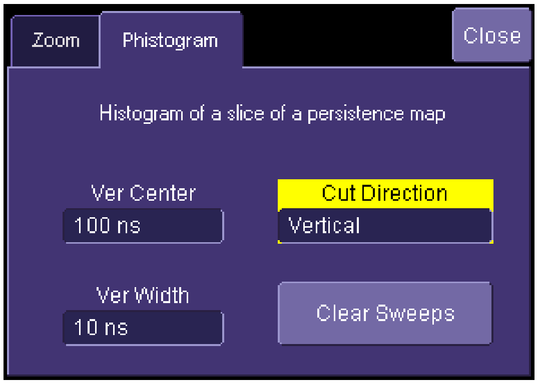
Note: By default, unused **Fx** positions are designated as zooms of C1. However, the traces are disabled, as indicated by an unchecked **On** box alongside the **Fx** button 

.

3. Touch the **Jitter Functions** button in the **Select Math Operator** menu 

, then touch the **Phistogram** button . A setup mini-dialog opens to the right of the main dialog.

Setting Up the History

The mini-dialog 

contains setup fields for your histogram.

Selecting the Cut

Touch inside the **Cut Direction** field 

and select either **Vertical** or **Horizontal**. If you choose to cut a vertical slice, the units of the center and width of the slice are given in nanoseconds. If you choose a horizontal cut, the units of the center and width of the slice are given in millivolts.

How to Trace Persistence

A persistence waveform created by turning on persistence is show here. rom this waveform, you can create three types of shapes on which waveform processing can be performed.F

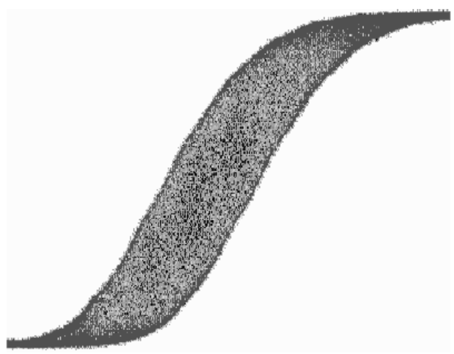


Figure 5-28.

Figure 5-30. From left to right: Average, Range, and Sigma.

An Innovative Visual and Processing Tool

With this timing function, not only can waveform noise and jitter be displayed but further processing can also be done.

Persistence Trace generates special graphic representations of the persistence waveform on which further processing, such as the application of parameters and even Pass/Fail testing, can be performed.

Displaying data acquired from multiple sweeps of the waveform, Persistence Trace computes a vector trace based on the bit map of the underlying signal acquisitions. Detail is then shown in a choice of three shapes: **average**, **range**, and **sigma**. These are created without destroying the underlying data, allowing the display of analytical results from raw data.

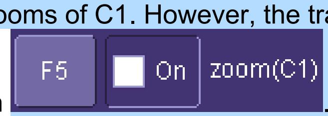
Typical applications of Persistence Trace are given in this table:

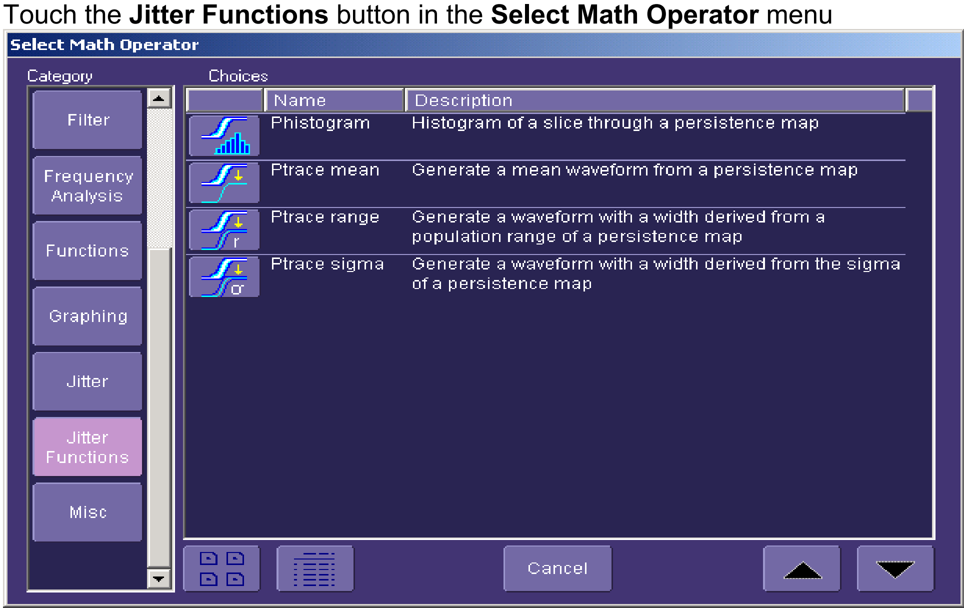
If you want to	Use
see edge detail in a fast signal	average
eliminate noise on a persistence trace	average
assess typical noise on a persistence trace	sigma
assess worst case noise on a persistence trace and use it to create a tolerance mask	range

To Set Up Trace Persistence

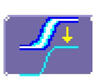
1. Touch **Math** in the menu bar, then **Math Setup...** in the drop-down menu.
2. In the "Math" dialog, touch an unused **Fx** button to simply make a selection from the **Select Math Operator** menu. Or, touch an **Fx**tab 

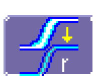
for more setup options.

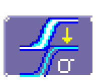
> **Note**: By default, unused **Fx** positions are designated as zooms of C1. However, the traces are disabled, as indicated by an unchecked **On** box alongside the **Fx** button 
>
> .

3. Touch the **Jitter Functions** button in the **Select Math Operator** menu 

, then touch one of the Persistence Trace buttons:


Ptrace Mean


Ptrace Range


Ptrace Sigma

A setup mini-dialog opens to the right of the main dialog, offering the following additional setup options:

Function	Options	How It Works
Ptrace Mean	Clear Sweeps	For each vertical time slice on the persistence map, Ptrace Mean calculates and plots a trace corresponding to the map's mean value. Single-shot signals sampled at or above 2 GS/s and accumulated in the persistence map can be traced at a resolution of 10 ps (100 GS/s equivalent sampling). The persistence trace average can be further analyzed using the instrument's standard parameters, such as rise time.
Ptrace Range	Clear Sweeps, **% population** range. A percentage of the population of the persistence map can be chosen from which the envelope will be formed, enabling exclusion of infrequent events (artifacts).	For each vertical time slice on the persistence map, Ptrace Range calculates and plots an envelope corresponding to the map's range. The range can then be used in further processing: for example, as a source for Pass/Fail masks.
Ptrace Sigma	Clear Sweeps, **Scale to** standard deviations. This allows you to select a sigma from 0.5 to 10.0, which expands those parts of the sigma envelope representing waveform regions with the most jitter. This is useful for making a tolerance mask.	For each vertical time slice on the persistence map, Ptrace Sigma calculates and plots an envelope corresponding to the map's standard deviation. Multiples of sigma can also be done using sigma. The sigma can be used in further processing; for example, as a source for Pass/Fail masks.

Choosing a Timing Parameter

This table lists the Jitter and Timing Analysis (JTA) parameters and the tasks that they can perform. Additional analysis and processing of the waveform can be carried out by activating Statistics and using histogram parameters. For some parameters, one of the variants of JitterTrack can perform the same task.

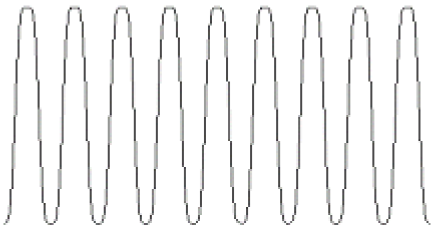
If You Want To	Use This Timing Parameter	For Further Processing	Or JitterTrack
measure accuracy of clock, period or frequency	p@lv freq@lv	Statistics On or use Histogram	Period Jitter Frequency Jitter
measure pulse width accuracy	wid@lv	Statistics On or use Histogram	Width Jitter
measure adjacent cycle deviation	Dp@lv	Statistics On or use Histogram	Cycle-to-Cycle Jitter
count number of edges in a waveform	edge@lv	---	---
measure duty cycle	duty@lv	Statistics On or use Histogram	Duty Cycle Jitter
measure time interval error	tie@lv	Statistics On or use Histogram	Interval Error Jitter
measure n-cycle	n-cycle@lv	---	N-Cycle Jitter
measure skew	skew	---	Clock Skew
measure setup	setup	---	Setup
measure hold	hold	---	Hold

How to Use the Trend Tool

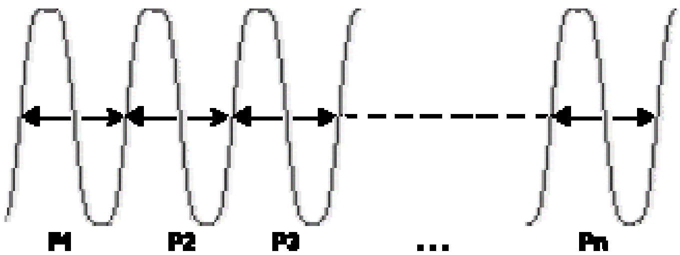
The Basic Idea

The Trend statistical tool displays the evolution of a timing parameter over time, in the form of a line graph. The graph's vertical axis is the value of the parameter; its horizontal axis is the order in which values are acquired.

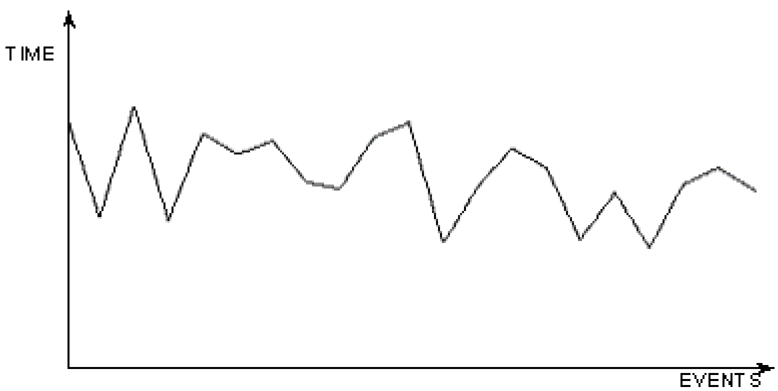
- **Display the waveform to be analyzed.**



- **Apply a timing parameter: period at level (p@lv), for example.**



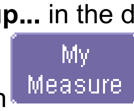
- **Plot the t**rend **of the parameter.**

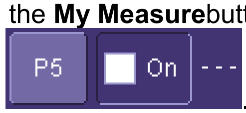


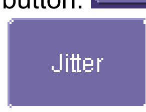
To Set Up and Configure Trend

Parameter Setup

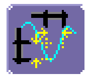
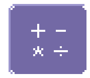
Before a Trend can be plotted, the timing parameter must be selected, as follows:

1. Touch **Measure** in the menu bar, then **Measure Setup...** in the drop-down menu.
2. In the "Measure" dialog, touch the **My Measure**button 

.
3. Touch an unused "Px" button: 

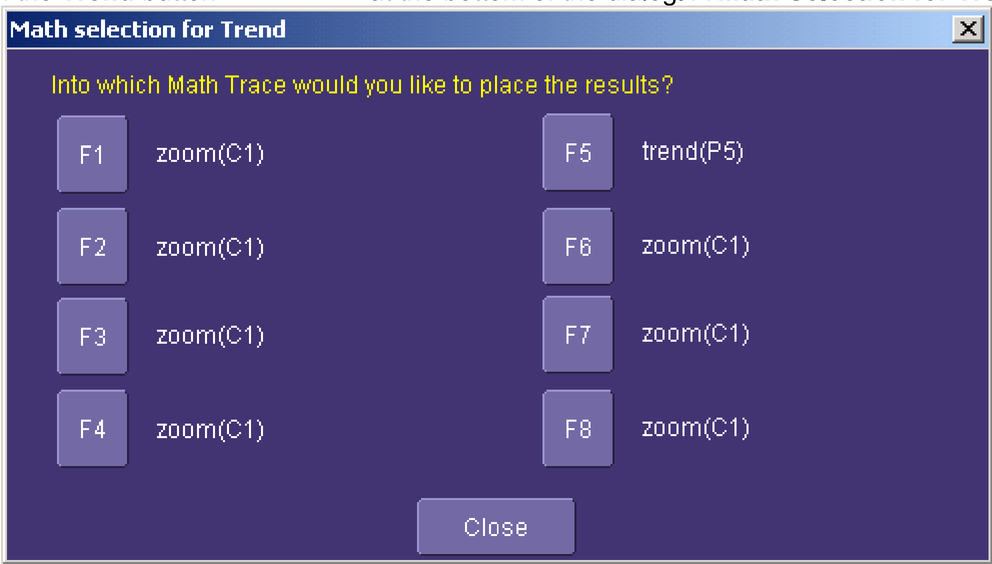
. The **Select Measurement** menu opens.
4. Touch the Jitter button 

on the **Select Measurement** menu and select a Jitter parameter. The setup dialogs for the Px position open.

5. Touch the **Measure On Waveforms** button  if you want to make a direct measurement on the source waveform. Or touch the **Math On Parameters** button 

if you want to make a measurement on the result of two other parameters that have been added, subtracted, multiplied, or divided. If you want to use this feature, you must have first set up those other two parameters.
6. Touch inside the **Source1** field and select a channel or memory waveform on which to make the parameter measurement. If you are performing **Math On Parameters**, Touch inside each of the **Source1** and **Source2** fields and select the source parameters.
7. Touch the **Trend** button 

at the bottom of the dialog. A **Math Selection for Trend**pop-up menu



opens from which you must select a math trace to display the trend.
8. A second setup dialog opens to the right of the main with more setup options. The options offered depend on the parameter you chose, but all include **Level is**, **Percent Level**, **Slope**, and **Hysteresis**. A **Find Level** button is also provided in this mini-dialog.

Option Field	Settings
Level Is	Percent Absolute
Percent Level	0 to 100%
Slope	Positive Negative
Hysteresis	0 div to 10 div

Note: The **Hysteresis** selection imposes a limit above and below the Level, which precludes measurements of noise or other perturbations within this band. The width of the band is specified in milli-divisions.

Guidelines for Use
1. Hysteresis must be larger than the maximum noise spike you want to ignore.
2. The largest value of hysteresis usable is less than the distance from the level to the closest extreme value of the waveform.
3. Unless you know the largest noise and closest extreme level that will ever occur on any cycle, leave some margin on both sides of the level.

Math Setup

Now that the parameter setups are done, you have to set up the Trend math function.

1. Touch **Math** in the menu bar, then **Math Setup...** in the drop-down menu.
2. In the "Math" dialog, touch the **Fx** tab 

for the math trace you chose to display the trend.
3. Touch the "Trend" tab 

in the setup dialog to the right of the main setup dialog. Touch inside the **Values to Trend** data entry field and enter a value from 20 to 20,000, using the pop-up numeric keypad.
4. You can touch the **Find Center and Height** button to automatically locate the center of the Trend waveform and to scale it to fit within the grid, without affecting zoom and position settings. Or you can enter specific values by touching inside the **Center** and **Height** data entry fields and typing in values, using the pop-up numeric keypad.
5. You can also touch the **Enable Auto Find** checkbox 

if you want the instrument to continuously self-adjust Center and Height.

Histogram and Trend Calculation

With the instrument configured for Histograms or Trends, the timing parameter values are calculated and the chosen function performed on each following acquisition. The Histogram or Trend values themselves are calculated immediately after each acquisition.

The result is a waveform of data points that can be used the same way as any other waveform. Other parameters can be calculated on it, it can be zoomed, serve as the **x** or **y** trace in an **XY** plot, or used in cursor measurements.

Acquisition Sequence

The sequence of events for acquiring Histogram or Trend data is:

1. Trigger
2. Waveform Acquisition
3. Parameter Calculations
4. Histogram Update
5. Trigger Re-arm

If the timebase is set in non-segmented mode, a single acquisition occurs prior to parameter calculations.

However, in segment mode an acquisition for each segment occurs prior to parameter calculations. If the source of the Histogram or Trend data is a memory, storing new data to memory effectively acts as a trigger and acquisition. Because updating the screen can take significant processing time, it occurs only once a second, minimizing trigger dead-time. (Under remote control, the display can be turned off to maximize measurement speed.)

Parameter Buffer

The instrument maintains a circular parameter buffer of the last 20,000 measurements, including values that fall outside the set histogram range. If the maximum number of events to be used in a histogram or trend is a number **N** less than 20,000, the histogram will be continuously updated with the last **N** events as new acquisitions occur. If the maximum number is greater than 20,000, the histogram or trend will be updated until the number of events equals **N**. Then, if the number of bins or the histogram or trend range is modified, the instrument will use the parameter buffer values to redraw the histogram with either the last **N** or 20,000 values acquired, whichever is the lesser. The parameter buffer thereby allows histograms or trends to be redisplayed using an acquired set of values and settings that produce a distribution shape with the most useful information.

In many cases the optimal range is not readily apparent, so the instrument has a powerful range finding function. If required, it will examine the values in the parameter buffer to calculate an optimal range and redisplay the histogram or trend using it. The instrument will also give a running count of the number of parameter values that fall within, below, and above the range. If any fall below or above the range, the range finder can then recalculate using these parameter values, while they are still within the buffer.

Parameter Events Capture

The number of events captured per waveform acquisition or display sweep depends on the type of parameter. Acquisitions are initiated by the occurrence of a trigger event. Sweeps are equivalent to the waveform captured and displayed on an input channel.

For non-segmented waveforms, an acquisition is identical to a sweep, but for segmented waveforms an acquisition occurs for each segment and a sweep is equivalent to acquisitions for all segments. Only the section of a waveform between the parameter cursors is used in the calculation of parameter values and corresponding histogram events.

The following table provides a summary of the number of Histogram or Trend events captured per acquisition or sweep for each parameter and for a waveform section between the parameter cursors.

Parameter	Number of Events Captured
Timing Parameters: p@lv, freq@lv, wid@lv, Δ p@lv, edge@lv,duty@lv, tie@lv, skew@lv, setup@lv, hold@lv	Unlimited number of events per acquisition
data	All data values in the region analyzed
duty, freq, period, width	Up to 49 events per acquisition
ampl, area, base, cmean, cmedian, crms, csdev, cycles, delay, dur, first, last, maximum, mean, median, minimum, nbph, nbpw, over+, over-, phase, pkpk, points, rms, sdev, delta dly, delta t@lv	One event per acquisition
f@level, f80-20%, fall, r@level, r20-80%, rise	Up to 49 events per acquisition

Zoom Traces and Segmented Waveforms

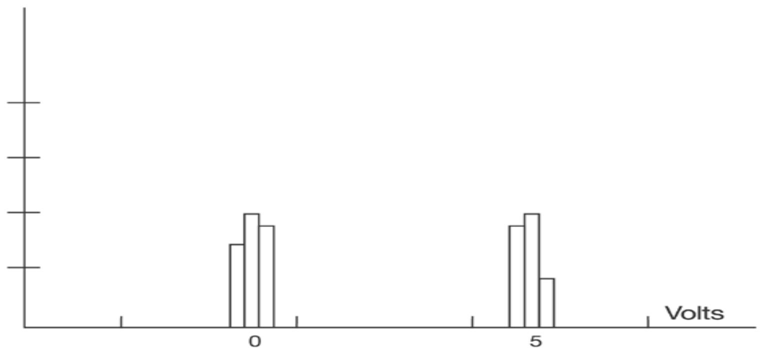
HTrends and istograms of zoom traces display all events for the displayed portion of a waveform between the parameter cursors. When dealing with segmented waveforms, and when a single segment is selected, the histogram or trend will be recalculated for all events in the displayed portion of this segment between the parameter cursors.

Histogram Peaks

Because the shape of histogram distributions is particularly interesting, additional parameter measurements are available for analyzing these distributions. They are generally centered on one of several peak value bins, known (with its associated bins) as a histogram peak.

Example

A histogram of the voltage value of a five-volt amplitude square wave is centered on two peak value bins: **0 V** and **5** *V* (see figure). The adjacent bins signify variation due to noise. The graph of the centered bins shows both as peaks.



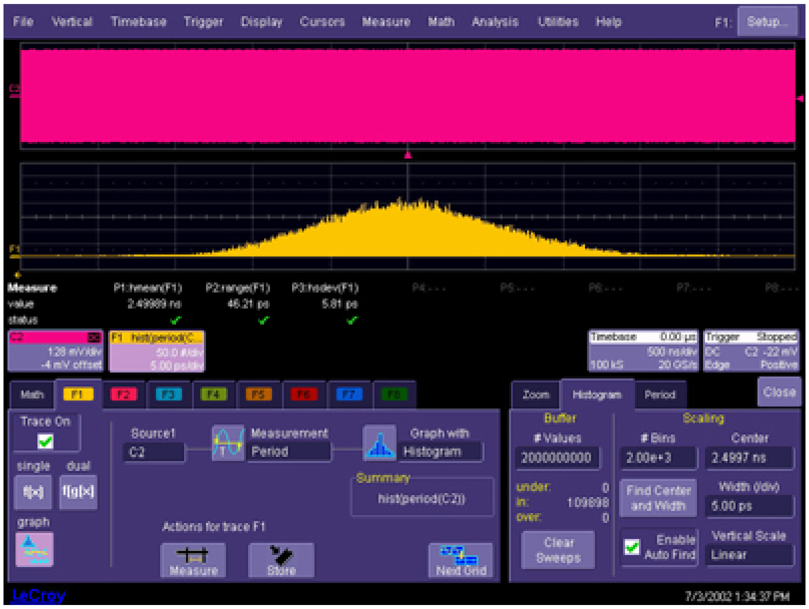
Determining such peaks is very useful because they indicate dominant values of a signal. However, signal noise and the use of a high number of bins relative to the number of parameter values acquired can give a jagged and spiky histogram, making meaningful peaks hard to distinguish. The instrument analyzes histogram data to identify peaks from background noise and histogram definition artifacts such as small gaps, which are due to very narrow bins.

Binning and Measurement Accuracy

Histogram bins represent a sub-range of waveform parameter values, or events. The events represented by a bin may have a value anywhere within its sub-range. However, parameter measurements of the histogram itself, such as **average**, assume that all events in a bin have a single value. The instrument uses the center value of each bin's sub-range in all its calculations. The greater the number of bins used to subdivide a histogram's range, the less the potential deviation between actual event values and those values assumed in histogram parameter calculations.

Nevertheless, using more bins may require a greater number of waveform parameter measurements to populate the bins sufficiently for the identification of a characteristic histogram distribution.

The next figure shows a histogram display of 17,999 parameter measurements divided or classified into 2000 bins. The standard deviation of the histogram sigma is 6.750 ps.



The instrument's parameter buffer is very effective for determining the optimal number of bins to be used. An optimal bin number is one where the change in parameter values is insignificant, and the histogram distribution does not have a jagged appearance. With this buffer, a histogram can be dynamically redisplayed as the number of bins is modified by the user. In addition, depending on the number of bins selected, the change in waveform parameter values can be seen.

In the next figure, the histogram shown in the previous figure has been recalculated with 100 bins. Note how it has become far less jagged, while the real peaks are more apparent. Also, the change in sigma is minimal (6.750 ps compared with 6.8 ps).

SDA Overview

Serial Data Analyzer Standard and Optional Capabilities

The Serial Data Analyzer is an instrument designed to provide comprehensive measurement capabilities for evaluating serial digital signals. In addition to the WaveShape Analysis features in the standard WaveMaster oscilloscope, the SDA provides eye pattern testing and comprehensive jitter analysis, including random and deterministic jitter separation, and direct measurement of periodic jitter, DDj, and DCD. The SDA also provides the capability to directly measure failed bits and to indicate their locations in the bit stream.

Note:
SDA - name of the instrument: Serial Data Analyzer
SDM - Serial Data Mask testing package, available on WaveMaster, WaveRunner Series, and WavePro Series oscilloscopes. Not available on the SDA.

SDA Capabilities

In addition to all the standard WaveMaster scope measurement functions, the SDA provides two other types of measurements: jitter and eye pattern. It also includes bit error rate analysis to the SDA. These measurements are available together in the summary screen, as well as in individual modes.

Measurements on the SDA are performed on long, continuous acquisitions of the signals under test. All jitter measurements and all displays are based on times of successive edges of the signal only; nothing is relative to the trigger. As a result, they are not affected by trigger jitter. Acquisitions should be long enough to include at least several thousand UI of the signal under test: 30,000 UI or more is optimal. To see low frequency jitter, it may be desirable to acquire longer records. Acquisitions can be up to the full available memory depth of the instrument (up to 258M samples with option SDA Zi-L-128), which may take considerable time to process.

Serial Data Analysis, which includes mask testing and jitter parameters (Rj, Dj, Tj, DDJ, Pj, DCD), is standard in the SDA. It is also available with option ASDA-J, which adds a major upgrade in capability over the standard SDA instrument. The different measurements available with each configuration are shown in Table 1.

SDM Capabilities

The capabilities of option SDM are standard in the SDA, so it is not available for purchase for the SDA. This option is only available for the WaveMaster, WaveRunner 6000 Series, and WavePro 7000 Series of oscilloscopes. SDM adds eye pattern testing to these oscilloscopes.

The option also adds other key components to the basic scope, including JTA2 with its TIE@lvl parameter. TIE@lvl is a JTA2 measurement that measures the time interval error of the crossing points of the signal under test and, with option SDM, also includes a golden PLL clock recovery module that is used for forming the eye pattern without an external trigger. Standard masks are included with option SDM as indicated in Table 2. Note that not all data rates can be tested with all oscilloscopes. The analog bandwidth limits the upper data rate that can be tested.

Advanced Capabilities

In its standard form, the SDA includes eye pattern testing with mask hit indication; Jitter testing, including jitter bathtub computation and separation of jitter into its random and deterministic components; as well as the breakdown of deterministic jitter into periodic, data dependent, and duty cycle distortion.

Standard advanced features include:

- Mask violation location This is the ability to list and view the individual bits that violate the selected mask. (Mask violation location takes much more time than jitter testing; acquisition size should be just a few thousand UI.)
- Filtered jitter Processes the time interval error trend vs. time with a user-selectable band-pass filter. This feature provides peak-to-peak and rms measurements of the jitter on the filtered waveform.
- ISI plot Generates an eye diagram including only those affects from data dependent sources. The user can select from 3 to 10 bit patterns for this test and can view the contribution from any individual pattern. The ISI plot is an alternate method for measuring data dependent jitter when the signal under test does not contain a repeating bit pattern.
- Bit error test with error map Measures the number of bit errors and error rate on the acquired waveform by converting the wave shape to a bit stream and comparing the result to a user-definable reference pattern. The data can be further divided into frames that can be arranged in a 3-dimensional map with frame number on the Y-axis, bit number on the X-axis, and failed bits shown in a light color.

Table of Standard Masks Included with Option SDM

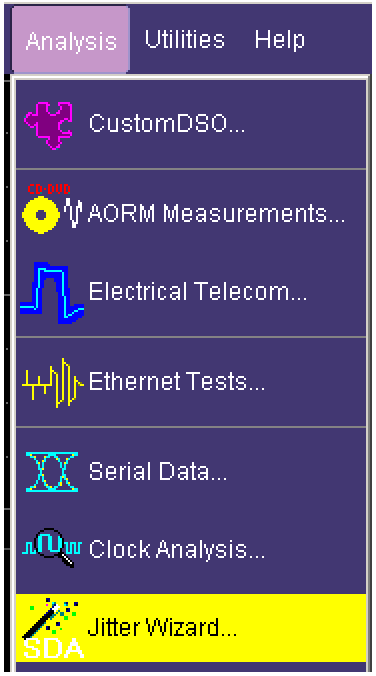
STANDARD	MODE
Gigabit Ethernet 1000Base-CX	TX normalized/absolute, RX
Gigabit Ethernet (optical long-haul) 1000Base-LX	TX
Gigabit Ethernet (optical short-haul) 1000Base-SX	TX
10GBase-LX4	TX normalized
10GFC-X4	Optical
Display Port 1.62 Gb/s	Source Connector TP2, Sink Connector TP3, Embedded TP3
Display Port 2.7 Gb/s	Source Connector TP2, Sink Connector TP3, Embedded TP3
DVI	Transmitter, receiver low, receiver high, cable test low, cable test high
Fibrechannel (optical) FC2125, FC1063	TX normalized
Fibrechannel (electrical) FC531, FC266, FC133	TX normalized, TX absolute, receiver
IEEE1394b	400 beta TP2 absolute, 400 beta TP2 normalized, 400 beta receive
Infiniband 2.5 Gb/s	Transmitter
RapidIO serial	3.125Gb/s TX (long haul/short haul), Rx 2.5Gb/s TX (long haul/short haul), Rx 1.25Gb/s TX (long haul/short haul), Rx
SONET	OC-1, OC-3, OC-12, OC-48, STS-1 eye, STS-3 transmit, STS-3 interface
SDH	STM-1, STM-4, STM-16
PCI Express	TX add-in transition, TX add-in de-emphasized, TX system transition, TX system de-emphasized, RX
Serial ATA 1.5 Gb/s	TX connector, RX connector
USB2.0	TX (near end/far end), RX (near end/far end), TX driver output, RX receiver input
XAUI	Driver far, Driver near

Table 5-2.Standard Masks Included with Option SDM.

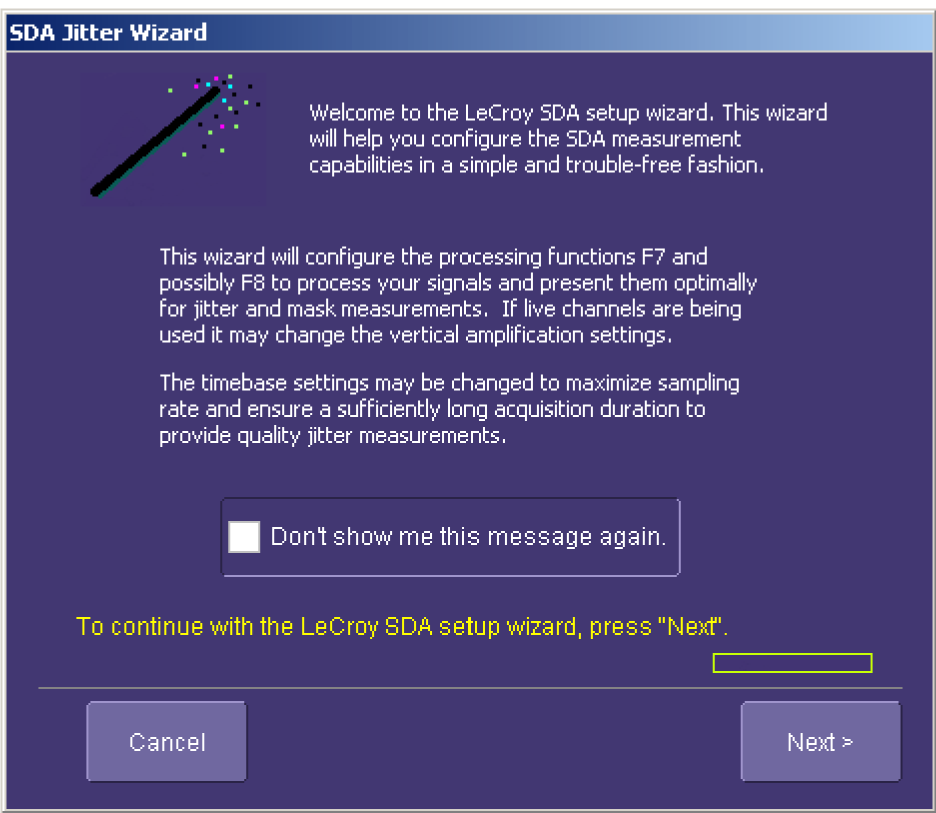
Jitter Wizard

The Jitter Wizard is a unique feature available with the ASDA-J option that automatically configures the SDA for optimal jitter measurement. As you provide information regarding the signal under test, the instrument selects the most appropriate settings. The wizard adjusts the sampling rate, memory depth, and vertical scale; and adds additional interpolation to the signal where appropriate.

The jitter wizard is accessed from the Analysis drop-down menu:



The initial dialog for the wizard gives you an overview of wizard operation. This dialog can be disabled after the first use of the wizard by checking the “Don’t show me this message again” checkbox. In this case, the dialog will not be displayed until the instrument is restarted.

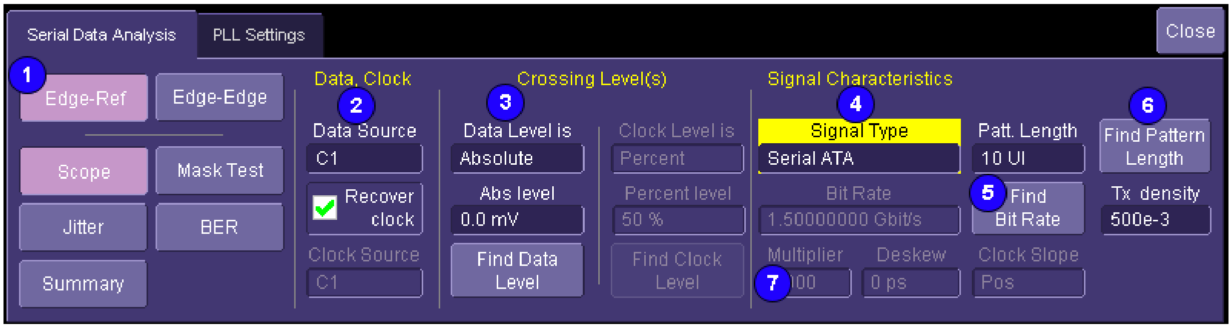


The remainder of the jitter setup wizard guides you through a series of dialogs that request input concerning the signal type and measurement method that is desired. Explanations of the various choices that can be made are also provided.

Each dialog has a default selection that represents the most common configuration. You can accept the defaults by simply touching the Next button in each dialog.

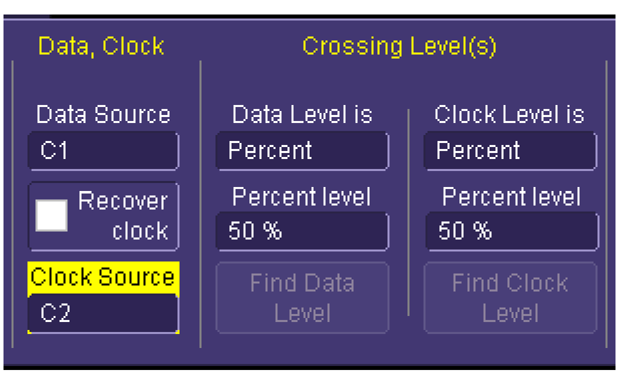
SDA Basic Setup

Note: Many of the measurements in the SDA require both a high sampling rate and long memory to compute accurately. Verify that you are in two channel mode (20 GS/s) and that at least 400k samples are being captured before performing any SDA measurements. Lower sampling rates can result in less accurate jitter measurements, and short record lengths can give incomplete eye patterns or jitter displays that diverge.



1. To access the SDA dialogs, press the **Serial Data Analyzer (SDA)** button on the Quick Access toolbar; or, touch **Analysis** in the menu bar, then **Serial Data** in the drop-down menu. The display defaults to the Scope dialog, which enables you to set up SDA measurements while you continue to view currently displayed waveforms. Touching the **Scope** button turns off SDA features.

   In the image above are shown two measurement modes: **Edge-Ref** and **Edge-Edge**. The Edge-Edge mode is explained separately in this manual; the information presented here applies mainly to the standard (Edge-Reference) mode.
2. Touch inside the **Data Source** field and select a data source, then touch inside the **Clock Source** field and select a clock input. If you want to recover the clock from the signal, touch the **Recover clock** checkbox; the **Clock Source** field will become inactive. The "Crossing level(s)" section in this dialog allows you to set the voltage level at which the signal timing is measured. The crossing level is set separately for the data and clock (if an external clock is selected) and can be either absolute or relative. The **Absolute** crossing level in volts (or watts for an optical signal) can be set directly, or can be found automatically by touching the **Find Data Level** button. The level is found by locating the midpoint between the highest and lowest signal levels in the current acquisition. The **Relative** level is automatically set to the selected percentage on each acquisition.
3. There are two crossing-level controls in the SDA main dialog: one for the data signal under test and another for the reference clock (if the **Recover clock** checkbox is unchecked). The crossing time used by both the jitter and eye pattern measurements is determined as the absolute time at which the signal level crosses the selected threshold. Touch inside the **Data Level Is** or **Clock Level Is** field and select either **Absolute** or **Percent** from the pop-up menu.



4. If you selected Absolute, touch inside the **Abs level** field and enter a voltage value. Alternatively, you can touch the **Find Data Level** or **Find Clock Level** button. If you selected Percent, touch inside the **Percent**

level field and enter a percentage value, using the pop-up keypad. The level is automatically set to the selected percentage on each acquisition.

Note: The absolute level setting should be used for jitter measurements over several acquisitions because the relative setting can potentially remove jitter by tracking slowly, thereby varying level shifts that could occur between acquisitions.

5. Touch inside the **Signal Type** field and select a standard signal type from the pop-up menu. The signal type defines the compliance masks and bit rates for the selected standard.

   If you selected **Custom** from the **Signal Type** menu, touch inside the **Bit Rate** field and enter a bit rate, using the pop-up keypad. By touching the **Find Bit Rate** button, you can have the bit rate determined automatically. This button is also available when a standard data type is selected.
6. The **Pattern Length**[The number of bits in the pattern.] control sets the repetition length of the data pattern in the signal under test. Serial data signals generally transmit a repeating data pattern of some sort. For example a PRBS7 pattern repeats every 127 bits while a K28.5 repeats every 20 bits. Enter the pattern length in the **Pattern Length** control. Alternatively, the SDA can automatically find the pattern length by means of the **Find Pattern Length** button. The **Find Pattern Length** button determines the pattern repetition length (if any) in the current acquisition and sets the **Pattern length** field to this value. The Pj measurement uses the pattern length value to determine which peaks in the jitter spectrum are caused by the pattern repetition. An incorrect value in the Pattern Length control will result in overestimating the Pj and, as a result, underestimating the random jitter.
7. The **Multiplier**, **Deskew**, and **Clock Slope** controls adjust the positioning of the clock relative to the data edges when an external clock source is selected. These controls are active only if the **Recover clock** checkbox is left unchecked. The **Multiplier** control is used when a sub-rate clock is applied. For example, if a clock signal at 1/10 the bit rate is used, the Multiplier control should be set to 10 in order to get a single eye pattern on the display.

The **Deskew** control enables positioning of the clock edges relative to the data signal. It shifts the clock signal relative to the data signal.

A clock signal goes through one complete cycle during each bit interval. The edge timing can be measured relative to the rising or falling slope of the clock by means of the **Clock Slope** setting.

The **Tx Density** control indicates the ratio of transitions in the data record to the total number of bits in the signal. Normally, an NRZ data stream does not have transitions during every bit interval. For example, there are cases where adjacent 1 or 0 values exist. On average, approximately 50% of the bit intervals have transitions so the Tx density is 500e-3. The actual acquired data set may have a transition density different from 50%, however. The total jitter measurement uses this transition density to normalize the measured histogram into a pdf[probability density function] . Many instruments simply assume 50% when performing this normalization which can lead to errors. These errors are eliminated in the SDA through the use of the measured transition density in the normalization process. The Tx density is automatically measured when the **Find Pattern Length** button is pressed.

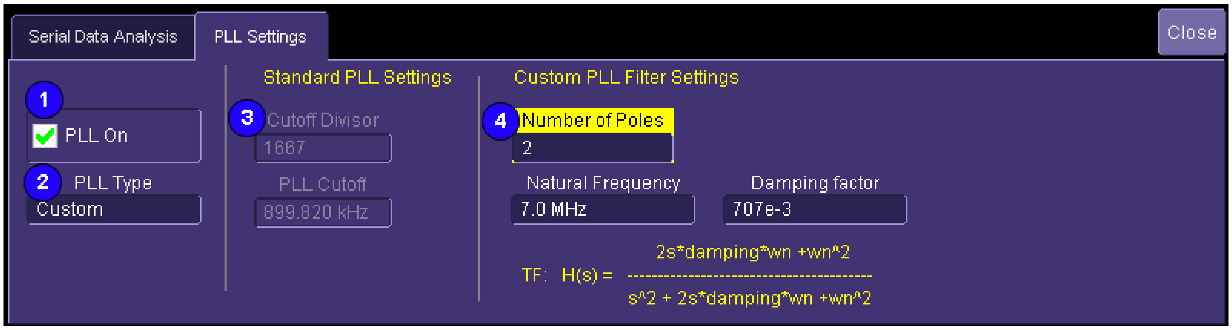
Note: If the pattern length exceeds the number of bits in the acquired waveform or there is no repeating pattern present, the control will indicate a value equal to the total number of bits in the current acquisition. In this case, the pattern length control should be set to 2 so that the Pj will not be underestimated

Note: An incorrect pattern length will result in an overestimated value for Pj. This overestimate will cause Dj to read high and Rj to read low.

PLL Setup

The **PLL Settings** dialog contains the controls to set the type and bandwidth of the digital PLL used in the jitter, eye pattern, and bit error rate tests. The PLL bandwidth limits the response of the recovered clock to high rate variations in the data rate. For example, a PLL bandwidth of 750 kHz will allow the recovered clock to track frequency variations below this rate, thereby removing their effect from jitter and eye pattern measurements. The software PLL implemented in the SDA allows you to choose among four types of PLL.

The selected PLL is applied to either the data stream under test or the selected clock source when the PLL On control is checked. The PLL recovers a reference clock from the selected source, which is used by all subsequent SDA measurements (jitter, eye pattern, and, bit error rate).



1. Place a check in the **PLL On** checkbox to enable it.
2. Touch inside the **PLL Type** field to select the type of PLL to be used in the clock recovery function. The choices are **FC Golden**, **PCI Express Gen1**, **PCI-Express G2 A 3dBpk 16 MHz fc**, **PCI-Express G2 B 3dBpk 8 MHz fc**, **PCI Express G2 C 1dBpk 5 MHz fc**, **DVI**, **FB-DIMM**, and **Custom**.
   A. **FC Golden** is the default selection and implements the “golden” PLL as defined in the Fibrechannel specification. By default, the golden PLL is set to a cutoff frequency of 1/1667 times the bit rate of the signal under test. This ratio can be adjusted from 1/10 to 1/1e6.
   B. The **PCI Express** PLL uses a filter that approximates the PCI-SIG compliance requirement. The PCI-SIG compliance procedure describes a processing algorithm that measures the average bit rate over 3500 consecutive unit intervals (UI or bit intervals). The 250 UI in the center of this 3500 UI window are then processed using the average bit rate as a reference clock. The 3500 UI window is then shifted by some number of UI and the process is repeated. Measurement continues until the end of the acquired data record is reached. The PCI Express PLL selection in the SDA models the sliding 3500 UI clock recovery and 250 UI processing windows using a digital low-pass filter whose cutoff frequency is approximately 1.5 MHz.
   C. The **DVI** selection follows the requirements of the DVI (Digital Video Interactive) and HDMI (High Definition Multimedia Interface) specifications. These specifications call out a clock recovery function that has a single-pole PLL loop response with a cutoff of 4 MHz.
   D. The **Custom** selection allows you to select either a first or second order loop response. The first order response allows you to select a pole frequency that sets the PLL cutoff, and a zero frequency that must be higher than the pole frequency that limits the stop-band attenuation.

      The second order PLL allows you to select the natural frequency and damping factor. The damping factor determines the transient behavior of the phase locked loop and is variable from 2 to 0.5. A damping factor above 0.707 results in an under-damped response in which the PLL will over-correct to a sudden change in frequency, but will react more quickly to the change. A damping factor below 0.707 will give an under-damped response that will react more slowly to sudden changes in frequency, but will not over-correct.

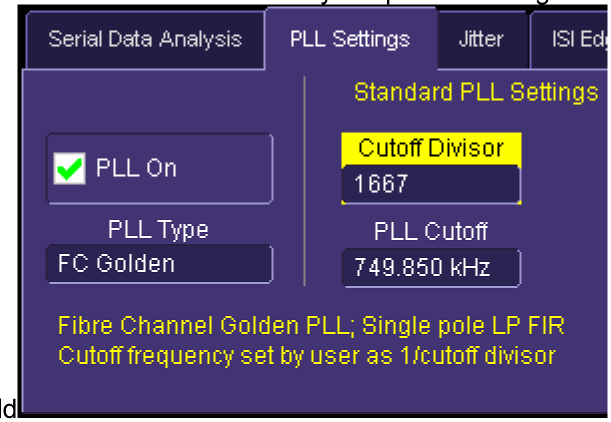
      The default value of 0.707 represents a critically damped response that will give the fastest reaction time without over-correcting.

      The second order PLL with a damping factor of 0.707 is specified in the serial ATA generation II document. This type of PLL is also very useful for measuring signals with spread-spectrum clocking because it can accurately track and remove the low-frequency clock spreading while allowing the signal jitter to be measured. The natural frequency is somewhat lower than the actual 3 dB cutoff frequency given by the following equation:

$$B_{3dB} = \frac{\omega_n}{2\pi}\left(2\zeta^2 + 1 + \sqrt{(2\zeta^2+1)^2 + 1}\right)^{1/2}$$

      The quantity ζ is the damping factor, and ω_n is the natural frequency. For a damping factor of 0.707, this relationship is $f_c = 2.06\ f_n$.
3. Settings for the **FC Golden** PLL: Touch inside the **Cutoff Divisor**[The PLL cutoff divisor is the value by which the bit rate is divided to compute the cutoff frequency for the loop bandwidth of the clock recovery operation for sequential eye pattern, jitter, and bit error rate functions. This control is variable from 10 to

1,000,000. A low PLL cutoff divisor means that the PLL will track and, therefore, attenuate jitter at higher frequencies. The default value of 1667 causes the clock recovery to operate as a "golden PLL," as defined



in the Fibrechannel specification.] field and enter a value by means of the pop-up keypad. The default value of 1667 is the industry standard for a "Golden PLL" and equals the ratio of the Bit Rate to the PLL Cutoff frequency. The **PLL Cutoff** frequency control reads the frequency corresponding to the Cutoff Divisor. Alternatively, the **PLL Cutoff** frequency may be entered and the nearest cutoff divisor will be computed from this entry.

4. Settings for the **Custom** PLL: Touch inside the **Number of Poles** control to select the order of the PLL. The number of poles can be 1 or 2. Touch inside the **Natural Frequency** control to set the natural frequency of the loop filter. The **Damping factor** control allows a damping factor setting between 2 and 0.5.

Summary

This is a quad-grid view of your signal: eye diagram, bathtub[The bathtub curve can be thought of as the "tails" of the TIE histogram, or the integral of the histogram. It is a logarithmic representation of bit error rate (BER). The greater the degree of separation between the sides of the bathtub curve, the better the BER. Deterministic jitter dominates the shape of the bathtub curve near the edges of the graph where the curves are at their highest amplitude, while random jitter determines the shape of the curve near the middle.] , TIE trend[This is a trend of the time interval error measurements between the data transitions and the reference clock. This is the time record of the information contained in the TIE histogram.] , and TIE histogram[A histogram of the time interval error for all bits in the waveform. Note that while the eye diagram and mask are displayed, mask hits are not detected in this view.] .

Note: The Summary screen does not allow mask testing.

To access the "Summary" dialog, do the following:

1. Press the **SDA** front panel button; or, touch **Analysis** in the menu bar, then **Serial Data** in the drop-down menu. The display will default to the "Scope" dialog, which enables you to set up SDA measurements while you continue to view currently displayed waveforms.
2. Touch the **Summary** button.

Mask Test

Press the **Serial Data Analyzer (SDA)** button on the Quick Access toolbar ; or, touch **Analysis** in the menu bar, then **Serial Data** in the drop-down menu. The display will default to the "Scope" dialog, which enables you to set up SDA measurements while you continue to view currently displayed waveforms. See the Basic Setup and PLL Setup sections for details on setting up the clock recovery.

Eye Setup

The PLL must be set up or an appropriate external clock must be supplied in order for the mask test function to operate.

1. Touch the **Mask Test** button. The "Eye Diagram" dialog opens.
2. Touch inside the **Mode** field and select a mode from the pop-up menu.

- **Traditional** – This conventional mode uses a trigger and one UI goes into the eye diagram per sweep. This is done using short acquisitions, is very slow, and adds trigger jitter to the signal jitter. This mode is likely to disappear and is only present because it's what people used to do in order to produce an eye diagram.
- **Sequential** – This is the typical SDA mode and uses long acquisitions. The SDA slices up the data and puts all the waveform UI's into the eye.
- **Transition** – Operates similar to Sequential mode, but divides the results into two separate eyes: ones comprised of UI's starting with a transition and ones that did not. PCIe refers to them as Transition and Non-Transition eyes. This display mode is useful for those serial data standards that utilize mask testing for both types of bit sequences (PCI Express and FB-DIMM Point to Point).
- **Gated** (Qualified) Eye Diagram – The Gated Eye Diagram mode utilizes a separate signal (the "Gate" or qualifier) to create dual eye diagrams based on the polarity of the Gate.
- **FSB** (Front Side Bus) – Operates similar to Sequential mode, but divides the results into two separate eyes based on which of a pair of signals occurred first.

3. Touch inside the **Persistence** field and select persistence mode: **Monochrome** or **Color graded**. The Monochrome selection will display frequency of occurrence in levels of intensity of the same color, while the Color graded selection will use a color spectrum to indicate frequency of occurrence.
4. If you would like to display another time-synchronized signal in its own grid, touch inside the **User Signal** field and select another signal source from the pop-up menu. This will enable you to see the signal correlated to the failed bits in the eye mask when using the eye mask failure locator.
5. The **Mask Type**[Each standard has a set of required tests. Some of the standards specify several types of masks. So a single standard can have a normalized mask and/or absolute mask for the transmitter. The standard requires you to make sure that the signal passes both the normalized and the absolute masks.] field allows you to select different modes for the selected standard. Individual modes for a given standard define specific masks; for example, transmit or receive.
6. Under **Measure** touch inside the **Type** field, then select a set of parameter measurements from the pop-up menu. The measurements made for each parameter set are as follows:

Amplitude	ehght(Eye)	Size of the vertical opening of the eye diagram. This parameter is defined as: (mean ones level - 3σ) - (mean zero level + 3σ)
	one(Eye)	Logic one level of an eye diagram. This is the mean value of the one (high) state.
	zero(Eye)	Logic zero level of an eye diagram. This is the mean value of the zero (low) state.
	eampl(Eye)	Difference between the mean one and zero levels of an eye diagram
Timing	rise	Transition time from 20% to 80% for a rising edge
	fall	Transition time from 80% to 20% of a falling edge
Eye	ewdth(Eye)	Size of the horizontal opening of an eye diagram, where σ is the standard deviation of the jitter. This value is the spacing between crossings minus 6σ.
	ER(Eye)	Ratio of the optical power levels in an eye diagram (extinction ratio), defined as log(p1/p0), where p1 is the power level for a "1" and p0 is the level of a "0" level.
	eyeBER(Eye)	Bit Error Rate estimated from an eye diagram
	ecross(Eye)	Level of the zero crossings in an eye diagram as a percentage of the eye amplitude
	avgpwr(Eye)	Average power level of a waveform. This is computed as the average of the sum of the ones and zeroes levels.

7. If you selected **Amplitude** or **Eye**, touch inside the **Slice Width**[The slice width is a percentage of the duration of a single bit, i.e., the part of the pattern over which the extinction ratio is measured. By setting a percentage value, you indicate how much of the central portion of the bit width to use.
Slice width is marked by the dashed vertical lines on the display.] field and enter a value from 0 to 100%.

Mask Margin

1. Touch inside the **X** field and enter a value from 0 to 100%. As you enlarge the mask's X margin, you lengthen the horizontal dimension, bringing the mask closer to your waveform. Consequently, you will have more failures.
2. Touch inside the **Y** field and enter a value from 0 to 100%. As you enlarge the mask's Y margin, you widen the vertical dimension, bringing the mask closer to your waveform. Consequently, you will have more failures.
3. Touch the **Vertical Auto fit** checkbox if you want to scale the eye pattern.

Note: The vertical autofit function sets the scaling of the eye pattern so that the one level is at the second vertical division above center, and the zero level is at the second division below center. The **Vertical Auto fit** checkbox is automatically checked or unchecked depending on the Signal Type that you selected. For example, when an absolute mask like XAUI is selected, the **Vertical Auto fit box** is unchecked; but, it is checked for a normalized mask like FC1063.

Scaling for absolute mask signals is accomplished by setting the vertical scale of the input signal.

Testing

1. Touch the **Mask Violation Locator** tab to display the mask testing dialog
2. Touch the **On** checkbox to begin testing the signal bits against the selected mask.
3. Failed bits are displayed in a table and as a waveform in the lower grid. The scale of the lower grid in which the failed bit waveform is set by the **Bits in Locator** field. The failed bit is centered around the middle two divisions and the vertical scale is the same as the eye pattern.
4. Touch the **Stop On Error** checkbox if you want to halt testing when an error occurs. But testing will continue until the end of the acquired waveform is reached.
5. Touch the **Show Location** checkbox, then touch inside the **Show** field and select **All** or **Near X-Y** from the pop-up menu. **All** means that all failures will be shown, regardless of where they occur in the eye pattern. **Near X-Y** means that if you select a failure marker, either by touching the marker on the waveform or by touching the bit number in the Failures list, only the subsequent failures near the selected marker will be shown.

Note: The X and Y coordinates indicate the position of the selected failure on a grid normalized to one, read from left to right and top to bottom.

6. Touch inside the **N Failures** field and, using the pop-up keypad, enter the number of failures after which data accumulation will end, up to 10,000. Multiple failures can occur on each bit, so the value of N failures will always be equal to or greater than the number of failed bits.
7. Touch inside the **Bits in Locater** field and, using the pop-up keypad, enter the number of bits to be displayed in the bad bits trace. This trace shows the part of the waveform that violates the mask.

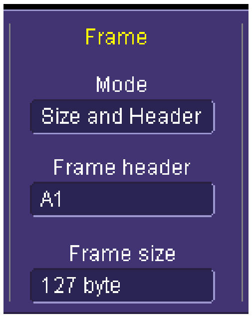
Bit Error Rate

1. Press the **Serial Data Analyzer (SDA)** button on the Quick Access toolbar; or, touch **Analysis** in the menu bar, then **Serial Data** in the drop-down menu. The display defaults to the "Scope" dialog, which enables you to set up SDA measurements while you continue to view currently displayed waveforms.
2. Touch the **BER** button. This brings you to the BER setup dialog.
3. Under **Pattern**, touch inside the **Method** field and select **Pattern** (expected bit pattern in the data stream), **PRBS** (a pseudo-random bit sequence), or **File** (a saved user ASCII hex file).
4. If you chose **Pattern**, use the pop-up keyboard to enter a hexadecimal number as a bit pattern.

   If you chose **PRBS**, touch inside the **PRBS Sequence** field and select a bit sequence from the pop-up menu. The digit at the end of PRBSx represents the number of shift registers to be used in the generation of the pseudo-random bit sequence used for BER testing. The sequences generated for each n value are as described in ITU-O.150.

   If you chose **File**, enter the path to the file, using the pop-up keyboard; or, touch the **Browse** button and navigate to the file.

5. Touch the **Show Error Map** checkbox to enable a display of the error map. The error map displays bit errors in a three-dimensional display, with errors shown in white and correct bits in dark gray. The display is a 2-color surface map with frame number in the Y direction and bit number in the X direction. A frame is indicated by the frame sync bit sequence and is composed of the bits from the end of one frame sync to the start of the next one. If frame sync bits are not specified, the bits are laid out in fixed-length rows, starting at the top left corner of the screen and proceeding from left to right and top to bottom in a raster pattern.
6. Touch the **Show Params** checkbox to display parameters BER, Nbits, False0, and False1.
7. You can use the fields under **Frame**, to align the same bits one on top of the other so that bit failures will appear as vertical lines.



Touch inside the **Mode** field and make a selection from the pop-up menu.

8. If you selected **Header** or **Size and Header**, touch inside the **Frame header** field and enter a hexadecimal number representing the prefix before the actual data. This prefix will be ignored and only the data will be examined.

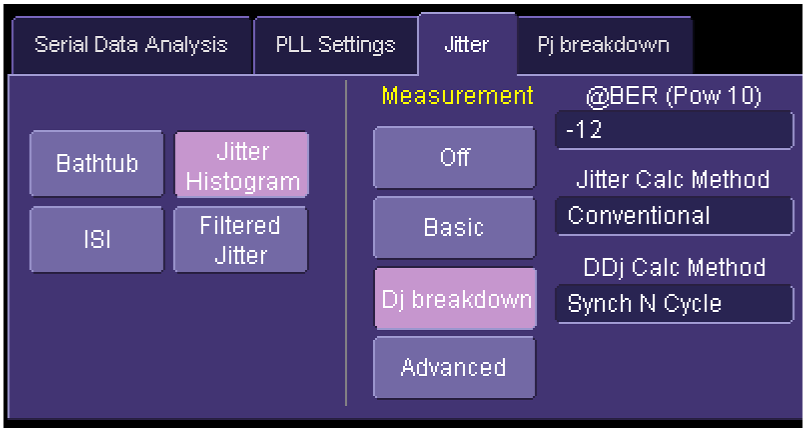
   If you selected **Size** or **Size and Header**, touch inside the **Frame size** field and enter the number of bytes in the frame, using the pop-up keypad. **Frame size** divides your waveform into equal pieces of the size that you enter.

Jitter Setup

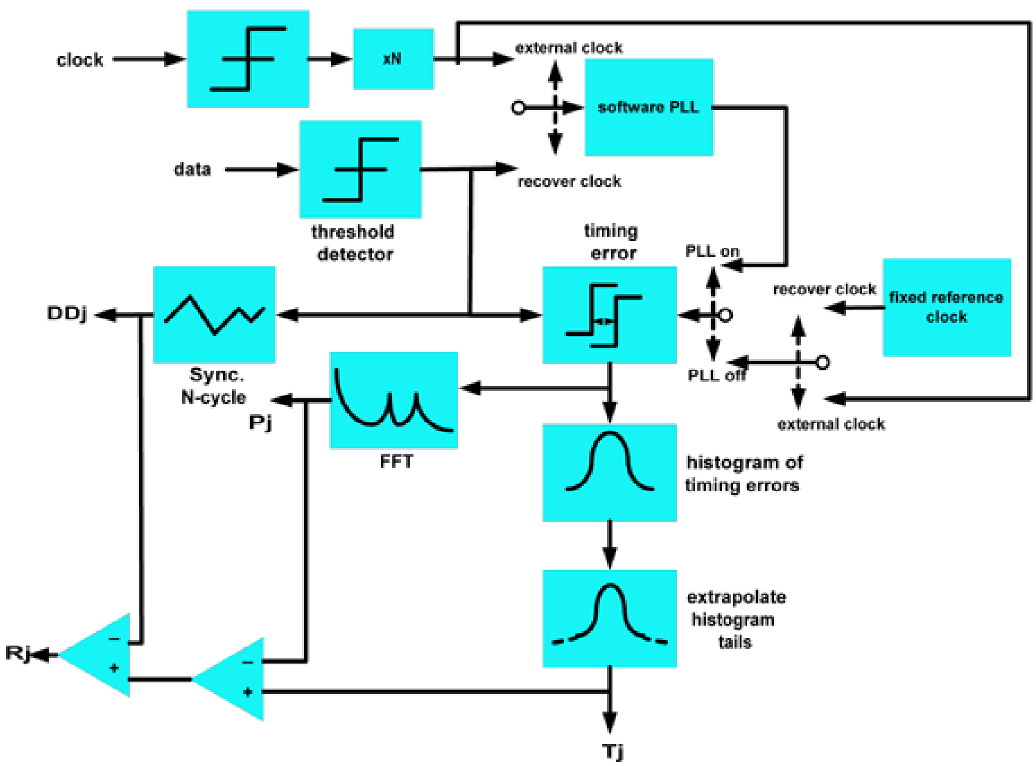
Jitter measurements are enabled by touching the Jitter button once the signal is set up. The Jitter dialog contains four display buttons and four measurement buttons. The displays include **Bathtub**, **Jitter histogram**, **ISI**, and **Filtered Jitter**. You can switch display modes without re-acquiring the signal.

The Measurement buttons control which jitter measurements are displayed. The **Basic** button displays the total jitter (Tj), Random jitter (Rj), and deterministic jitter (Dj). The **Dj Breakdown** button displays the components of Dj --- periodic jitter (Pj), duty cycle distortion (DCD), and data dependent jitter (DDj) --- in addition to the basic Rj, Dj, and Tj. The **Advanced** button replaces the Tj, Rj, Dj display with the peak-to-peak and rms values of the filtered jitter.

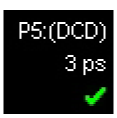
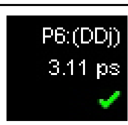
The SDA measures jitter by determining the timing error between the edge crossings of the data signal under test and a reference clock, which can be derived either from the data or an external clock. The measurement can also be performed directly relative to the edges of the external clock.



Total jitter is derived from the histogram of timing error measurements. The histogram of measurements is accumulated and the tails of the distribution are extrapolated to form an estimate of the histogram for a population of 10^{16} measurements. The extrapolated histogram is then integrated to determine the cumulative distribution function which forms the sides of the bathtub curve. The bathtub curve gives the eye opening (or 1UI - Tj(UI)) for all bit error rates down to 10^{-16}. The total jitter for a given bit error rate is found by measuring the width of the bathtub curve at the y-axis value corresponding to the desired bit error rate. The periodic jitter and data dependent jitter are measured directly on the sequence of timing error measurements, and the random jitter is found by measuring the difference between the total jitter and the components of deterministic jitter.



Jitter Measurements

	**Tj** Total jitter at a specific BER. The value is determined by the width of the curve expressing the confidence limits of the extrapolated TIE histogram. The value represents the expected range of values observed for a number of observed measurements equal to 1/BER. The Tj parameter is accumulated over all acquisitions since the start of the measurement or the last clear sweeps operation. The total population of the TIE histogram is indicated in the waveform descriptor box for the bathtub curve and Htie (the TIE histogram).
	**Rj** Random jitter is obtained indirectly by first determining deterministic jitter through various direct analyses of the TIE trend. Once all of Dj is found, the Rj value is deduced from the equation Rj = (Tj(BER) - Dj)/ $Tj_{(1\text{-}sigma)}$(BER), where the value for BER is typically 10e-12 (i.e., Rj is not a function, but a single value). The bit error rate is selected in the **@BER (pow 10)** control in the jitter menu. Rj is expressed in terms of an rms value, whereas Tj and Dj are expressed as peak-to-peak values.
	**Dj** Deterministic jitter is the peak-to-peak non-random part of the total jitter. This parameter is the sum of the measured peak-to-peak values of periodic jitter (Pj) and data dependent jitter (DDj). The DDj includes the effects of both inter symbol interference and duty cycle distortion.
	**Pj** Peak-to-peak magnitude of the periodic components of the TIE trend. It is measured by analyzing peaks in the Fourier transform of the trend of the time interval error. It is necessary to specify the pattern repetition length (if there is one) so that spectral lines harmonically related to this pattern rate do not contribute to the estimate of Pj, since the spectral energy associated with data pattern-related spectral components is included in the DDj measurement. The total periodic jitter is the complex sum of the spectral components listed in the table under the Pj breakdown tab.
	**DCD** Duty Cycle Distortion is the mean difference between the width of positive going pulses (low to high to low) and negative going pulses (high to low to high) measured over all pulses in the acquired waveform. The widths are measured at the same amplitude as specified for TIE (i.e., not necessarily at 50% of the signal amplitude). This measurement is a component of DDJ and included in the DDj value.
	**DDj** The peak-to-peak jitter caused by systematic effects related to the sequence of data transitions.

Pj Breakdown

The Pj Breakdown tab reveals a table of components of periodic jitter. This table lists the peak-to-peak amplitude and rate (frequency) of each Pj component. The components are listed from largest to smallest. The Pj readout below the grid on the display is the complex sum of the components listed in this table.

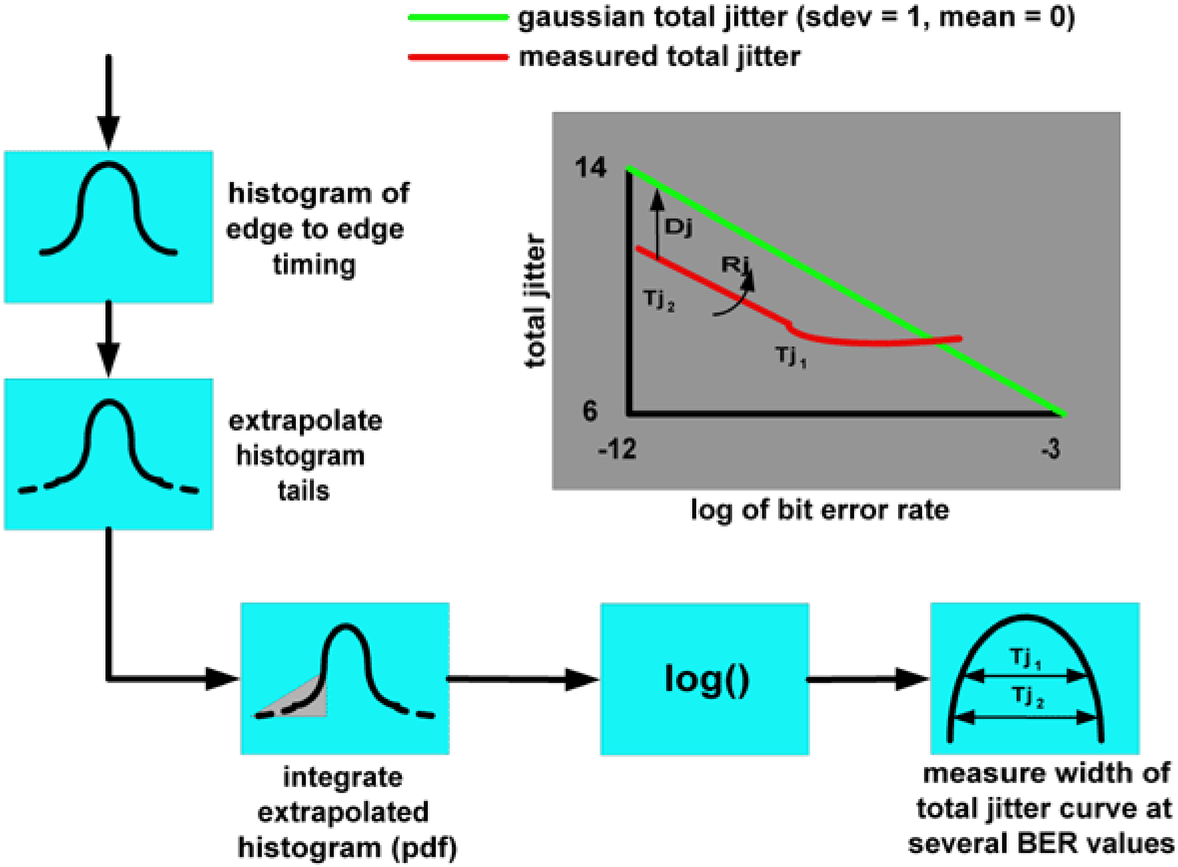
Alternate Jitter Breakdown Methods

There are two additional jitter breakdown methods. These methods are termed **Effective** and **MJSQ** and are selected in the **Jitter Calc. Method** control. The Effective and MJSQ methods provide alternate ways of determining the random and deterministic jitter but do not include the breakdown of deterministic jitter into periodic and data dependent parts. When either of these modes is selected, the jitter breakdown button the jitter menu becomes grayed (unavailable) and only the basic jitter display is shown. The DDj plot can still be viewed in this mode; however, the information from this plot is not used in the computation of Dj.

Effective Jitter

The effective jitter mode is entered when **Effective** is selected in the **Jitter Calc Method** control. Effective jitter is determined from the measured total jitter by evaluating the total jitter at several bit error rate values and solving Tj = $Tj_{(sigma=1)}$(BER)*Rje + Dje. The term $Tj_{(sigma=1)}$ is the total jitter of a Gaussian (normal) distribution of jitter with a standard deviation of 1 second. The two unknowns in this equation (Rje and Dje) are found by solving for several

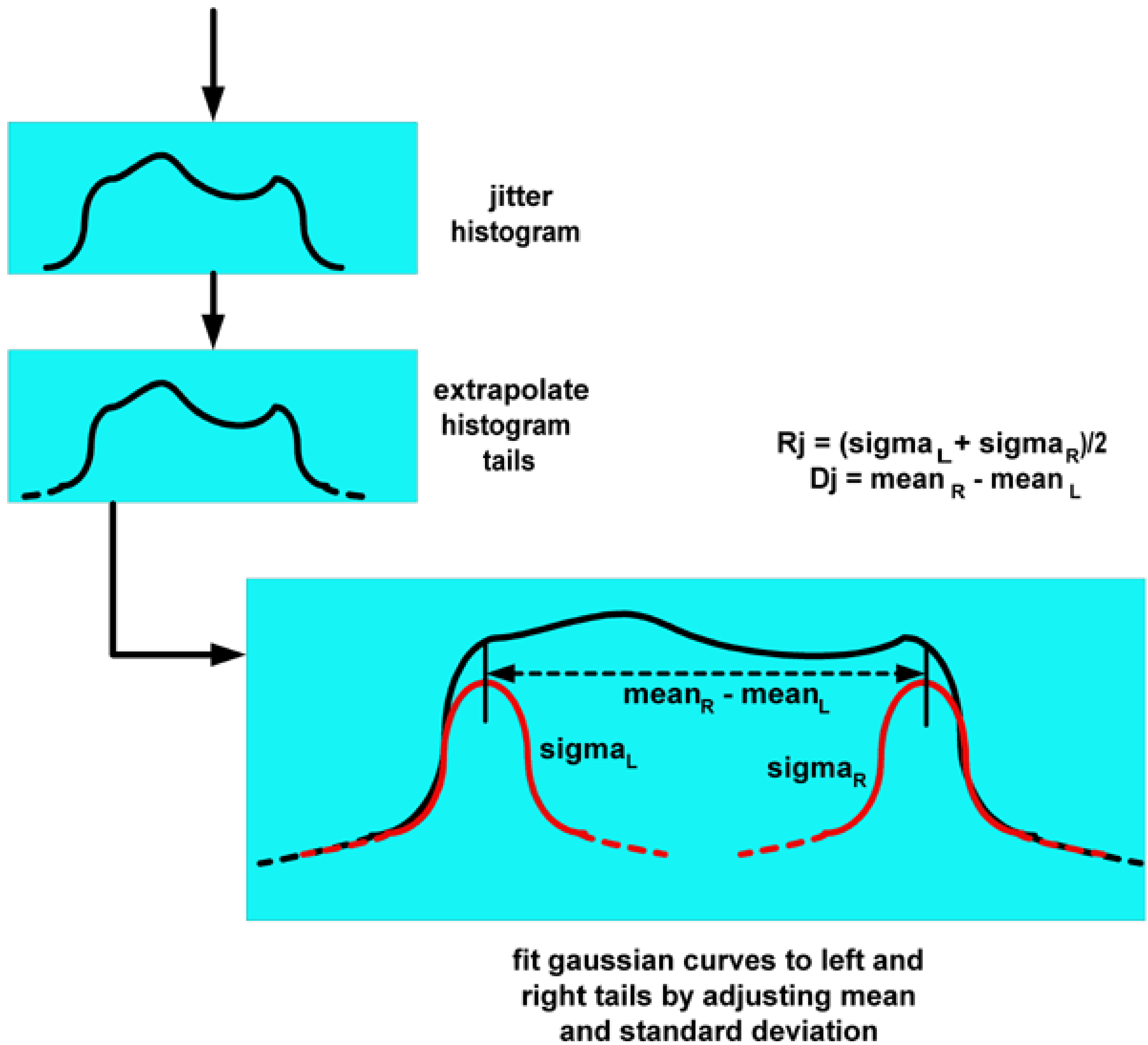
Tj values at BER levels below 10^{-10}. The jitter breakdown is the best-fit to the bathtub curve for very low BER values, but does not take into account the jitter contribution at the very top of the bathtub curve. The figure below shows the flow of the effective jitter measurement.



P2:(EffeRj) 1.466 ps	**EffeRj** Effective Random Jitter is one of the two constants used to estimate the function Tj(BER) for very low BER. Rje is obtained through the analysis of the growth of the extrapolated histogram of TIE as a function of BER, fitting the form: Tj(BER) = Dje + $Tj_{(sigma=1)}$(BER)*Rje
P3:(EffeDj) 3.536 ps	**EffeDj** Effective Deterministic Jitter is one of the two constants used to estimate the function Tj(BER) for very low BER. Rje is obtained through the analysis of the growth of the extrapolated histogram of TIE as a function of BER, fitting the form: Tj(BER) = Dje + $Tj_{(sigma=1)}$(BER)*Rje

MJSQ Jitter

The other method of determining the random and deterministic components is based on the procedure described in the Fibrechannel MJSQ (Methods for Jitter and Signal Quality) document. This method is similar to the effective jitter described above except that it operates directly on the jitter probability density function (pdf). A pair of normal distributions (one for each side) is found by adjusting their standard deviations so that they best fit the tails of the measured pdf. The difference between their mean values is the Dj and the average of their standard deviations is the Rj. This method is shown below.



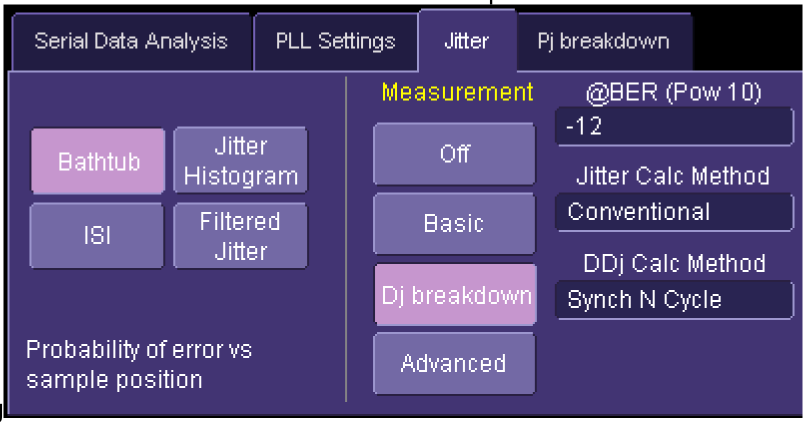
P2:(MJSQRj) 1.5 ps	**MJSQRj** MJSQ Random Jitter is determined by finding the average of the two standard deviations of the best-fit Gaussian distributions to the left and right sides of the measured jitter probability density function.
P3:(MJSQDj) 1.9 ps	**MJSQDj** MJSQ Deterministic Jitter is the difference between the mean values of the best-fit Gaussian distributions to the left and right sides of the measured jitter probability density function.

Bathtub Curve

The bathtub curve is the integral of the jitter probability density function (PDF -- derived by normalizing the extrapolated TIE histogram -- see the Theory section) for all possible sampling points within one unit interval (UI). The right side of the bathtub curve is the integral of the jitter pdf from approximately -1/2 of a UI to zero and the left side is the integral from +1/2 of a UI to zero, where the jitter PDF is centered about zero, and zero is defined as the ideal crossing time of a bit. The bathtub curve is normalized to 1 UI. It is a logarithmic representation of bit error rate (BER) vs. the sampling point. The greater the degree of separation between the sides of the bathtub curve, the better the BER.

The horizontal distance between the sides of the bathtub curve at a given Y value (bit error rate) is a measure of the eye width at that bit error rate.

1. Press the **Serial Data Analyzer (SDA)** button on the Quick Access toolbar; or, touch **Analysis** in the menu bar, then **Serial Data** in the drop-down menu. The display defaults to the "Scope" dialog, which enables you to set up SDA measurements while you continue to view currently displayed waveforms.
2. Touch the **Jitter** button. The Jitter Bathtub setup dialog



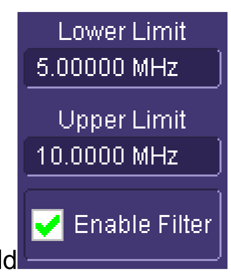
is displayed. You can enable several parameter measurements from this dialog.
3. For basic measurements, touch the **Basic** button, then touch inside the **@BER(Pow 10)**[The bit error rate (BER) is the percentage of bits that have errors relative to the total number of bits received in a transmission. Thus, a BER of 10^{-6} means that out of 1,000,000 bits transmitted, one bit was in error. The BER is an indication of how often a packet of data must be retransmitted because of error. The setting of the @BER control affects the indicated jitter values by selecting the vertical level at which the width of the total jitter (integrated TIE histogram) is measured. In general the Tj value will increase with decreasing bit error rate (more negative power of ten).] field. Enter an exponent value, using the pop-up keypad.

Jitter Filter

The jitter filter is a band-pass filter that is applied to the TIE-vs.-time data measured on the signal under test. The filtered waveform of the trend of TIE over time is displayed in the waveform grid when Filtered Jitter is selected. The filter is implemented as an FIR (Finite Impulse Response) digital filter with a maximum tap length of 2000. Not all high-pass filters are realizable, owing to the tap length restriction or the data record size or both. The instrument will apply a low-pass filter if the band-pass filter cannot be realized. The filter is applied only to the filtered jitter trace and does not affect the other jitter measurements: Tj, Rj, Dj etc. The peak-to-peak and rms value of the filtered jitter trace will be displayed below the trace.

The clock recovery PLL results in a jitter spectrum with a high-pass characteristic determined by the type and loop bandwidth of the PLL (see the Theory section). The combination of the low-pass jitter filter and the PLL cutoff will result in a band-pass filter for cases where the jitter filter function cannot realize the high-pass filter selected.

1. Touch the **Filtered Jitter** button.



2. Touch inside the **Lower Limit**field and enter a value, using the pop-up keypad. To realize a low pass-filter, select **Set to min.** in the pop-up keypad. In this case the lower frequency cutoff will be set by the PLL cutoff frequency.
3. Touch inside the **Upper Limit** field and enter a value, using the pop-up keypad.

TIE Histogram

A TIE histogram is often useful to display the raw measured histogram of the measured jitter values because it can give clues as to the sources of deterministic jitter and it also gives a good indication of whether the extrapolated total jitter value is likely to be accurate. The bins in the histogram at the extreme tails (the last 5 to 10 bins on each side) are used to determine the best-fit curve (see the Theory section for more details). Generally, if the histogram is smooth between the two tails, that is, there are no deep troughs in the shape, the curve fitting and extrapolation will give an accurate and stable Tj value.

There are some cases where very large amounts of ISI or periodic jitter cause deep troughs in the shape of the histogram. In these cases, a very large population is required in the histogram to ensure a sufficient population in the tail regions to achieve a reliable fit. Generally 1.5 to 2 million measurements in the histogram are sufficient to resolve the tails, but more data is always helpful. A histogram that appears smooth is a good indicator that the total jitter will be accurate.

The TIE histogram shows the measured jitter distribution that is the source for the bathtub curve and total jitter parameter.

1. Touch the **Jitter Histogram** button. The histogram will be displayed in the center of the bathtub curve.
2. Touch the **Bathtub** button to toggle the histogram display off.

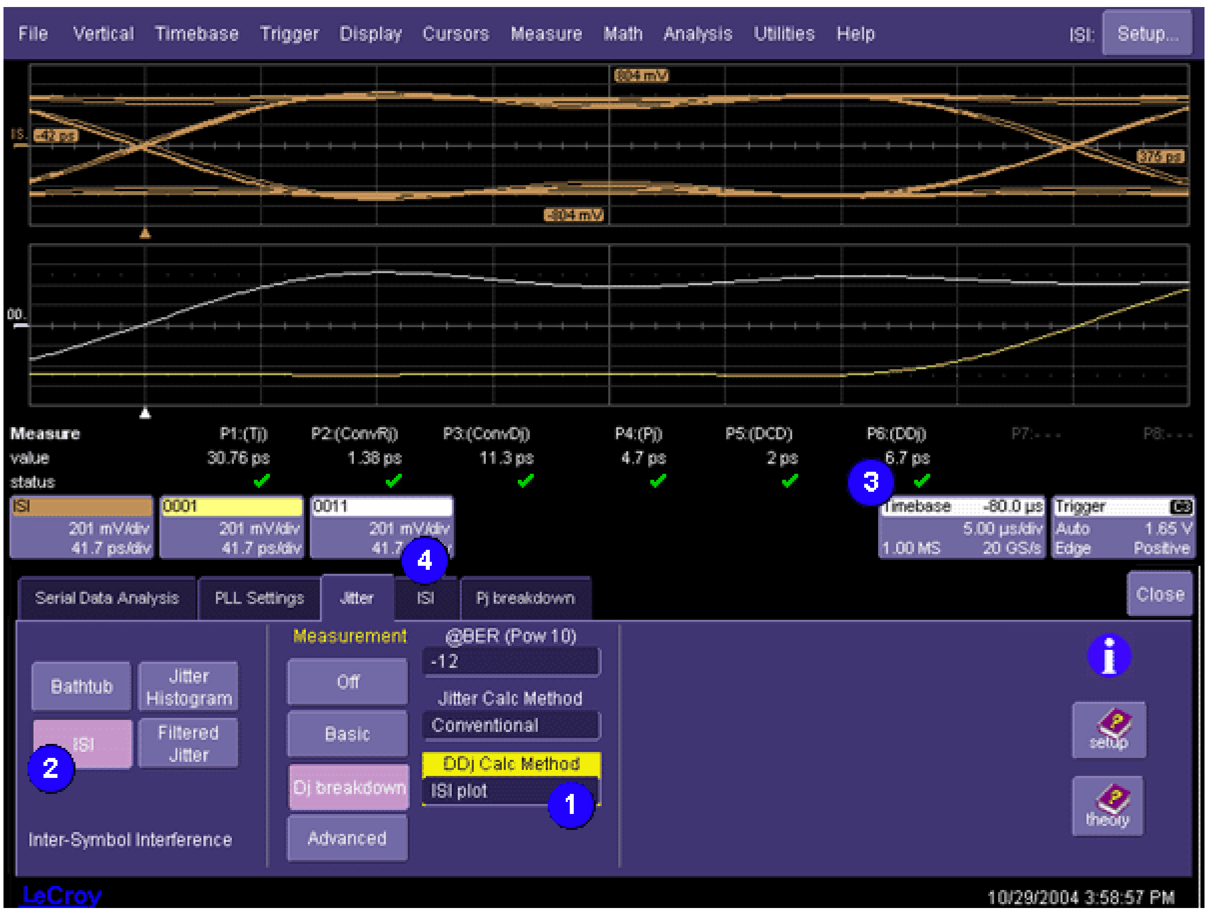
DDJ (Synchronous N-Cycle Plot)

The DDj (data dependent jitter) is measured on the data stream when the **Conventional** jitter breakdown method is selected. The DDj is measured by finding the average crossing time for each edge in the data pattern and comparing this to the nominal crossing time, based on the expected bit rate of the signal under test. This measurement requires a repeating data pattern to operate. If the data pattern in the signal does not repeat at least once in the acquired data record, the DDj parameter will indicate zero even though this type of jitter is present. A graphical display of the DDj can be viewed by clicking the **ISI** button in the **Jitter** menu. This display shows the position error of the average crossing point for each edge in the pattern.

1. Touch inside the **DDj Calc Method** control and select **Synch N Cycle** for a repeating pattern (available for Edge-Ref or Edge-Edge method).
2. Touch the **ISI** button.
3. Note the change in the DDj measured value.
4. Touch the **ISI Edge-Edge** tab.
   - A. Touch the **Detect Pattern** button to find the pattern and length. Normally this is done automatically when the signal is set up in the SDA main menu. The **Patt. Length** control in the SDA main dialog indicates the repetition length of the pattern.
   - B. The pattern is indicated in Hexadecimal in the **Pattern** window. This window allows you to view and edit the pattern. The **Find Pattern Length** button in the SDA main dialog searches the pattern for specific hex values.
   - C. Check the **Error Relative to First edge** checkbox to display the plot of edge position errors relative to the first transition in the data pattern.
   - D. Check the **Show Graphic Bit Pattern** checkbox to display a trace showing the bit pattern in the grid with the synchronous n-cycle plot. The transitions of the pattern line up with the measurement points in the plot.

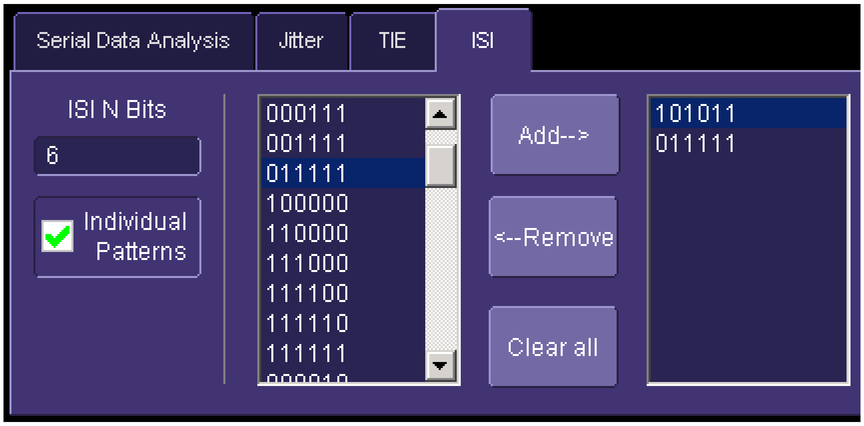
DDj (ISI plot)

A repeating data pattern is not always present in cases where there is no ability to program the PHY under test. There is an alternate method of measuring data dependent jitter that does not rely on a repeating pattern. This plot shows an eye pattern consisting of the averaged waveform trace from each pattern in the data stream of a specified length, which can be adjusted from 3 to 10 bits. The value of DDj is the width of the zero crossing at the right side of the ISI eye at the threshold level set by the DDj threshold value.



1. Select ISI Plot in the **DDj Calc. Method** control for a random pattern (available only for Edge-Ref jitter method).
2. Touch the **ISI** button; the ISI plot will be displayed.
3. Note the change in the DDj measured value.
4. Touch the **ISI** tab in the **Jitter** dialog.

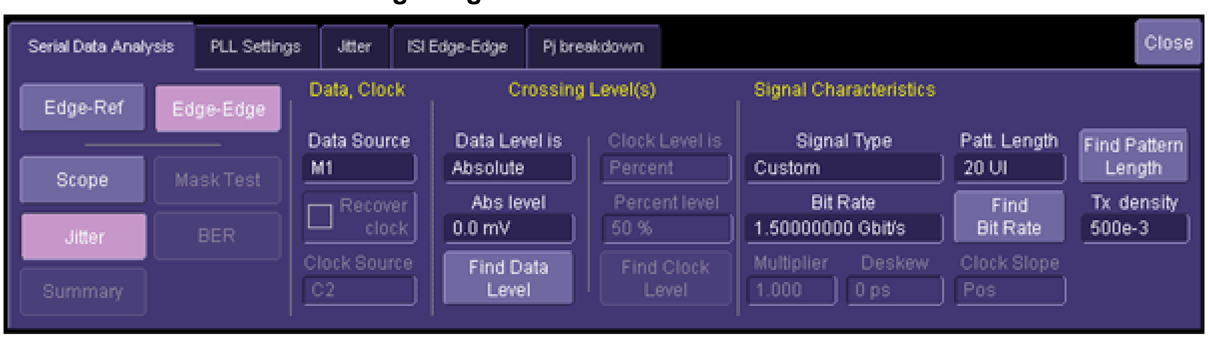
A. Touch inside the **ISI N Bits** field to set the pattern length for the ISI plot.
B. Touch the **Individual Patterns** checkbox to display the bit patterns of the length specified in the **ISI N Bits** field. Only the patterns that exist in the acquired data stream will be displayed:



C. Touch any bit pattern in the list, then touch **Add** to display that trace. Up to seven patterns can be placed in the box at the right and displayed as traces.
D. To clear any pattern and trace from the screen, touch that pattern in the list box on the right, then touch **Remove**. Touch **Clear all** to remove all patterns.
E. The averaged traces become clearer as more occurrences of each pattern are observed. If one of the traces does not seem to get clearer, try reducing the number of bits considered **ISI N Bits** from 6 to a lower value.

Edge-to-Edge Basic Setup

In this mode, all jitter measurements are made between adjacent edges of the data pattern and no reference clock is used. Only jitter measurements are available in this mode, so the mask test, bit error rate, and summary buttons are inactive when the **Edge-Edge** mode is selected.



1. Touch inside the **Data Source** field and select a data source.
2. The **Crossing Level(s)** section in this dialog allows you to set the voltage level at which the signal timing is measured. It can be set to either absolute or percent. The absolute crossing level --- in volts (or watts for an optical signal) --- can be set directly or can be automatically found by touching the **Find Data Level** button. This level is determined by locating the midpoint between the highest and lowest signal levels in the current acquisition. The percent level is automatically set to the selected percentage on each acquisition.
3. If you selected absolute, touch inside the **Abs level** field and enter a voltage value. Alternatively, you can touch the **Find Data Level** or **Find Clock Level** button. If you selected Percent, touch inside the **Percent level** field and enter a percentage value, using the pop-up keypad. The level is automatically set to the selected percentage on each acquisition.

Note: The absolute level setting should be used for jitter measurements over several acquisitions because the relative setting can potentially remove jitter by tracking slowly, thereby varying level shifts that could occur between acquisitions.

4. Touch inside the **Signal Type** field and select a standard signal type from the pop-up menu. The signal type defines the bit rates for the selected standard.

5. If you selected **Custom** from the **Signal Type** menu, touch inside the **Bit Rate** field and enter a bit rate, using the pop-up keypad. By touching the **Find Bit Rate** button, you can have the bit rate determined automatically. This button is also available when a standard data type is selected.
6. The **Pattern Length** control sets the repetition length of the data pattern in the signal under test. Serial data signals generally transmit a repeating data pattern of some sort. For example a PRBS7 pattern repeats every 127 bits while a K28.5 repeats every 20 bits. Enter the pattern length in the **Pattern Length** control.

   Alternatively, the SDA can automatically find the pattern length by means of the **Find Pattern Length** button. The **Find Pattern Length** control determines the pattern repetition length (if any) in the current acquisition and sets the **Pattern length** control to this value. The Pj measurement uses the pattern length value to determine which peaks in the jitter spectrum are caused by the pattern repetition. An incorrect value in the Pattern Length control will result in overestimation of the Pj and, as a result, underestimation of the random jitter.
7. The **Tx Density** control indicates the ratio of transitions in the data record to the total number of bits in the signal. Normally, an NRZ data stream does not have transitions during every bit interval. For example, there are cases where adjacent 1 or 0 values exist. On average, approximately 50% of the bit intervals have transitions, so the Tx density is 500e-3. The actual acquired data set may have a transition density different from 50%, however. The total jitter measurement uses this transition density to normalize the measured histogram into a pdf[probability density function] . Many instruments simply assume 50% when performing this normalization, which can lead to errors. These errors are eliminated in the SDA through the use of the measured transition density in the normalization process. The Tx density is automatically measured when the Find Pattern Length button is pressed.

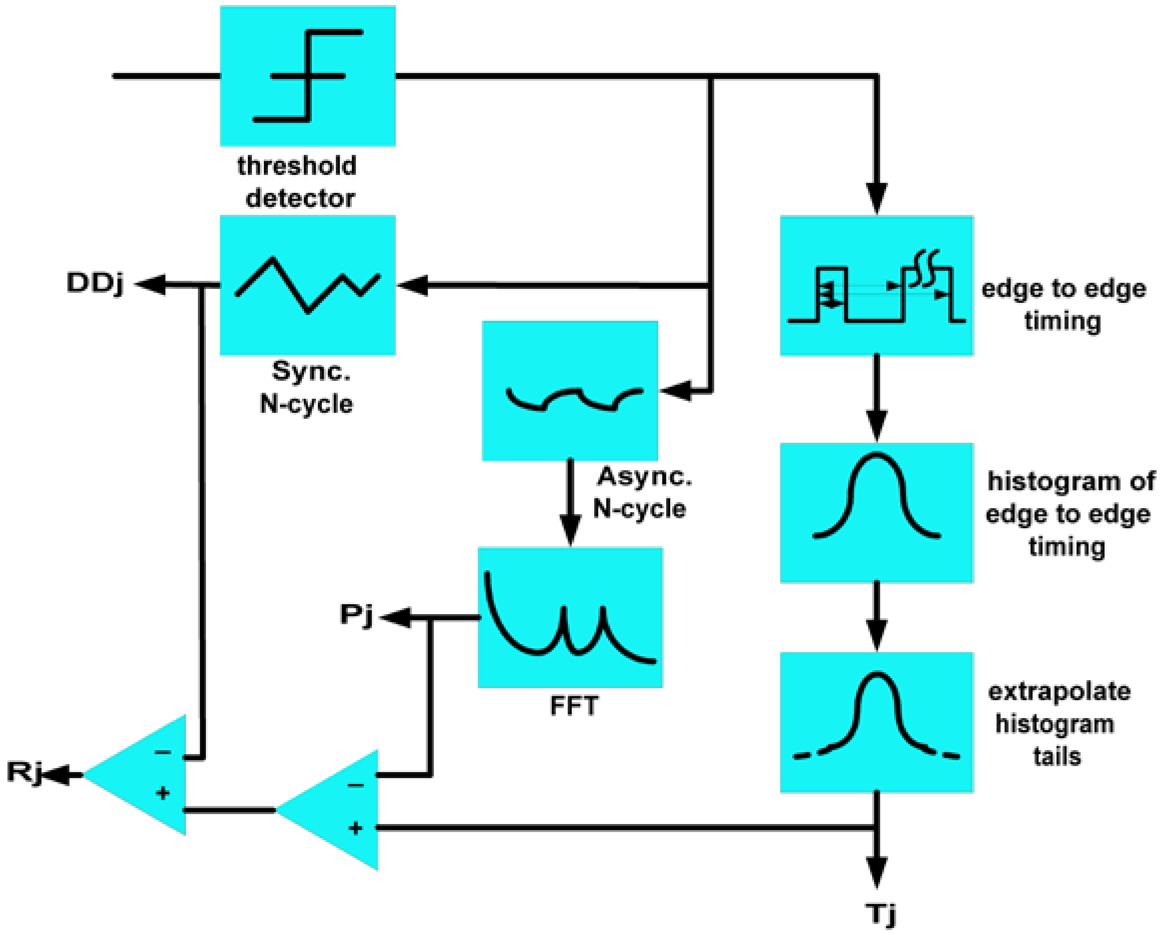
Edge-Edge Jitter Measurements

Jitter measurements are enabled by selecting **Jitter** in the SDA main menu when the mode is set to Edge-Edge. The available displays include a jitter bathtub curve, TIE histogram, ISI display, and N-cycle vs. N plot.

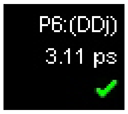
The Measurement buttons control which jitter measurements are displayed.

Basic	The **Basic** button displays the total jitter (Tj), Random jitter (Rj), and the deterministic jitter (Dj).
Dj breakdown	The **Dj Breakdown** button enables the display of the components of Dj --- periodic jitter (Pj), duty cycle distortion (DCD), and data dependent jitter (DDj) --- in addition to the basic Rj, Dj, and Tj.
Advanced	The ASDA-J option adds the **Advanced** button which, when selected, replaces the Tj, Rj, Dj display with the peak-to-peak and rms values of the filtered jitter.

The SDA measures jitter by determining the timing between the edge crossings of the data signal under test and comparing these measurements with the nominal unit interval (UI). A flow diagram of the edge to edge jitter measurement is shown below. The total jitter is derived from the histogram of timing error measurements. The histogram of measurements is accumulated and the tails of the distribution are extrapolated to form an estimate of the histogram for a population of 10^{16} measurements. The extrapolated histogram is then integrated to determine the cumulative distribution function, which forms the sides of the bathtub curve. The bathtub curve gives the eye opening (or 1UI - Tj(UI)) for all bit error rates down to 10^{-16}. The total jitter for a given bit error rate is found by measuring the width of the bathtub curve at the y-axis value corresponding to the desired bit error rate. The periodic jitter and data dependent jitter are measured directly on the sequence of timing error measurements and the random jitter is found by measuring the difference between the total jitter and the components of deterministic jitter.



	**Tj** Total jitter at a specific BER. The value is determined by the width of the curve expressing the confidence limits of the extrapolated TIE histogram. The value represents the expected range of values observed for a number of observed measurements equal to 1/BER. The Tj parameter is accumulated over all acquisitions since the start of the measurement, or the last clear sweeps operation. The total population of the TIE histogram is indicated in the waveform descriptor box for the bathtub curve and Htie (the TIE histogram)
	**Rj** Random jitter is obtained indirectly by first determining deterministic jitter through various direct analyses of the TIE trend. Once all of Dj is found, the Rj value is deduced from the equation Rj = (Tj(BER) - Dj)/ $Tj_{(1\text{-sigma})}$(BER), where the value for BER is typically 10e-12 (i.e., Rj is not a function, but a single value). The bit error rate is selected in the **@BER (pow 10)** control in the jitter menu. Rj is expressed in terms of an RMS value, whereas Tj and Dj are expressed as peak-to-peak values.
	**Dj** Deterministic jitter is the peak-to-peak non-random part of the total jitter. This parameter is the sum of the measured peak-to-peak values of periodic jitter (Pj) and data dependent jitter (DDj). The DDj includes the effects of both inter-symbol interference and duty cycle distortion.
	**Pj** Pj is the peak-to-peak magnitude of the periodic components of the TIE trend. It is measured by analyzing peaks in the Fourier transform of the trend of the time interval error. It is necessary to specify the pattern repetition length (if there is one) so that spectral lines harmonically related to this pattern rate do not contribute to the estimate of Pj, since the spectral energy associated with data pattern-related spectral components is included in the DDj measurement. The total periodic jitter is the complex sum of the spectral components listed in the table under the **Pj breakdown** tab.

	**DDj** The peak-to-peak jitter caused by systematic effects related to the sequence of data transitions.

Edge-Edge Jitter Measurement Controls

Adjust Rj

The TIE measurements in the edge-to-edge mode are measured by determining the timing between transitions of the data signal and comparing this to the nominal UI. The measurement is essentially the difference between two random values (the location of each transition in time). The random part of this TIE measurement (random jitter) is the sum of the random jitter values from each of the two edges and, as a result, is larger by a factor of the square root of 2 compared to the actual TIE (measured relative to a reference). The **Adjust Rj** control divides the measured edge-edge Rj by the square root of 2 to convert it to the equivalent edge-reference value. At the same time, the total jitter is recomputed when this control is selected by multiplying the adjusted Rj by the appropriate number of standard deviations determined by the selected BER (14 for 10e-12 BER). The result is that Tj and Rj more closely approximate the edge-reference measurements when the **Adjust Rj** control is checked.

Async N Cycle Plot

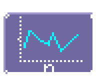
In the absence of a reference clock, periodic jitter cannot be directly measured from the data record because there is no timing reference on which to base the Fourier transform. The SDA employs a function called the Alternate N Cycle plot to measure periodic jitter on edge-to-edge TIE values. The plot displays the standard deviation of a histogram of the timing variations for transitions in the data signal that are spaced *n* UI apart. Each Y-value in this plot represents the standard deviation of the histogram for a given value of *n* as *n* varies from 1 to the maximum value set in the **Max N** field, the upper limit of which is determined by the number of UIs in the analyzed signal. The **HPF Corner Freq.** readout displays the minimum frequency jitter component that can be measured.

Edge-Edge Spacing Controls

The histogram and bathtub curve of TIE measurements are derived by determining the time interval between transitions of the data signal. The timing is measured between adjacent edges that are not always 1 UI apart, depending upon the data pattern. The **For Edge-Edge Use All Edges** checkbox, when checked, includes all adjacent transitions in its measurement. When this control is un-checked, only transitions that are the selected number of UI apart are measured. In this case, the measurement compares transitions 1 with N, 2 with N+2, 3 with N+3, etc. The **Use only UI spacing** control sets the UI spacing to be used in the jitter measurement.

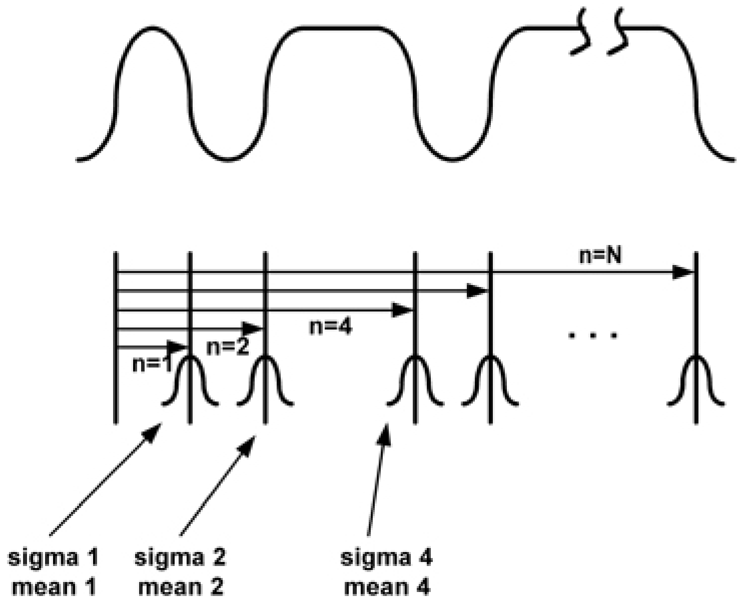
SDA Function Reference

There are several specialized math functions that appear in the SDA math menu. These functions make up the building blocks of the SDA jitter and eye pattern measurements and are available from the **Select Math Operator** menu. The following table presents an overview.

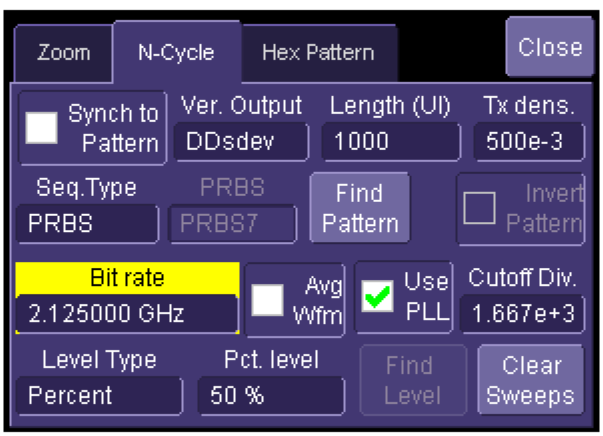
	AltNcycle	Alternate N-cycle jitter plot -- this function displays a plot of the standard deviation of the jitter for all edges spaced N UI apart for 1 < N < Nmax. Where Nmax can be set up to 2M.
	Htie to BER	Extrapolates a histogram of time interval error measurements to the desired bit error rate. This function creates the bathtub and total jitter curves. Total jitter, as well as the effective and MJSQ breakdowns, use the output of this function.
	Jitter filter	Filter the trend of jitter values with the specified band-pass filter. This function expects a trend of jitter values as an input. It is further required that the measurements be time-continuous, that is, there is one measurement for each consecutive UI in the data signal.
	Slice2Persist	This function takes a waveform and divides it up into sequences that are 1 UI long based on a separate clock input. The output is a persistence map showing the eye pattern.

AltNcycle

Timing of the transitions in the data waveform is measured for each transition and plotted as a function of the number of unit intervals over which the timing is measured. The **N-Cycle** dialog is used to control how this measurement is performed, and the diagram and table below describe the function in detail. The N-cycle plot displays the mean or standard deviation of the edge placement in the waveform relative to each other (data to data) or to a reference clock (clock to data).



The N-cycle plot displays the mean or standard deviation of the distribution of timing for edges spaced n UI apart as a function of n. The value of n can vary from 1 to some maximum value, including all intermediate values, or can be synchronous with the transitions in a repeating pattern.



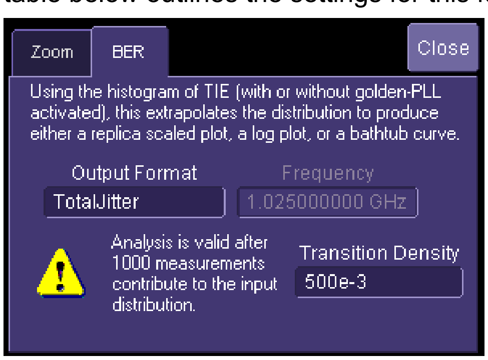
AltNcycle Control Summary

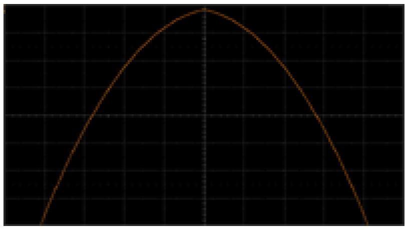
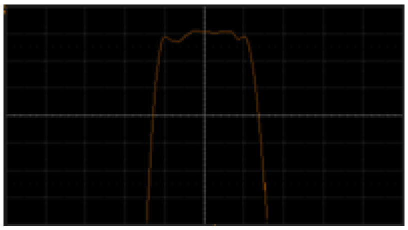
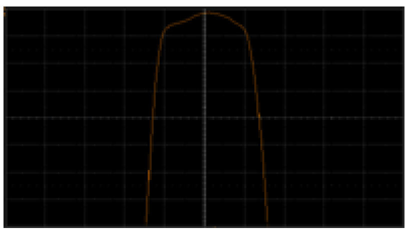
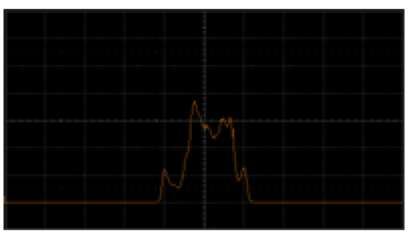
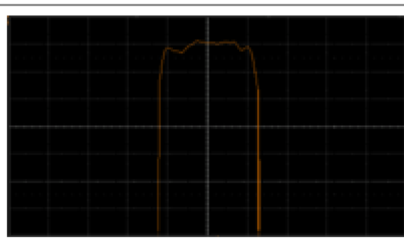
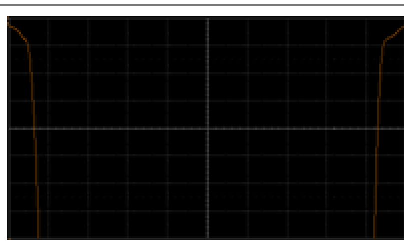
Control	Values	Description
Synch to pattern	On/off	Synchronizes the measurement to a repeating data pattern (if present) in the signal when this control is checked. The timing is measured between the first transition in the pattern and each subsequent transition. For an N-bit pattern, each transition is measured in succession and measurements are accumulated by shifting the measurement window by N bits and repeating the measurement.
Ver. Output	Mean, sdev, DDmean, DDsdev	These are the vertical units of the measurement. Mean gives the average position of edges for each value of N relative to a reference clock at the bit rate. Sdev gives the standard deviation for each edge for each value of N relative to the reference clock. DDmean gives the mean value of each edge relative to the

Control	Values	Description
		first edge normalized to a unit interval for each N. DDsdev gives the standard deviation for each edge relative to the first edge normalized to a UI for each N.
Length (UI)	The maximum N value	This field displays the pattern length if **Sync to Pattern** is selected. Otherwise, this is the maximum N value displayed in the plot.
Tx dens.	0.0 to 1.0	Transition density of the data stream. This control reads out the measured transition density of the signal when the Find Pattern button is pressed. This field is not used by the N-cycle plot.
Seq. Type	PRBS or Pattern	Selects whether the repeating pattern is a standard PRBS or a general pattern. This control is only used when snyc to pattern is selected.
PRBS	5 to 15	When PRBS is selected, sets the length.
Find Pattern		Pressing this button searches the signal for a repeating pattern and loads it into the function when it is found. The Length (UI) and Tx dens. controls are then updated.
Invert Pattern	On/off	Checking this box inverts the found pattern bit by bit.
Bit rate		The symbol rate of the signal under test. This value must be entered by the user in order to set the nominal UI.
Avg Wfm	On/off	Averages the N cycle plots measured on successive acquisitions. This is especially useful for removing random effects from standard deviation plots.
Use PLL	On/off	Use the software golden PLL to generate the reference clock for edge timing. This setting is ignored when DDmean or DDsdev is selected.
Cutoff Div.	PLL cutoff divisor	When the PLL is selected, this control sets the loop bandwidth as a ratio of the bit rate (loop BW = data rate/cutoff divisor).
Find Level		When the level type is absolute, clicking this control finds the mean signal level.
Clear Sweeps		Pressing this button clears the sweeps that have been accumulated in the averaging mode.

Htie to BER

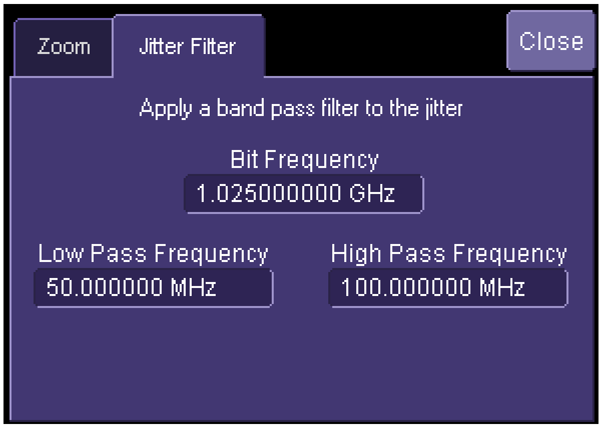
This function takes as an input a histogram and converts it to one of 6 output waveforms. Normally, the histogram contains a set of time interval error (TIE) measurements from which a total jitter measurement is desired. The histogram should have a population of at least 100k and a minimum of 500 bins. The histogram should also be completely contained on the instrument screen (it may be necessary to set the horizontal scale to do this). The table below outlines the settings for this function.



Output Format		Waveform Type
TjGaussian		The total jitter curve for a Gaussian with zero mean and unity standard deviation
LogEstTIE		The log of the estimated (extrapolated) time interval error histogram
Total Jitter		The total jitter curve which, shows the cumulative probability of an edge being greater (right-hand side) or less (left-hand side) than its nominal location in the center. The total jitter at a particular bit error rate is measured from the width of this curve at the desired bit error rate given by the vertical location over which the width is measured.
EstTIE		Estimated tie histogram. This shows the original histogram of TIE measurements that has been extrapolated to a population of 10^{16}.
LogTIE		The log of the TIE histogram
Bathtub		The bathtub curve. This is a an alternate version of the total jitter curve scaled horizontally to 1 UI. The right-hand side of the curve is taken from the left-hand side of the total jitter curve, and the left-hand side comes from the right-hand side of the total jitter curve.
Frequency	100 kHz to 4.5 GHz	The bit rate of the signal. This control is only active when the bathtub output is active.
Transition density	0.1 to 1.0	The transition density in the data signal. Usually assumed to be 0.5. This represents the number of bit transitions compared to the total number of bits in the data stream. This value is used to normalize the extrapolated histogram.

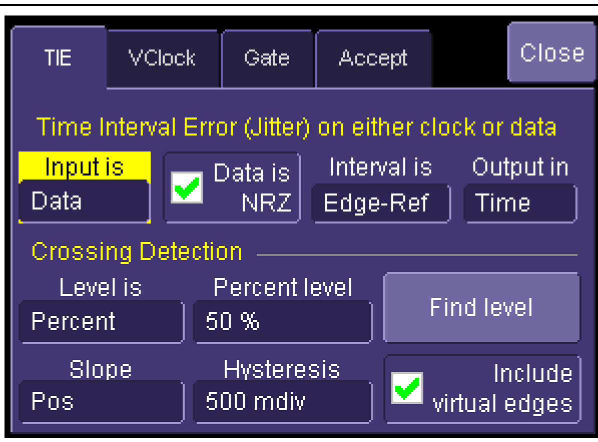
Jitter Filter Function

The jitter filter function generates a time sequence of jitter measurements that are filtered by the selected band-pass filter. Low-pass and high-pass frequencies are set in the function's menu. You must enter the bit frequency (data rate) to set the sampling rate for the digital filter. This function is only valid when operating on the TIETrend trace generated by the SDA. The low frequency corner is limited by the record length (longer records are required for lower cutoff frequencies). If an invalid lower cutoff frequency is chosen, the filter reverts to low-pass.



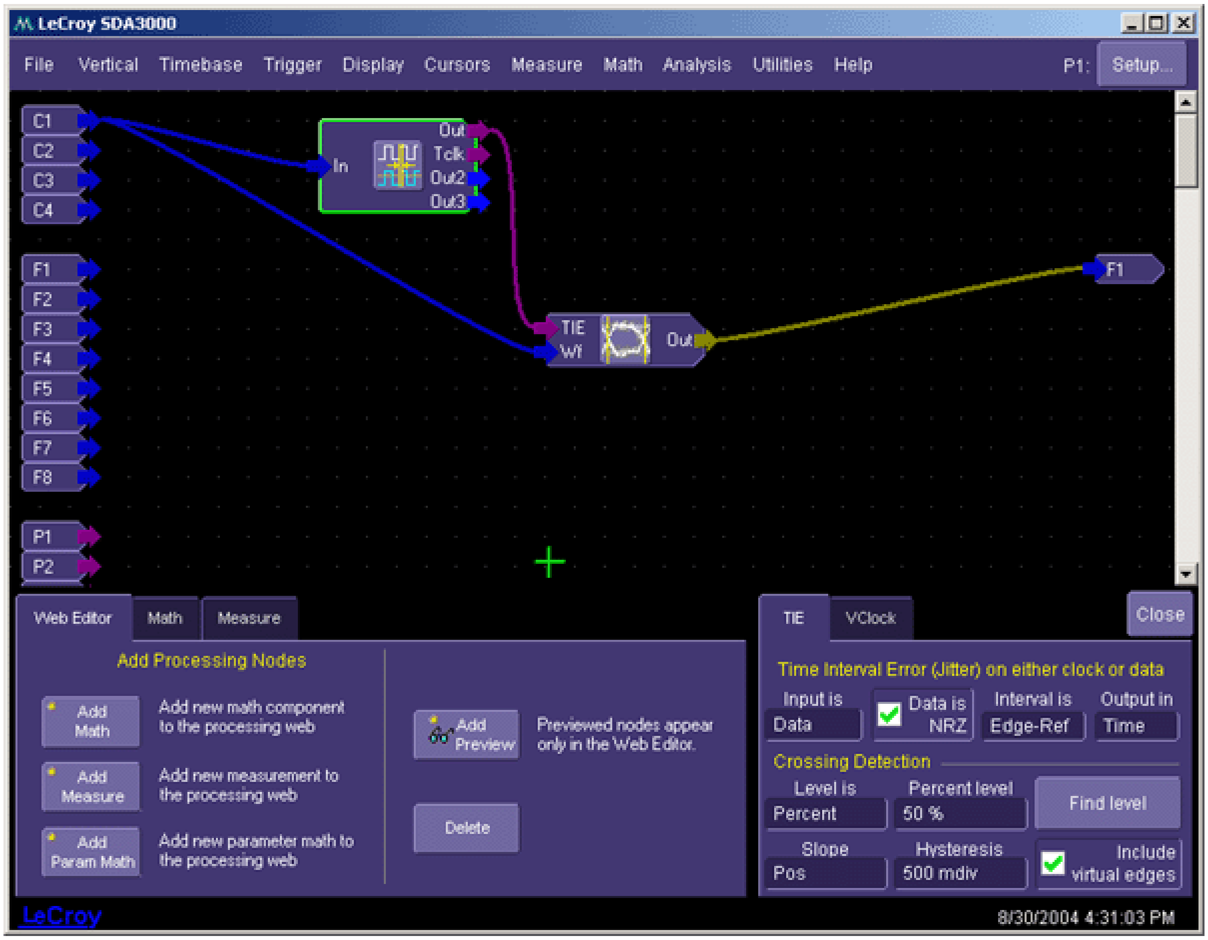
Bit Frequency	100 kHz to 4.5 GHz	The bit rate of the signal under test
Low Pass Frequency	Set by record length and sample rate (bit rate)	The lower corner frequency of the band-pass filter
High Pass Frequency	Set by record length and sample rate (bit rate)	The upper corner frequency of the band-pass filter

Slice2Persist



The **Include virtual edges** checkbox must be checked in the TIE@lvl parameter dialog to include both transition and non-transition edges in the eye pattern.

The Slice2Persist function takes as its input a data waveform and a clock signal. The clock must consist of a sequence of time values that increase at the nominal unit interval for each successive sample. For example, a 2.5 Gb/s bit stream should have a clock input consisting of the numbers 0, 4e-10, 8e-10, 12e-10, …, which increase at a 400 ps per UI rate. The clock signal is derived from the output of the TIE@lvl parameter, which contains a sequence of adjustment values that are added to the nominal bit period for each UI. These correction values allow the TIE@lvl parameter to act as a phase locked loop. The web edit setup is shown below.



Frequency	100 kHz to 4.5 GHz	The bit rate of the signal under test. This number can also be set to a sub-multiple of the bit rate to show more than one eye. For example, 2.5 GHz will show one eye of a 2.5 Gb/s signal while setting the control to 1.25 GHz will display two eyes.
Auto Scale Vertical		Adjusts the vertical scale so that the one level is centered on 2.5 divisions above the screen center and the zero level is set to -2.5 divisions.
Auto clear sweeps		Clears the eye pattern display on each new acquisition.
Clear Sweeps		Clears the current persistence map from the display.
Scale Top to	4 V to -4 V	Scales the one level to a specific voltage on the display.
Scale Bottom to	4 V to -4 V	Scales the zero level to a specific voltage on the display.

SDA Theory

The SDA operates by processing a long signal acquisition. The processes include clock recovery, eye pattern computation, jitter measurement, and bit error testing. All of these operations are performed on the same data record. The processes are described in detail in this section.

Clock Recovery Theory

An accurate reference clock is central to all of the measurements performed by the SDA. The recovered clock is defined by the locations of its crossing points in time. Starting with zero, the clock edges are computed at specific time intervals relative to each other. A 2.5 GHz clock, for example, will have edges separated in time by 400 ps.

The first step in creating a clock signal is the creation of a digital phase detector. This is simply a software component that measures the location in time at which the signal crosses a given threshold value. Given the maximum sampling rate available, 40 GHz, interpolation is necessary in most cases. Interpolation is automatically performed by the SDA. Interpolation is not performed on the entire waveform; rather, only the points surrounding the threshold crossing are interpolated for the measurement. A cubic interpolation is used, followed by a linear fit to the interpolated data, to find the crossing point. This is shown in the following figure .

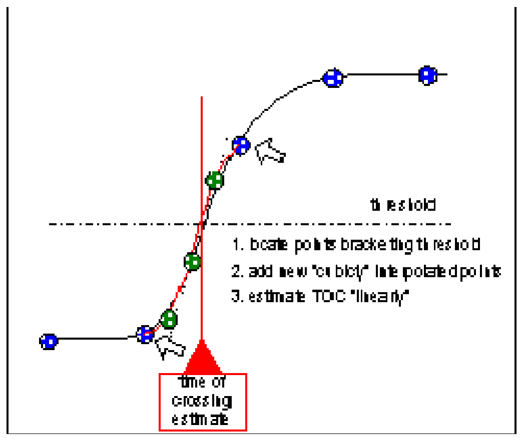


Figure 5-31. SDA Threshold Crossing Algorithm.

Clock recovery implementation in the SDA is shown in the following figure. This algorithm generates time values corresponding to a clock at the data rate. The computation follows variations in the data stream being tested through the use of a feedback control loop that corrects each period of the clock by adding a portion of the error between the recovered clock edge and the nearest data edge.

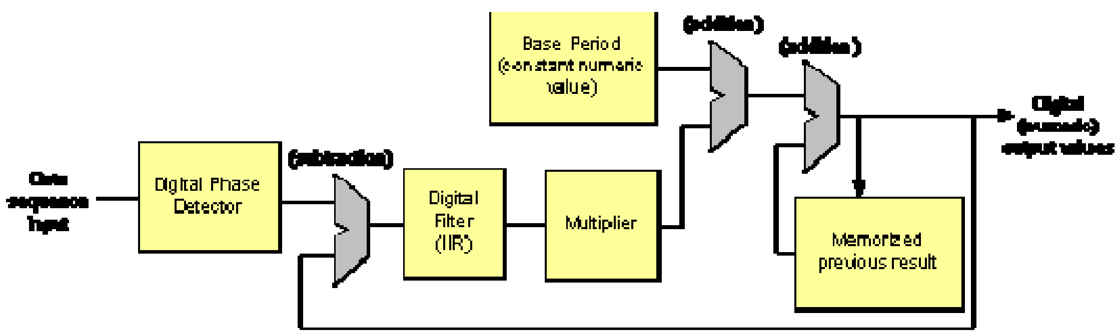


Figure 5-32. Clock Recovery Implementation.

As shown in Figure 2, the initial output and the output of the digital phase detector are set to zero. The next time value output is equal to the nominal data rate. This value is fed back to the comparator on the far left which compares this time value to the measured time of the next data edge from the digital phase detector. The difference is the error between the data rate and the recovered clock. This difference is filtered and added to the initial base period to generate the corrected clock period. The filter controls the rate of this correction by scaling the amount of error that is fed back to the clock period computation. This filter is implemented in the SDA as a single-pole infinite impulse response (IIR) low-pass filter. The equation of this filter is:

$$y_k = \frac{1}{N}x_k + \left(1-\frac{1}{N}\right)y_{k-1}$$

The value of y_k is the correction value for the k^{th} iteration of the computation and x_k is the error between the k^{th} data edge and the corresponding clock edge. Note that the current correction factor is equal to the weighted sum of the current error and all previous correction values. The multiplier value is set to one in the SDA, and the value of n is the PLL cutoff divisor that is set from the SDA main menu. The cutoff frequency is Fd/n where Fd is the data rate. This filter is related to its analog counterpart through a design process known as impulse invariance and is only valid for cutoff frequencies much less than the data rate. For this reason, the minimum PLL cutoff divisor setting is 20 in the SDA.

The factor *n* determines the number of previous values of the correction value *y* that is used in the computation of the current correction value. This is theoretically infinite; however, practically there is a limit to the number of past values included. One can define a “sliding window” equivalent to a number of UI (unit intervals) of the data signal for a given value of *n*. This is useful for measuring signals such as serial ATA and PCI-Express, where the specifications call for clock recovery over a finite window. The equivalent bandwidth of the sliding window is given by a sin(x)/x function. The first null of this function occurs at $x = \pi$ or ½ the bit rate (the digital equivalent of the frequency of a signal at the sampling rate is 2π and the sampling rate for clock recovery is the data rate). This is scaled by the window size to be $2\pi / N$ where N is the window in UI. The 3 dB point of the sin(x)/x function is at $0.6\pi / N$ or 0.3Fd/N for a window length of *N*. This gives us a relationship between *N* and *n*:

Fd/n = 0.3Fd/N or n = N/0.3

For a sliding window size of 250, the equivalent value of *n* would be 833.

Eye Pattern Theory

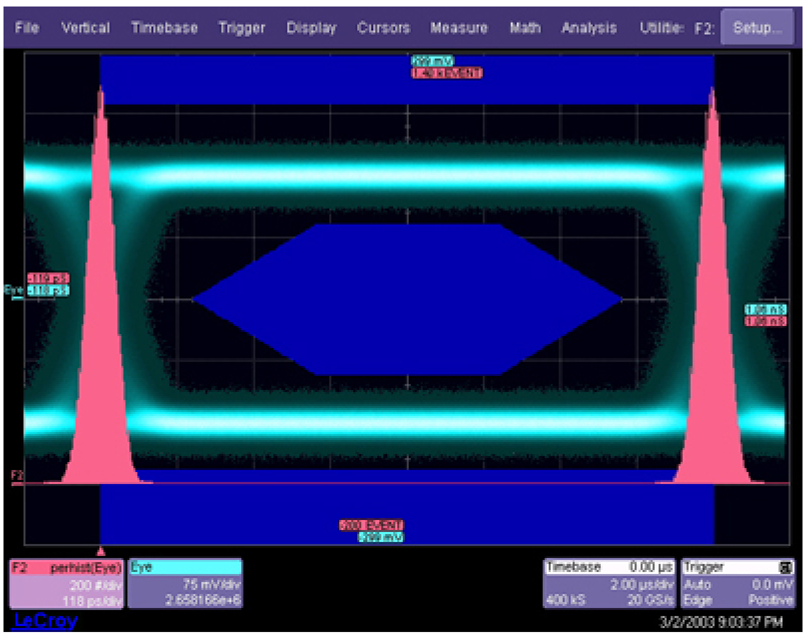


Figure 5-33. Histogram of Zero Crossing in Eye Pattern Showing Jitter Distribution

An eye diagram shows all values that a digital signal takes on during a bit period. A bit period (or UI) is defined by the data clock, so some sort of data clock is needed in order to measure the eye pattern. The traditional method of generating an eye pattern involves acquiring data on an oscilloscope, using the data clock as a trigger. One or more samples are taken on each trigger. The samples are stored in a persistence map with the vertical dimension equal to the signal level, and the horizontal position equal to the sample position relative to the trigger (or data clock). As many data points are collected, the eye pattern fills in with multiple occurrences of time and amplitude values counted by incrementing counters in each x,y “bin.” Timing jitter is indicated by the horizontal distribution of the points around the data crossings. The histogram of the bins around the crossing points gives the distribution of jitter amplitude.

A recovered clock is used if there is no access to a data clock. The recovered clock is normally a hardware PLL designed to operate at specific data rates and with a cutoff frequency of Fd/1667. One of the major drawbacks of a hardware clock recovery circuit is that jitter associated with the trigger circuit adds to the measured jitter by creating uncertainty in the horizontal positioning of the eye pattern samples.

The SDA measures eye patterns without using a trigger. It does this by using the software clock recovery discussed above to divide the data record into segments along the time values of the clock. For the purpose of dividing the time line into segments, the time resolution in the waveform record is infinite. The samples occur at fixed intervals of 50 ps/pt (for a 20 GS/s sampling rate). The samples are positioned relative to the recovered clock timing points and the segments delimited by the clock samples are overlayed by aligning the clock samples for each segment. A monochrome or color persistence display is used to show the distribution of the eye pattern data. The jitter added by the measurement system in this case is from the sampling clock which, for the SDA is very low: on the order of 1 ps rms.

Eye Violation Locator (ASDA Option)

The eye pattern is measured by overlaying segments of a continuous acquisition. Since the complete data record is available, the location of individual bits can be determined by comparing each bit interval in the original waveform to the selected mask. The mask is aligned horizontally along the mean bit interval, and vertically along the mean one and zero level in the case of a relative mask. Absolute masks exist for some standards and are defined in the vertical dimension by specific voltage values. Figure 4 below shows this alignment. When mask testing is turned on, the entire waveform is scanned bit-by-bit and compared to the mask. When a mask hit is detected, the bit number is stored and a table of bit values is generated. The table is numbered starting with the first bit in the waveform. This table can be used to index back to the original waveform to display the waveform of the failed bit.

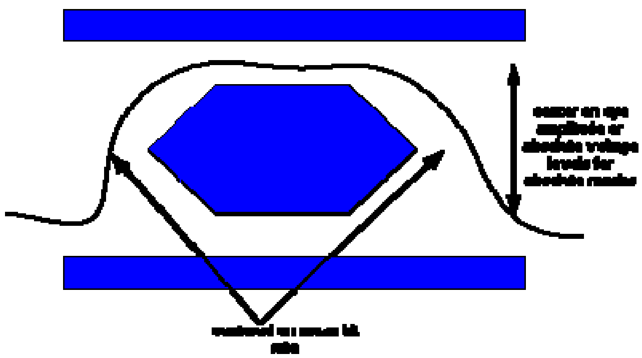


Figure 5-34. Eye Mask Alignment for Violation Locator

Eye Pattern Measurements

There are several important measurements that are made on eye patterns. These are specified as required tests for many standards. Eye measurements mainly deal with amplitude and timing, which are outlined below.

Eye Amplitude

Eye amplitude is a measure of the amplitude of the data signal. The measurement is made using the distribution of amplitude values in a region near the center of the eye (normally 20% of the distance between the zero crossing times). The simple mean of the distribution around the “0” level is subtracted from the mean of the distribution around the “1” level. This difference is expressed in units of the signal amplitude (normally voltage).

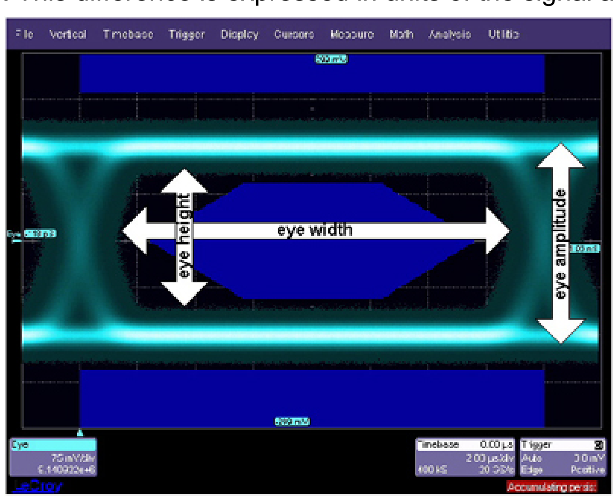


Figure 5-35. Eye Measurements.

Eye Height

The eye height is a measure of the signal-to-noise ratio of a signal. The mean of the “0” level is subtracted from the mean of the “1” level as in the eye amplitude measurement. This number is modified by subtracting the standard deviation of both the “1” and “0” levels. The measurement basically gives an indication of the eye opening.

Eye Width

This measurement gives an indication of the total jitter in the signal. The time between the crossing points is computed by measuring the mean of the histograms at the two zero crossings in the signal. The standard deviation of each distribution is subtracted from the difference between these two means.

Extinction Ratio

This measurement, defined only for optical signals, is the ratio of the optical power when the laser is in the ON state to that of the laser in the OFF state. Laser transmitters are never fully shut off because a relatively long period of time is required to turn the laser back on, thus limiting the rate at which the laser can operate. The extinction ratio is the ratio of two power levels (one very near zero) and its accuracy is greatly affected by any offset in the input of the measurement system. Optical signals are measured using optical-to-electrical converters on the front end of the SDA. Any DC offset in the O/E must be removed prior to measurement of the extinction ratio. This procedure is known as dark calibration. The output of the O/E is measured with no signal attached (i.e., dark), and this value is subtracted from all subsequent measurements.

Eye Crossing

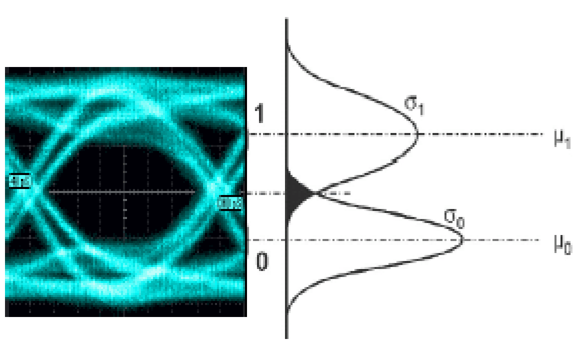
Eye crossing is the point at which the transitions from 0 to 1 and from 1 to 0 reach the same amplitude. This is the point on the eye diagram where the rising and falling edges intersect. The eye crossing is expressed as a percentage of the total eye amplitude. The eye crossing level is measured by finding the minimum histogram width of a slice taken across the eye diagram in the horizontal direction as the vertical displacement of this slice is varied.

Average Power

The average power is a measure of the mean vale of all levels that the data stream contains. It can be viewed as the mean of a histogram of a vertical slice through the waveform covering an entire bit interval. Unlike the eye amplitude measurement where we separate the ones and zeroes histograms, the average power is the mean of both histograms. Depending on the data coding that is used, the average power can be affected by the data pattern. A higher density of ones, for example, will result in a higher average power. Most coding schemes are designed to maintain an even ones density resulting in an average power that is 50% of the overall eye amplitude.

Q Factor or BER

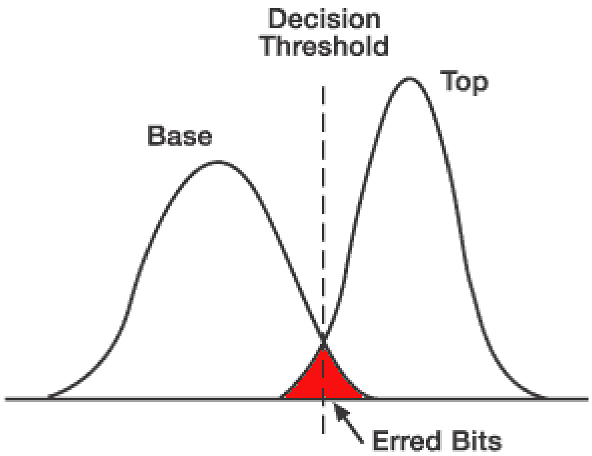
The Q factor is a measure of the overall signal-to-noise ratio of the data signal. It is computed by taking the eye amplitude (the difference between the mean values of the "1" and "0" levels) and dividing it by the sum of the noise values (standard deviations of the "1" and "0" levels). All of these measurements are taken in the center (usually 20%) of the eye.



$$Q-factor = \frac{\mu_1 - \mu_0}{\sigma_1 + \sigma_0}$$

eyeBER

EyeBER is the estimated bit error rate from an eye diagram. It is calculated using the intersection of the distribution of the one and zero levels. EyeBER differs from BER in that eyeBER is calculated from Q-factor and is, therefore, based on signal-to-noise ratio; BER, on the other hand, is based on jitter.



$$eyeBER = \frac{e^{-(Q^2/2)}}{Q\sqrt{2\pi}}$$

Jitter Measurement

Jitter is measured by the relative variation in the location in time of the transitions of the signal level across a specific level. For clock signals, the relative time between threshold crossings (rising-to-rising or falling-to-falling) is measured. Data signals, on the other hand, generally require the measurement of the relative positioning of the data signal to the sampling clock, which is related to setup and hold time. Because of its random nature, jitter is normally described in terms of its probability density function (PDF).

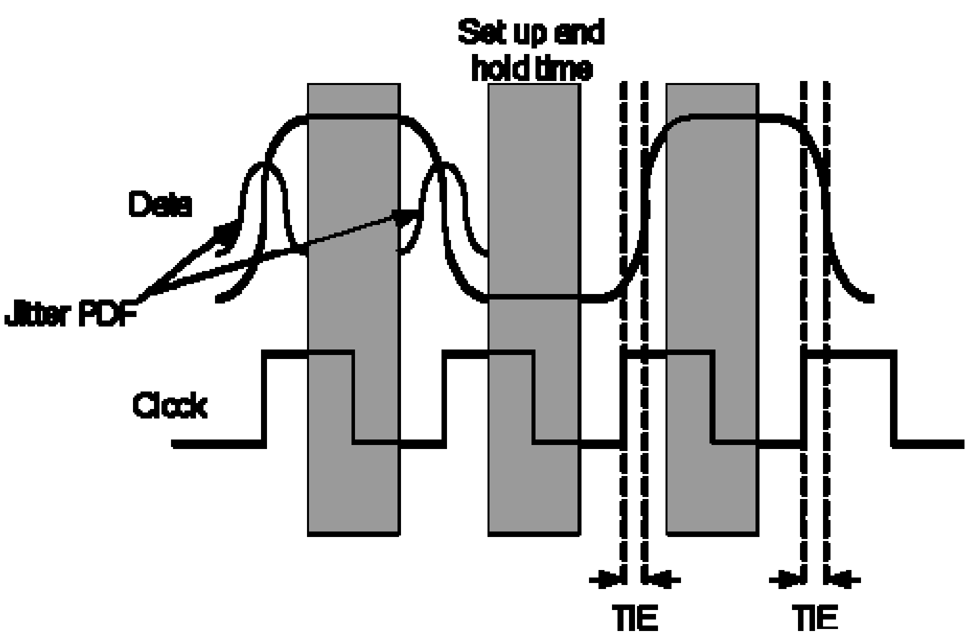


Figure 5-36. Setup and hold time requirement for error-free operation. Data transitions within the setup and hold time (gray area) will result in bit errors. Time interval error (TIE) is the time difference between clock and data edges. The PDF of TIE is a measure of the probability of an edge occurring during the setup and hold time.

The processes that make up jitter are complex and come from many different random and non-random (deterministic) sources. The PDF of the jitter is the convolution of all individual component PDF's. Measurements are able to estimate the jitter PDF but are not able to determine the distributions of the random and deterministic parts of the overall distribution. The lack of exact measurements for the jitter distributions of Rj and Dj has lead to the use of a simplified model for the total jitter. Equation 1 describes this model which was first presented in the Fibrechannel MJSQ document.

$$Tj(BER) \approx \psi(BER) * Rj + Dj \quad (1)$$

Equation 1 is a heuristic that describes total jitter as a function of bit error rate (BER) and is related to a distribution consisting of a Gaussian convolved with a pair of impulses as shown in Figure 7. The constants Rj and Dj represent all of the components of random and deterministic jitter. The Greek letter Psi is a function of BER and represents the total peak-to-peak jitter of a unit normal distribution (i.e., a Gaussian with zero mean and a standard deviation of 1 at the specified bit error rate. The process of determining Rj and Dj involves finding the "best fit" values that solve equation 1. There are many possible ways to fit Rj and Dj to (1) and since it is a simplification, no single set of solutions can completely describe the behavior of actual jitter completely. It is for this reason that the SDA uses two separate methods to measure Rj and Dj, effective and direct, and presents these to the user.

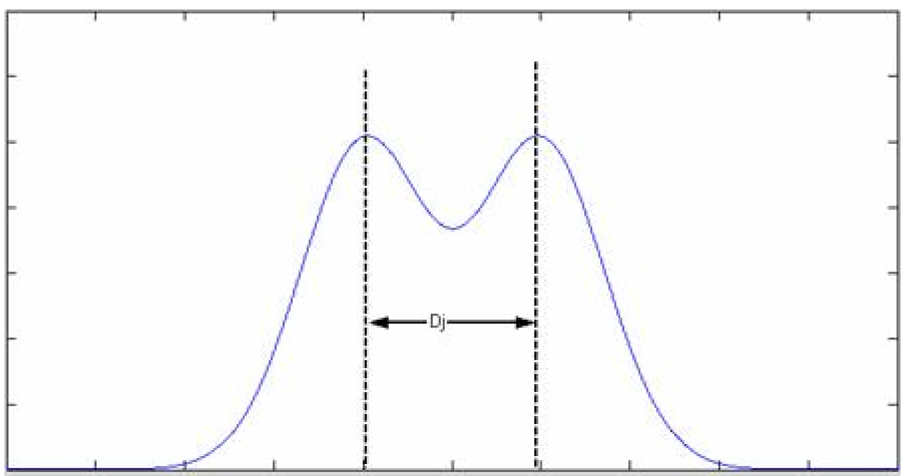


Figure 5-37. Jitter PDF model corresponding to the heuristic in equation 1. The random jitter is modeled by a Gaussian and the deterministic jitter by a pair of impulses separated by the value of the parameter Dj. The curve shown is the convolution of Rj and Dj.

Bit Error Rate and Jitter

Equation 1 shows that the total jitter is a function of bit error rate. This relationship is based on the effect that jitter has on the bit error rate of a system. The bit error rate is influenced by other parameters of the system, such as noise, so it is not correct to say that BER and jitter are equivalent. It is the contribution to the overall bit error rate caused by jitter that is shown in Equation 1. A bit error will occur when the data signal transitions from one state to another during the setup and hold time, as shown in Figure 6. Since jitter has a random component, the location in time of the transitions varies over a range of values. The longer the transitions are observed, the greater this range will be. Now, if we think of each transition in the data signal as the change in a bit value, then a transition at the wrong time (i.e., outside the setup and hold window) will lead to a bit error. The probability of this event is equivalent to the bit error rate contribution due to jitter. The total jitter gives a confidence interval for the jitter in that it will not exceed a certain value to a confidence of (1-BER). The term "bit error rate" is commonly used in this context to refer to the jitter confidence interval in many specifications.

EyeBER differs from BER in that eyeBER is calculated from Q-factor and is, therefore, based on signal-to-noise ratio; BER, on the other hand, is based on jitter.

Total Jitter

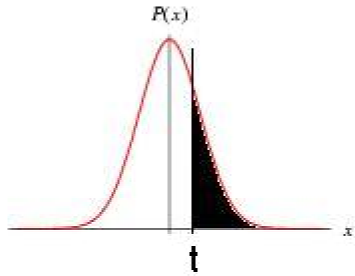


Figure 5-38. Probability of a data edge displacement greater than time t from the sampling clock. The mean value of the distribution is 0 which represents perfect alignment.

The total jitter is the peak-to-peak jitter in a clock or data signal within a specified confidence, equal to 1-BER. An example of a normally distributed jitter PDF is shown in Figure 8. In order to determine the total jitter from the PDF, the probability of the jitter exceeding a certain value *t* must be evaluated. This is done by integrating the PDF from a time *t* to +∞ which will give the total probability of an edge occurring at or after this time. The probability can be computed for all values of *t* by integrating the PDF separately for *t* > 0 and *t* < 0. The resulting curve, shown in Figure 9, gives the total probability of an edge being greater than *t* (or less than -*t*). The contribution to the system BER by jitter is given by the probability that an edge occurs at a time greater than *t* as we mentioned earlier. In order to guarantee a BER contribution from jitter below a certain value, the positive and negative values of *t* are chosen so that the probability of an edge at a time greater and less than these times is equal to the desired bit error rate. These jitter values can be measured by finding the intersection between a horizontal line at the bit error rate and the total jitter curve. The horizontal spacing between these two points is the total jitter.

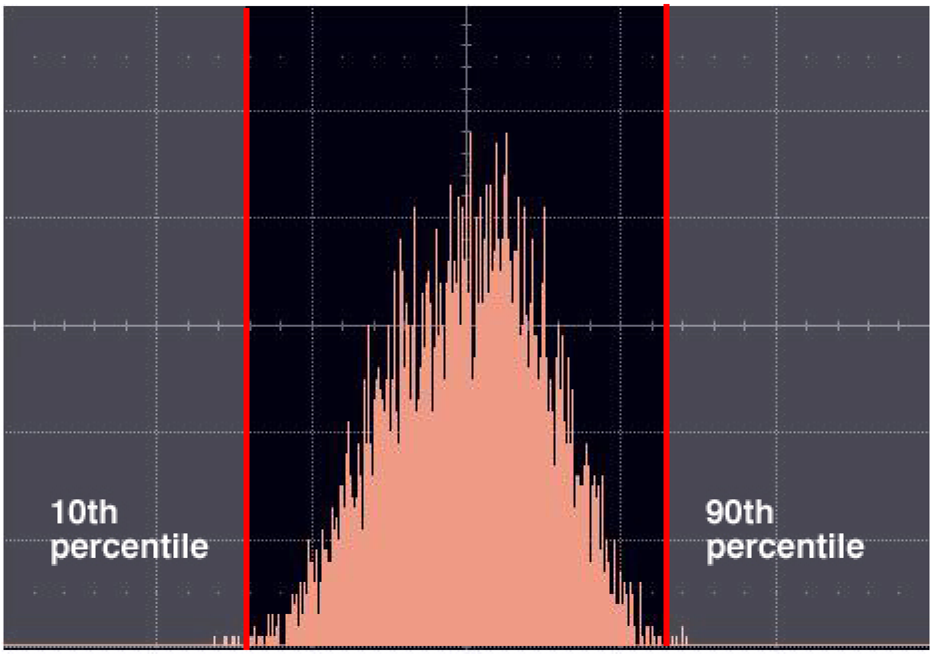
Figure 5-39. Total jitter curve. The vertical values of this curve represent the probability of a data transition occurring at a time represented by the horizontal axis. The horizontal center of the plot is 0 ps. The two markers are placed at the vertical level corresponding to a bit error rate of 10 e-12 and the horizontal distance between these two points is the total jitter at this bit error rate.

Figure 5-40. The bathtub curve is constructed by rescaling the total jitter curve in Figure 9 to one unit interval, and centering the right side of the total jitter curve at 0 UI and the left side at 1 UI (the left and right sides of the bathtub curve).

A common way to view the total jitter is by plotting the bit error rate as a function of sampling position within a bit interval. This curve, commonly referred to as the “bathtub” curve is derived from the total jitter curve by scaling it to one bit interval (UI). The right half of the bathtub curve is taken from the left half of the total jitter curve and the left half is taken from the right half of the total jitter curve. The bathtub curve corresponding to the total jitter curve in Figure 9 is shown in Figure 10.

Extrapolating the PDF

Measuring the total jitter requires that the probability density function (PDF) of the jitter be known exactly. The SDA measures the jitter PDF by collecting a histogram of TIE measurements. The following histogram approximates the PDF by counting the number of edges occurring within the time period delimited by each bin in the histogram. In order to accurately measure jitter contributions at very low bit error rates such as 1012, the histogram must contain measurements with populations that are below 1 in 1016 (one TIE measurement out of 1016 at a certain value). This number of data transitions would take approximately 38 days at 3 Gb/s. Measuring this number of edges is clearly impractical.



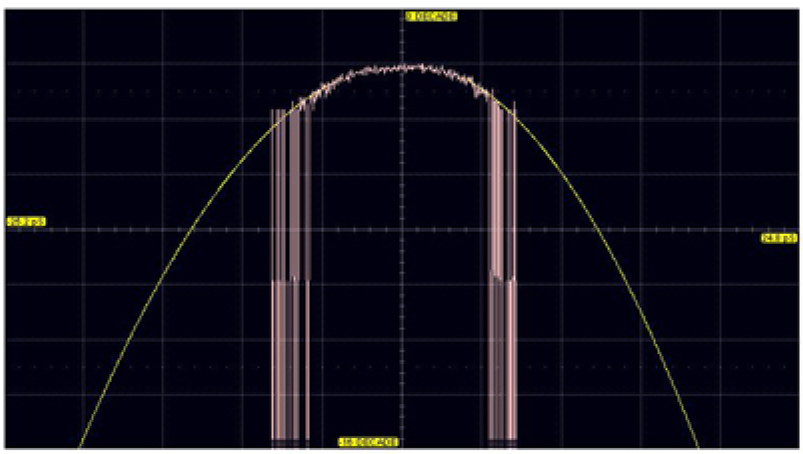


Figure 5-41. Logarithm of the measured TIE histogram superimposed on the extrapolated curve (in yellow). The extrapolation uses a quadratic curve fit to the histogram bins at the extremes.

A smaller data set is extrapolated in order to estimate the data for the larger sample size. The extrapolation of the measured histogram of TIE values uses the random nature of jitter at the extremes of the histogram to extrapolate the bins below the 10th percentile and above the 90th percentile. The central part of the distribution is dominated by deterministic jitter, while the extremes are entirely random. The bins in these ranges behave as a normal distribution as the jitter range is increased, that is, their populations fall as exp(-t2). Taking the logarithm of the histogram makes this relationship quadratic, so the extrapolation is simply a quadratic curve fit to the extremes of the log of the jitter histogram. The extrapolated histogram is used to compute the total jitter curve described above and is normalized so that the sum of the populations of all of the bins is one. The integrals described above are implemented by summing the extrapolated histogram bins.

Separating Rj and Dj – Two Methods

The total jitter curve is the basis for estimating the magnitude of Rj and Dj. Since the total jitter curve is derived directly from the signal under test, its value is the most accurate representation of the jitter for a given bit error rate. There are basically two ways of separating the random and deterministic jitter. The first method, which models the growth of total jitter as BER is decreased, leads to the effective jitter parameters Rje and Dje. These values are effective in the sense that they provide an equivalent total jitter model for low bit error rates. Starting with the total jitter curve, the growth in the total jitter as a function of decreasing BER is plotted. The curve described by equation 1 is fitted to the measured curve by selecting the Rj (slope) and Dj (intercept) to minimize the error in the fit.

The second method of fitting Rj and Dj to the measured data is based on direct measurements of the deterministic jitter. Random jitter is the difference between this value and the total jitter at the selected bit error rate measured from the total jitter curve. This, of course, exactly matches the measured total jitter at the selected bit error rate, but is a poor predictor of the jitter for bit error rates below this level. The motivation behind employing this method is to better represent the contribution of deterministic jitter in the overall jitter at the specified bit error rate.

Each method of measuring Rj and Dj results in different values for the standard deviation and spacing between the Gaussian curves in the distribution in Figure 7. The total jitter at the specified bit error rate, however, is the same for either distribution.

Effective Random and Deterministic Jitter

The effective jitter components Rje and Dje represent the best fit values for equation 1 to the behavior of the measured total jitter as the observation time is increased or, equivalently, the bit error rate is decreased. For a given bit error rate, the total jitter is measured from the width of the total jitter curve. The value of the total jitter as the bit error rate is decreased can be plotted as shown in Figure 13. The vertical axis of the plot on the left is the log of bit error rate. The Gaussian nature of the jitter at the extremes of the distribution results in a total jitter that grows approximately linearly with the log of BER, as shown in the upper curve in the plot on the right. The function N(BER)□in equation 1 represents the width of a normal distribution with a variance of one at a given confidence level equal to 1-BER. The lower curve on the plot on the right shows the variation of N(BER) with the log of BER which is approximately linear. The values of Rje and Dje are chosen so that the lower curve lies on top of the upper one. From equation 1, it can be seen that Rje is a slope parameter while Dje adjusts the intercept point.

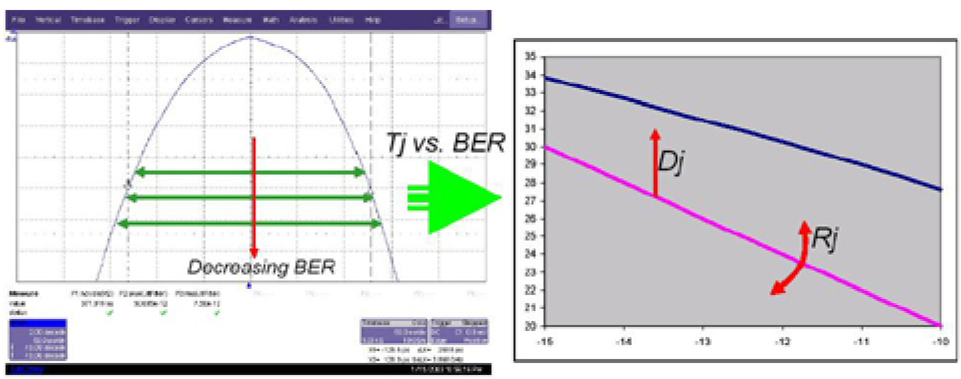


Figure 5-42. The change in total jitter with BER is represented by the values Tjn in the total jitter curve on the left. These values are plotted vs. bit error rate in the upper line in the chart on the right while the lower line shows the variation vs. BER for a pure Gaussian. Rje and Dje are chosen such that the curves line up.

The jitter computed using this method allows equation 1 to accurately model the jitter behavior of systems as a function of bit error rate. This model is especially useful when computing jitter margins in system applications.

Direct Measurement of Deterministic Jitter

Deterministic jitter can be completely characterized by measuring the threshold crossing times of the data signal over a finite time period. The two classes of deterministic jitter are periodic and data dependent.

Data dependent jitter is caused by system effects that are dependent on the data pattern. A common source of data dependent jitter is the frequency response of the channel through which the serial data signal is transmitted. In this case, data patterns with many transitions (such as a 101010… pattern) contain more high frequencies in their spectrum than patterns containing fewer transitions (11001100…, for example). The patterns with higher frequency content will be attenuated and phase shifted relative to the lower frequency patterns. In addition to data dependent jitter, the rise and fall times of the data bits can be different. The detection threshold in the receiver is normally set to the 50% amplitude (midway between the “1” and “0” levels) so unequal rise and fall times will generate jitter. This type of jitter is known as Duty Cycle Distortion (DCD).

When **DDj Calc. Method** is set to **ISI Plot**, the SDA uses a patent-pending method to measure both forms of data dependent jitter. The method uses the history of a number of bits in the waveform to determine their effect on the transition of a given bit. A user-selectable number of bits (from 3 to 10) is used in the measurement. The acquired waveform is processed in segments the size of the selected number of bits. For example, if 5 bits are selected, segments 5 UI long are examined. For each segment, the value of the 5 bits is determined and each group of 5 bits is averaged with segments of similar value. When the entire waveform is scanned, a set of up to 32 (for 5 bits) waveforms are created. The averaging process removes all random jitter, noise and periodic jitter from the segments. The waveform segments are overlaid by lining up the first transition of each of the segments and measuring the relative timing of the transitions to the last (5th in this example) bit.

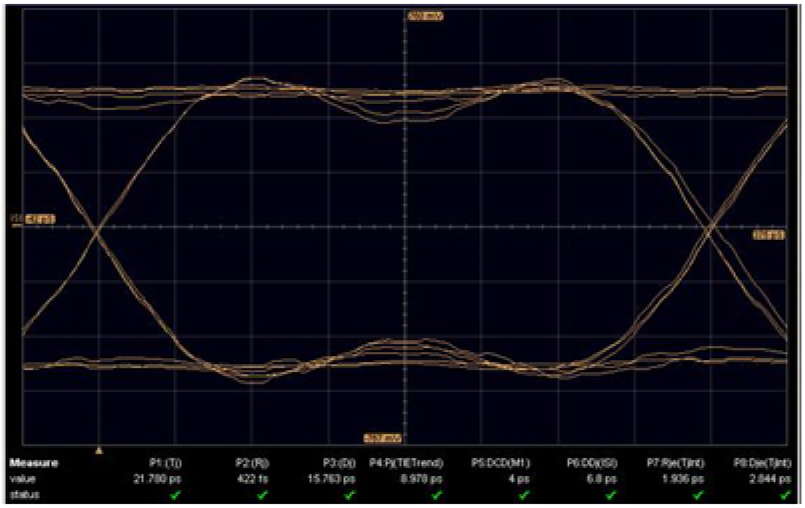


Figure 5-43. DDj measurement procedure. Averaged waveform segments for each pattern in the data stream are overlaid by lining up the first data transition. The curve above shows all of the transitions between the next-to-last bit and the last bit on the right side. The DDj is measured by examining the width of this crossing point at the selected threshold level.

Periodic jitter is measured by examining the spectrum of the trend of TIE values. The time interval error is measured for each edge in the data stream. Where no edges are present, as is the case for consecutive “1” or “0”' values, edges are inserted at the expected data transition times. These inserted edges do not add any additional jitter since they are placed at the ideal edge locations for the data rate. The trend of TIE is continuous and the spectrum can be computed. Periodic jitter is the complex sum of the spectral components excluding those associated with the repetition frequency of the data pattern and its harmonics.

The deterministic jitter is computed by adding up the periodic (Pj) and data dependent (DDj) components. The random jitter is computed using Equation 1 and subtracting the measured deterministic jitter from the total jitter at the selected bit error rate.

$$Rj = \frac{Tj(BER) - Dj}{\Psi(BER)} \quad (2)$$

Comparing Models

The plot in the following figure shows the bathtub curves for the measured values, as well as both estimates. Viewed in this way, it is clear why both the effective and direct measurements for Rj and Dj are used. Both estimates arrive at the same total jitter at the specified BER (10-12 in this case) but they give different values of Tj at other BER values. The effective jitter values give a very accurate prediction of total jitter for bit error rates below about 10-10, which is where they are fitted. The direct measurement underestimates the total jitter at error rates below the specified one and overestimates the jitter above this. Note that the effective parameters underestimate the jitter for high bit error rates. The three curves in the following figure show the resulting bathtub curves from the measured signal (blue line) and the two models: Hj(BER) for the direct Dj method (red line) and Hje(BER) for the effective jitter method (green line).

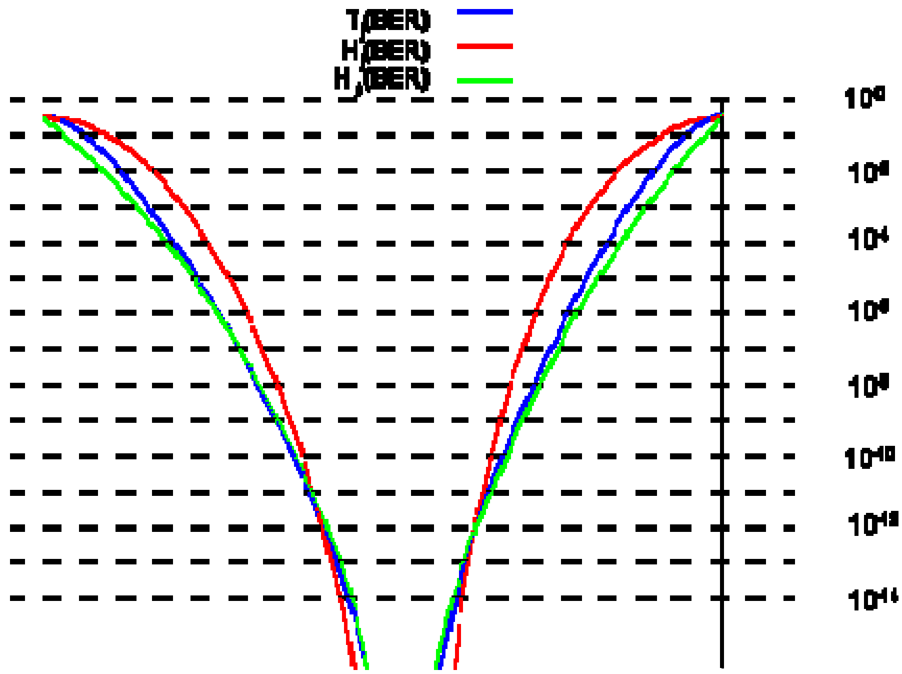


Figure 5-44. Measured jitter bathtub curve (blue curve) based on the extrapolation of the measured TIE histogram. The red and green curves are the estimated bathtub curves based on the direct Dj and effective measurements respectively. Note that the direct Dj method underestimates the total jitter below the BER at which it is computed.

Bit Error Rate

The SDA measures bit error rate directly on the captured bit stream by using the recovered clock to sample the waveform, and a user selectable threshold. The data are assumed to be NRZ so a high level is interpreted as a "1" and a low level is interpreted as a "0". The bit stream that is decoded in this process is compared, bit-by-bit with a user-defined known pattern. Since the instrument does not have any information as to which bit in the pattern it has received, a searching algorithm is used to shift the known pattern along the received data until a match is found. A match is determined when more than half of the bits are correct for a given shift of the known pattern. No match can be found if the bit error rate is over 50%, or if the wrong pattern is selected. In this case, the bit error rate will indicate 0.5 meaning that exactly ½ of the bits are in error which, of course, is the worst case.

Bit Error Map

A further level of debugging is available through the bit error map. This display is a view of the bit errors in the data stream relative to any framing that may be present in the signal. There are several options for framing that may be set. The general form of the data signal is shown here.

The header portion is a fixed pattern that can be set to any pattern. The header must be one or more bytes if it is present. The software searches for the header and treats the bits between headers as a frame. Each frame is displayed as a line of pixels in an x-y map and each successive frame is displayed below the previous one in a raster fashion. Bit errors are computed only on the payload sections of the data stream.

Framing can also be defined by a specific number of bits without a header. An example of this is a pseudorandom bit sequence (PRBS) of a specific length, 127 bits for example. In this case, setting the frame size to 127 will display one repetition of this sequence per line of the error map. Bit errors are displayed in a lighter color, whereas non-errors are shown in dark blue. By displaying bit errors on a frame-by-frame basis, pattern dependent errors can be clearly seen as lightly colored vertical lines in the error map.

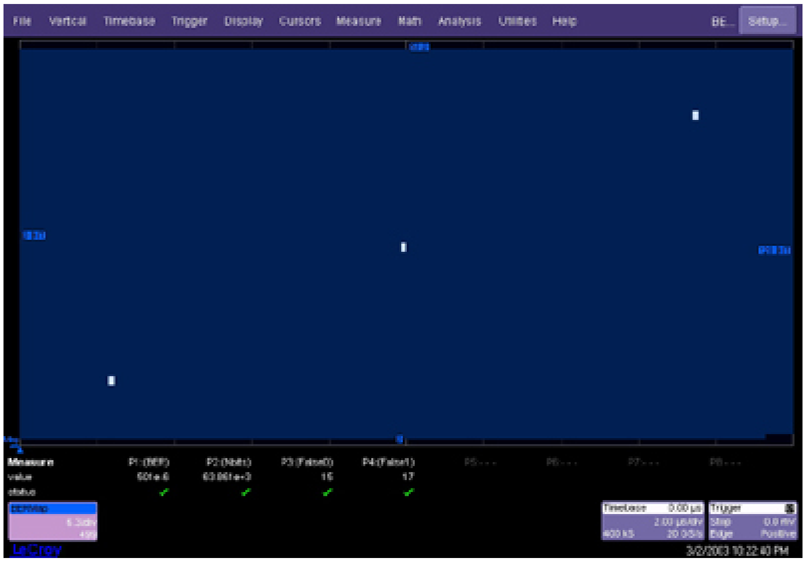


Figure 5-45. Bit Error Map for 127-Bit Pattern Containing Random Errors (White Squares).

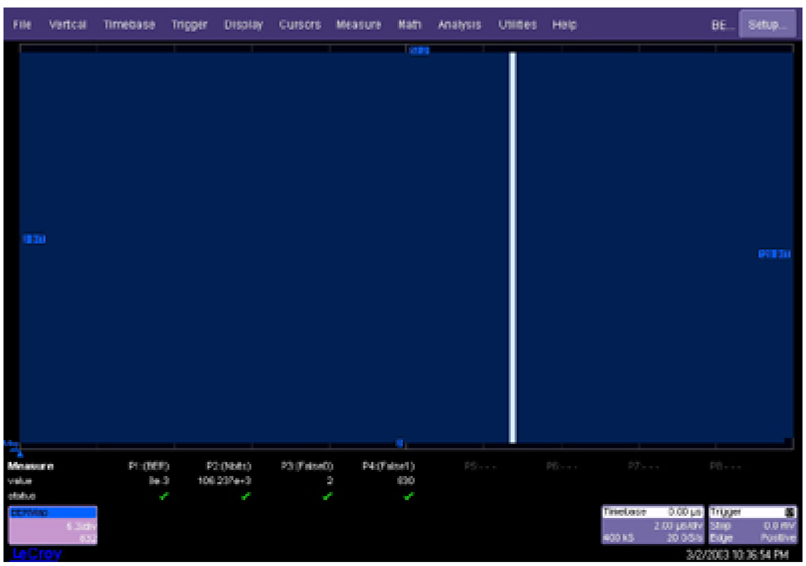


Figure 5-46. Bit Error Map for 127-Bit Pattern Containing Pattern Dependent Errors.

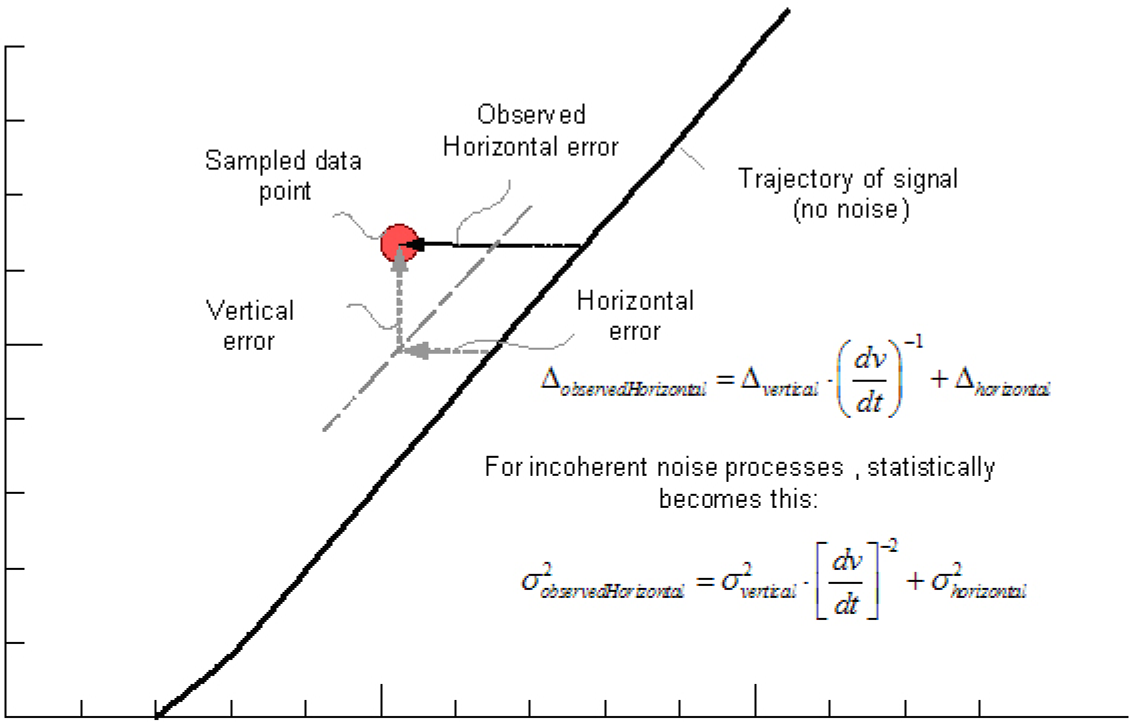
Vertical Noise Compensation

Introduction

Jitter is a measure of the timing error of data or clock signal transitions. The error is determined by measuring the time at which the signal crosses a given amplitude level and comparing it to the “expected” transition time. In a perfect world, this error would be entirely due to timing jitter but, as is often the case, things are not so simple. All signals contain amplitude noise and this noise can be interpreted as timing jitter in certain circumstances leading to higher overall jitter values. The noise on the signal under test is, and should be, included in any jitter measurement, since the data receiver would also interpret this as jitter. Any noise added by the test system should, however, not be included as this will overestimate the jitter. The SDA includes a feature that removes the system noise from the instrument, which uses the rms noise of the oscilloscope and probe, if present. This section explains the theory of operation of the noise compensation.

Signal Slew Rate and How Vertical Noise Converts to Jitter

In RF electronics, this process is known as AM to PM conversion. The basic principle is that the slew rate of the signal serves as a mechanism for vertical noise to be confused with horizontal (timing) noise, and vice versa. The following image illustrates:



It is easy to see how if the slew rate is very high (i.e., very low rise-time) that the effect of vertical noise is low and, conversely, that if the slew rate is very low, the vertical noise can dominate.

The Relative Impact of Noise on Jitter (Quadratic Addition of Noise)

The “quadrature addition” used to “add” the noise components is really only precise for Gaussian distributions of variations (even though this works well for many combinations of incoherent phenomena).

What’s important in this application is that (at least for current instruments) the instrument’s contribution to vertical noise is almost always Gaussian in nature. Furthermore, this can be confirmed by in-situ calibration procedures.

How the Measurement System Noise Is Subtracted from the Jitter Measurement

The timing noise can be very complex, exhibiting periodic jitter (*Pj*) and, for data streams, ISI induced data dependent jitter *DDj*. Other forms of bounded uncorrelated jitter (*BUj*) can contribute, as well, to make the jitter distribution shape quite non-Gaussian.

If, however (as is usually the case), the jitter analysis breaks down the distribution characteristic into *Rj* and *Dj*, the systematic effects are lumped into the *Dj* part. The *Rj* part of this jitter behaves as a Gaussian, and the vertical noise contribution (from the instrument) can be compensated as follows:

$$Rj_{compensated} = \sqrt{Rj^2_{observed} - \left[\frac{Rv_{instrument}}{dv/dt}\right]^2}$$

So, if the *dv/dt* can be well estimated by the instrument, and if the instrument’s contribution to vertical noise is known (and incoherent with the signal, which is nearly always the case), this compensated Rj figure can be reported.

As such, the LeCroy SDA software has been equipped to perform this compensation, and applies it to the *Rj* figure and not the *Dj* figure; however, the corresponding effect on *Tj* is reported as well.

Per model noise figures are supplied as a built-in database for each SDA oscilloscope, and a procedure for updating the values for a specific oscilloscope channel and probe are provided.

Q-scale Theory

Introduction

Jitter is an important aspect of signal integrity for both optical and electrical serial data streams (and clocks). The SDA (serial data analysis) software is designed to measure the jitter and its components: random jitter (Rj), deterministic jitter (Dj), data dependent jitter (DDj) duty cycle distortion (DCD), and periodic jitter (Pj). The SDA uses a powerful method called "Normalized Q-scale Analysis" to estimate/measure the random and bounded, uncorrelated jitter components. The following section presents the technical background underlying this method.

Interpretation of TIE Histogram – the Distribution of Edge Transition Times vs. Ideal (Expected) Transition Times

For the purposes of this discussion (in connection with jitter measurements[The "normalized Q-scale" method can be applied to other statistical studies and measurements; in particular, for examining the nature of vertical noise distributions.]) the entire subject surrounds the matter of interpreting the observed distribution of timing errors. This observed distribution is the histogram of Time Interval Error (TIE) values, obtained through analysis of either clock or NRZ data waveforms acquired by a digital recording instrument (such as a digital oscilloscope).

Relationship between Histograms and PDF

A histogram is nothing more than a form of data representation that expresses the frequency of occurrence of measurement values sorted or "binned" into adjacent, equal width contiguous intervals (or bins). When the timing errors (TIE) are collected as a histogram, the histogram serves as an approximation to the Probability Density Function (PDF) of this statistically based phenomenon (jitter). The PDF is (in theory) a smooth function determined by the underlying physics of the measured phenomenon (and of course what we actually observe includes the physics of the instrumentation as well).

The PDF is a continuous function, and reflects integrals of the probability (see http://en.wikipedia.org/wiki/Probability_density_function) over each interval of measurement value, *x*.

$$\Pr(a \le x < b) = \int_{x=a}^{x=b} PDF(x) \cdot dx$$

That is to say, the density of probability as a function of the measured quantity when integrated over a given region gives the probability that any measurement value will be within that region.

The process of forming a histogram is based upon a pre-specified set of bin boundaries (meeting the above conditions of contiguity and equal width). A further constraint, which is usually unspoken, is that the histogram range must cover all possible observation values if it is to be useful.

$$h_i = h_{leftmost} + i \cdot h_{width}$$

The set of observations (of measured quantity *x*) are "binned" or counted for each range of histogram bin. The resulting histogram is a collection of populations (or counts) for each bin region.

$$H_i = Population_i = \#observations(h_i \le x < h_{i+1})$$

Now a measurement histogram (like the one we will analyze to estimate jitter) represents a single experiment, with some number of trials or individual TIE measured values. It is only an approximation of the PDF insofar as the true PDF plays its probabilistic role and so is reflected in the resulting observations.

$$\Pr(h_i \le x < h_{i+1}) = \int_{x=h_i}^{x=h_{i+1}} PDF(x) \cdot dx = \frac{\#observations(h_i \le x < h_{i+1})}{\#observations(-\infty \le x < \infty)} = \frac{H_i}{\sum H_i}$$

For this case (jitter) the observed distribution is the histogram of time interval error (TIE) values, obtained through analysis of either clock or NRZ data waveforms acquired by a digital recording instrument (like a digital oscilloscope).

Integrating the PDF's

The Cumulative Distribution Function (CDF or c.d.f.) expresses the probability that an observation will fall between $-\infty$ and the value *x*.

$$CDF(x) = \Pr(-\infty < x' < x) = \int_{x'=-\infty}^{x'=x} PDF(x') \cdot dx'$$

Of course this is a purely theoretical value. We can, however, calculate the Empirical Distribution Function (EDF or e.d.f.) by summing the histogram from the left extreme to some value *x*.

$$EDF(x = h_i) = \frac{\sum_{k=0}^{k=i-1} H_k}{\sum_{k=0}^{k=N-1} H_k} = \frac{1}{P_{total}} \sum_{k=0}^{k=i-1} H_k$$

Now, for our purposes and in keeping with tradition for discussions of jitter, we make a small change of representation. We are interested in variations in timing (jitter) before and after the mean timing value (or, to left or right of the mean). As such, and in order to view the timing errors in a symmetric fashion, we paste together two halves, the right-hand and the left-hand parts of the EDF, where the left-hand part is summed from the left (increasing bin index, starting with leftmost bin) and the right-hand part is summed from the rightmost bin with decreasing bin index, noting that these are approximations of the right-hand and left-hand parts of the “bathtub curve.”

$$EDF_{left}(x = h_i) = \frac{1}{P_{total}} \sum_{k=0}^{k=i-1} H_k$$

$$EDF_{right}(x = h_i) = \frac{1}{P_{total}} \sum_{k=N-1}^{k=i} H_k$$

In practice, these two functions are joined at the median of the histogram (the bin containing the median, or the bin for which 50% of the total population is in that bin or those with lower index, and consequently 50% of the total population also falls within that bin and those bins with higher index.

To the point of all of this, in telecommunications or data communications, the relative rate of bit errors is called BER[Sometimes called bit error ratio, BER is the relative rate of bit errors compared to the bit rate, often expressed as a power of ten. For example: the BER is “ten to the minus 15th.”] .

Extrapolation of the Distribution Tails (Extremes)

Historical Note: Before this approach was taken by LeCroy, the extrapolation was applied to the histogram tails, whereas now the extrapolation is applied to the EDF; in fact, to each half of the EDF, as described previously.

While the EDF is well defined in the central region of the histogram, where events are populous, it is poorly defined at the extremes. This is of course the nature of the problem of analyzing jitter, since we are most interested in learning about the nature of those most rare (timing error) events.

So we strive to obtain an estimation of how this extremal behavior is even beyond where we have real observed data. To this end we strive to fit the data at the extremes of the EDF to plausible mathematical forms (suggested by the underlying physics). One such mathematical form is the “error function” which is closely related to the CDF of a Gaussian distribution.

The Error Function erf(x), Inverse Error Function $erf^{-1}(x)$ and Related Functions

$$CDF(x) = \frac{1 - erf(x)}{2}$$

Where the error function itself is

$$erf(x) = \frac{2}{\sqrt{\pi}} \int_0^x e^{-t^2} dt$$

And the complementary error function

$$erfc(x) = \frac{2}{\sqrt{\pi}} \int_x^\infty e^{-t^2} dt = 1 - erf(x)$$

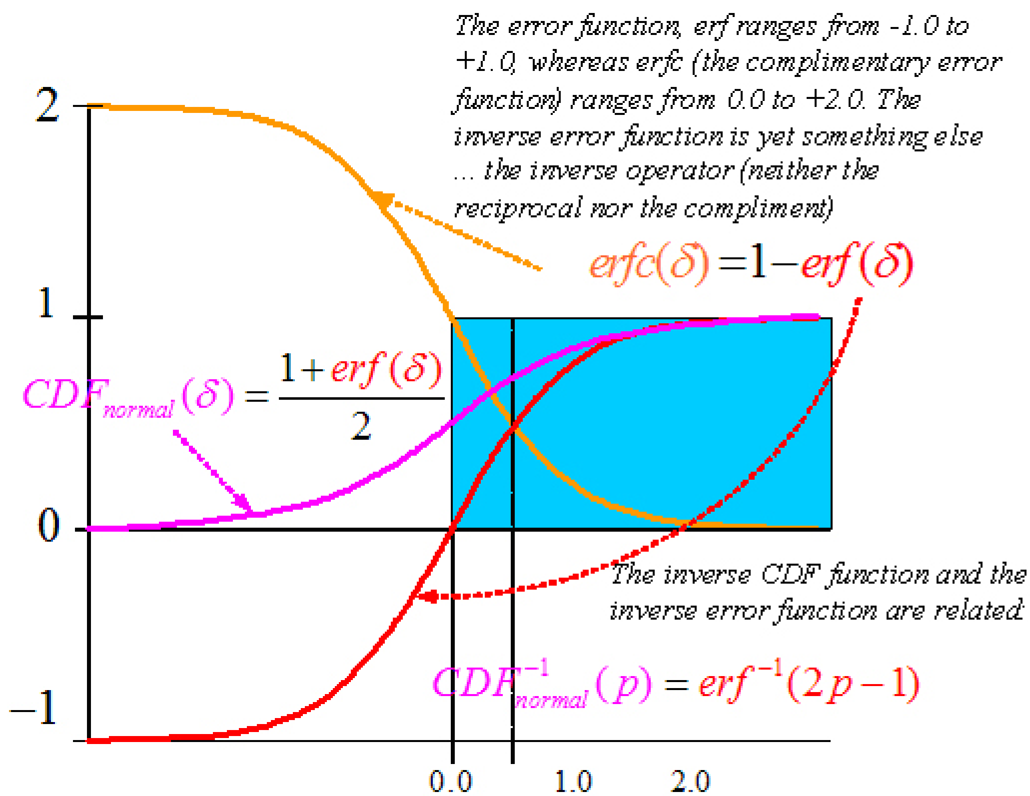
But more importantly, the inverse error function *erf-1(x)* is the inverse function (neither reciprocal nor complementary). This function gives the displacement *x*, for which a given value is the error function of that displacement. (See http://mathworld.wolfram.com/InverseErf.html for more detail on the inverse error function.)

$$erf^{-1}(erf(x)) = x$$

And

$$erf(erf^{-1}(z)) = z$$ for -1< z >1

The figure below can help to put the general shapes of these functions in perspective. In particular, remember not to confuse the complementary error function *(erfc)* with the inverse error function (erf^{-1}).



The Relationship between the Inverse Error Function and Total Jitter

It is noteworthy that the inverse error function has no closed analytical form. It is also noteworthy that the heuristic jitter equation is linearly related to this inverse error function. Recall this equation:

$$Tj(BER) = Dj + \alpha(BER) \cdot Rj$$

Usually when this equation is presented, a qualifying remark might be "and the function *α(BER)* is the number of standard deviations for a Gaussian with sigma of 1, that corresponds to the specified bit error ratio (BER)." Now, it turns out that *α(BER)* is to within a constant factor, exactly the inverse error function, $erf^{-1}(1-BER)$. That is:

$$\alpha(BER) = 2 \cdot \left| CDF^{-1}_{Gaussian}\left(\frac{BER}{2\rho_{tx}}\right) \right|$$

$$\alpha(BER) = 2 \cdot \left| erf^{-1}\left(\frac{BER}{\rho_{tx}} - 1\right) \right|$$

Where in both cases we explicitly incorporate the transition density[This is the ratio of transitions between bit values to the total number of bits (so upper bounded by 1) but normally about 0.5 for standard test patterns and, in particular, for PRBS patterns. See white paper from LeCroy on this subject.] , since the purpose of the alpha factor is to calculate *Tj*, and transition density is required for this purpose (since jitter is only pertinent for bit errors that have transitions).

Application of Error Function to Measured Jitter CDF (on Q-scale)

There is a notion in science of "preferred coordinates." The notion goes a bit like this: "a physical-mathematical relationship can be most simply expressed when the coordinates of the problem are well chosen." An example is Kepler's Laws. When expressed in the usual Cartesian coordinate system, those laws are rather obscure; but, expressed in polar coordinates, they are quite simple.

The problem of analyzing the EDF in order to predict a CDF is best served by a coordinate transformation for the variable BER.

The subject of Q-Scale has been proposed and described by several sources[Fibre Channel – Methods for Jitter and Signal Quality Specification – MJSQ, T11.2 project 1316-DT Rev. 14, June 9, 2004
Jitter Analysis: The Dual-Dirac Model, RJ/DJ, and Q-scale, Ransom Stephens, Agilent Technologies white paper, December 31, 2004] .

The desired transformation is to a new variable *Q* obtained from BER as follows:

$$Q(BER) = CDF^{-1}_{Gaussian}(BER / 2)$$

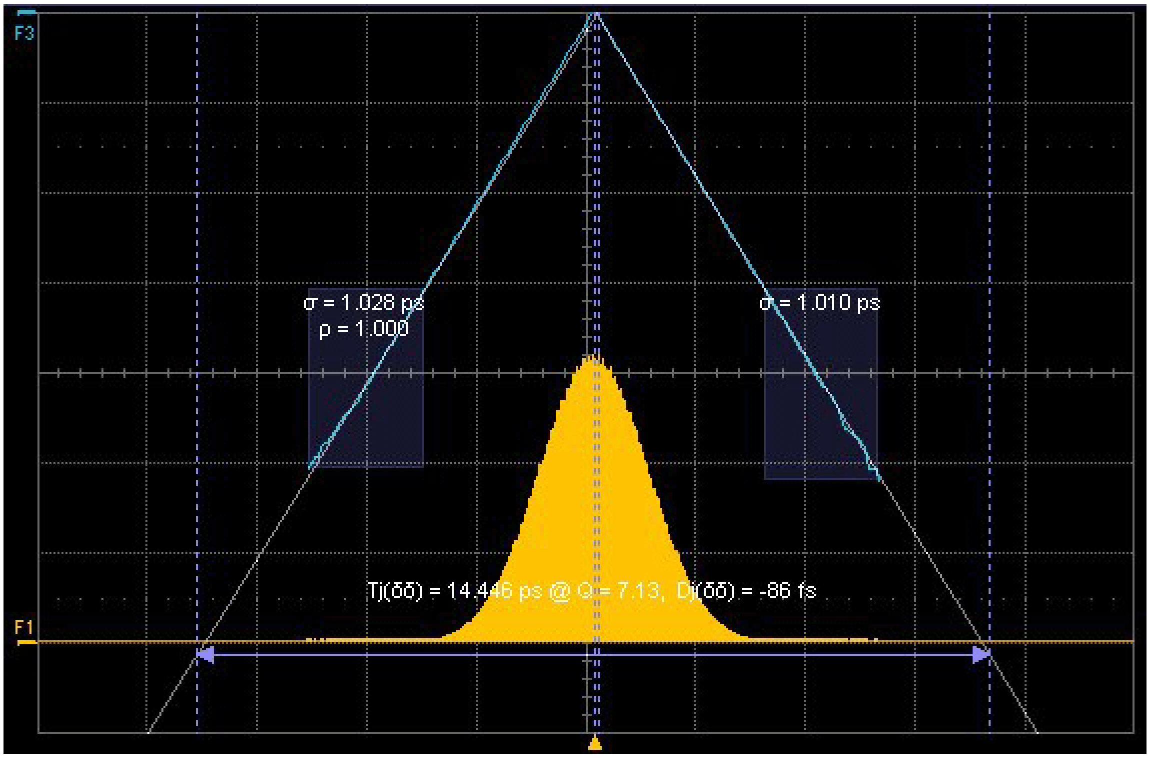
The Cumulative Distribution Function, for a normal Gaussian is related to the error function in the following way (see graphic).

$$CDF(x) = \frac{1 - erf(x)}{2}$$

The *CDF* being the function that provides the probability for an event (for a system obeying a Gaussian probability density function) occurring to the left of the value *x*. Note also, by inspection:

$$CDF^{-1}(p) = erf^{-1}(2p - 1)$$

The reason this is such an interesting (and thus preferred) representation for the CDF and EDF is that on this scale the CDF of a Gaussian PDF is a straight line. When the CDF or EDF is of the modified symmetric form, then their graphs appear as the upper lines of a triangle. Below is a plot of an EDF (simulation) for a Gaussian PDF with a sigma of 1 picosecond.

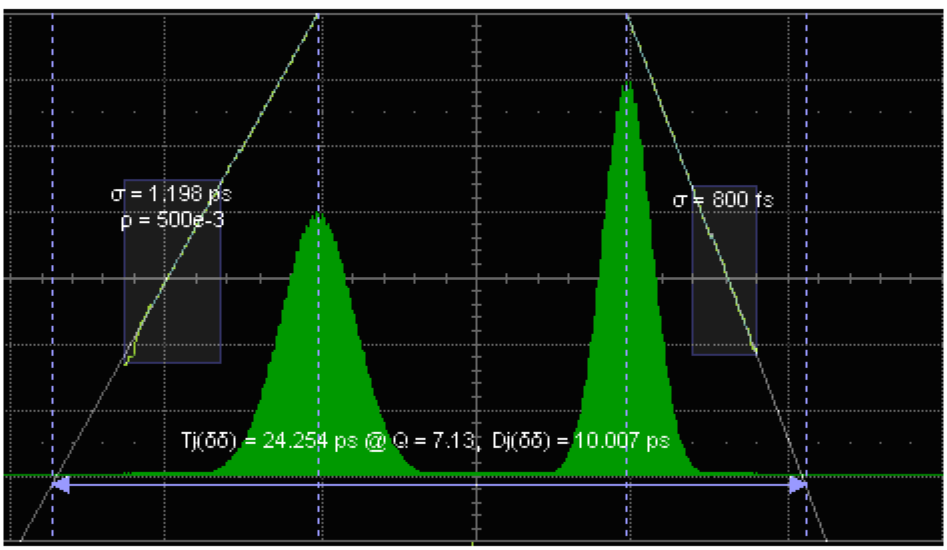


The other interesting attribute of this representation is that the slope of the lines gives the sigma of the distribution. This is all common treatment so far. All is well for a single Gaussian distribution.

Now enter the idea that this coordinate transformation should have a variable normalization factor. This normalization is such that when the area of an un-normalized Gaussian is ρ_{norm}, then that resulting CDF manifests as straight lines with slope revealing sigma, and intercept with Q=0, giving the mean of the Gaussian PDF. This is the "normalized Q-Scale" where:

$$Q_{norm}(BER) = CDF^{-1}\left(\frac{BER}{2\rho_{norm}}\right)$$

As an example, when two different Gaussian distributions are analyzed in this way, their EDFs appear as below. The linear fits on this coordinate scale yields the proper (simulated) values for their sigmas and means.



Automatic Renormalization of the Q-scale

This is the heart of the (patent pending) method for analysis of EDF, yielding best estimates of the underlying PDFs defining the measured system's behavior.

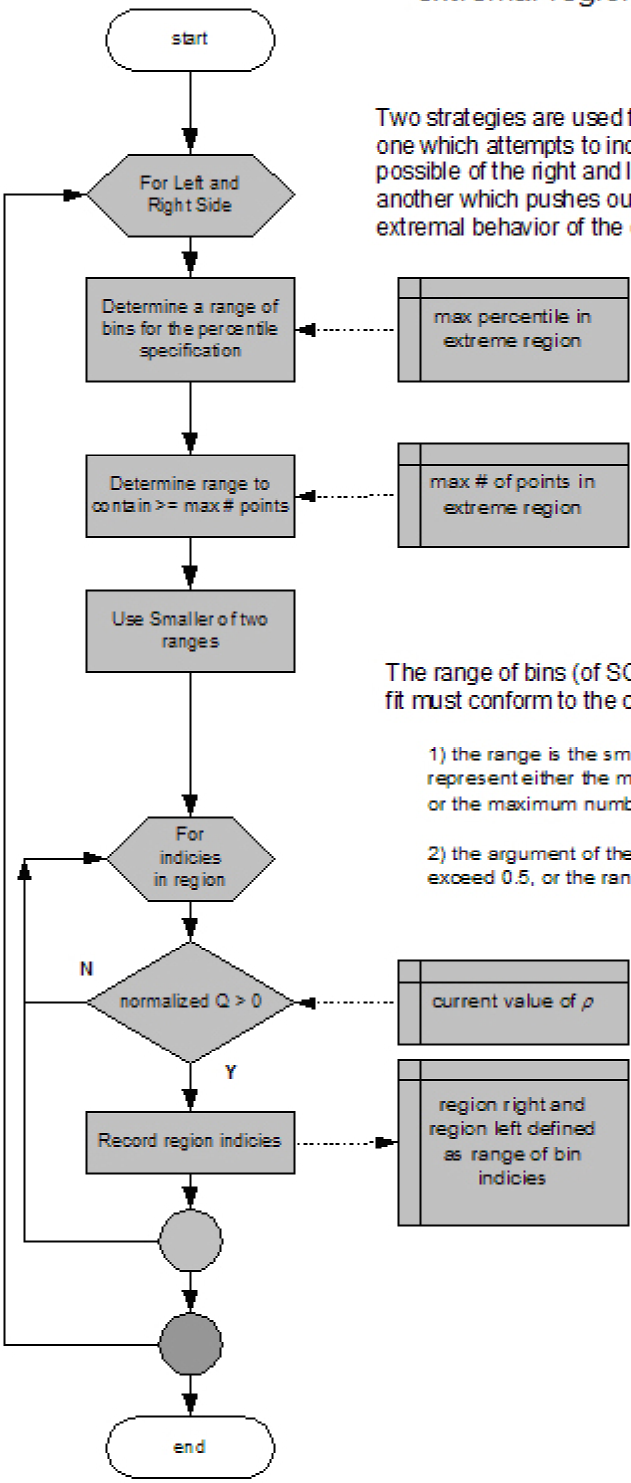
The procedure is straightforward: create the symmetric EDF for each side of the distribution.

The method can be described in steps as:

1. Forming a set of elements to be fitted, which are 3 values each, a vertical (Q-scale) value, a horizontal coordinate (for the case of time variable jitter, but the method is extensible to vertical noise analysis or, in fact, any variable), and an associated error. Statistically, the error of the input variable $\Sigma\rho_n$ is furnished as the square root of the total population of observations (the same sum) contributing to the estimate (of EDF in this case). Upper and lower values of BER are obtained for these variations, and an error inversely proportional to the error is assigned to the data point.
2. Varying the Q-Scale normalization factor over a range of plausible values, the "best" fit to linear behavior is determined.
3. Using this value to correctly scale the probability, an intercept (with $Q_{norm}(BER)=0$) is obtained for both the right-hand side and the left-hand side of the $CDF(Q_{norm}(BER)$. This value may be interpreted as the mean-value associated with the noise/jitter contributor with sigma equal to the reciprocal of the slope obtained from the best linear fit, and amplitude (area) equal to the best normalization factor itself.
4. Under the assumption of symmetric noise/jitter contributions a single ideal normalization may be obtained and used, however, ideally both extremes of the EDF must be treated independently, yielding strengths (ρ_{norm}) and sigmas (reciprocal slope) to characterize the extremal behavior of each side of the EDF distribution. In this case we show only an example of one side of the EDF.

Selection Method for points in extremal regions to be fitted

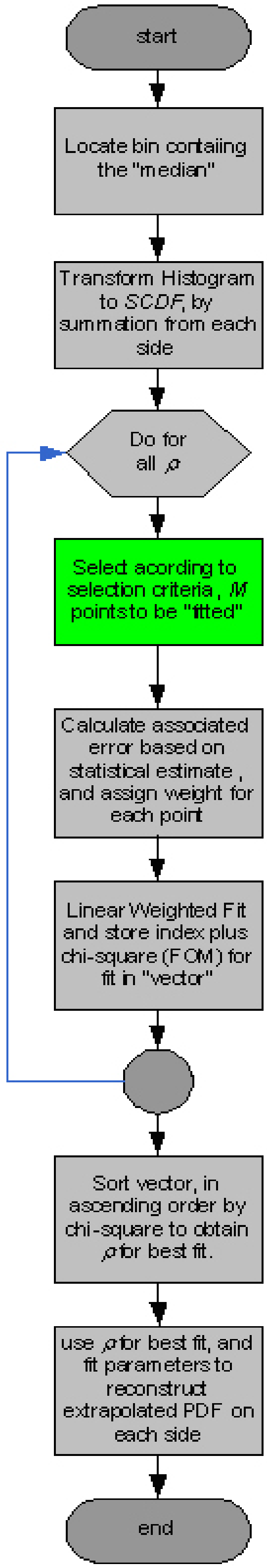
Two strategies are used for this procedure, one which attempts to include as much as possible of the right and left shoulders, and another which pushes out to find the extremal behavior of the distribution.



The range of bins (of SCDF) used for the fit must conform to the conditions:

1) the range is the smaller of the ranges required to represent either the maximum percentile of the PDF or the maximum number of points specified.

2) the argument of the inverse CDF must not exceed 0.5, or the range is reduced.



The smallest integer value k, for which $\frac{N_{total}}{2} \le \sum_{i=0}^{i=k} N_i$

$$SEDF_{m\le k}(N_i, N_{total}) = \left(\frac{1}{N_{total}} \left(\sum_{i=0}^{i=m} N_i \right) \right)$$ let *SEDF*

$$SEDF_{m>k}(N_i, N_{total}) = \left(\frac{1}{N_{total}} \left(\sum_{i=M-1}^{i=m} N_i \right) \right)$$ right *SEDF*

$$\rho_{norm} = 1.0, 0.95, 0.90, \ldots 0.05$$

$$Q_{norm}(BER) = CDF^{-1}_{Gaussian}(BER / (\rho_{norm} \cdot 2))$$

$$Q_i(N_i, N_{total}) = CDF^{-1}_{Gaussian}\left(\frac{1}{(\rho_{norm} \cdot 2N_{total})} \left(\sum_{i=0}^{i=k} N_i \right) \right)$$

$$if (N == 1)$$

$$w_i = 1/\Delta_i = 1/\left(2Q((N_i + \sqrt{N_i})/N_{total}\right)$$

$$elseif (N > 1)$$

$$w_i = 1/\Delta_i = 1/\left(\begin{array}{l} (Q((N_i + \sqrt{N_i})/N_{total}) - \\ (Q((N_i - \sqrt{N_i})/N_{total}) \end{array} \right)$$

$$x_i = x_0 + (i \cdot \Delta x)$$

Weighted Linear Fit based upon the minimization of chi-square wrt. the variable m and b (slope and intercept)

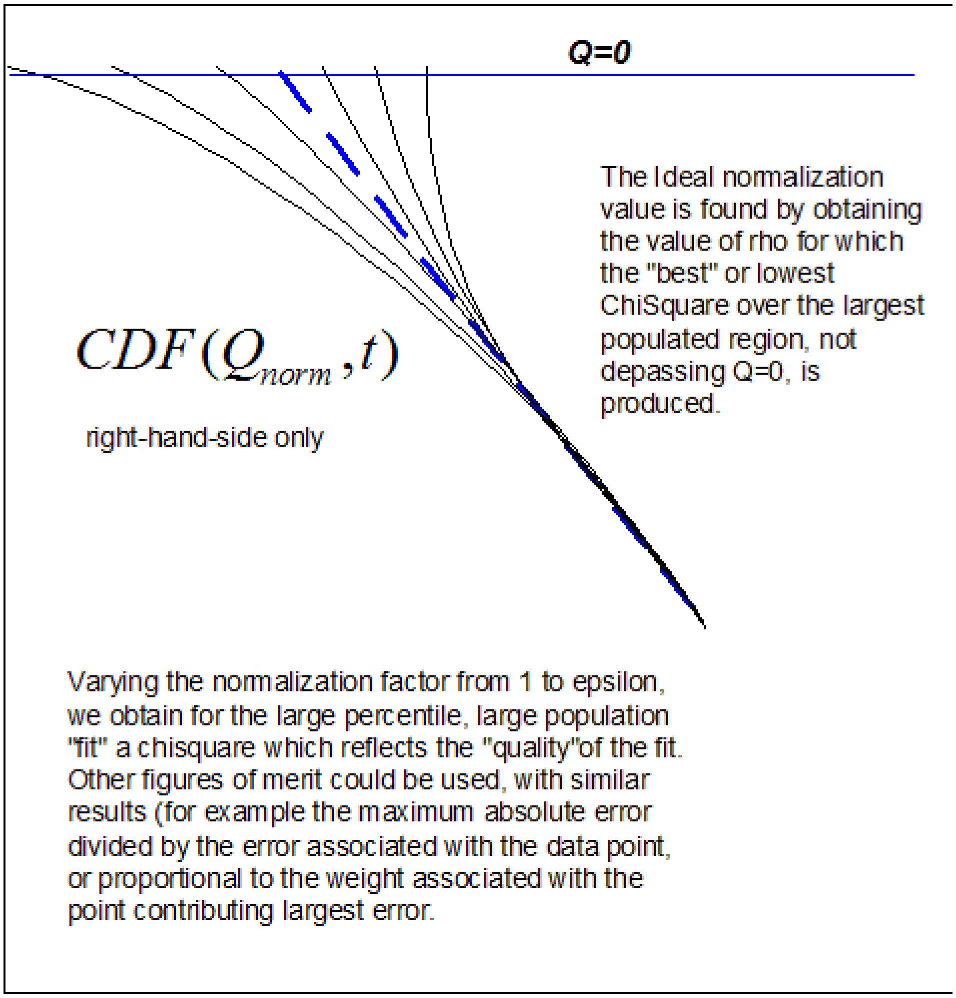
$$\chi^2 = \frac{\sum_{j=0}^{M-1} w_j^2 (Q_j - mx_j - b)^2}{\sum_{j=0}^{M-1} w_j^2}$$

Sort from lowest chi-square to highest, to determine the ideal vlaue for ρ

$$\sigma = |1/m| = |1/slope| \qquad \mu_L = line_L(x @ y = 0) = -b/m$$

$$PDF_i' = \left(\frac{\rho_{norm}}{\sigma\sqrt{2\pi}} e^{-\frac{(x_0 - j\Delta x - \mu_L)^2}{2\sigma^2}} \right)$$

To fill in PDF bins to the left (including region fit from left), and rather similarly fror the right hand side, using results of the righthand fit.



Obtaining Deterministic and Random (Gaussian) Components from the Normalized Q-scale Diagram

While there are detailed differences between this and other approaches for obtaining Rj and Dj, the most important figure we need to establish is total Jitter. Referring roughly back to the heuristic jitter equation, we rewrite this as:

$$Tj(BER) = Dj + 0.5\alpha(\frac{BER}{W_{left}}) \cdot Rj_{left} + 0.5\alpha(\frac{BER}{W_{right}}) \cdot Rj_{right}$$

where Dj is the separation between the two means of the distributions.

If the distribution is governed by only two Gaussians of equal weight, this equation degenerates into

$$Tj(BER) = Dj + 0.5\alpha(2BER) \cdot Rj_{left} + 0.5\alpha(2BER) \cdot Rj_{right}$$

And if, in addition, the two Gaussians have the same sigma (or Rj), then

$$Tj(BER) = Dj + \alpha(2BER) \cdot Rj$$

This equation is disturbingly different from

$$Tj(BER) = Dj + \alpha(BER) \cdot Rj$$

This is because in the traditional Dual-Dirac discussion, we typically “forget” that the two Gaussians are only half-strength.

In general, to approximate the traditional Dj and Rj value, we are still using the separation of the “outermost means, and approximating Rj by the mean of the two sigmas.

We do, however, calculate Tj from the most precise estimation of the CDF from the renormalized Q-scale fit, using correct weights and sigmas to reconstruct a theoretical PDF, using the central region of the histogram "as is" and the extremes replaced by theoretical Gaussians of the correct strength, sigma, and mean to estimate each extreme region. This is then reconstructed into a CDF, so Tj(BER) becomes a simple calculation on that curve.

Overview of Multi-Eye Measurement Tools

The Multi Eye Measurement Tool is designed to improve the analysis and processing of serial data signals in high-speed buses and interfaces commonly used in desktop, server, and storage applications. It is available with version 4.3 of X-Stream software in SDAs and WaveMaster DSOs. Its main features are:

- Three Additional Modes of Eye Diagram display:
  - Front Side Bus (FSB) Application
  - Transition/Non-Transition Bit Application (PCI Express)
  - Gated (Qualified) Eye Diagram
- Eye Diagram Measurements available in every mode:
  - Amplitude measurements (Height, Amplitude, Zero Level, One Level)
  - Timing Measurements (Rise Time, Fall Time)
  - Eye-Specific (Crossing, Eye BER)
- Seamless Integration with Standard masks:
  - User-Selectable, Dual mask inputs
  - Dual Mask Violation Testing

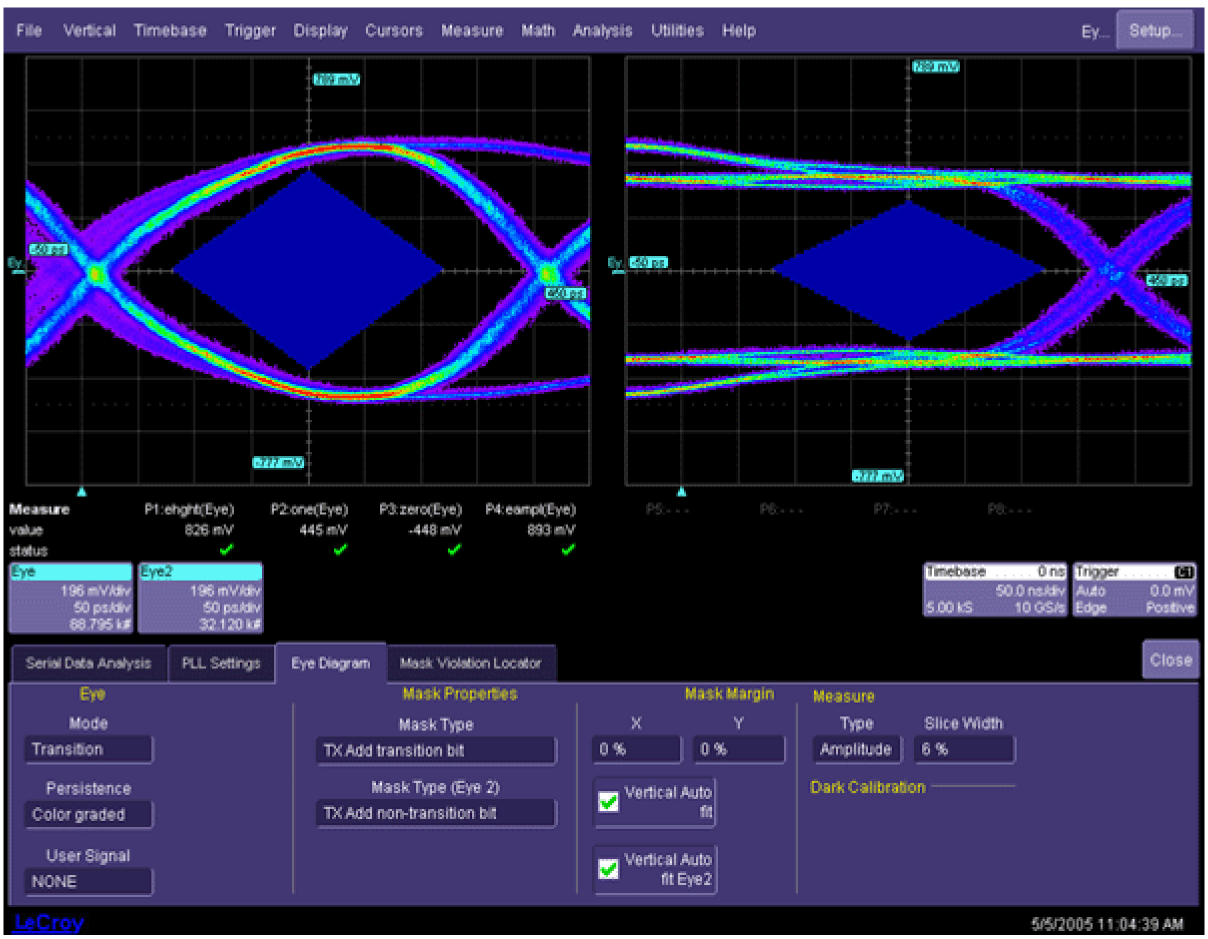


Figure 5-47. Transition Mode Display.

Multi-Eye Setup and Installation

The Multi Eye Measurement Tool does not require any additional software installation, as it is fully integrated into the SDA software. However, depending on the specific mode of operation selected, each of the three new modes of operation requires distinct instrument and channel setups.

For example, the FSB mode requires setting up multiple channels to carry out the eye diagram measurements, as described in the example below. The Gated Eye Diagram requires at least two inputs: The Data and the Gate signal. The Transition Mode requires at least one input (the Data Stream, but depending on the specific serial data standard, a clock input may also be required to recover the clock information).

These signals (except for the strobes) are differential signals, so the use of differential probes is often required to carry out the measurements. As with any high-speed precision measurements, all probes and SMA cables should be calibrated and deskewed prior to any data collection.

Example Setups

FSB Eye Mode Configuration

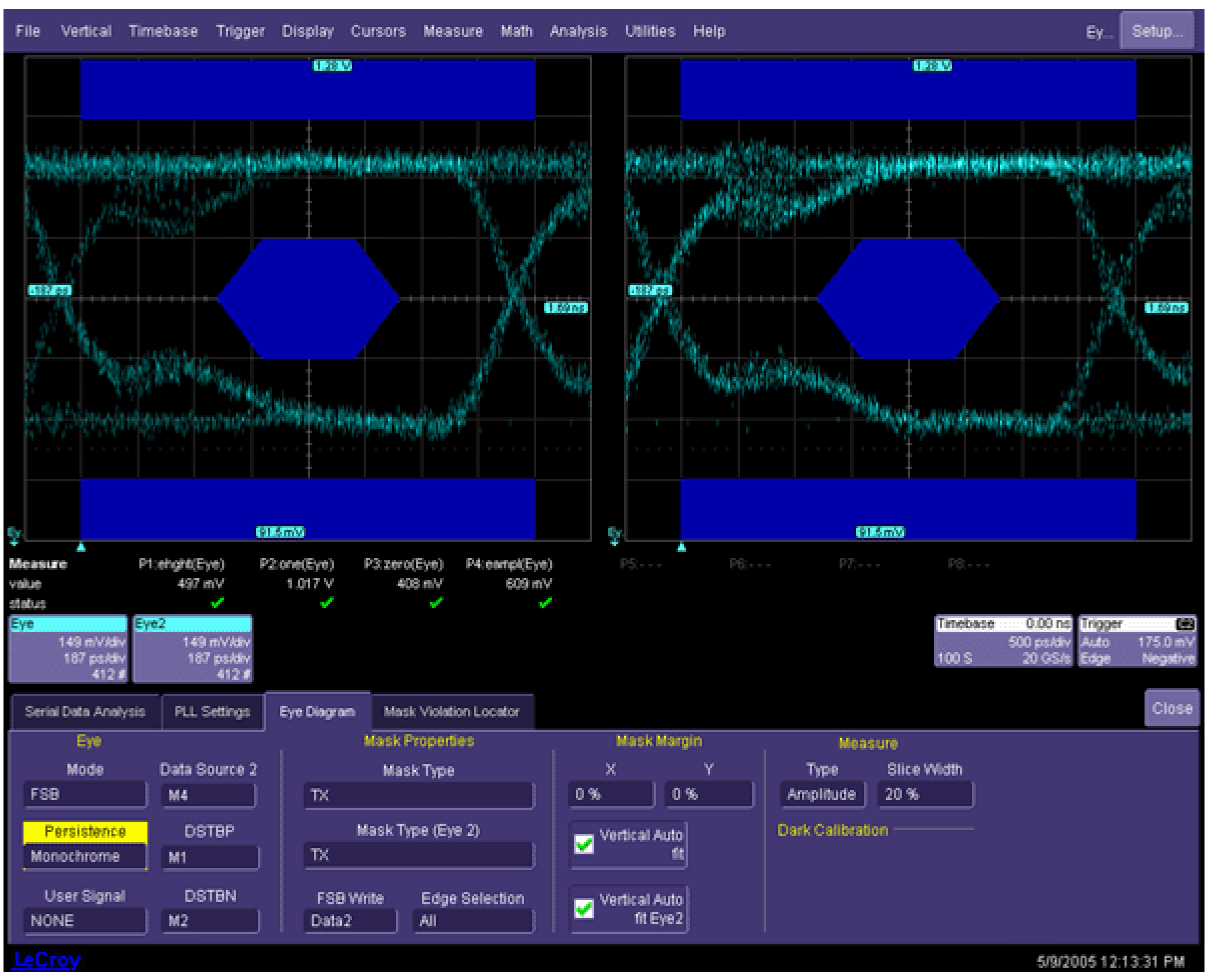
Channel Setup

- DataStrobep: Channel 1 (M1)
- DataStroben: Channel 2 (M2)
- Data1: Channel 3 (M3)
- Data2: Channel 4 (M4)

Serial Data Analysis Setup

1. Select Signal Type: FSB 533MHz, 667MHz or 800MHz
2. Uncheck Recover Clock -- No clock recovery is required in FSB case
3. Select Eye Diagram Tab, then configure the screen as follows:
   - Mode: FSB
   - Data Source 2: Channel 4 (M4)
   - DSTBP: Channel 1 (M1)
   - DSTBN: Channel 2 (M2)

The screen should look like this.



Eye Diagram Setup and Measurements

1. Select Mask for each Eye Diagram -- Depending on the Standard selected, there may be one or several types of serial data masks available.
2. Select a Mask Margin (as % of the nominal mask size) if desired.
3. Select a Measurement Type. There are three groups of parameters:
   - Amplitude
     - Eye Height
     - One Level (Eye)
     - Zero Level (Eye)
     - Eye Amplitude
   - Timing
     - Eye RiseTime
     - Eye FallTime
   - Eye
     - Eye Width
     - Eye RMS Jit
     - Eye BER
     - Eye Crossing
     - Avg Pwr (Eye)

Mask Test (Mask Violation Locator) Setup

1. Select the "Mask Violation Locator" tab.
2. Check the **Testing On** checkbox to detect mask violations.
3. Check the **Show Location** checkbox to select display of bits where mask violations occur.
4. Check the **Stop on Error** checkbox after **N Failures**.

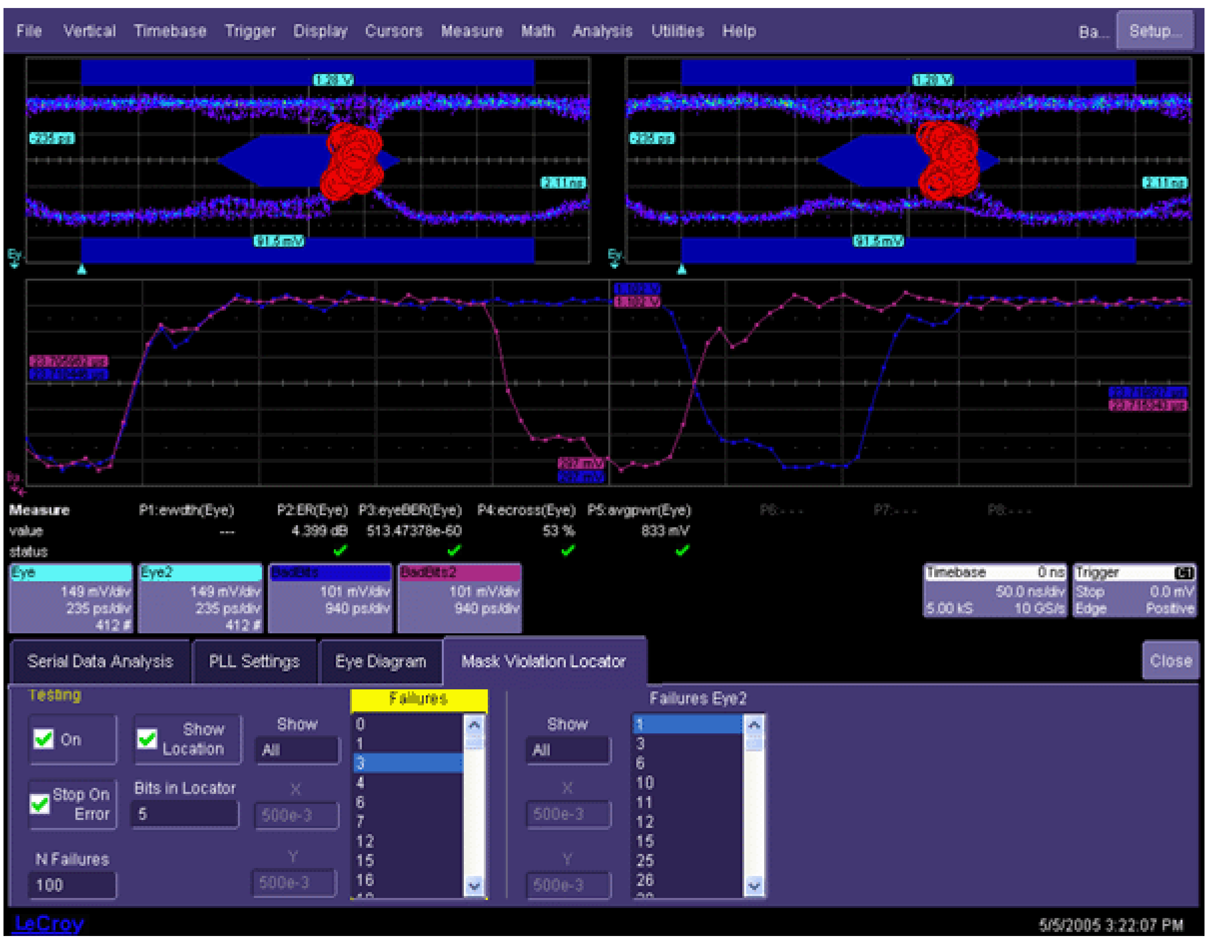


Figure 5-48. Mask Violation Setup.

Front Side Bus (FSB)

Introduction to FSB

Front Side Bus is also known as the processor bus, memory bus, or system bus. The FSB connects the processors to the main memory, and is used to connect to other resources within the computer. The FSB can range from speeds of 66 MHz, 133 MHz, 100 MHz, 266 MHz, 400 MHz, and higher.

The FSB speed can generally be set either using the system BIOS or with jumpers located on the computer motherboard. FSB can accommodate one or two CPUs. This section of the manual describes single CPU operation.

FSB Theory of Operation

There are three sub-buses in the FSB: data bus, address bus, and common clock bus. All these buses are bidirectional; information can flow in either direction. The direction of the bit flow will be determined by the relative position of the high-to-low transition in the strobe pins (address, data or clock) when simultaneously probed at both ends of the applicable bus. For example, when signal A transition is delayed relative to signal B transition, this means that B is the transmitter and A is the receiver. The bus clock (BCLK) signal is formed by a differential pair, BCLK0 and BCLK1, running at 533 MHz. This is the trigger signal for the bit transactions, with the strobe signals as the qualifiers for the actual bit transfers.

Data Bus Characteristics (Refer to the Following Figure)

- During every period of BCLK, 4 bits are sent or there's an idle state: high voltage on both strobes and relevant data pins.
- Bit transfers occur in quad multiples of bits only (4, 8, 12…).
- Data Strobe positive (DSTBp#) samples bits 0 and 2 on consecutive falling edges
- Data Strobe negative (DSTBn#) samples bits 1 and 3 on consecutive falling edges

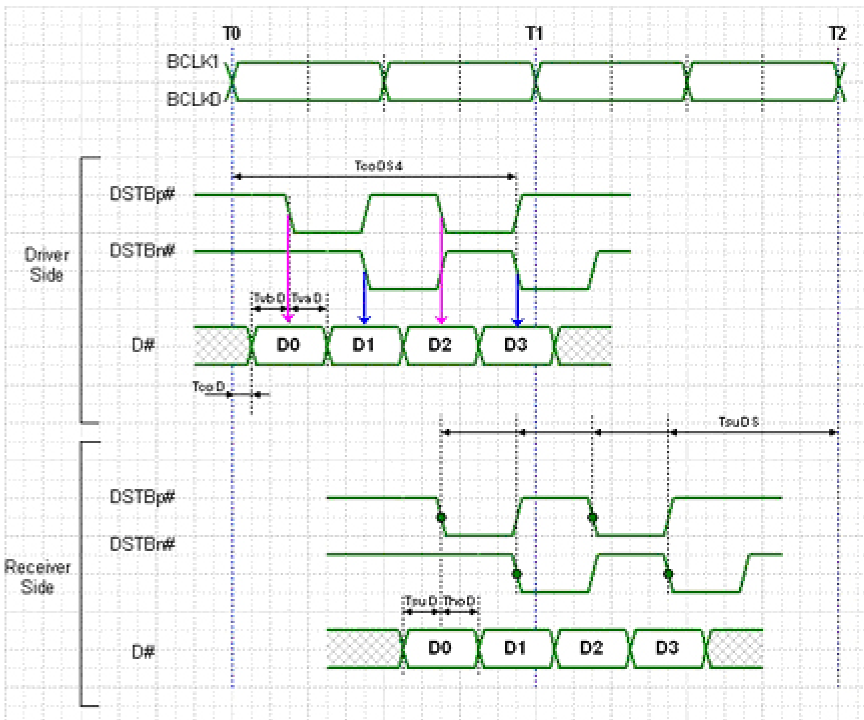


Figure 5-49. Data Bus Timing Diagram ©Intel Corporation.

Address Bus Characteristics (Refer to the Following Figure)

- During every period of BCLK, 2 bits are sent or there's an idle state (high voltage on both strobes and relevant data).
- Bit transfers occur in double multiples of bits only (2, 4, 6…).
- There's sampling of address bits on both falling and rising edges of ADSTB
- Falling edge (ASTB#) samples address bit 0

- Rising edge (ASTB#) samples address bit 1

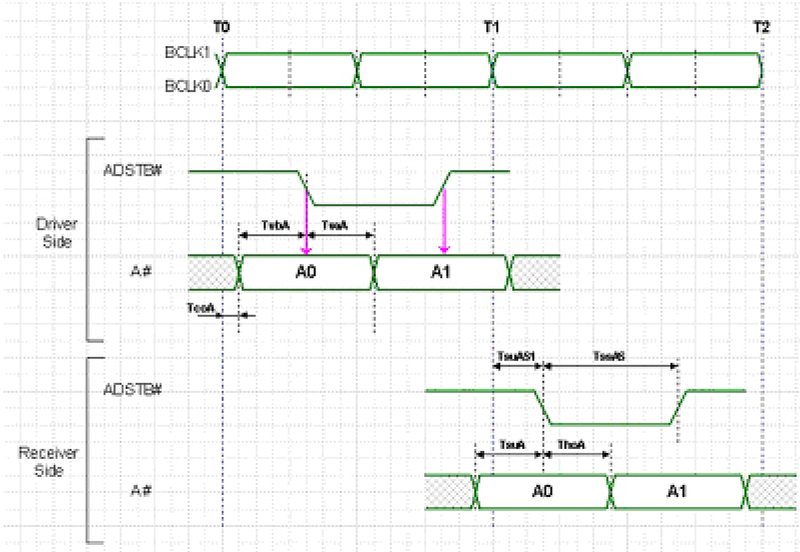


Figure 5-50. Address Bus Timing Diagram ©Intel Corporation.

Common Clock Bus Characteristics (Refer to the Following Figure)

- During every period of BCLK, 1 bit is sent or there's an idle state meaning high voltage on the signal.
- BCLK is a differential signal.

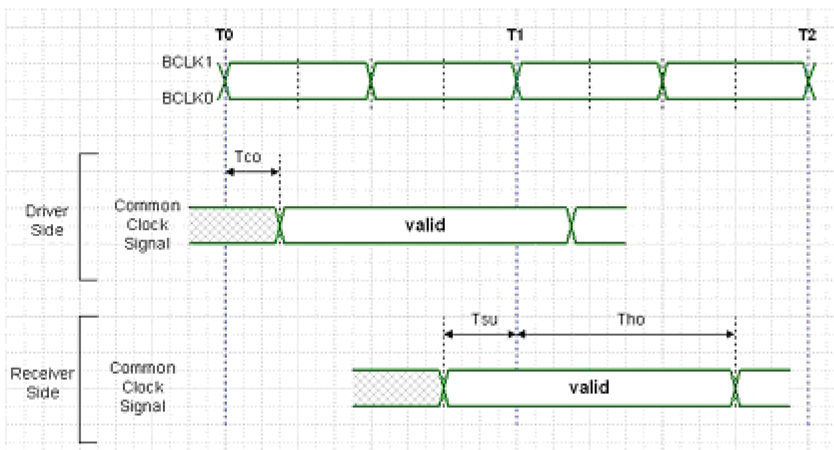


Figure 5-51. Common Clock Bus Timing Diagram © Intel Corporation.

Transition / Non-Transition Eye Diagram

A dual display eye diagram is formed with bits that are of non-changing polarity (non-transition) and changing polarity (transition). This display mode is useful for those serial data standards that utilize mask testing for both types of bit sequences (PCI Express and FB-DIMM Point to Point).

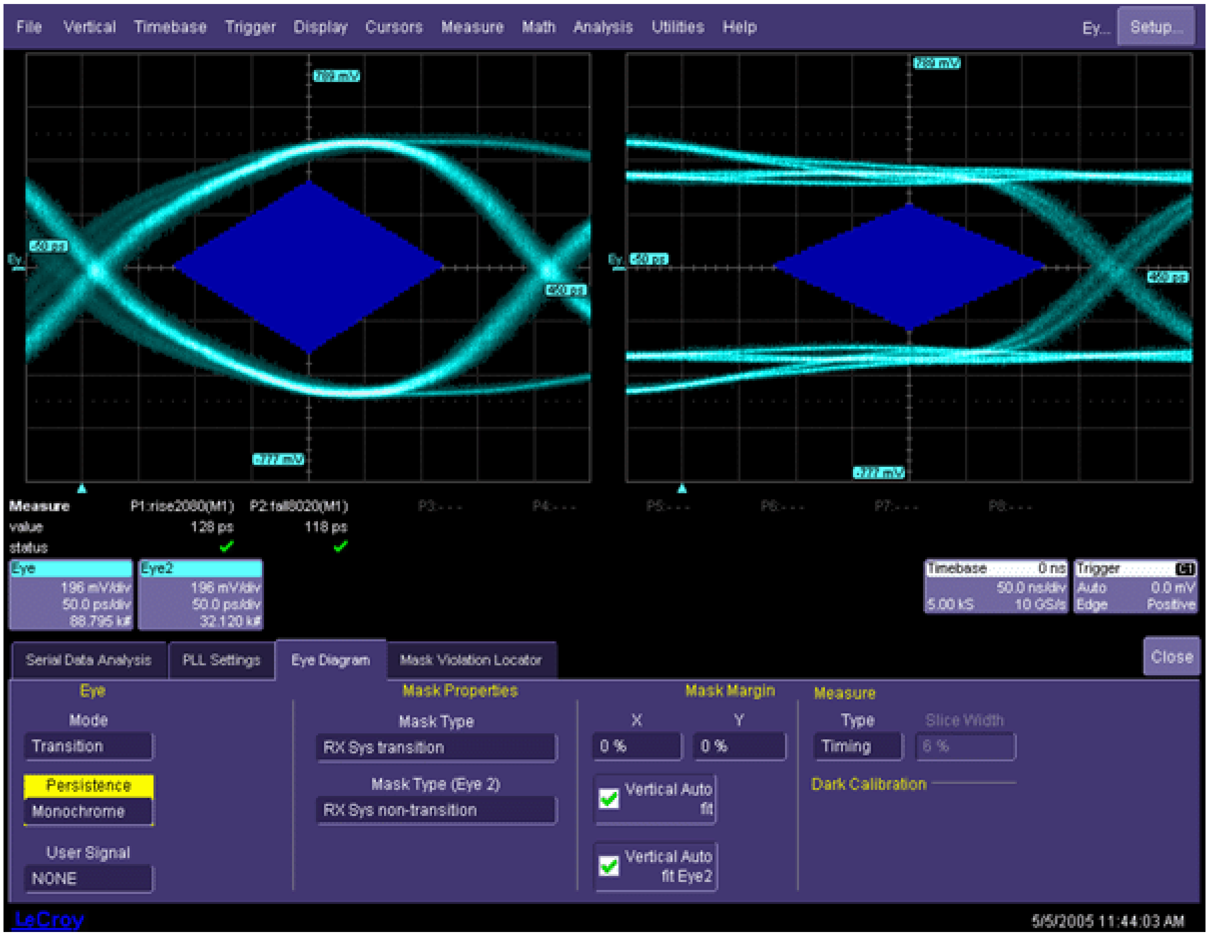


Figure 5-52. Transition Mode.

For example, Section 4.3.3.1 of PCI Express Specification defines the requirements for Transition and Non-transition eye diagrams:

> "There are two eye diagrams that must be met for the Transmitter. Both eye diagrams must be aligned in time using the jitter median to locate the center of the eye diagram. The different eye diagrams will differ in voltage depending whether it is a transition bit or a de-emphasized bit. The exact reduced voltage level of the de-emphasized bit will always be relative to the transition bit."

Gated (Qualified) Eye Diagram

The Gated Eye Diagram mode utilizes a separate signal (the "Gate" or qualifier) to create dual eye diagrams based on the polarity of the Gate.

Here's an example of a gated eye diagram. Channel 1 (M1) contains the raw data and Channel 2 (M2) contains a Gate signal. While the Gate signal is high, the UIs will transfer to Eye display; When the Gate signal goes low, the UIs will transfer to Eye2 display

Figure 5-53. Data (M1) and Gate (M2) signal sources.

The following display is the Eye diagram resulting from applying the Gate signal. When the Gate is high, the bit data goes to one eye diagram. When the Gate is low, data goes to the other eye.

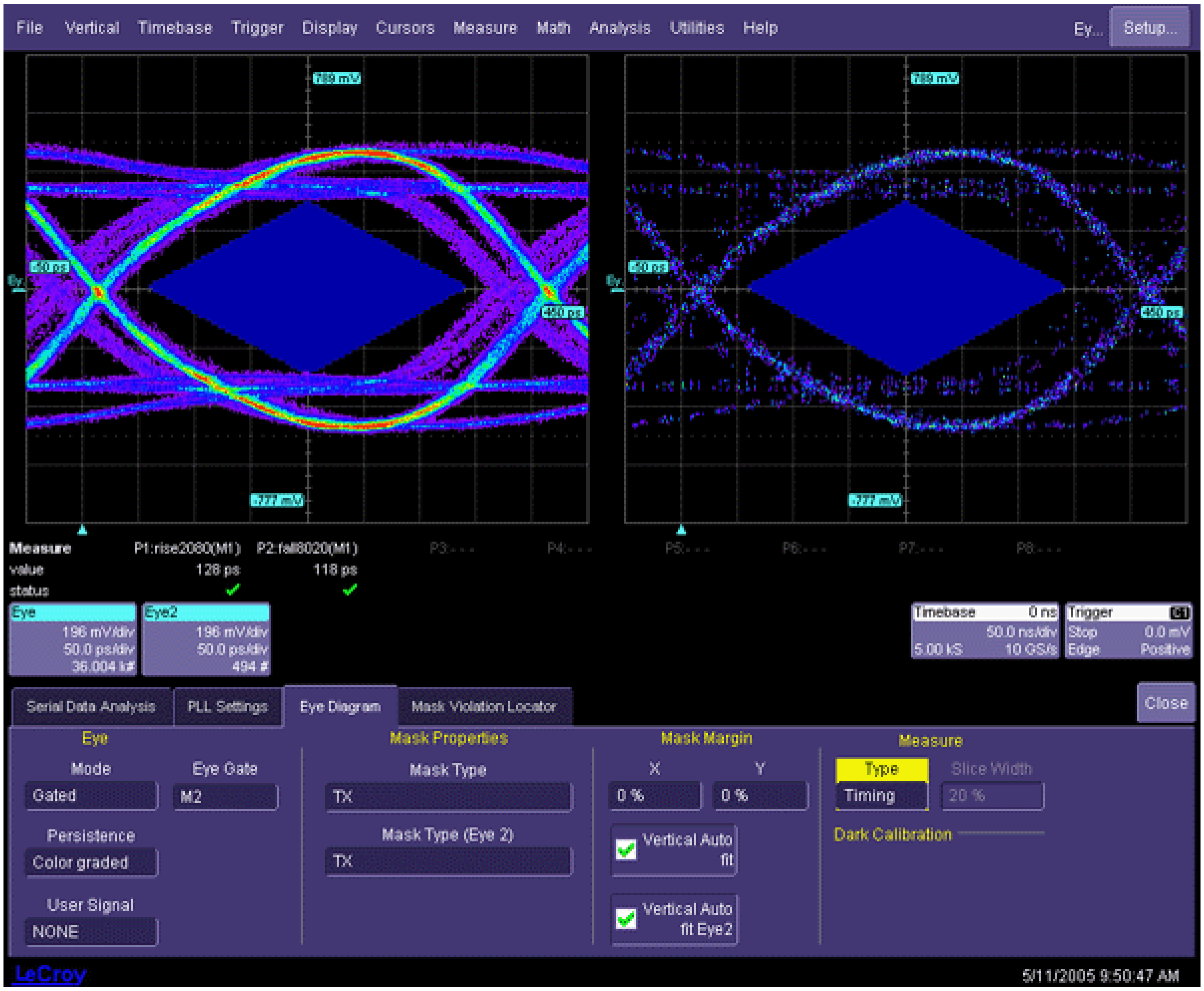
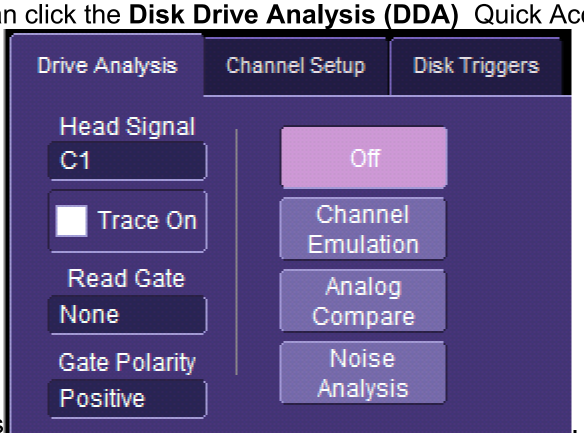


Figure 5-54. Gated Eye Diagram Example.

DDA Quick Access Toolbar Button

You can click the **Disk Drive Analysis (DDA)** Quick Access toolbar button to display the Drive Analysis setup dialogs .

DDA Specifications

Additional DDA Triggers

Sector Pulse: Triggers on the n^{th} sector pulse (1 to 50) after index. Index and sector pulse polarity and sector pulse number are selectable.

Servo Gate: Triggers on the n^{th} servo gate after index and every m^{th} thereafter. Index and servo gate pulse polarity are selectable.

PES Trigger: Triggers on Position Error Signal (PES) exceeding an adjustable voltage window. Servo gate can be selected as qualifier.

Read Gate Trigger: Triggers on any read gate longer than an adjustable Sector ID field length (100 ns to 50 μs).

Disk Drive Measurement Package (DDM2)

This package provides disk drive parameter measurements and related mathematical functions for performing disk drive WaveShape Analysis.

- Disk Drive Parameters:

amplitude symmetry	local time over threshold
auto correlation s/n	local time trough-peak
local base	local time under threshold
local baseline separation	narrow band phase
local maximum	narrow band power
local minimum	non-linear transition shift
local number	overwrite
local peak-peak	pulse width 50
local time between events	pulse width 50-
local time between peaks	pulse width 50+
local time between troughs	resolution
local time at minimum	track average amplitude
local time at maximum	track average amplitude-
local time peak-trough	track average amplitude+

- Correlation function
- Trend (datalog) of up to one million events
- Histograms expanded with 19 histogram parameters and up to 2 billion events

Automated DDA Measurements

ACSN	local time under threshold
local base	msnr
local baseline separation	m_to_r
local maximum	NLTS
local minimum	narrow band phase
local number	narrow band power
local peak-peak	overwrite
local time between events	pulse width 50
local time between peaks	pulse width 50-
local time between troughs	pulse width 50+
local time at minimum	resolution
local time at maximum	rsnr
local time peak-trough	track average amplitude
local time over threshold	track average amplitude-
local time trough-peak	track average amplitude+

Advanced DDA Analysis

- Head Filter/Equalizer Emulation
- Channel Emulation
- SAM Histograms
- Plot of SAM Values
- PES Runout Analysis
- Analog Compare
- Correlation
- Trend
- Histogram

Drive Analysis Overview

Obstacles that Can be Overcome using the DDA's Channel Analysis

Disk Drive engineers who are analyzing channels to determine where and why data errors occur face important obstacles. But the DDA's Channel Analysis feature can be used to overcome these obstacles.

Lack of Synchronization

The first obstacle is the lack of integration or synchronization between the computers used to identify and locate data errors, and the instruments that analyze the channel signals. It is, therefore, difficult to capture the signal that may be responsible for an error at the same point in time at which the error occurs. If an error is repetitive, its signal can be captured and viewed. But if the error is intermittent, capturing it at the correct time may be impossible.

Unknown Sectors

Another obstacle is that the data written to a particular sector may not be known. And because no reference is available, the exact location of an error in a particular sector cannot be determined. In order to have a known data set, data may subsequently be written to the sector concerned. Nevertheless, there is no guarantee that this will recreate the error. For many, writing to a sector with errors is the last resort.

Problematic PRML

Another problem is the difficulty of analyzing partial response maximum likelihood (PRML) head signal quality and identifying both the problem locations and the margin available before errors occur. Head signals for PRML channels have complex waveshapes, which are very difficult or even impossible to analyze by visual inspection. Analysis of these signals with an oscilloscope is often limited to looking for gross abnormalities such as significant thermal asperities or dropouts of sufficient duration. Furthermore, because of the sophistication of PRML signal processing, even some visible anomalies may not necessarily cause an error.

Time and Effort

Lastly, data in the sectors is complex and lengthy, and it can take significant time and effort to scroll through and visually inspect signal locations. The problem is of data presented in a time-linear fashion, rather than in the order of signal areas most deficient in PRML channel related signal integrity.

In an integrated environment, the Channel Analysis feature analyzes the head signal. Designed to overcome the obstacles just described, it combines powerful analytical tools with the ability to capture and analyze the head signal in a synchronized way.

What Channel Analysis Provides

The DDA's Channel Analysis feature provides insight into channel data quality and errors, even while data is being processed through the channel. It can also do the following:

- Filter and display the head signal with automatic or selectable settings for -3dB cutoff and boost
- Select head signal sections to be viewed by byte number
- Provide a display of PRML target levels annotated on the head signal trace to provide an intuitive visual indication of head signal quality
- Identify head signal locations of the poorest quality from a PRML standpoint, quantify the quality of the signal at these locations, and display in order of poorest quality the head signal location and the byte number of each error

- Identify areas where two head signal differences are greatest and provide a benchmark for the difference and the byte locations where these occur

The Channel Analysis feature can be operated in either of these basic modes:

- Channel Emulation
- Analog Compare

Because each requires a similar setup, both share the same Channel Setup menu system, which lets you switch between modes without the need to redefine setup details.

Channel Emulation

Often when examining a partial response maximum likelihood (PRML) data signal, it may be desirable to view those signal locations where the quality is poorest and the signal is likely to be the most problematic for a PRML channel.

Channel Emulation enables analysis of PRML head signal quality from the point of view of the PRML channel. This is made possible by the emulation of PRML channel processing by means of signal equalization, automatic gain control (AGC), phase lock loop (PLL), sampling, and Viterbi detection.

For each PRML sample, a quality sequenced amplitude margin (SAM) benchmark is determined. The lower the value of SAM the greater the difficulty for the drive channel to produce the right data value for that sample, and the less margin there will be. A SAM value of less than zero indicates that an incorrect data value will be selected at that sample.

PRML signals characteristically have a set of target-level values that the signal samples should meet at the sampling times. For example, for a PR4 signal, the targets are **1 0 -1**. In order to achieve these desired levels, the DDA’s channel emulation equalizes the head signal, either automatically or using selected values for -3 dB cutoff and boost. The equalized head signal is then displayed. This lets you examine the head signal after it is processed, when it will have the characteristic PRML waveshape.

The distance of the waveform samples from the PRML target values is a first-order indication of the quality of a PRML signal. The channel emulation annotates the head signal after equalization with the target values. This makes visual interpretation of the quality of the head signal possible and intuitive. As a result, you can visually inspect the head signal to an extent beyond the more obvious indications of problems such as thermal asperities and dropouts.

In addition, several powerful ways are available for selecting how the section of the equalized, annotated head signal will be viewed:

- You can scroll through the head signal in the traditional time sequential mode using the Auto Scroll feature, which provides "hands free" scrolling at a rate and in the direction that you specify.
- You can select which part of the head signal is to be displayed by byte number. This is particularly useful if a data error is known to exist at a particular byte location.
- You can review the head signal in order of poorest SAM value. This capability is particularly practical because the areas of poorest quality are generally of greatest interest.

The ways in which Channel Emulation can be operated are with or without a reference signal, and Stop on SAM (on/off).

With or Without Reference

Using a reference head signal, where available, provides two major benefits. First, the head signal under analysis can be viewed with a head signal that has a reference for improved interpretation of waveform misshapes. The DDA will equalize both reference and head signals. It will also auto-align the two signals when selecting the head signal to be viewed by SAM, even if they were captured at different spindle speeds (up to 1%).

An additional benefit is that the SAM calculation can then assume values less than zero and indicate a data error likely at that point. Without a reference signal, it would be impossible to determine whether an error occurs for a particular sample, only the confidence (SAM) of the Viterbi detector in selecting between a data ‘1’ or data ‘0’ for a particular sample. In this case, the minimum confidence level possible is zero SAM, indicating no confidence in making a selection. The correct data can be specified with a reference, which allows SAM values of less than zero to be detected when the signal quality is so poor that an incorrect data value would be selected by the DDA's Viterbi detector.

Stop On SAM

The second selection mode of operation is Stop on SAM. When this mode is enabled, the DDA is placed in what is essentially a test mode for PRML signal quality based on SAM. When in Normal acquisition mode, the DDA will continuously acquire and analyze head signals until it finds a PRML sample value with a SAM value below a user-specified level. When this occurs, acquisition will be stopped, and the user can directly view the locations below the selected SAM threshold. This mode of operation is particularly useful for capturing intermittent errors.

Analog Compare

Often the head signal locations requiring examination are a header or other non-PRML data section. You may wish to compare these signal sections to a reference signal in order to obtain visual clues to possible problems. However, several issues may have to be addressed before this can be done.

The reference and head signals might differ in time due to spindle speed variations. This could make alignment and comparison of the two signals difficult.

Another issue is that differences may be so subtle that they are not very apparent, especially so if the signals being compared are lengthy.

A third issue is that the problem may be intermittent.

What is generally required for addressing all three issues is an automatic comparison method that adjusts for spindle speed variations and identifies where the two signal differences are the greatest. In the case of intermittent problems, comparisons should be made continuously until a difference greater than a selected threshold is seen. This is just what the Analog Compare feature provides.

With Analog Compare, you select a head signal to act as the reference, and it gets stored in memory. The maximum allowable difference between the two signals is then selected. Analog Compare automatically aligns the two signals and identifies where a mismatch occurs. It also counts each mismatch, storing its byte location for further review. Up to 100 mismatches can be identified and stored in largest-to-smallest order.

As a general-purpose test method, Analog Compare can be applied to finding problems in practically any signal, including other head signals.

General Steps of Analog Compare

1. **Tell the DDA the signals you are providing:** Identify in the Channel Setup the signals that are being provided and their source: the particular input channel or memory. Also identify the section of the head signal you want analyzed.
2. **Set up the channel characteristics:** Each of the Channel Analysis methods has a required set of head signal characteristics that need to be provided in order to perform the necessary analysis. These may include bit cell time, code rate, PRML type, and other characteristics.
3. **Select a channel analysis method:** Specific configuration requirements differ depending on the method used. See “Setting up for Using Drive Channel Analysis” in the Disk Drive Analyzer Reference Manual for more information.
4. **Turn on and configure the method selected:** Once a channel analysis method is selected, it needs to be configured and turned on.
5. **Review the problems the DDA identifies:** Review the head signal areas that the DDA identifies as having the poorest quality or differences from a reference. If the method selected is Channel Emulation with Stop on, SAM on, or Analog Compare, head signal sections that exceed a user-specified threshold will be displayed. If no sections exceed the selected threshold, it may be desirable to adjust the threshold until violations are identified.

Head Signal Filtering

When you are visually analyzing or measuring a servo head signal, it is usually desirable to first filter the head signal to remove noise. In addition, if boost is provided in the servo system, adding the corresponding boost to the displayed or measured head signal may also be beneficial. This allows analysis of the head signal as the servo processing system sees it.

The ability to set values for the –3 dB cutoff, boost and group delay is provided by this Servo Analysis feature. In addition, the filter provided can be easily set up to view or measure any head signal.

Analog Compare

When you are attempting to locate a problem in a wedge signal, for example, the same complications are encountered as with Channel Analysis, with similar solutions.

Noise Analysis

Disk noise parameters enable parameter measurements of media signal-to-noise (msnr), residual (electronics) signal-to-noise (rsnr), and the ratio of media to residual signal-to-noise (m_to_r). The calculation of all three parameters is based on the distribution of the averaged Viterbi input samples.

- **msnr** can be applied to any single-frequency, sector-based data pattern. The single-frequency data will be sampled at the peaks (maxima), zero crossings, and troughs (minima). Any deviations from the ideal sample points will be a result of noise. By performing multiple reads, random noise can be averaged away. With this measurement, the repeating media noise level can be derived by msnr.
- **rsnr** can be applied to any single-frequency, sector-based data pattern. The single-frequency data will be sampled at the peaks (maxima), zero-crossings, and troughs (minima). Any deviations from the ideal sample points will be a result of noise. By performing multiple reads, random noise can be quantified. With this measurement, the non-repeating residual (electronics) noise level can be derived by rsnr.
- **m_to_r** can be applied to any single-frequency, sector-based data pattern. The msnr is compared with the rsnr. The resulting ratio indicates whether the signal is dominated by media noise if it is greater than 1.00, or dominated by residual (electronics) noise if less than 1.00.

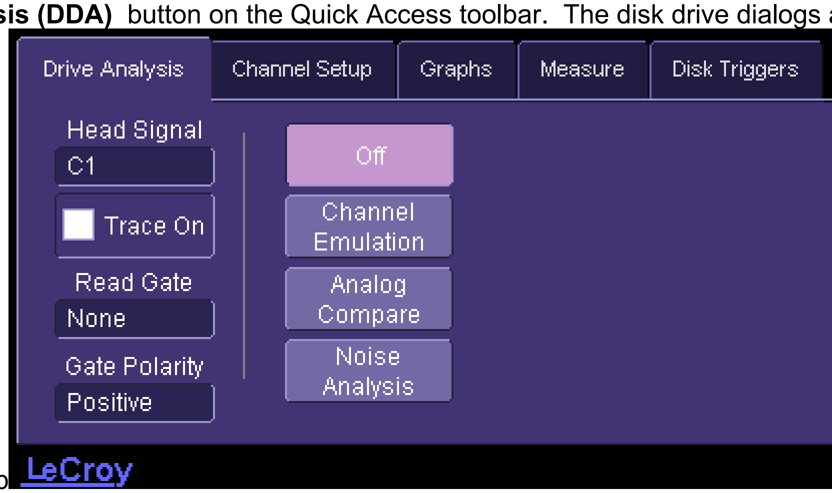
Measure's Drive Parameters

The buttons accessed via the "Measure" tab allow quick and convenient setup of the most common sets of measurements made on a head signal. These include:

- Track Average Amplitude (TAA), PW50
- Amplitude Asymmetry, PW50+, PW50-, Baseline Separation, and TAA
- Auto-Correlation Signal-to-Noise (ACSN)
- Non-Linear Transition Shift (NLTS)
- Jitter (period at level)

Setting Up Channel Emulation

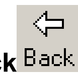
Drive Analysis Setup

1. Press the **Disk Drive Analysis (DDA)** button on the Quick Access toolbar. The disk drive dialogs appear.
2. Touch the **Drive Analysis**tab 

.
3. Touch the **Channel Emulation**button 

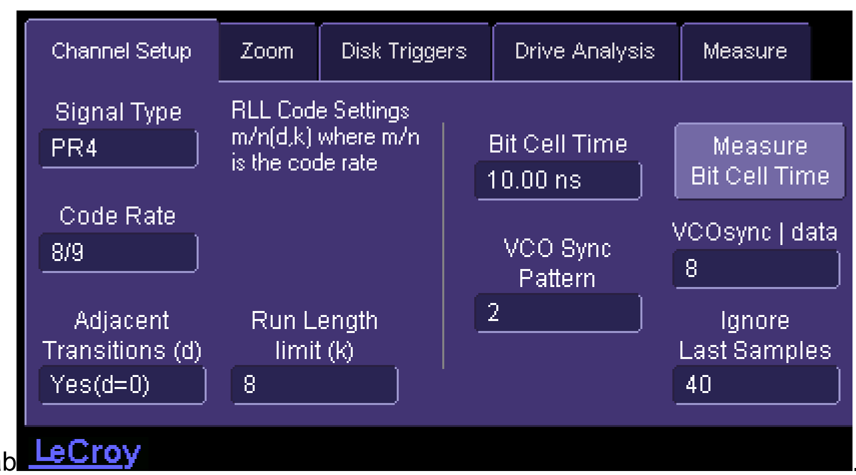
. The Channel Emulation setup checkboxes, buttons, and data entry fields appear.
4. Touch inside the **Head Signal** field and select a signal source from the pop-up menu. The choices comprise channel inputs C1-C4, math traces F1-F8, memories M1-M4, or Reference. See Channel Emulation with Reference and Channel Emulation without Reference for guidelines on using a Reference.
5. Touch the **Trace On** checkbox to turn the Head Signal on.
6. Touch inside the **Read Gate** field and select a source from the pop-up menu. The choices include **Ref** and **none**. Read GateIf the Read Gate signal is connected to a DDA channel and specified, it will be used to determine the regions of the signal to be analyzed. Since the **VCO Synch** field is required for Channel Emulation with Reference and is normally present in the head signal in every block just after Read Gate

goes true, it is recommended that Read Gate be used. If Read Gate is not present, the entire waveform will be used unless the Analyze Region cursors are enabled.

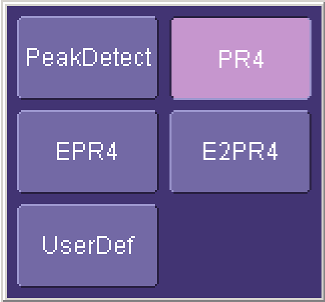
7. If for Read Gate you selected other than **none**, Touch inside the **Gate Polarity** field and select positive or negative polarity.
8. Touch inside the Compare to Reference checkbox to enable comparison of the Head Signal to the Reference. Then touch the **Store Reference** button.
9. Touch the **Stop on SAM** checkbox if you want to stop the acquisition when the signal falls below a user-defined SAM value. Stop on SAMWhen this mode is enabled, the DDA is placed in what is essentially a test mode for PRML signal quality based on SAM. When in Normal acquisition mode, the DDA-5005 will continuously acquire and analyze head signals until it finds a PRML sample value with a SAM value below a user-specified level. When this occurs, acquisition will be stopped, and you can directly view the locations below the selected SAM threshold. This mode of operation is particularly useful for capturing intermittent errors.
10. Touch inside the **SAM Threshold** data entry field and enter a value from 0 to 2 , using the pop-up numeric keypad. See SAM in "PRML Channel Emulation" then touch **Back** 

to complete this setup.
11. Touch the **Show ML Markers** checkbox to enable the markers. ML MarkersThe ML Markers indicate the location of the ideal PRML sample values based on the DDA-5005’s channel emulation.
12. Touch the **Show Level Markers** checkbox to enable the markers. Level MarkersThe Level Markers indicate the vertical position of the PRML levels based on the amplitude of the acquired PRML signal. The Level Markers indicate the vertical position of the PRML levels based on the amplitude of the acquired PRML signal. They reflect the levels at the center of the display.
13. Touch the **Specify Region** checkbox to specify a start and end time, if desired. This may be necessary if you are not using a Read Gate. If you are not using Read Gate, the Analytical Region must start with a preamble for VCO synchronization. Touch inside the Start and End data entry fields and enter starting and ending time values from -1.0 ks to +1.0 ks, using the pop-up numeric keypad.
14. To jump to a position in the head signal, touch inside the "Position" **Segment** field and enter a value from 1 to 999, using the pop-up numeric keypad. Then touch inside the **Byte** field and enter a value from 50 to 50,000.
15. To jump to a "worst SAM" area, after channel emulation, touch inside the **Worst Error #** field and enter a value from 1 to 100, using the pop-up numeric keypad.

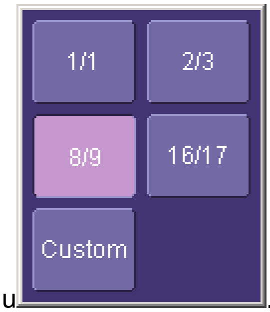
Channel Setup

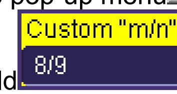
1. Touch the **Channel Setup**tab 

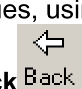
. The Channel Setup buttons and data entry fields appear.

2. Touch inside the **Signal Type** field and make a selection from the pop-up menu 

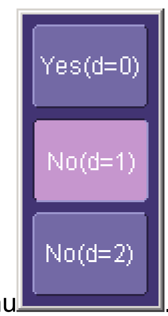
.

3. Touch inside the **Code Rate** field and make a selection from the pop-up menu 

. If you select **Custom**, also touch inside the **Custom m/n**data entry field 

and enter new *m* and *n* values, using the pop-up keyboard. See PRML Encoding in "PRML Channel Emulation" then touch **Back** 

to complete this setup.

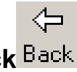
4. Touch inside the **Adjacent Transitions (d)** field and make a selection from the pop-up menu 

. In general, PR4 and EPR4 use d=0; 2/3(1,7) encoded E2PR4 uses d=1.
5. Touch inside the **Run Length Limit (k)** field and enter a value from 0 to 99 using the pop-up numeric keypad.
6. Touch inside the **Bit Cell Time** field and enter a value from 1.00 ns to 1.00 µs using the pop-up numeric keypad. Then press the **Measure Bit Cell Time** button. See Principle of Equalization in "PRML Channel Emulation" then touch **Back** 

to complete this setup.
7. Touch inside the **VCO synch | data** field and enter a value from 1 to 32 using the pop-up numeric keypad.
8. Touch inside the **Ignore Last Samples** field and enter a value from 0 to 999.
9. Touch the **Filter head signal** checkbox to enable filtering of the head signal; then touch the **Train Filter** button to automatically set up equalization on the currently acquired head signal. See Train Filter in "PRML Channel Emulation" then touch **Back** 

to complete this setup. If you do not automatically train the filter, you need to perform steps 10 and 11 to manually set up the equalization filter.
10. Touch inside the **–3 dB Frequency** field and enter a value from 1.0 MHz to 800 MHz using the pop-up numeric keypad. See –3 dB Frequency in "PRML Channel Emulation" then touch **Back** 

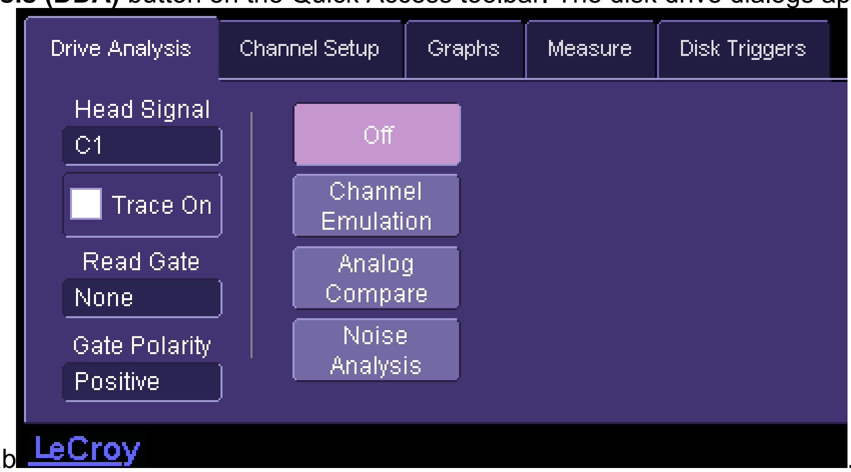
to complete this setup.

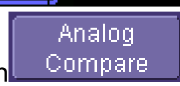
11. Touch inside the **Boost** field and enter a value from 0 to 13 dB using the pop-up numeric keypad. See Boost at fc in "PRML Channel Emulation" then touch **Back** 

to complete this setup.
12. Touch inside the **Group Delay** field and enter a value from –30.0% to +30.0% using the pop-up numeric keypad. See Group Delay in "PRML Channel Emulation." This parameter is not set by the "Train Filter" function.

Setting Up Analog Compare

Drive Analysis Setup

1. Press the **Disk Drive Analysis (DDA)** button on the Quick Access toolbar. The disk drive dialogs appear.
2. Touch the **Drive Analysis**tab 

.
3. Touch the **Analog Compare**button 

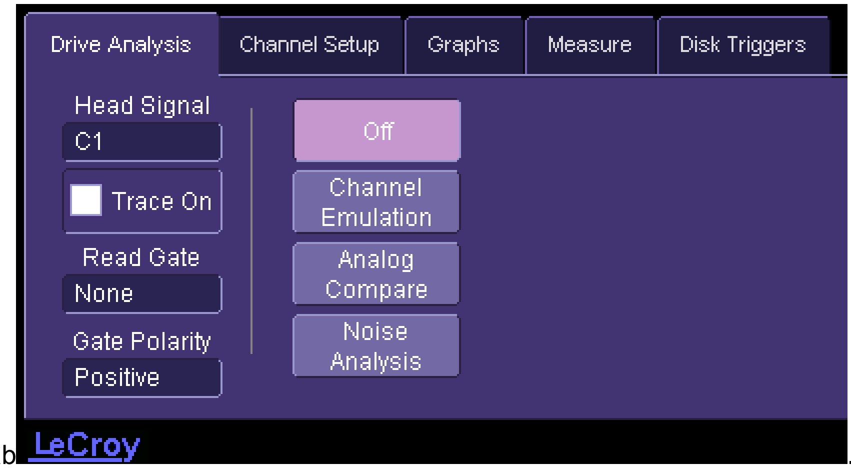
. The Analog Compare setup checkbox, buttons, and data entry fields appear.
4. Touch inside the **Head Signal** field and select a signal source from the pop-up menu. The choices comprise channel inputs C1-C4, math traces F1-F8, memories M1-M4, or Reference. See Using Analog Compare for guidelines on using a Reference.
5. Touch the **Trace On** checkbox to turn the Head Signal on.
6. Touch inside the **Read Gate** field and select a source from the pop-up menu. The choices include **Ref** and **none**. Read GateIf the Read Gate signal is connected to a DDA-5005 channel and specified, it will be used to determine the regions of the signal to be analyzed. If Read Gate is not present, the entire waveform will be used unless the Analyze Region cursors are enabled.
7. If for Read Gate you selected other than **none**, Touch inside the **Gate Polarity** field and select positive or negative polarity.
8. Touch the **Store Head Reference** button.
9. Touch the **Specify Region** checkbox to specify a start and end time, if desired. (This may be necessary if you are not using Read Gate. If you are not using Read Gate, the Analysis Region must start with a preamble for VCO synchronization.) Then touch inside the **Start** and **End** data entry fields and enter starting and ending time values from -1.0 ks to +1.0 ks, using the pop-up numeric keypad.
10. To jump to a position in the head signal, touch inside the "Position" **Segment** field and enter a value from 1 to 999, using the pop-up numeric keypad. Then touch inside the **Byte** field and enter a value from 50 to 50,000.
11. Touch inside the **Worst Error #** field and enter a value from 1 to 100, using the pop-up numeric keypad.
12. Touch inside the **Analog Threshold** field and enter a value from 0 mV to 1 V.

Channel Setup

Channel Setup is required for Analog Compare, which is the same as for Channel Emulation.

Setting Up Noise Analysis

1. The disk drive dialogs appear.



2. Touch the **Drive Analysis** tab.
3. Touch the **Noise Analysis** button. The Noise Analysis setup checkbox, buttons, and data entry field appear.
4. Touch inside the **Head Signal** field and select a signal source from the pop-up menu. The choices comprise channel inputs C1-C4, math traces F1-F8, memories M1-M4, or Reference.
5. Touch the **Trace On** checkbox to turn the Head Signal on.
6. Touch inside the **Read Gate** field and select a source from the pop-up menu. The choices include **Ref** and **none**. Read GateIf the Read Gate signal is connected to a DDA channel and specified, it will be used to determine the regions of the signal to be analyzed. Since the **VCO Synch** field is required for Noise Analysis with Reference and is normally present in the head signal in every block just after Read Gate goes true, it is recommended that Read Gate be used. If Read Gate is not present, the entire waveform will be used unless the Analyze Region cursors are enabled.
7. If for Read Gate you selected other than **none**, Touch inside the **Gate Polarity** field and select positive or negative polarity.
8. Touch the **Setup for Single Frequency** button. This action automatically selects parameters **msnr**, **rsnr**, and **m_to_r** as **Source1** in "Measure" dialog positions P1 to P3, respectively. **ParamPassThru** is the **Measure** selection made for each, which means that the output is the same as the input.
9. Touch the **Avg. Samples** checkbox to enable averaging.
10. Touch inside the **Max Averages** data entry field and enter a value from 1 to 32,000 using the pop-up numeric keypad. Then touch the **Reset Average** button to clear the previous average.

Setting Up disk Triggers

Read Gate

Read Gate triggers on a pulse of a specified minimum width on the Read Gate source. To set it up, proceed as follows:

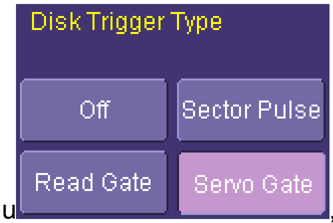
1. Press the **Disk Drive Analysis (DDA)** button on the Quick Access toolbar. The disk drive dialogs appear.
2. Touch the **Disk Triggers** tab.
3. In the "Disk Trigger Types" menu 

, touch **Read Gate**. The Read Gate data entry fields appear. See Read Gate in "Channel Analysis Concepts" then touch **Back** 

to complete this setup
4. Touch inside the **Minimum Width** data entry field and enter a value from 100 ns to 100 µs.
5. Touch inside the Read Gate **Source** field and select an input channel or external source from the pop-up menu.
6. Touch inside the Read Gate **Polarity** field and select positive or negative polarity from the pop-up menu.
7. Touch inside the Read Gate **Level** field and enter a value, using the pop-up numeric keypad.

Sector Pulse

Sector Pulse triggers on the specified (n^{th}) sector start pulse after the Index mark. To set it up, proceed as follows:

1. Press the **Disk Drive Analysis (DDA)** button on the Quick Access toolbar. The disk drive dialogs appear.
2. Touch the **Disk Triggers** tab.
3. In the "Disk Trigger Types" menu 

, touch **Sector Pulse**. The Sector Pulse data entry fields appear. See Sector Pulse in "Setting Up to Use Drive Channel Analysis then touch **Back** 

to complete this setup.
4. Touch inside the **Trigger On Sector x after Index** data entry field and enter a value from 0 to 999 using the pop-up numeric keypad.
5. Touch inside the Sector Pulse **Source** field and select an input channel or external source from the pop-up menu.
6. Touch inside the Sector Pulse **Polarity** field and select positive or negative polarity from the pop-up menu.
7. Touch inside the Sector Pulse **Level** field and enter a value from -250 mV to 250 mV using the pop-up numeric keypad.
8. Touch inside the Index **Source** field and select an input channel or external source from the pop-up menu.
9. Touch inside the Index **Level** field and enter a value from -250 mV to 250 mV using the pop-up numeric keypad.

Servo Gate

Servo Gate triggers on all or selected Servo Gate pulses starting after the Index mark. To set it up, proceed as follows:

1. Press the **Disk Drive Analysis (DDA)** button on the Quick Access toolbar. The disk drive dialogs appear.
2. Touch the **Disk Triggers** tab.
3. In the "Disk Trigger Types" menu 

, touch **Servo Gate**. The Servo Gate data entry fields appear.
4. Touch inside the **After index wait x servo gates** field and enter a value from 0 to 999 using the pop-up numeric keypad.
5. Touch inside the **then skip x gates between triggers** field and enter a value from 0 to 999 using the pop-up numeric keypad.
6. Touch inside the Servo Gate **Source** field and select an input channel or external source from the pop-up menu.
7. Touch inside the Servo Gate **Polarity** field and select positive or negative polarity from the pop-up menu.
8. Touch inside the Servo Gate **Level** field and enter a value from -250 mV to 250 mV using the pop-up numeric keypad.
9. Touch inside the Index **Source** field and select an input channel or external source from the pop-up menu.
10. Touch inside the Index **Level** field and enter a value from -250 mV to 250 mV using the pop-up numeric keypad.

Setting Up Zoom

You can zoom on a particular segment and byte of your signal as follows.

1. Press the **Disk Drive Analysis (DDA)** button on the Quick Access toolbar. The disk drive dialogs appear.
2. Touch the **Zoom** tab.

3. Touch inside the Position **Segment** data entry field and enter a value from 1 to 999 using the pop-up numeric keypad.
4. Touch inside the Position **Byte** field and enter a value from -50 to 50,000 using the pop-up numeric keypad.
5. If you want to increase or decrease your horizontal or vertical zoom in small increments, touch the **Var.** checkbox in the right-hand dialog to enable variable zooming. Now with each touch of the zoom control buttons  , the degree of magnification will change by a small increment.

   To zoom in or out in large standard increments with each touch of the zoom control buttons, leave the **Var.** checkbox unchecked.

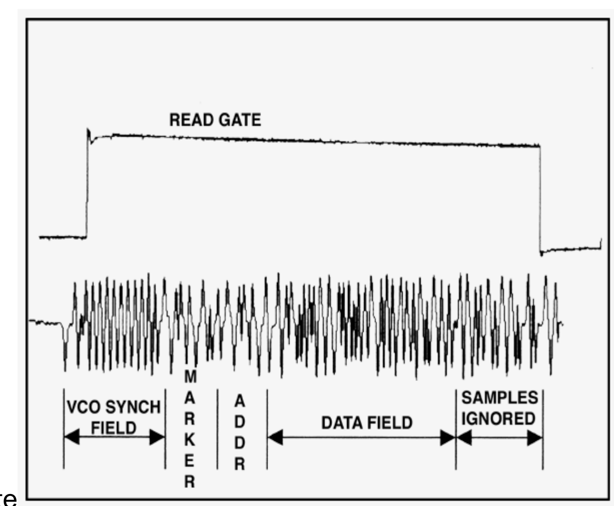
   To set exact horizontal or vertical zoom factors, touch inside the Horizontal **Scale/div** data entry field and enter a time-per-div value, using the pop-up numeric keypad. Then touch inside the Vertical **Scale/div** field and enter a voltage value.
6. To reset the zoom to x1 magnification, touch **Reset Zoom** in the dialog or press the front panel zoom button .

Channel Analysis Concepts

The purpose of Channel Analysis is to find signal quality problems in the head signal. Thus, both methods used by the DDA (Channel Emulation and Analog Compare) require that you first specify the DDA channel or memory on which the head signal is to be carried.

The instrument utilizes the head signal and, optionally, the Read Gate and the Analyze Region cursors to determine which part of the head signal will be analyzed. What follows is an explanation of how the instrument does this and the manner in which it processes the head signal.

The following figure shows the fields of the head signal and their relationship to Read Gate.



Head Signal Fields and Read Gate

In a normally operating disk drive, every time Read Gate goes true (at the beginning of every segment to be read) there must be a repetitive signal called the VCO Synch (voltage-controlled oscillator). It is needed to adjust the phase of the PLL (phase locked loop), which generates the sampling clock, and to adjust the AGC (automatic gain control).

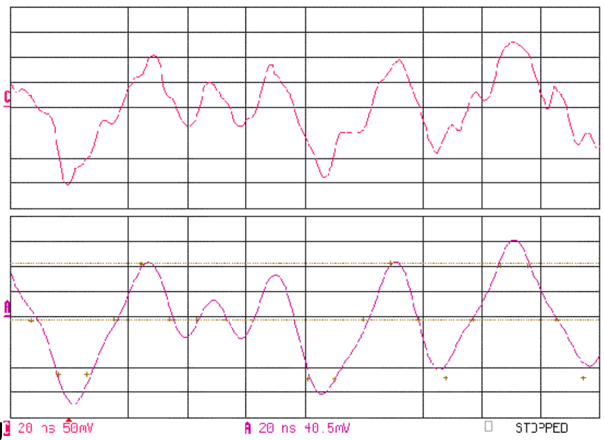
The DDA will try to identify the VCO Synch signal in a similar fashion in order to perform channel emulation. In order to identify VCO Synch, you will need to provide a Read Gate signal or to set the Analyze Region cursors (see Selecting the Waveform Section to be Analyzed). After the DDA analyzes the VCO Synch signal, it is ready to analyze the data field for problems. Since the data field generally does not start immediately after the end of the VCO Synch field, the DDA needs to know where the data field starts. You must specify the number of bytes between the VCO Synch field and the data field. The DDA can then determine the location of the first byte of data.

If the channel analysis method selected is **Channel Emulation**, the DDA will perform PRML channel emulation to determine the location of problems in the data field. Often, there is a delay between the end of valid data and the disabling of the Read Gate. It is generally not meaningful to analyze this area of a sector for problems. In order to avoid analyzing this area, you can specify the number of PRML samples between the end of valid data and Read Gate being disabled in the **Ignore Last Samples** data entry field in the "Channel Setup" dialog.

Using the DDA's Equalization Filter

Disk drives generally have filters to remove noise and to shape the head signal. Without these filters, data would not be properly recovered. The DDA provides a similar filter capability. The equalization filter available in the instrument’s channel emulation can be used with all the channel analysis methods to clean up and shape the signal, much like the channel chip will do.

It is recommended that unless the head signal has been equalized before being acquired by the DDA, the filter should be applied. Otherwise false problems may be reported. If you have access to the equalized signal from the drive, this signal can be provided to the DDA. The filter should not be used in this case. The following figure shows the same signal before and after using the DDA’s filter.



Before and after filtering

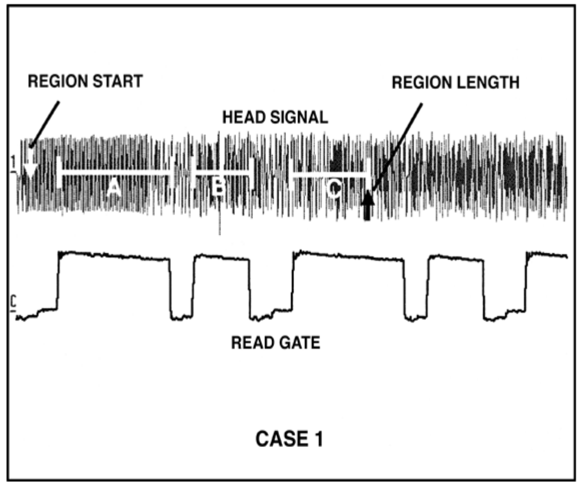
As you can see, it would be very difficult to perform meaningful analysis of the "before" picture, because of noise. Note that Maximum Likelihood (ML) Markers ("+" signs) and Level Markers (horizontal lines) are displayed in the "after" picture. The ML Markers indicate the location of the ideal PRML sample values based on the DDA’s channel emulation. The Level Markers indicate the vertical position of the PRML levels based on the amplitude of the acquired PRML signal.

Selecting the Waveform Section to Be Analyzed

For the Analog Compare and Channel Emulation channel analysis methods, you can specify selective areas of the head signal for analysis by using the Analyze Region cursors or the drive’s Read Gate signal, or both.

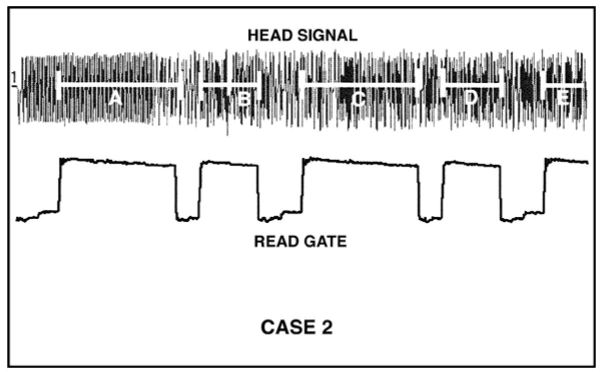
As discussed above, it is important for the DDA to be able to analyze the VCO Synch signal at the beginning of the region it is analyzing. If Read Gate is used to determine which head signal sections to analyze, this will not be a problem, since the DDA will use Read Gate to identify the location of VCO Synch. If Read Gate is not used, it is important, by means of the Analyze Region cursors, that the beginning of the head signal section analyzed be very near the beginning of VCO Synch.

The following figures show the areas of the waveform to be analyzed, with the different combinations of the Analyze Region cursors or Read Gate, or both, enabled. The Analyze Region cursors, if enabled, generally specify the outer-most boundaries of the data to be analyzed. They are particularly useful if Read Gate is not available or if only a subset of several Read Gate true sections captured is to be analyzed; for example, a single sector within several captured, or excluding the ID field. Within the area designated by the markers, if Read Gate is specified, only the regions of the Head Signal during which Read Gate is true are analyzed (as shown in Case 1) while all others are ignored.



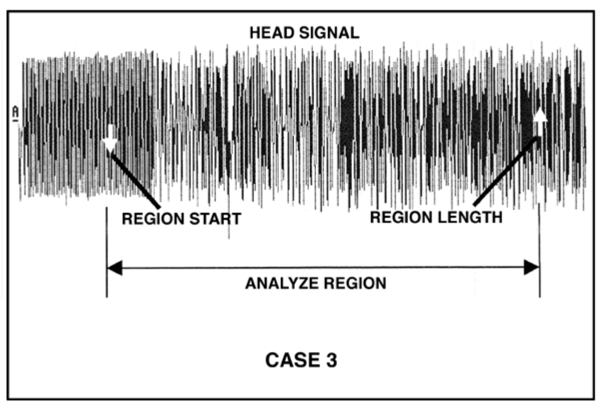
Case 1: Only Read Gate True regions are analyzed

Similarly, if Analyze Region is disabled and Read Gate enabled, the area of the Head Signal during which Read Gate is true is analyzed, as shown in sections A, B, C, D, and E in Case 2



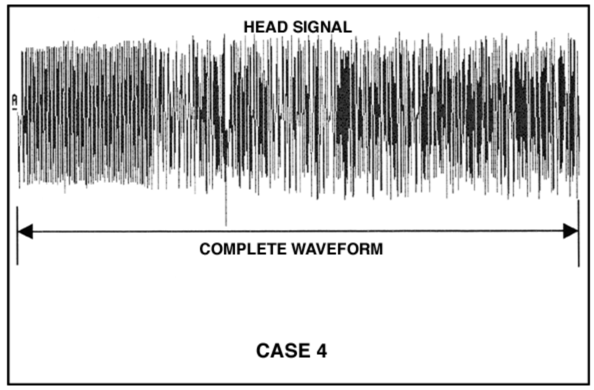
Case 2: Head Signal Read Gate True regions analyzed

If Analyze Region is enabled and Read Gate is not available, only the area of the Head Signal within the region is analyzed, as shown in Case 3.



Case 3: Only the area within the Region is analyzed

If neither the markers nor Read Gate are enabled, the entire Head Signal is analyzed (Case 4).



Case 4: The entire head signal is analyzed

DDA Markers

If Read Clock is acquired by the DDA, the "sample time" markers can be drawn. Because the phase relation of RCLK to the head signal is not known, it is adjustable. It is worth noting here that the ML markers are at positions determined by the instrument's channel emulation; the sample time markers are at positions determined by the channel's sample clock signal with ML markers ("+" signs) and level markers (horizontal lines).

Setting Up to Use Drive Channel Analysis

There are several basic considerations when you are preparing to use Drive Channel Analysis. Which signals are to be provided to the DDA? What waveform section or sections will be analyzed? What will be used as the reference waveform? What trigger and time/div settings should be used?

Which Signals to Provide

The following is an overview of the signals that can be used by the DDA.

- Head Signal: The drive head signal is the only signal the DDA requires. The drive head signal can either be the signal after the pre-amplifier, or the one after the drive filter if available.
- Read Gate: Read Gate is "optional" for all error find methods; however, it is strongly recommended that you use Read Gate.

Choosing the Waveform Section to Be Analyzed

Before using Drive Channel Analysis, you should determine which part of the head signal will be analyzed.

The options you have are to analyze:

- Part of a single Read Gate cycle
- A single Read Gate cycle
- Multiple Read Gate cycles.

There are additional considerations for each of these choices.

If you choose to analyze a part of a single Read Gate cycle, it is important that it include the VCO Synch signal. Because it will be difficult to set up a reference signal if it is less than a single Read Gate cycle, it is recommended that this not be done with either the Analog Compare or Channel Emulation methods.

If you want to analyze a single Read Gate cycle, the easiest approach is to use the Read Gate signal and, if necessary, the Analyze Region cursors, to determine which section is analyzed. If you are using Channel Emulation or Analog Compare, you must make sure that the reference signal is the same length (see the following Selecting the Reference Waveform).

If you wish to analyze multiple Read Gate cycles, the setup is the same as for a single Read Gate cycle. The Read Gate signal and, if necessary, the Analyze Region cursors are used to determine which sections are analyzed. If you are using Channel Emulation or Analog Compare, you need to ensure that the reference signal is of similar makeup.

Except for Analog Compare, there are limits as to how much data the channel analysis methods can analyze at the same time:

- Channel Emulation **without** a reference saves ML sample results for up to 20,000 bits, which are used to draw the ML sample (+) markers.

- Channel Emulation **with** a reference saves up to 20,000 bits as its reference, while also saving ML sample results; there can be up to 250 separate Read Gate true sections in the data analyzed by Channel Emulation with reference.
- Analog Compare can handle one block of data of any length up to the maximum that can be acquired and stored by the DDA. It is the most computationally intensive error find method, so its use on maximum length waveforms will not be typical. Another indirect constraint on waveform size arises if Read Gate is used. If the source of Read Gate is set to something other than NONE, Read Gate will break the data into multiple blocks. In that case, there is a requirement that the start of each block, relative to the first block in the reference and the acquisition, be the same within the DDA's realignment window of about 12 bit cells.

Selecting the Reference Waveform

The Analog Compare and Channel Emulation with reference signal channel analysis methods compare the captured head signal to a stored reference head signal. For these methods, the appropriate menus will only appear after a head signal has been selected.

The stored reference should be acquired in the same manner (same time/div, sample rate, trigger position, etc.) and with the same timing as the acquisitions to which it is to be compared. It is not possible to compare a reference having only one sector to every sector in an acquisition containing many; and a split sector cannot be compared with an unsplit or differently split sector.

Generally, it is best that the reference signal be from the same drive section as the one with which it is to be compared. This will ensure that any splits that occur are in the same location. Another reason for selecting the same drive section is to avoid scrambler problems. If a drive uses a scrambler, two sections with the same logic data may have different head signals. Since Channel Emulation with reference and Analog Compare both analyze the head signal, they may see differences between the reference and acquired signals due to the presence of the scrambler and not because real problems exist. If a scrambler is not present, it is reasonable to use other drive sections from the same zone as a reference.

The head signal can be stored as a reference simply in order to have a stored trace automatically overlapped with later acquisitions. This can be done even if the selected channel analysis method does not require a reference, or if you do not intend to use the method. This might be done just to scroll through waveforms using Auto Scroll to see how well they match.

Changing the source of the Head Signal or Read Gate invalidates the stored reference. You must store another valid reference for the new setup.

Time/Div Settings

The DDA allows you to keep acquiring a head signal until it finds a problem, at which point it will stop the acquisition so that errors can be analyzed. In order for Drive Channel Analysis to operate properly the DDA trigger must be set up so that the correct waveform section or sections are reliably acquired. The trigger setting must be selected to obtain the desired data on the display, in approximately the same position, in each acquisition.

In addition, the time/div must be set properly. Two factors should be taken into account when setting this. First, the time/div should be set to a sufficient duration so that the head signal sections that will be analyzed are captured. Second, in order for the DDA to correctly analyze a head signal it should have at least five ADC samples per bit cell.

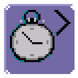
For drives without ID, Read Gate only goes true to read the data of interest. If you can manually initiate a read operation of the desired data, this is easy: trigger on the leading edge of Read Gate, and every time the data is read, the DDA will trigger. Wait for the channel analysis to be done (the instrument rearms if no problems are found) before performing another read.

A precaution should be taken if more than a single Read Gate block of data is to be analyzed. For this situation, if reads are being performed repeatedly it is important to ensure that the trigger mode selected is not one that triggers solely on the occurrence of any Read Gate. Otherwise, the Read Gate block triggered on might sometimes be the second or later Read Gate true section.

If the drive provides Sector Pulses at the beginning of each sector it is recommended that the DDA's Sector Pulse trigger be used to avoid this problem.

Otherwise, a SMART Trigger can easily solve this problem, by triggering on the first (or nth) Read Gate edge after Index.

If Read Gate is not available or its use is not desired, but looking at the head signal is, try triggering on Index and using a trigger delay to move out to the region of interest. Use the Analyze Region markers to delimit the area to analyze: for channel emulation methods the start marker should always be near the beginning of the VCO Synch field. However, even 0.1% speed variance becomes 0.8 µs of jitter after 8 ms of delay. That is probably several hundred bit cells (far more than Analog Compare can compensate for) which may cause the channel emulation methods to miss the VCO Synch field entirely. Therefore this technique will not be able to look at sectors more than 1 or 2 ms delayed from Index.

If Read Gate is unavailable and you want to look at something other than sectors of data on the head signal, some repetitive signal such as Servo/Data can be used as Read Gate. Analog Compare will have to be used then. The SMART Trigger can be set up as outlined previously, except that the bottom **Qualify by:** field should be set to time greater than , and the amount of time from index to just before the data of interest should be entered. This can be used to look at a particular servo burst, for example.

Automatically Shown Traces

When the source of the head signal is changed from **None** to an input channel or memory, a trace showing a zoom of the head signal is automatically created and displayed. If the **Filter head signal** checkbox is checked, the data in the trace is filtered. All channel markers selected will appear on this trace. All channel analysis methods analyze the (possibly filtered) data in the trace waveform (the entire waveform, not just the part displayed). **Byte** offset and **Worst Error #** change the horizontal position of this trace.

When **Store Reference** is selected, a trace showing a zoom of the stored reference is automatically established where possible. This means that whenever **Byte** offset or **Worst Error #** is changed, the reference will be realigned with the Head Signal at the new position. The reference signal will also be filtered if the **Filter head signal** checkbox is checked.

Setting Bit Cell Time

The value in the bit cell time menu is used as a starting guess for analyzing VCO Synch. It is also used by Byte offset and Analog Compare to determine how much time a byte occupies, and to set the -3 dB frequency of the DDA's equalizing filter (if enabled). It should be set correctly for the zone being read; it does not need to be set on each read within the same zone.

Changing the bit cell time changes the filter's -3 dB frequency. The filter's settings should therefore be updated, or "trained" after the bit cell time has been changed. If the filter is on, the bit cell time must be fairly close for **Find Bit Cell Time** to work.

Retraining the Filter

Training the filter adjusts the boost and the -3dB frequency to optimize the mean of the worst 100 SAM values. The training starts by setting the -3dB frequency to 0.5/bit cell time. It then alternates optimizing the boost with optimizing -3 dB frequency. The 3 dB, boost and group delay values can also be independently set without having the DDA automatically train.

Choosing an Analysis Method

Which method or methods should be used? Deciding this depends on the type of problem and the information available.

By definition, a data error occurs in a drive when a bit is interpreted as a "1" when it should be a "0" or vice-versa. In order to gain insight into the cause of the error, it is generally desirable to examine the head signal at the location or locations where data errors occur. The DDA provides tools to help identify the likely location of a data error. In addition, with the instrument's channel emulation capability you are able to gain further insight into the cause of an error or errors once location is determined.

Before choosing a method, there are some basic questions that need to be answered:

- Is the byte position of the error known?
- Is a reference signal easily available?
- Is the error repetitive or intermittent?

Recommendations on the appropriateness of each analysis method are provided below.

Analog Compare

Analog Compare is the most general of the channel analysis methods, since it can be applied to all parts of the head signal, including VCO Synch, servo burst and data field. It is recommended for identifying the location of errors for peak detect signals. It can also be used for PRML signals, although Channel Emulation with a reference signal is the recommended method if a reference signal is available. See Using Analog Compare for a full description of this method.

Channel Emulation without Reference

Since it does not require a reference, Channel Emulation without a reference signal is the easiest of the channel analysis methods to use for PRML signals. This method performs channel emulation to determine the locations in the head signal that gave the Viterbi detector the most trouble in its decision as to the maximum likelihood sequence of samples. This means the state at a particular time, in the final surviving sequence, survived by the smallest margin. (The "state" corresponds to deciding on a "1" or "0" bit at each position.) The margin between keeping and rejecting what turned out to be the final choice for the state at that time is called the Sequenced Amplitude Margin (SAM). Unlike the compare methods, Channel Emulation without reference is based only on analyzing the quality of the head signal. It is generally assumed that the quality of the head signal is strongly correlated to the bit error or errors in nonreturn to zero (NRZ) data. Head signal sections that produce incorrect NRZ data are much more likely to be marginal when analyzed by the Viterbi detector. The most marginal sections are the first flagged by Channel Emulation as having the worst SAM. See Channel Emulation Without Reference for a full description of this method.

Sections with no signal transitions are one exception to this rule. These may not be identified as problematic by the Viterbi detector. However, if the Limit Run Length feature is enabled, sections with no transitions will be identified as errors if they exceed the run length limit.

Because it is the easiest of all methods, Channel Emulation without reference is generally the best to use first.

Channel Emulation with Reference

Channel Emulation with a reference is very useful if the error being looked for occurs at least fairly frequently and a reference head signal is available. If the error is not "hard," that is, it does not occur all the time, any acquisition may be stored as the reference; Channel Emulation with reference will catch acquisitions for which the DDA's channel emulation detects a different bit sequence. If the error is hard, a separate reference must be available. To have no errors detected, the reference must have the same bit pattern as the acquisitions with which it will be compared. This means each sector of the reference must be either unsplit or split in the exact same place as the matching sectors in the acquisitions with which it will be compared.

Channel Emulation without reference must assume that the final surviving sequence of bits is correct and only points out poor quality in the head signal. But Channel Emulation with reference has a reference "correct" path and will catch differences even if both the reference and the comparison acquisition are high quality PRML signals. For Channel Emulation with a reference, the SAM values can be negative, indicating that a different decision was made (less than 0 margin to the "correct" state). The threshold below which an "error" is flagged is adjustable down to -1, to permit you to flag places where the DDA's channel emulation determines a different bit value for the head signal as compared to the reference head signal. See Channel Emulation With Reference for a full description of this method.

Channel Emulation without Reference

Channel Emulation without Reference finds a single trace and predicts the bits where the Sequenced Amplitude Margin (SAM) -- the distance or margin the Viterbi detector has for making a decision -- is poorest. It uses a full disk drive channel emulation to indicate how the signal ought to appear when a good reference signal is not available. It measures the SAM of all the samples (PRML clock locations).

The software emulates a PRML channel and ranks errors by SAM value. A distance or SAM value of "0" indicates no margin for a decision and the detector's lack of certainty as to whether the digital bit should be "1" or "0." The positions of the 100 worst margins are identified and can be displayed along with the SAM value of each.

Using complete disk drive channel emulation, Channel Emulation without reference predicts where the head signal quality is the poorest in respect of a PRML channel's ability to confidently select a "1" or "0" value.

Channel Emulation without reference starts by finding the beginning of the sector. The algorithm looks at the head signal beginning at the Read Gate true transition (or analyze region start if Read Gate is not available) and tries to synchronize to the VCO Synch pattern in order to establish sampling phase and expected sample levels. To accomplish this, it is required that **VCO Synch Pattern** be set correctly, and that the "Bit Cell Time" be approximately correct. The data is then passed through the emulated channel where it is appropriately sampled. The sampled output enters the Viterbi detector, which chooses the "sequence" of bits (history) that is the most likely when the new bit due to this sample is appended. The difference between the mean squared distance (msd) of the selected sequence and the other possible sequence leading to the selected state (SAM) is then calculated. For each sample in the Viterbi detector, Channel Emulation without reference determines a SAM indicating the confidence it has in making a decision between the two most likely sequences.

If **Run Length Limit** has a non-zero value, run length limit will also be detected and violations reported.

If Read Gate is present, it does not necessarily go false immediately after the last byte of valid information, usually error-correcting code (ECC). The delay is due to the propagation time through the channel chip and any delays from the controller. Therefore, you can specify the amount of 'garbage' data to ignore after the end of the written data and before Read Gate goes false. Errors detected in this area will be ignored.

When Channel Emulation without reference runs, the ML markers ("+" signs) are automatically displayed. These show the Maximum Likelihood sample sequence that the channel emulation chose, based on the signal and the possible sequences. They are drawn at the expected level at the time the channel sampled the head signal. If the "+" signs are all very close to the waveform, the signal is good. If the "+" signs are not close to the waveform, the signal may not be good. The ML markers do not appear if more than 500 are needed on the display. If they are on but not visible, zoom in on the head signal.

Additionally, you can turn on Level Markers, horizontal lines that show the expected levels at the center of the screen.

The ML markers do not appear if more than 50 are needed on the display; if they are on but not visible, they can be zoomed.

The Channel Emulation without reference method can be used either on newly acquired channel data or data previously saved in a memory. The following information is required:

- **Head Signal:** The VCO Synchronization field is needed at the beginning of the area to be analyzed. If the VCO Synch field is not found, Channel Emulation without reference will not be able to analyze the signal.
- **Signal Type:** Specifies the type of PRML channel.
- **VCO Synch Pattern:** In a normally operating disk drive, every time Read Gate goes true (at the beginning of every segment to be read) there must be a repetitive signal called the VCO Synch. It is required for adjustment of the phase of the PLL (phase locked loop), which generates the sampling clock, as well as adjustment of the AGC (automatic gain control). Most commonly, the VCO Synch is 2T (a transition every other bit cell).
- **Bit Cell Time** is used as a starting estimate of the VCO Synch signal. If the value is not known and the VCO Synch field is at the beginning of the area to be analyzed, touching the **Measure Bit Cell Time** button will determine it automatically.
- **Ignore Last <n> Samples:** Specifies the amount of 'garbage' data to ignore after the end of the written data and before Read Gate goes false.
- **SAM Threshold:** This is the user-specified threshold used by Channel Emulation without Reference to decide when to record an error. SAM is the difference in mean squared distance between the samples observed from the sequence of samples decided upon as the likely sequence and the one discarded at that decision point. See Note 1 below.
- **Run Length Limit (k):** If RLL encoding is set to zero, no check is made and Channel Emulation without Reference only reports errors due to difficulty deciding on a sequence of samples (small SAM). If set to a non-zero value, it specifies the number of permitted non-transitions in a row. See Note 2 below.
- **Adjacent Transitions**: Corresponds to the "d" in RLL code specified as m/n(d,k). See Note 3 below.

Additionally, these parameters are useful (though not essential) for Channel Emulation without reference:

- **Read Gate:** If the Read Gate signal is connected to a DDA channel and specified, it will be used to determine the regions of the signal to be analyzed. Since the VCO Synch field is required for Channel Emulation, and it is normally present in the head signal in every block just after Read Gate goes true, it is recommended that Read Gate be used. If Read Gate is not present, the entire waveform will be used unless the Analyze Region cursors are enabled.
- **Read Gate Polarity:** Typically, Read Gate is a positive true signal. However, there are drives that use a negative true signal for Read Gate. If Read Gate is enabled, this setting allows you to select whether "True" is high or low.
- **Specify Region:** As previously described, markers can be used to define a subset of the head signal for analysis.
- **Filter Head Signal:** If enabled, an equalization filter is applied to the head signal before processing and display. This feature should be used if the DDA does not have access to the drive's equalized head signal.

Notes on Using Channel Emulation without Reference

1. Sequenced Amplitude Margin (SAM) is used by the Viterbi detector to decide whether a bit should be "1" or "0". As the Viterbi detector receives each sample, the detector must choose between two possible sequences, which lead to the state it selects. The sequence with the best fit determines which state is selected. SAM indicates how much better the surviving sequence is than the one that is discarded from further consideration. "0" means the Viterbi Detector has no preference at all between the sequence it chose and one it threw away at a particular sample. The most positive value is the square of the "minimum error distance" for the selected Signal Type, and implies excellent certainty about which sequence to keep. SAM values are associated with the sample leaving the Viterbi detector's trellis, that is, the sample about which the final decision is being made. The SAM includes the effect of all previous samples in the block and of the following samples that are already in the trellis.

   Near-zero SAM values are due to distortions in the shape of the head signal. Problems causing distortions, which Channel Emulation without Reference can find, include media defects and dropouts, asperities, noise bursts, or a bad head. (Noise bursts are sometimes obscured by the equalization filter, and result in a distorted shape of a pulse. To see if this is the cause of a distorted shape, look at the unfiltered waveform also.) Without a reference Channel Emulation without Reference does not know if the decisions it makes actually correspond to what was written, it only knows how good the head signal looks when the disk is read. As an example, suppose a disk occasionally simply fails to write several transitions, resulting in a flat area in the head signal. All the pulses are the correct shape, it's just that some aren't there at all. This can still be caught if it causes a Run Length Limit violation and that check was enabled, but it might not be a problem for the Viterbi detector.
2. When checking for run length limit (RLL) violations, "k" of the RLL code "m/n(d,k)" must be specified: "k" is the maximum number of non-transitions between transitions. When **Run Length Limit** is set to a non-zero value, Channel Emulation without Reference also counts the number of non-transitions in a row and checks against the user specified limit. Examples: we recommend that k be set to 8 to handle 8/9(0,4,4) codes; set k to 7 for 2/3(1,7) modulation code.
3. **d** is the minimum number of non-transitions between transitions. When d=0, adjacent transitions are allowed. When d=1, adjacent transitions are not permitted. For E2PR4 it can also be set to d=2. A non-zero **d** eliminates several possible sequence choices (states and transitions in the Viterbi detector). If set to **1** when it should be **0**, Channel Emulation without Reference will almost certainly detect a large number of "errors." If set to **0** when it should be **1**, it is possible that Channel Emulation without Reference will pass a few bits that should have been flagged as errors.

Channel Emulation with Reference

Whereas Channel Emulation without Reference acquires a single trace and makes a prediction of problems based on signal quality, when the method is used with a reference signal, it calculates the Viterbi output of the reference, as well as that of the acquired trace, and compares the two to find mismatches.

Channel Emulation with Reference starts by finding the beginning of the sector. The algorithm looks at the head signal beginning at the Read Gate true transition (or analyze region start if Read Gate is not available) and tries to synchronize to the VCO Synch pattern in order to establish sampling phase and expected sample levels. To accomplish this, Channel Emulation with reference requires that **VCO Synch Pattern** be set correctly, and that the **Bit Cell Time** be approximately correct. Because the position of Read Gate relative to the head signal can jitter from read to read, the method aligns the two bit streams it is comparing based on the end of the repetitive VCO Synch pattern. If there is an error within the VCO Synch pattern, this will be found, but the method may also find many other mismatches if incorrectly aligned.

The data is then passed through the emulated channel where it is appropriately sampled. The sampled output enters the Viterbi detector, which chooses the "sequence" of bits (history) that is the most likely when the new bit due to this sample is appended.

SAM is the margin between keeping and rejecting the correct state in the sequence, at any point. Channel Emulation **without** reference essentially assumes that the output of the Viterbi detector is "correct," and gives the margin between that and a different decision at each bit position. It must do this because it does not have a reference. Therefore SAM results are always greater than zero from Channel Emulation without reference; they are the margin from the final decision to some other decision.

Channel Emulation **with** reference performs channel emulation on the reference signal when it is stored and saves the resulting bit sequence. When "Emulation" is turned on it performs channel emulation on each subsequent acquisition and looks at the margin for the states in the final surviving state of the reference bit sequence. As long as the bit sequence for the current acquisition and the reference acquisition match, the margin is > 0. When a different decision is actually made the margin reported is less than zero: there was less then zero margin to making a different decision; a different decision was actually made. Even if both signals are of excellent quality, but different, Channel Emulation with a Reference signal will catch it.

This divergence of paths through the detector may be referred to as an "error event." After a very few bits, typically, the paths converge again. The SAM reported for the bit where the paths converge will also be negative. This is because the reference and acquisition disagree on the previous bit, and the state of the detector reflects the current bit and at least one previous bit (for PR4).

Note: Unless a drive is in a diagnostic mode (direct write), data is scrambled and encoded when written. In the drive, after the Viterbi detector finds the bit pattern that was written, the bit pattern is decoded and de-scrambled to recover the user data. The DDA does not do the decoding or de-scrambling; it does not need to recover the user data in order to find problems in the quality of the head signal or differences from a reference. This means, however, that each bit in error in the head signal may turn into multiple bits in error in the user data (for example, if it resulted in a different valid 9 bit pattern in 8/9 encoding).

In addition, a normally operating drive tries to correct short bursts of errors in the user data using an Error Correcting Code syndrome, so an error in the head signal may result in no bits after Error Correction if it was within the capability of the ECC to correct it. Usually the bit error rate is measured without Error Correction to check drive operation. The ECC is only capable of correcting a limited number of errors in a sector, so keeping the number of raw errors low is a requirement.

The SAM values shown by Channel Emulation with a reference signal directly reflect where the DDA’s channel emulation believes an error (without Error Correction) would occur. The SAM threshold determines the value of SAM below which a bit position is recorded on the Channel Emulation’s "error" list. For Channel Emulation with a reference, you can set the SAM down to -1. This allows the use of either inclusion or exclusion of positions with close to zero SAM, which might or might not actually be an error. If the threshold is set near -1, only places where the decision to reject the reference bit sequence was made with considerable certainty will appear in the DDA error list. If the threshold is greater than zero, all the places where Channel Emulation calculates the bit sequences will differ, and some places close to making an error (small positive margin) will be included in the error list.

If Read Gate is present, it does not necessarily go false immediately after the last byte of valid information, usually error-correcting code (ECC). The delay is due to the propagation time through the channel chip and any delays from the controller. Therefore, you can specify the amount of "garbage" data to ignore after the end of the written data and before Read Gate goes false. Errors detected in this area will be ignored.

When Channel Emulation with a reference signal runs, the ML markers ("+" signs) are automatically displayed. These show the Maximum Likelihood sample sequence that the DDA’s channel emulation chose, based on the head signal and the possible sequences. They are drawn at the expected level at the time the channel sampled the head signal. Since Channel with reference requires a stored reference signal, the reference trace will be overlapped with the head signal.

The ML markers do not appear if more than 500 are needed on the display; if they are on but not visible, zoom in on the head signal.

Channel Emulation with reference uses the same channel emulation as Channel Emulation without reference signal.

Channel Emulation with reference requires the following setup, including a stored reference signal:

- **Head Signal:** The VCO Synchronization field is needed at the beginning of the area to be analyzed. If the VCO Synch field is not found, Channel Emulation with reference will not be able to analyze the signal.
- **Reference:** A reference waveform must be acquired and processed by selecting "STORE REFERENCE"; subsequent acquisitions will be processed and then compared to this.
- **Signal Type:** Specifies the type of PRML channel.
- **Adjacent Transitions (d):** Corresponds to the "d" in RLL code specified as m/n(d,k); "d" is the minimum number of non-transitions between transitions. If d=0 then adjacent transitions are allowed. If d=1, then adjacent transitions are not permitted. For E2PR4 this field can also be set to d=2.
- **VCO Synch Pattern:** In a normally operating disk drive, every time Read Gate goes true (at the beginning of every segment to be read) there must be a repetitive signal called the VCO Synch. It is a requirement to

adjust the phase of the PLL (phase locked loop), which generates the sampling clock, and to adjust the AGC (automatic gain control). Most commonly the VCO Synch is 2T.

- **Bit Cell Time:** is used as a starting estimate of the VCO Synch signal. If the value is not known and the VCO Synch field is at the beginning of the area to be analyzed, selecting **Measure Bit Cell Time** will determine it automatically.
- **Ignore last <n> samples:** Specifies the amount of "garbage" data to ignore after the end of the written data and before Read Gate goes false.

Additionally, the parameters below are useful (but not essential) for Channel Emulation with reference:

- **Read Gate:** If the Read Gate signal is connected to a DDA channel and specified, it will be used to determine the regions of the signal to be analyzed. Since the **VCO Synch** field is required for Channel Emulation with Reference and is normally present in the head signal in every block just after Read Gate goes true, it is recommended that Read Gate be used. If Read Gate is not present, the entire waveform will be used unless the Analyze Region cursors are enabled.
- **Read Gate Polarity:** Typically, Read Gate is a positive true signal. However, there are drives that use a negative true signal for Read Gate. If Read Gate is enabled, this setting allows you to select whether "True" is high or low.
- **Read Clock:** If present, the Read Clock signal is used only to determine where the "sample time" markers (vertical lines) are drawn.
- **Specify Region:** As previously described, markers can be used to define a subset of the head signal for analysis.
- **Filter Head Signal:** If enabled, an equalization filter is applied to the head signal before processing and display. This feature should be used if the DDA does not have access to the drive’s equalized head signal.

After the setup is complete, a reference waveform should be acquired and saved by selecting **Store Reference**.

Notes on Using Channel Emulation with Reference

1. Time misalignment between the reference and the current acquisition is made up for based on end of VCO Synch in the first Read Gate true section. If the reference contains multiple Read Gate true sections, we assume that their relative timing is approximately the same as in the current acquisition.
2. It is not possible to compare a reference of one sector to each sector in an acquisition containing many. It is not possible to compare a split sector with an unsplit or differently split sector.

Using Analog Compare

This method compares a reference signal to subsequent acquisitions and looks for large changes in the waveform. It is a general purpose test method that can be applied to finding errors in practically any signal, including VCO Synchronization fields, data and servo-information.

The user stores a "good" reference analog waveform. On subsequent acquisitions the filter, if selected, equalizes the newly acquired waveform. Analog Compare then attempts to measure the bit cell time and to find the end of VCO Synch, on both the stored reference and the acquired waveforms. If that succeeds the comparison starts, aligned by the marks at the end of VCO Synch. If the waveforms being compared do not contain VCO Synch, or if Read Gate is not used, the message "Could not analyze VCO Synch at start of signal" is displayed. This is just a warning, in case it is believed that VCO Synch ought to have been found. Analog Compare will proceed anyway.

Analog Compare computes the mean squared distance (msd) between the stored reference waveform and the head signal, over a 3-byte-wide window. The two waveforms are aligned to the nearest sample at the start. As the window is moved across the waveforms, alignment to the nearest sample is maintained as long as the timing in the reference is within 1% of the timing of the head signal. When the normalized msd exceeds the user-set Analog Threshold, an error will be recorded on the error list; the position of the maximum difference in the reference and the head signal is recorded. If the difference remains above threshold for more than the length of an encoded byte, a new error is recorded at each byte. Errors are ordered in the error list from largest to smallest difference.

Analog Compare requires some setup, and a stored reference signal to compare to before it can be used. The following must be provided.

Head Signal: As with the other methods, the head signal is essential for Analog Compare. Unlike the other methods, however, Analog Compare will work on any part of the head signal.

Reference: The stored reference should be acquired in the same manner (same time/div, sample rate, trigger position, etc) as the acquisitions that will be compared to it. See Note 3 below.

Analog Threshold: This is the user-specified threshold used by Analog Compare to decide when to record an error. A typical value for the Analog Threshold might be about 0.025. See Note 1 below.

Additionally, these parameters are useful (but not essential) for Analog Compare:

Read Gate: If the Read Gate signal is available, it will be used as described in Note 2, below.

Read Gate Polarity: Typically, Read Gate is a positive true signal. However, there are drives that use a negative true signal for Read Gate. If Read Gate is enabled, this setting allows you to select whether "True" is high or low.

Read Clock: If present, the Read Clock signal is used to determine where the "sample time" markers are drawn.

Use Analyze Region: As previously described, markers can be used to define a subset of the head signal for analysis.

Bit Cell Time: If specified, will be used to determine window size for comparison, otherwise a default value is used.

Code Rate: If provided, this is used to determine the size of an encoded byte, which is needed in determining the window size for Analog Compare.

Filter Head Signal: If enabled, an equalization filter is applied to the head signal before processing and display. This feature should be used if the DDA does not have access to the drive's equalized head signal.

After the setup is complete, a reference waveform should be acquired and saved by touching **Store Reference**.

Notes on Using Head/Analog Compare

1. Setting Analog Threshold: The "Analog Threshold" sets how large the difference (mean squared distance) between the reference and the acquisition must be to be recorded as an "error". A threshold of zero means that even the slightest difference will be enough. The threshold is normalized to full scale, which means that a setting of 1.000 specifies that the signals must mismatch by full scale over the entire three byte wide analysis window to be recorded as an error. DC mismatch is compensated and does not contribute to "mismatch".

   If the waveform amplitude can be -full scale to +full scale, the formula relating threshold to number of divisions mismatch (without vertical zoom) -- average over the analysis window -- is: sqrt(thresh). For example, a threshold of 0.015 corresponds to 0.49 of a division average mismatch within the window.
2. Analog Compare works best if it can analyze VCO Synch at the beginning of its analysis region for both the reference and the head signal. If Read Gate is available it should be able to do that.

   If Read Gate is not available, the "Analyze Region" cursors can be turned on and the "start" marker positioned near the beginning of a VCO Synch field. This also will let Analog Compare analyze the VCO Synch field and align reliably.

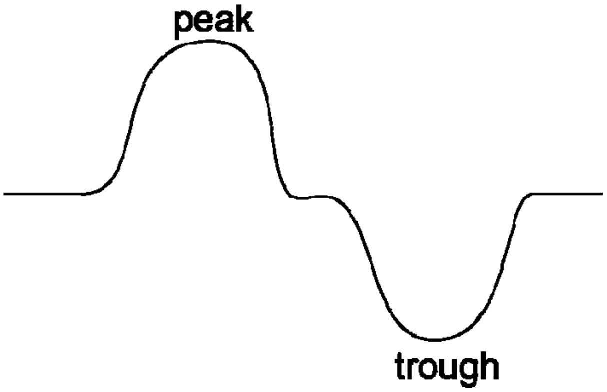
   If Read Gate is not available, and VCO Synch is not available, then Analog Compare attempts to align the signals up to about 12 bit cells difference. Note that if the start of the analysis region is far from the t=0 (trigger) position then small speed variations can cause the waveforms to be too misaligned for Analog Compare to be correct. For example, if the start of the analysis region is 15,000 bit cell times after the trigger, a 0.1% speed variation gives 15 bit cell times of jitter at the start of the analysis region. No matter how it achieves alignment, Analog Compare attempts to maintain alignment for up to a 1% speed mismatch.

   Analog Compare compensates for misalignment in the reference and acquisition based on the first Read Gate edge (if a source of Read Gate has been specified), and if it can analyze VCO Synch, on the end of the VCO Synch in that first section. If Read Gate is being used and the reference contains multiple Read Gate true sections, then the timing between them in the reference and the acquisitions must match within the alignment tolerance of Analog Distance, about 12 bit cells.
3. The reference waveform must be the same as the waveforms it is compared to. That means it must be the same length, acquired at the same horizontal and vertical scales. The same regions of the new and reference waveform will be compared. That means it is not possible to store a reference of a single sector and compare it to subsequent acquisitions containing multiple sectors. It is also not possible to compare a split sector to an unsplit or differently split sector. The reference and the acquisition must basically match.

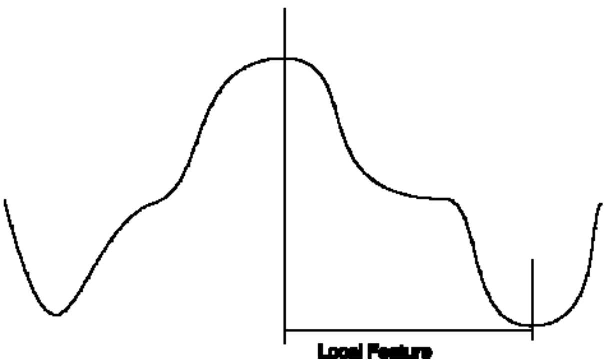
Local Feature Concepts

Overview

The term "local feature computation" indicates that a parameter computed on a waveform is determined only by information in the immediate vicinity of a specified feature of that waveform. The DDA defines a local feature as a waveform peak followed by a trough, like this:

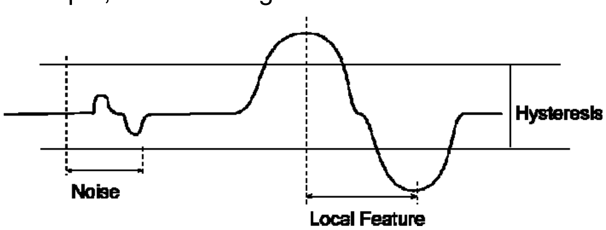


However, it is not the opposite: a trough followed by a peak. The diagram below shows a single local feature: the first peak and the trough that follows it.



Peak-Trough Identification

The key to identifying peak-trough pairs is the ability to discriminate between real pairs and false ones. For example, noise in a signal can be mistaken for a local feature, as here:



Similarly, 'bumps' in a waveform may also be mistaken for peak-trough pairs.

In order to avoid such misidentification, a hysteresis argument is provided for many local feature parameters. This essentially enables you to set a voltage band, which a peak-trough pair must exceed in order not to be considered noise or a "bump."

The hysteresis setting is also essential to the way peaks and troughs are identified by the DDA.

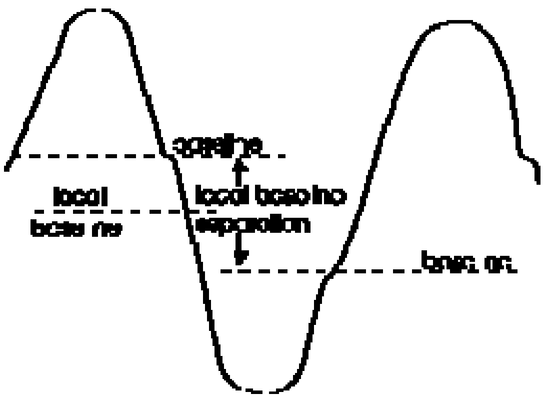
The search for local features extends from the left to the right parameter cursor. But first a peak must be found; and a waveform must rise to at least the value of the hysteresis setting in order to be positively identified as a peak.

This peak search starts with the first waveform sample, whose voltage value is used as an initial reference value for locating the peak. If a following waveform sample is found to be higher than the first by an amount greater than the hysteresis setting, a peak is said to exist. Any sample lower than the reference value, made prior to determination of a peak's existence, is used as a new reference point.

When a waveform rises by an amount that is more than the hysteresis, compared to the lowest prior waveform sample, the criterion for the existence of a peak is met. Then the search for its exact location and voltage value is initiated. Successive samples are compared to find the highest sample. Next, two points are found, one on either side of this highest sample and down from it by at least 25 % of the distance to the previous trough amplitude. A quadratic interpolation is then performed on these three samples to find the new peak location and amplitude. The same approach, using a sample lower than the highest sample by more than the hysteresis setting, is used to locate the trough.

Local Baselines

Many parameter measurements require that the baseline of a local feature be identified. In order to account for asymmetries due to MR heads, baselines are identified between the peak and trough, and between the trough and the following peak.



The baselines are found by locating a point at which the waveform 'rests' between the peak and trough and peak. These resting points are identified by statistically measuring the area of least change in voltage value between the peak and trough or trough and peak, with internal tolerance levels set to ensure against false baseline identification.

Another condition for identification is that the resting points must fall within a band, centered around the midpoint of the peak and trough extremes, whose height is 2/3 of the peak-to-trough height.

If one of the baselines cannot be identified, the local baseline is set to the found value. If neither baseline can be identified, then the local baseline is set halfway between the extremes of the local feature's peak and trough.

If baselines can be identified on both falling and rising slopes, the local-feature baseline is an average of the two baselines and 1 bsep is the distance between them.

If the local feature is the last to be identified before arriving at the end of the region being analyzed, it will not be possible to identify the trough-to-peak baseline of the following local feature. But when the peak-to-trough baseline is identified, then these two baselines are assumed to be separated by the same distance as the baselines for the previous local feature. And if this baseline cannot be identified, then the local baseline becomes the midpoint of the local peak and trough.

The separation between the baselines (local baseline separation) can also be of interest in determining the validity of certain measurements.

The following table summarizes the determination of the local baseline and its separation when the local feature is and is not the last identified before the right parameter cursor:

Local baseline and local baseline separation if last local feature		
Baseline identified peak-to-trough (PTBase)	**Local Baseline**	**Baseline Separation**
yes	(PTBase + [PTBase + previous separation])/2	previous local feature's baseline separation
no	midpoint of local peak and trough	0

Local baseline and local baseline separation if not last local feature			
Baseline identified peak-to-trough (PTBase)	**Baseline identified trough-to-peak (TPBase)**	**Local Baseline**	**Baseline Separation**
yes	yes	average of PTBase + TPBase	PTBase – TPBase
yes	no	PTBase	0
no	yes	PTBase	0
no	no	midpoint of local peak and trough	0

Setting Hysteresis

Hysteresis must be set for all local parameters. The determining factors for a hysteresis value are:

1. The maximum peak-to-peak noise in the waveform
2. The minimum local feature amplitude

The value should be somewhere between the first and second factors to ensure that noise is not mistaken for a local feature, and that all local features are recognized.

Local Parameters

The local parameters group offers measurements of common disk drive waveform parameters.

Local Parameter	Definition
lbase	baseline of local feature
lbsep	separation between peak-to-trough and trough-to-peak baselines
lmax	maximum value of local feature
lmin	minimum value of local feature
lnum	number of local features displayed
lpp	local feature’s peak-to-trough amplitude
ltbe	time between peak-to-trough or trough-to-peak
ltbp	local feature’s time between peaks
ltbt	local feature’s time between troughs
ltmn	time of local feature’s minimum value
ltmx	time of local feature’s maximum value
ltot	local feature’s time over a % threshold
ltpt	time between local feature peak-to-trough
lttp	time between trough-to-following peak
ltut	local feature’s time under a % threshold

Note: The DDA’s variable hysteresis setting is essential to identifying peak-trough pairs.

Local Feature Parameters

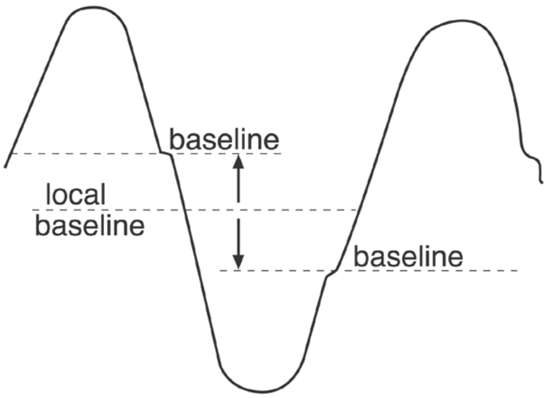
lbase **Local Base**

Definition The value of the baseline for a local feature.

Description The average value of the local baselines for all local features between the parameter cursors is displayed as lbase. For histograms, each individual baseline value for all local features between the parameter cursors is provided.

Parameter Settings Selecting this parameter from the **Measure** menus prompts you for a hysteresis setting, which allows you to set the hysteresis value to a specified number of vertical divisions.

Example



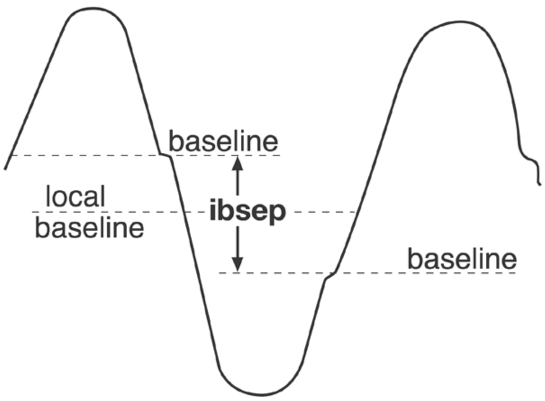
lbsep **Local Baseline Separation**

Definition The value of the baseline separation for a local feature.

Description The average value of the separation of the two baselines used to calculate a local baseline is displayed for all local features between the parameter cursors. For histograms, each individual baseline separation value for all local features between the parameter cursors is provided.

Parameter Settings Selecting this parameter from the **Measure** menus prompts you for a hysteresis setting, which allows you to set the hysteresis value to a specified number of vertical divisions.

Example



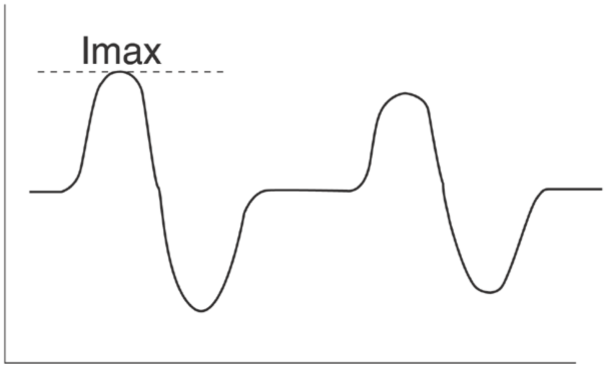
lmax **Local Maximum**

Definition The maximum value of a local feature.

Description The maximum value of all local features between the parameter cursors is determined, and the average value is displayed as lmax. For histograms, the maximum value of each local feature between the parameter cursors is provided.

Parameter Settings Selecting this parameter from the **Measure** menus prompts you for a hysteresis setting, which allows you to set the hysteresis value to a specified number of vertical divisions.

Example



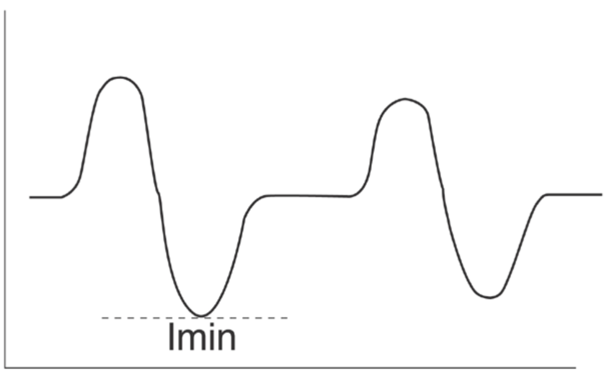
lmin **Local Minimum**

Definition The minimum value of a local feature.

Description The minimum value of all the local features between the parameter cursors is determined, and the average value is displayed as lmin. For histograms, the minimum value of each local feature between the parameter cursors is provided.

Parameter Settings Selecting this parameter from the **Measure** menus prompts you for a hysteresis setting, which allows you to set the hysteresis value to a specified number of vertical divisions.

Example



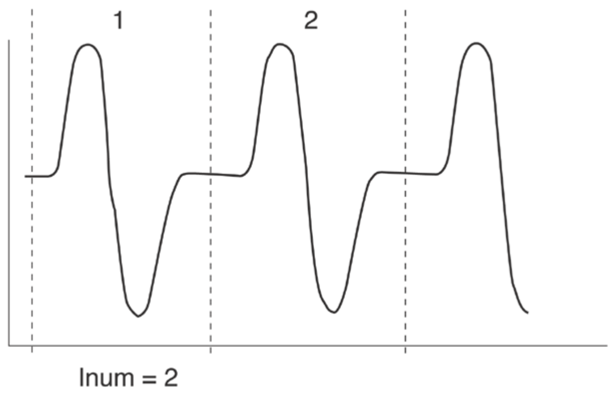
lnum **Local Number**

Definition The number of local features in the input waveform.

Description The number of local features between the parameter cursors is determined and displayed as lnum. One value of lnum each sweep is provided for histograms.

Parameter Settings Selecting this parameter from the **Measure** menus prompts you for a hysteresis setting, which allows you to set the hysteresis value to a specified number of vertical divisions.

Example



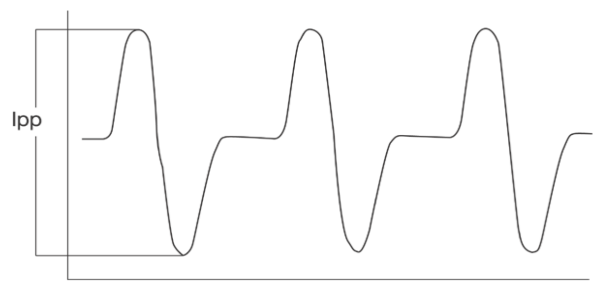
lpp **Local Peak-to-Peak**

Definition The vertical difference between the peak and trough for a local feature.

Description The peak-to-trough voltage difference is determined for all local features in a waveform, and the average is displayed as lpp. Provided for histograms is the peak-to-peak value of each local feature between the parameter cursors.

Parameter Settings Selecting this parameter from the **Measure** menus prompts you for a hysteresis setting, which allows you to set the hysteresis value to a specified number of vertical divisions.

Example



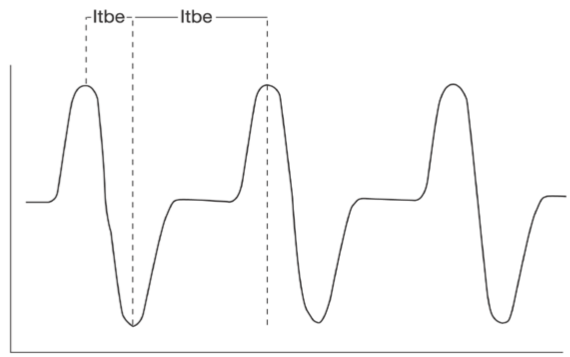
ltbe **Local Time Between Events**

Definition The time between a local feature peak and trough or a local feature trough and the next local feature peak.

Description Events are defined as either peaks or troughs. The average time between successive events in a waveform is displayed as ltbe. Provided for histograms is the time between each successive event between the parameter cursors

Parameter Settings Selecting this parameter from the **Measure** menus prompts you for a hysteresis setting, which allows you to set the hysteresis value to a specified number of vertical divisions.

Example



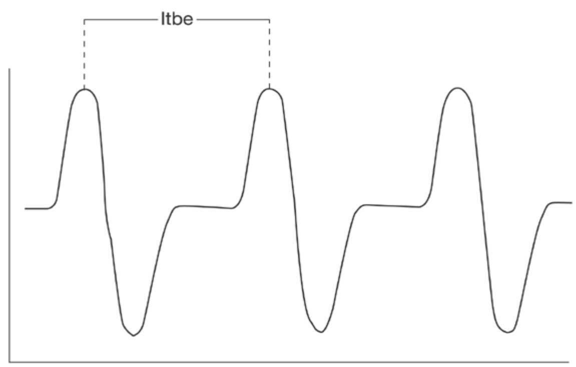
ltbp **Local Time Between Peaks**

Definition The time between a local feature peak and the next local feature peak.

Description The average of the time between successive local feature peaks is determined, and its value displayed as ltbp. Provided for histograms are the times between successive peaks for all peaks between the parameter cursors.

Parameter Settings Selecting this parameter from the **Measure** menus prompts you for a hysteresis setting, which allows you to set the hysteresis value to a specified number of vertical divisions.

Example



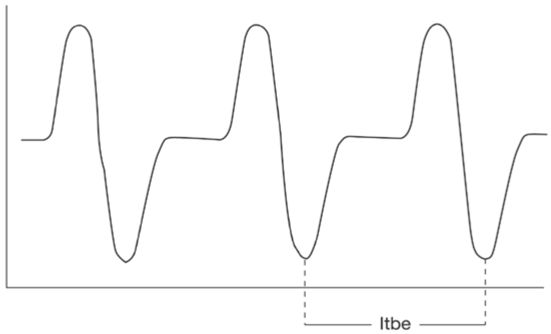
ltbt **Local Time Between Troughs**

Definition The time between a local trough and the next local trough.

Description The average of the time between successive troughs is determined, and its value displayed as ltbt. Provided for histograms are the times between successive troughs for all troughs between the parameter cursors.

Parameter Settings Selecting this parameter from the **Measure** menus prompts you for a hysteresis setting, which allows you to set the hysteresis value to a specified number of vertical divisions.

Example



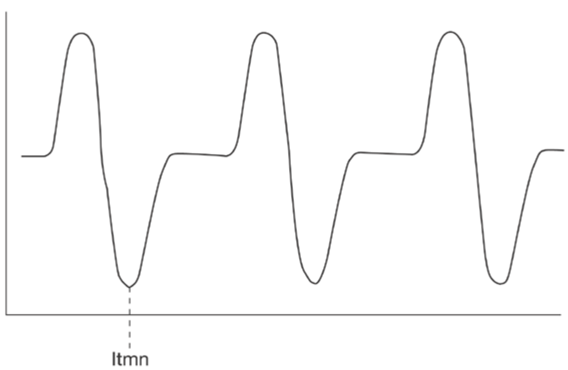
ltmn **Local Time at Minimum**

Definition The time of the minimum value of a local feature.

Description The time of the minimum value of the first local feature in a waveform after the left parameter cursor is determined. The time is returned as ltmn. Provided for histograms are all times for local feature minimums between the parameter cursors.

Parameter Settings Selecting this parameter from the **Measure** menus prompts you for a hysteresis setting, which allows you to set the hysteresis value to a specified number of vertical divisions.

Example



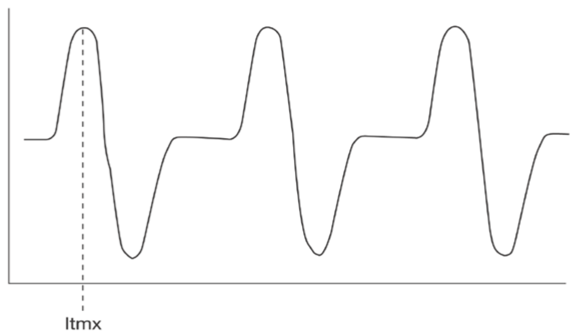
ltmx **Local Time at Maximum**

Definition The time of the maximum value of a local feature.

Description The time of the maximum value of the first local feature in a waveform, after the left parameter cursor, is determined and returned as ltmx. Provided for histograms are all times for local feature maximums between the cursors.

Parameter Settings Selecting this parameter from the **Measure** menus prompts you for a hysteresis setting, which allows you to set the hysteresis value to a specified number of vertical divisions.

Example



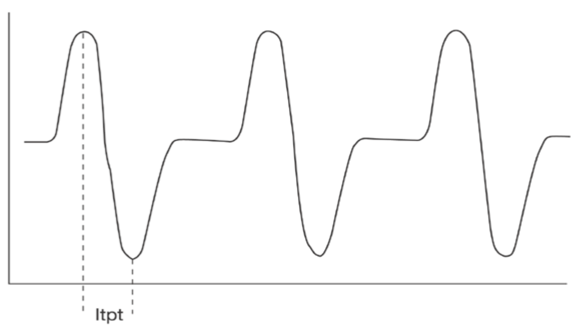
ltpt **Local Time Peak-to-Trough**

Definition The time between a local feature peak and trough.

Description The average of the time between all local feature peaks and troughs is displayed as ltpt. Provided for histograms are the times between peak-trough pairs for all local features between the parameter cursors.

Parameter Settings Selecting this parameter from the **Measure** menus prompts you for a hysteresis setting, which allows you to set the hysteresis value to a specified number of vertical divisions.

Example



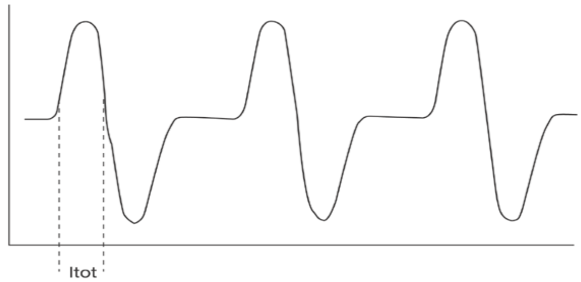
ltot **Local Time Over Threshold**

Definition The time a local feature spends over a user specified percentage of its peak-to-trough amplitude.

Description The peak-to-trough height of a local feature is measured. The time the local feature spends over a user specified percent of the peak-to-trough height is then determined. The average for all local features in a waveform is displayed as ltot. Provided for histograms is the time spent over the threshold by each local feature between the parameter cursors.

Parameter Settings Selecting this parameter from the **Measure** menus displays hysteresis and level percent menus. The associated fields allow you to set values in those menus: a specified number of vertical divisions or a percentage of the peak-to-peak height of the local feature.

Example



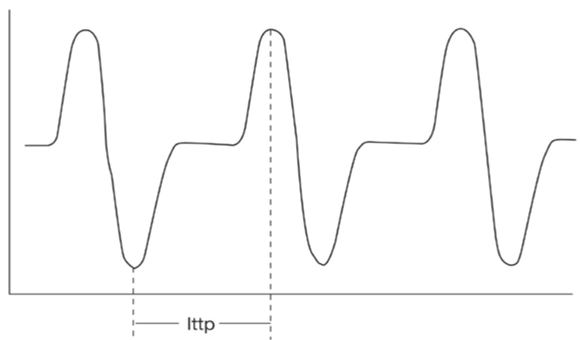
lttp **Local Time Trough-to-Peak**

Definition The time between a local-feature trough and the next local-feature peak.

Description The average of the time between all local feature troughs and the following local feature peak is displayed as lttp. Provided for histograms are the times between trough and following peak for all local features between the parameter cursors.

Parameter Settings Selecting this parameter from the **Measure** menus displays hysteresis and level percent menus. The associated fields allow you to set values in those menus: a specified number of vertical divisions or a percentage of the peak-to-peak height of the local feature.

Example



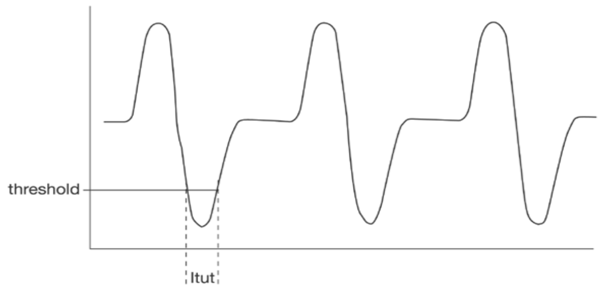
ltut **Local Time Under Threshold**

Definition The time a local feature spends under a user-specified percentage of its peak-to-trough amplitude.

Description The peak-to-trough height of a local feature is measured. The time the local feature spends under a user-specified percentage of this height is determined, and the average for all the waveform's local features is displayed as ltut. Provided for histograms is the time spent under the threshold by each local feature between the parameter cursors.

Parameter Settings Selecting this parameter from the **Measure** menus displays hysteresis and level percent menus. The associated fields allow you to set values in those menus: a specified number of vertical divisions or a percentage of the peak-to-peak height of the local feature.

Example



Disk Standard Parameters

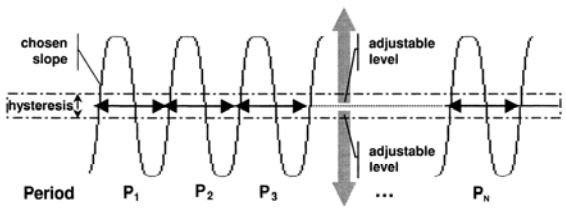
These parameters enable standard disk drive waveform parameter measurements. These parameters are accessible by touching **Measure** in the menu bar, then touching the **My Measure** button, and then a parameter button: P1 to P8.

DISK STANDARD PARAMETER	DEFINITION
aasym	amplitude asymmetry between taa+ and taa-
p@lv	period of each cycle in an acquired waveform, called Jitter in the parameter menu
nbph	narrow band phase of waveform Discrete Fourier Transform (DFT)
nbpw	narrow band power of waveform Discrete Fourier Transform (DFT)
owrt	overwrite
pw50	pulse width of peaks at 50% amplitude from baseline
pw50+	pulse width of positive peaks at 50% amplitude from baseline
pw50-	pulse width of negative peaks at 50% amplitude from baseline
res	resolution
taa	track average amplitude
taa+	track average amplitude of positive peaks from baseline
taa-	track average amplitude of negative peaks from baseline

All disk standard parameters except nbph, nbpw, owrt and p@lv make their measurements on waveform peak-trough pairs. In addition, several of the parameters determine the baseline of peak-trough pairs in order to perform their calculations.

Note: The DDAs variable hysteresis setting is essential for identifying peak-trough pairs.

ampl asym	**Amplitude Assymetry**
Definition	aasym = 1 [|(taa+ - taa-)| / (taa+ + taa-)]
Description	For a perfectly symmetric waveform, aasym=1. If any one side of the waveform is missing, aasym=0.
Parameter Settings	Selecting this parameter displays a hysteresis-dialog for setting the hysteresis value to a specified number of vertical divisions.

p@lv	Period at Level
Definition	Calculates the period of each cycle in an acquired waveform
Description	For each cycle in a waveform, p@lv determines the time from the beginning of the cycle as defined by a user specified threshold and slope to the end of the cycle as shown in the diagram below.
Parameter Settings	Selecting this parameter accesses a dialog to set hysteresis, level and slope. A hysteresis value defines the hysteresis, in divisions. That is, a voltage band is extended equidistantly above and below the selected level. In order for the signal to be considered valid, and not as noise, the signal must exceed the upper or lower limits of this band by half the hysteresis-division setting. 

nbph	Narrow Band Phase
Definition	Provides a measurement of the phase at a specific frequency for a waveform.
Description	nbph is the phase of the Discrete Fourier Transform (DFT) computed on a waveform at a specific frequency. The result is the phase of the corresponding frequency sine wave component of the waveform at the first data point between the parameter cursors. The nbph parameter calculates one bin of a DFT centered at the frequency provided. The bin width is 0.105% of the frequency selected if the waveform trace displayed by the DDA is 960 x (1/frequency) or more in length (i.e., the trace is equal to or longer than 960 cycles of a waveform at the selected frequency). Otherwise, the bin width is: 100 / integer[(trace length)/(1/frequency)] %, where integer[ ] designates discarding any fractional portions in the result. Thus, if the waveform trace is 48.5 times longer than 1/frequency, the bin width will be: 100/48 = 2.1% of the selected frequency. nbph is very sensitive to frequency, and it is important that the frequency value provided be as accurate as possible if accurate results are to be obtained.
Parameter Settings	Selecting this parameter accesses a frequency setting dialog.

nbpw	Narrow Band Power
Definition	Provides a measurement of the power at a specific frequency for a waveform.
Description	nbpw is the magnitude of the Discrete Fourier Transform (DFT) computed on a waveform at a specific frequency. nbpw calculates one bin of a DFT centered at the frequency provided. The bin width is 0.105% of the frequency selected if the waveform trace on the DDA is 960* (1/frequency) or more in length (i.e. the trace is equal to or longer than 960 cycles of a waveform at the selected frequency). Otherwise, the bin width is: 100 / integer[trace length/(1/frequency)] %, where integer[ ] designates discarding any fractional portions in the result. Thus, if the waveform trace is 48.5 times longer than 1/frequency then the bin width will be: 100/48 = 2.1% of the selected frequency.

	A BlackmanHarris window is applied to the input data to minimize leakage effects. The net result is that nbpw will provide excellent results even if frequency changes occur due to spindle speed variations. If the actual frequency differs from the specified frequency, and the bin width is 0.105% (minimum resolution bandwidth), the resulting power will be reduced from the actual as in this table: Frequency Difference dB Reduction 0.03% 0.3 dB 0.06% 1.1 dB 0.1% 3 dB If the bin width is greater than 0.105%, the frequency difference for which a specified dB reduction will occur will scale proportionally to the bin width/0.105. nbpw results are presented in dBm.
Parameter Settings	Selecting this parameter accesses a frequency setting dialog.

owrt	**Overwrite**
Definition	The ratio of residual to original power of a low-frequency disk waveform overwritten by a higher frequency waveform.
Description	owrt measures the residual power of a low-frequency LF waveform after it has been overwritten by a high-frequency HF waveform. The LF waveform should be stored to memory (M1-M4). The HF waveform can then be input to the DDA, and overwrite calculated where: owrt = 20 log (V_r/V_o), where V_r is the residual V_{rms} of the sine wave component of the HF waveform at the LF base frequency after the HF waveform write, and V_o is the V_{rms} of the sine wave component of the LF waveform at the LF base frequency. The calculation is performed by the DDA making a narrow-band power measurement (see nbpw parameter description) at LF, for both the HF and LF waveforms, and subtracting the second result from the first. A menu enables the choice of which waveform, HF or LF, is assigned to which DDA channel or trace (1, 2, 3, 4, M1 to M4, or F1 to F8). The menu button is used to set the input for HF or LF, while the input for the selected waveform is set with the associated knob. The owrt results are presented in dB. All averaging, including statistics and trend average, is performed on linear units. Average results are converted to dB. **Note**: In typical use, it is preferable to use nbpw to measure the LF waveform, and then the residual LF in the HF separately, instead of the owrt parameter. Overwrite is the difference between the nbpw readings in dB. There are two reasons why this is preferable: 1) nbpw, with statistics on, provides average power readings. With owrt the low frequency signal is typically a stored single-shot acquisition due to the difficulty finding a suitable trigger for time domain averaging of a head signal. 2) owrt computes both nbpw results each time. If the LF is stored, this is not necessary. So nbpw will take twice as many acquisitions as owrt and achieve a more stable average result in the same amount of time.
Parameter Settings	Selecting this parameter accesses a frequency setting dialog. This frequency is used to calculate nbpw for both the HF and LF waveforms. If a large number of digits is used, selection of the exact frequency may be difficult. In this case, a number with fewer digits and less precision should be chosen for the approximate frequency, then the precision should be increased as desired and the exact value chosen.

pw50	**Pulse Width 50**

Definition	The average pulse width at the 50% point between a local baseline and the local-feature peak, and between the local baseline and the local feature-trough.
Description	All local features between the parameter cursors for an input waveform are identified. The local baseline is identified for each feature, and the height between the local baseline and the peak is determined. The pulse width is measured at 50% of the peak. The same measurement is then performed for the trough. The average of all width measurements is displayed as pw50. Provided for histograms is the average pw50 value for each local feature between the parameter cursors.
Parameter Settings	Selecting this parameter displays a hysteresis-dialog for setting the hysteresis value to a specified number of vertical divisions.

pw50-	**Pulse Width 50-**
Definition	The average pulse width measured at the 50% point between the local feature baseline and the local feature trough.
Description	All local features between the parameter cursors for an input waveform are identified. The local baseline is identified for each feature, and the height between the local baseline and trough is determined. The pulse width is measured at 50% of the trough amplitude. The average of all width measurements is displayed as pw50. Provided for histograms is the average pw50 value for each local feature between the parameter cursors.
Parameter Settings	Selecting this parameter displays a hysteresis-dialog for setting the hysteresis value to a specified number of vertical divisions.

pw50+	**Pulse Width 50+**
Definition	The average pulse width at the 50% point between the local feature baseline and the local feature peak.
Description	All local features between the parameter cursors for an input waveform are identified. The local baseline is identified for each feature, and the height between the local baseline and peak is determined. The pulse width is measured at 50% of the peak amplitude. The average of all width measurements is displayed as pw50+. Provided for histograms is the average pw50+ value for each local feature between the parameter cursors.
Parameter Settings	Selecting this parameter displays a hysteresis-dialog for setting the hysteresis value to a specified number of vertical divisions.

res	**Resolution**
Definition	The ratio of the track average amplitude for a high and low frequency waveform.
Description	res returns, as a percentage, the ratio of track average amplitude (see taa parameter description) for a low frequency LF and high frequency HF waveform: res = (taa(LF) / taa(HF)) * 100%.
Parameter Settings	Selecting this parameter displays a hysteresis-dialog for setting the hysteresis value to a specified number of vertical divisions.

taa	**Track Average Amplitude**
Definition	The average peak-to-trough amplitude for all local features.

Description	All local features between the parameter cursors for an input waveform are identified. The peak-to-trough amplitude is determined for each feature, and the average is returned as taa. Provided for histograms is the peak-to-trough amplitude for each local feature between the parameter cursors.
Parameter Settings	Selecting this parameter displays a hysteresis-dialog for setting the hysteresis value to a specified number of vertical divisions.

taa-	**Track Average Amplitude-**
Definition	The average local baseline-to-trough amplitude for all local features.
Description	All local features between the parameter cursors for an input waveform are identified. The local baseline-to-trough amplitude is determined for each feature, and the average is returned as taa. Provided for histograms is the local baseline-to-trough amplitude for each local feature between the parameter cursors.
Parameter Settings	Selecting this parameter displays a hysteresis-dialog for setting the hysteresis value to a specified number of vertical divisions.

taa+	**Track Average Amplitude+**
Definition	The average local baseline-to-peak amplitude for all local features.
Description	All local features between the parameter cursors for an input waveform are identified. The local baseline-to-peak amplitude is determined for each feature, and the average is returned as taa+. Provided for histograms is the local baseline-to-peak amplitude for each local feature between the parameter cursors.
Parameter Settings	Selecting this parameter displays a hysteresis-dialog for setting the hysteresis value to a specified number of vertical divisions.

Disk PRML Parameters

These enable parameter measurements of auto-correlation signal-to-noise (ACSN) and non-linear transition shift (NLTS). The calculation of both parameters is based on a correlation math function.

- **ACSN** can be applied to any periodic waveform. Since these waveforms are by definition identical in every period, any deviation is due to uncorrelated noise sources. By performing an auto-correlation calculation of the waveform over successive periods, the level of less-than-perfect correlation can be measured. With this measurement, the noise level can be derived by ACSN. (See ACSN description.)
- **NLTS** offers the ability to measure all echoes in the auto-correlation calculation of a disk waveform. This includes the NLTS (adjacent location), second adjacent location, and overwrite (initial magnetization) echoes. The parameter performs NLTS averaging, pattern length searching, and limit checking to reduce the effects of noise and to ensure accurate measurements. (See NLTS description.)

Correlation Theory of Operation

The DDAs correlation function measures the correlation between one section of a waveform and other sections of the same waveform having the same length, or between a section and sections of equal length belonging to another waveform.

When the correlation is performed on the same waveform it is called an auto-correlation. If the shape of two waveform sections are identical, the correlation value will be maximized.

The DDA normalizes correlation values to ±1, with 1 indicating that the waveform sections are identical, -1 that the sections are inverted from each other, and 0 that there is no correlation.

Noiseless periodic waveforms will have perfect correlation (a correlation value of 1) when performing auto-correlation, and when the start of the second section is an integer number of periods later than the start of the first section.

Correlation values can be calculated as a function of various amounts of time shift between two waveform sections used in calculating a correlation. This calculation, as a function of the starting point of the second section being the ith waveform sample, is determined by:

$$\mathrm{mean}() = \sum_{x=a}^{b} \mathrm{wave}_x \Big/ (b - a)$$

$$\mathrm{variance}(\mathrm{wave}_a^b) = \frac{\sum_{x=a}^{b} (\mathrm{wave}_x)^2}{b - a} - \mathrm{mean}(\mathrm{wave}_a^b)^2$$

$$\mathrm{Corr}_i = \frac{\left(\sum_{j=0}^{N} \mathrm{wave1}_j * \mathrm{wave2}_{i+j} / (N+1)\right) - \mathrm{mean}(\mathrm{wave1}_0^N) * \mathrm{mean}(\mathrm{wave2}_i^{N+i})}{\sqrt{\mathrm{variance}(\mathrm{wave1}_0^N) * \mathrm{variance}(\mathrm{wave1}_i^{N+i})}}$$

where Corr_i is the ith sample point (starting from 0) of the correlation waveform, $\mathrm{wave1}_j$ is the jth sample of the first waveform, wave2 is the second input waveform (wave1 in an auto-correlation), and wave_a^b is a section of a waveform from sample *a* to sample *b*. The upper bound *N* in the summations determines the length (length is N+1 sample points, since the first sample is point 0) of the waveform sections on which the correlation calculation is performed. The divisor in the correlation function:

$$\sqrt{\mathrm{variance}(\mathrm{wave1}_0^N) * \mathrm{variance}(\mathrm{wave1}_0^N)}$$

normalizes the correlation calculation to ±1, while the

$$\mathrm{mean}(\mathrm{wave1}_0^N) * \mathrm{mean}(\mathrm{wave2}_i^{N+i})$$

term in the dividend removes any effect due to DC offset of the input waveforms in the correlation function.

Essentially, the correlation waveform function takes a section of the first waveform and calculates how it correlates with an equal-length section of a second waveform, using different starting points in the second waveform. This can be visualized as taking a section of waveform 1, sliding it over waveform 2, and calculating the correlation value for the area that overlaps. The bounds of the starting point are from the beginning of the second waveform to its length, minus the section length. At the upper bound, the end of the first waveform section lies at the last sample point of the second waveform. Because the length of waveforms in the DDA is limited to 10 divisions, the upper bound of the correlation function is 10 divisions minus the section length in divisions.

acsn	**Auto-Correlation Signal-to-Noise**
Definition	Provides a signal-to-noise ratio for periodic waveforms.
Description	Using the DDAs correlation function, **acsn** provides a measurement of the auto-correlation signal-to-noise for a repetitive waveform. At least two waveform repetitions need to be acquired in order to calculate **acsn**. In addition, the period of the waveform must be specified. Selecting this parameter displays a hysteresis-dialog for setting the hysteresis value to a specified number of vertical divisions. The parameter then verifies, and may adjust, the period based on the value provided. This is crucial because variations in disk rotation speed make the exact length of time for a disk waveform difficult to determine. Using the period as a starting point, the DDA performs an auto-correlation and looks for an auto-correlation peak at the period. At the top of the peak, the pattern repeats. The DDA locates the top and notes the corresponding time so that it can determine the period. Then it recalculates the auto-correlation using this period. The value of the auto-correlation at the period peak, R, is used to calculate the ACSN as: S/N = R/(1R), ACSN = 10* log10 S/N. For greater accuracy, the instrument averages several ACSN measurements when calculating

	acsn. An ACSN measurement is performed for each pattern, and the results averaged. All individual ACSN measurements can be observed by histogramming the **acsn** parameter. ACSN is limited to measuring signal-to-noise ratios of 9.6 dB or greater. ACSN results are presented in dB. All averaging, including statistics and trend average, is performed on linear units. Averaged results are then converted to dB.
Parameter Settings	When you select **acsn** from the **PRML** dialog's **With parameter** field (Drive Analysis  → Measure → Parameter Set → **PRML**) an acsn **Pattern Length** field appears also. The pattern length should be set as an integral number of waveform periods, and must be at least 50 samples at the DDAs sample rate. (The sample rate is shown in the **TimeBase** label box.) Because these periods will be correlated with the same number of following periods, the pattern length must be no more than half the number of full periods available in the sweep. **Notes on How Settings Affect ACSN Measurement** Pattern length should be close enough to the signal period (or multiple thereof) that it is on the correlation peak. The code measures correlation at 3 positions: the length, 6 samples less and 6 samples more. It will walk up a slope until the center sample is highest, tighten the spacing until the two side positions have correlation above half the correlation at the center position, and eventually perform a parabolic fit. But if pattern length misses the peak entirely then this search may not work; if the smallest of the three correlations is at the center and the two side position correlations are equal, it stops. For a sinusoidal signal, almost any setting of pattern length should work. One limitation to know about at this stage: the code checks for a "reasonable" peak. The correlation at the center position must be > 0.9, or the parameter computation is canceled. This is done without the benefit of the parabolic fit, but the intent is only to make sure of a reasonable peak. The total number of cycles in the acquisition does not affect the ACSN result for a stationary signal. Note that horizontal scale and the **Max Sample Points** entry on the TIMEBASE "Smart Memory" dialog interact. As you increase the time of an acquisition, the sample rate may decrease to keep the acquisition smaller than **Max Sample Points**. Although ACSN is not affected by horizontal scale, it is affected by sample rate. Be sure to keep it high enough. For the same signal, ACSN will fall as vertical scale is increased, because the signal fills less of the code space of the ADC and is therefore more contaminated by quantization noise and any other internal noise. In general, when displayed on a channel (or a reset trace), the signal should fill as much of the grid vertically as possible, but should never clip against the top or bottom of the grid. If high ACSN readings (greater than or equal to 30 dB) are expected, this is especially important. The pattern length should be kept to the minimum number of cycles that will satisfy the 50 samples per pattern requirement. In general, with a somewhat noisy signal, as the number of cycles in one pattern length is increased, ACSN is slightly reduced. For example, with an 80 MHz 75 mV sine wave with 0.6% (of p-p) rms noise, sampled at 1GS/s, **acsn** result varied with pattern length as follows: As ACSN readings get higher, small amounts of noise cause greater changes in dB. This is because high readings mean correlation was very close to 1. Examples:

Pattern Length (Seconds)	Cycles in One Pattern Length	ACSN
5.00E-8	4	27.54 dB

6.25E-8	5	26.97 dB
7.50E-8	6	26.78 dB
8.75E-8	7	26.57 dB
1.000E-7	8	26.47 dB
1.125E-7	9	26.41 dB
1.250E-7	10	26.30 dB

Correlation	ACSN
.9995	33.0 dB
.9990	30.0 dB
.9985	28.2 dB
.9980	27.0 dB
.9975	26.0 dB
.9970	25.2 dB

nlts	**Non-Linear Transition Shift**
Definition	Provides a measurement of the nonlinear transition shift for a disk drive signal.
Description	Using the DDAs correlation function, **nlts** measures the nonlinear transition (adjacent location) shift. At least two full cycles of the test sequence are required for the auto-correlation. In addition, the period of the waveform must be specified. The parameter then verifies, and may adjust, the pattern length based on the value provided. This is crucial, because variations in disk rotation speed make the exact pattern length for a disk waveform difficult to determine. Using the pattern length as a starting point, the DDA looks for an auto-correlation peak at the length. At the top of the peak, the pattern repeats. The DDA locates the top, and notes the corresponding time so as to determine the exact pattern length. Then it recalculates the auto-correlation, using this length. If the value of the auto-correlation peak at the pattern length is less than 0.9, the NLTS is not calculated. This is because the pattern-length sections will be too uncorrelated to provide a meaningful result. Otherwise, the pattern length value is used to calculate nlts. Using the pattern delay value, the DDA measures the auto-correlation coefficient for the first pattern-length chunk of the input waveform with a second pattern-length chunk, starting from the beginning of the input waveform at the delay value. In order to correctly calculate **nlts**, the disk drive waveform must be a pseudorandom sequence that will create an echo in an auto-correlation calculation, corresponding to the non-linear transition shift. Typically, this waveform is a 127-bit pattern based on an $x^7 + x^3 + 1$ polynomial; and the NLTS echo appears at a pattern delay of 20.06% of the input pattern length. Ideally, the value of NLTS is: NLTS(%) = 200* Correlation Coefficient (at delay). However, because noise in the input waveform can affect the correlation coefficients value, the DDA averages several NLTS measurements to reduce the effect of noise. An NLTS measurement is performed for each pattern, and the results averaged. All the individual NLTS measurements can be observed by histogramming the **nlts** parameter. The greater the number of pseudorandom pattern periods in the input waveform, the greater the reduction in the effect of noise on the **nlts** result. In order to further reduce the impact of noise,

<table>
<tr>
<td></td>
<td>
the NLTS calculations are adjusted by dividing their value by the correlation coefficient value at an integral number of pattern-length delays.
The following table gives the standard deviation of the nlts parameter for varying amounts of auto-correlation signal-to-noise, and numbers of repetitions of the pseudorandom sequence in the input waveform. The sampling rate used was four samples/bit cell, and the input waveform had 20% NLTS.
</td>
</tr>
<tr>
<td>Parameter Settings</td>
<td>
When you select nlts from the PRML dialog's With parameter field (<u>Drive Analysis</u> → Measure → Parameter Set → PRML) an nlts Pattern Length field and pattern Delay field appear also. You can adjust the mantissa, exponent or number of mantissa digits using the pop-up numeric keypad. The pattern length should be set to the pattern period.
Although the DDA searches for the correct pattern length, the value provided needs to be sufficiently close to the actual pattern length for nlts to perform the search. A 1 µs pattern may, for example, accept a range of 1 µs ± 40 ns. Within this range, a value for nlts will be provided. Otherwise "---" appears on the screen, indicating that no measurement can be made.
The pattern Delay setting is a percentage of the pattern length. The DDA will internally scale the delay value entered by the ratio of the pattern length calculated internally to the pattern entered by you. Several disk drive waveform attributes can be measured by using different delay values. The following table provides delay values to enter for the commonly used 127-bit pseudo-random sequence ($x^7 + x^3 + 1$ polynomial) when measuring various waveform attributes:
<h2>Notes</h2>
The pattern Delay tells the DDA where in the repeating pattern to measure NLTS. NLTS is calculated from the correlation coefficient at that time. Correlation is calculated at 3 sample times: the nearest sample time and one on each side. A curve fit is performed to calculate a better estimate of the true peak height.
As described above, the nlts parameter only requires one waveform. Each acquisition must contain only the pseudorandom repeating sequence (PRS), not a servo wedge or a preamble. It does not matter where in the PRS the acquisition begins; its echo properties are independent of starting point. There must be at least two repetitions of the pattern in each acquisition. A reasonable number would be about 25 repetitions (32 repetitions of a 127-bit sequence should fit in a sector).
NLTS requires at least 5 samples per PW50 for acceptable accuracy. More is better.
The PRS data must correspond to the transitions on the media. This means it must be written in direct write mode; it must not be scrambled or encoded.
NLTS calculated by correlation techniques and by the fifth harmonic elimination techniques do correlate, but they are not identical. Fifth harmonic elimination uses a pattern including only dibits and widely spaced transitions. Of course, a dibit is the worst case for NLTS. The pseudorandom pattern has some dibits, some tribits, some transitions 1 apart, etc. The reading that any method based on a PRS comes up with is affected by all of the pattern. This tends to make the correlation method come up with a somewhat lower value than fifth harmonic. That is the better value to look at when determining write precomp, since the PRS is a lot like real data.
The fifth harmonic method is not sensitive to amplitude asymmetry. Its drawbacks are

If the fifth harmonic is small, noise will tend to inflate the value read: for 20 dB SNR it may not be possible to read below 10% nlts
Amplitude loss due to partial erasure will contribute to the fifth harmonic about as much as actual NLTS.

The correlation method is less sensitive to amplitude loss due to partial erasure: one reference cites experiments showing that 25% nlts and 25% amplitude loss only inflates the nlts reading to 30%.
However, it is more sensitive to PW50/T ratio (it is most accurate at high ratio, approaching 3.0).
</td>
</tr>
</table>

	Partial erasure and hard/easy transition shift (due to DC erase) creates separate correlation echo peaks which, depending on the pattern used, may be close to the adjacent transition **nlts** echo and interfere. It is important to pick a good PRS.

ACSN	#Pattern Repetitions	nlts Standard Deviation
26 dB	2	0.44 %
	10	0.28 %
	25	0.20 %
23 dB	2	0.59 %
	10	0.32 %
	25	0.26 %
20 dB	2	0.65 %
	10	0.42 %
	25	0.28 %
17 dB	2	1.08 %
	10	0.57 %
	25	0.35 %

Waveform Attribute	Bit Cell Location	Delay (%)
Adjacent Location	25.5	20.08%
2nd Adjacent Location	-30.5	-24.02%
Initial Magnetization	45.5	35.83%
Interaction Interference	-60.5	-47.64%

Noise Parameters

Disk noise parameters enable parameter measurements of media signal-to-noise (msnr), residual (electronics) signal-to-noise (rsnr), and the ratio of media to residual signal-to-noise (m_to_r). The calculation of all three parameters is based on the distribution of the averaged Viterbi input samples.

- **msnr** can be applied to any single-frequency, sector-based data pattern. The single-frequency data will be sampled at the peaks (maxima), zero crossings, and troughs (minima). Any deviations from the ideal sample points will be a result of noise. By performing multiple reads, random noise can be averaged away. With this measurement, the repeating media noise level can be derived by msnr.
- **rsnr** can be applied to any single-frequency, sector-based data pattern. The single-frequency data will be sampled at the peaks (maxima), zero-crossings, and troughs (minima). Any deviations from the ideal sample points will be a result of noise. By performing multiple reads, random noise can be quantified. With this measurement, the non-repeating residual (electronics) noise level can be derived by rsnr.
- **m_to_r** can be applied to any single-frequency, sector-based data pattern. The m
- snr is compared with the rsnr. The resulting ratio indicates whether the signal is dominated by media noise if it is greater than 1.00, or dominated by residual (electronics) noise if less than 1.00.

> **Note:** Although the math on the following pages is simplified by assuming a large number of sweeps, inside the DDA it calculates what the final value should be from 2 or more sweeps. The value will become more stable as more sweeps are taken, but its mean should not change. Therefore, a large number of sweeps is not required for unbiased values.

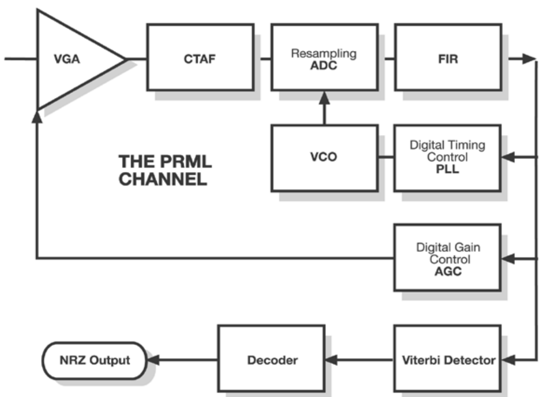
msnr	Media Signal-to-Noise Ratio
Definition	Provides a signal-to-noise ratio for single-frequency, sector-based waveforms.
Description	Granularity in magnetic media produces zigzag transitions. The exact location of the zigzags changes from write to write causing media noise (also known as zigzag noise). During read operations, the read head effectively averages across the track the location of all the zigzags. If the recorded track width is wide, there are many zigzags and very little variability in the averaged transition location. If the track width is narrow, there are fewer zigzags, and the variability in the averaged transition location increases. Head signal amplitude is approximately proportional to track width. Media noise is approximately proportional to the square root of the track width. Therefore, the ratio of the head signal to the media noise (msnr) is proportional to the square root of the track width. As track widths continue to decrease, msnr will get worse. Although advanced head technology (i.e., MJR and GMR) can increase the head signal output for a given track width, the effect of media noise on the head signal is also increased so there is no improvement in msnr. As track widths decrease, msnr will continue to increase despite greatly improved head technology. By using single-frequency data, the algorithm is able to focus on the peaks, the zero-crossings, and the troughs. The ideal Viterbi input samples are therefore +1, 0, and -1. Deviations from the ideal are quantified and graphed. The squared sigma of the total noise distribution is equal to the squared sigma of the media noise distribution. This relationship is always true: $\sigma_t^2 = \sigma_m^2 + \sigma_r^2$ For a large number of samples: $\sigma_m^2 = \sigma_a^2$ $\sigma_r^2 = \sigma_t^2 - \sigma_a^2$ Once the distribution of the media noise has been calculated, the msnr is then calculated by: $msnr = 20 \log(v_{0-p}/\sigma_m)$
Parameter Settings	After specifying the location of the “Head Signal” and the optional “Read Gate,” selection of msnr is automatic after you touch the **Noise Analysis** button. Touch the **Setup for Single Frequency** button to initialize the measurement.

rsnr	Residual Signal-to-Noise Ratio
Definition	Provides a residual signal-to-noise ratio for single-frequency, sector-based waveforms.
Description	Residual noise is the random noise present on a disk drive signal from read to read. By using single-frequency data, the algorithm is able to focus on the peaks, zero-crossings, and the troughs. The ideal Viterbi input samples are therefore +1, 0, and -1. Deviations from the ideal are quantified and graphed. The squared sigma of the total noise distribution is equal to the squared sigma of the media noise distribution and the squared sigma of the residual noise distribution. This relationship is always true: $\sigma_t^2 = \sigma_m^2 + \sigma_r^2$ For a large number of samples: $\sigma_m^2 = \sigma_a^2$ $\sigma_r^2 = \sigma_t^2 - \sigma_a^2$ Once the distribution of the media noise has been calculated, the rsnr is then calculated by: $rsnr = 20 \log(v_{0-p}/\sigma_r)$
Parameter Settings	After specifying the location of the “Head Signal” and the optional “Read Gate,” selection of rsnr is automatic after you touch the **Noise Analysis** button. Touch the **Setup for Single Frequency** button to initialize the measurement.

m_to_r	MSNR-to-RSNR Ratio
Definition	Provides a media signal-to-noise residual to signal-to-noise ratio for single-frequency, sector-based waveforms.
Description	The m_to_r ratio provides a quick measurement to compare the media noise with the residual noise. If this ratio is greater than 1.00, the signal is residual, or electronics noise dominated. The measurement is calculated by: $m\_to\_r = \sigma_m / \sigma_r$
Parameter Settings	After specifying the location of the "Head Signal" and the optional "Read Gate," selection of m_to_r is automatic after you touch the **Noise Analysis** button. Touch the **Setup for Single Frequency** button to initialize the measurement.

PRML Channel Emulation

This introduction to PRML and its concepts explains the role of the PRML channel chip components



. It describes how the Channel Emulation feature of the Disk Drive Analyzer works together with them using equalization, clock and gain recovery, maximum likelihood detection, sequenced amplitude margin, and encoding and error detection.

Why PRML?

For the remarkable gains in disk drive capacity to continue, media and head performance improvements are no longer enough. Faced with equally impressive advances in semiconductor technology, disk drive engineers have been working to create a new read-channel architecture that will allow capacity to grow unimpeded.

The answer lies in the construction of the disk itself. The disk's magnetic poles, with two orientations possible along the track, store the bits as "0" and "1". When the drive reads, the head detects the transition from one pole to another, as bit "0" to bit "1", for instance. If such transitions are "far away," or low-density, the drive will see isolated pulses. But to increase density, the pulses can be made shorter and placed closer together or kept wide but overlapping. While the first of these alternatives, represented by Peak-Detect systems, has reached its limits, the second, Partial-Response Maximum Likelihood (PRML), has allowed the industry to go on boosting capacity.

The overlapping pulses of partial-response systems allow much greater density. PRML systems have more samples per pw50, which is defined as the width of an isolated pulse at 50% of its amplitude. And the more complex, or higher-order, the PRML system, the greater the density that can be obtained. Comparing typical values achieved by available PRML systems with Peak-Detect we find:

Density of Samples per pw50	
Peak Detect	1
PRML	
PR4	1.65

EPR4	2
E2PR4	2.31

However, the higher order PRML schemes need very complex circuits and decoders. While the Class IV partial response (PR4) system works with three vertical levels of samples, extended partial response 4 (E2PR4) has seven levels, and requires not only a higher resolution of ADC, but a complicated timing and gain recovery circuit and sophisticated ML detector as well. Another disadvantage of the more complex PRML schemes is that they are more sensitive to noise.

Principle of Equalization

The process of taking the more-or-less Lorentzian shaped head response to a magnetic transition and turning it into a correctly shaped pulse is called equalization. This is of great importance, due to the need of the Viterbi detector inside the PRML channel chip for correctly shaped pulses. Essentially, equalization is performed in the read channel chip by a continuous time analog filter (CTAF).

Noise must be eliminated before sampling occurs, or else it becomes impossible to separate it from the head signal. Since the head signal is typically noisy, and contains pulses that are not quite the desired shape, the DDA provides an equalization filter to reduce much of the noise and reshape the pulses before it processes the waveform. This filter is a digital implementation of a seven-pole, two-zero equiripple filter.

When you are using the filter, there are a number of parameters to be set. If the head signal has already been acquired, the filter parameters can be set automatically by pressing the **Train Filter** button. When this button is pressed:

- If the signal type is Peak Detect, the boost is set to zero and the -3 dB frequency is set to:

$$\frac{1}{\text{bit cell time}}$$

- If the signal type is PRML (PR4, EPR4 or E2PR4) the -3 dB frequency is set to:

$$\frac{0.5}{\text{bit cell time}}.$$

The best boost and -3 dB frequency are found by optimizing boost at the default -3 dB frequency, then optimizing -3 dB at the better boost. Then optimize boost at the new -3 dB frequency, and optimize -3 dB frequency again at the new boost. And then, if -3 dB frequency has changed by more than a small amount, optimize the boost one final time. The goal for optimization is to maximize the mean of the 100 worst SAM values. A typical run will recompute the filter, apply it, and run the Viterbi detector on the filtered waveform fifteen to twenty times. While the filter training is in progress, a message is displayed showing the last boost and -3 dB settings and the mean of the 100 worst SAM values at that setting.

The result of training is a close approximation to the best settings for our digital version of a CTAF on the current acquisition, using the current setup of the FIR. The cleaner the waveform, the better the approximation will be. The filter should be trained on a signal from a good read, those settings can be used for all reads in the same zone.

Train Filter should be done with the acquisition stopped (press STOP on the DDA), so that the same waveform is worked on each time; otherwise the search may be slow to converge. **Train Filter** can take a significant amount of time, and it is recommended that it be done on relatively short waveforms of 50 or 100 kpoints. Once trained, the memory length can be adjusted to the desired length. To ensure that the group delay is flat, the filter requires at least five samples per bit cell. This is not a hard limit, but performance will degrade with fewer than five samples per bit cell.

Alternatively, you can adjust the filter settings manually.

-3 dB Frequency

This is the actual -3 dB frequency of the filter. In most implementations of frequency cutoff (fc), the -3 dB point, if Boost is 0 dB and group delay is 0%, is controlled by you. It is understood that the real -3 dB frequency will be higher by some factor that depends on Boost and Group delay settings. Therefore changing Boost or Group delay requires changing "fc" in order to keep the -3 dB point in the same place.

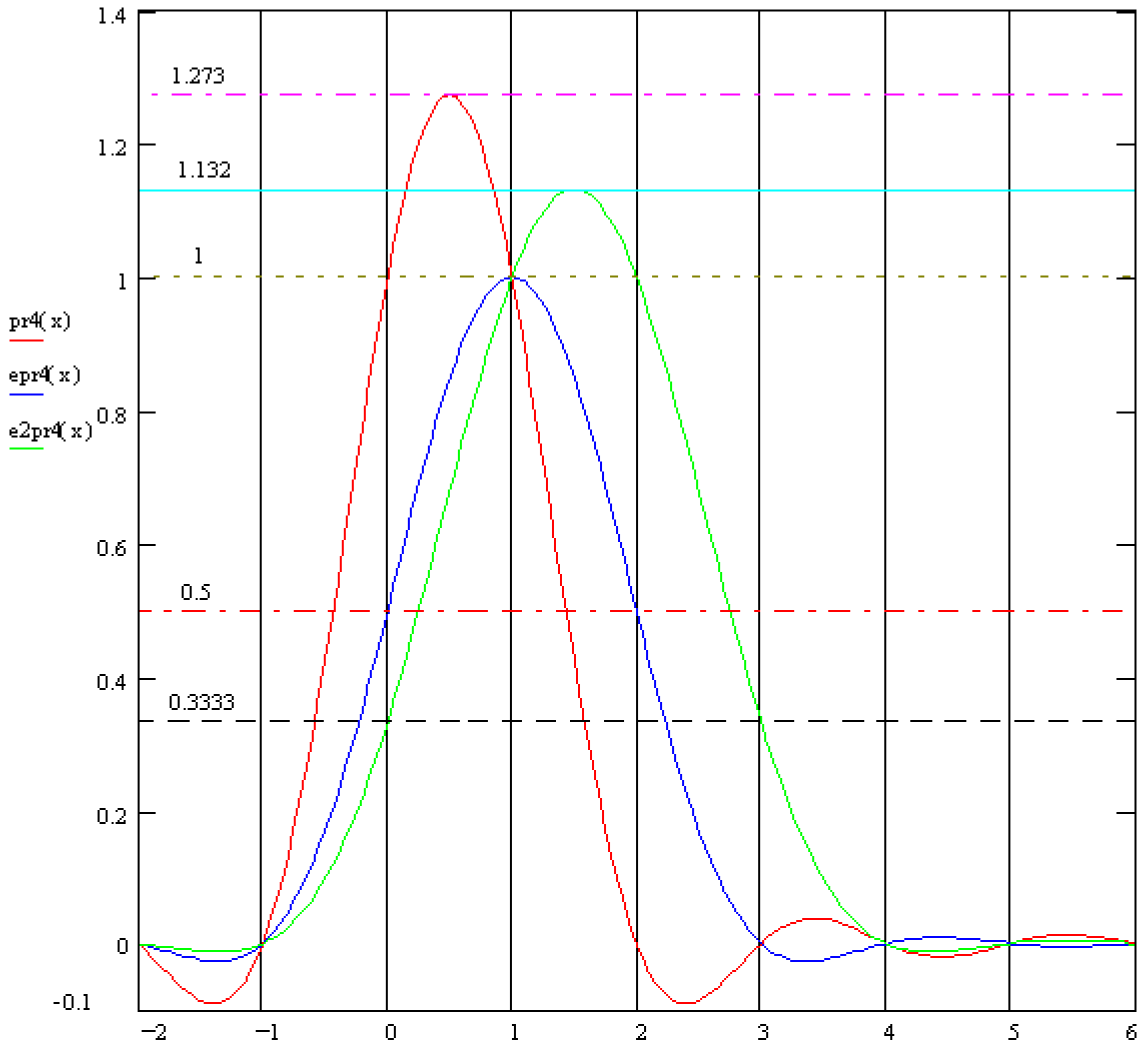
The DDA does this work automatically: you set the desired -3 dB frequency, and the DDA calculates fc for the current Boost and Group delay.

The frequency cutoff accuracy (alpha = 0) is < 2% of the setting.

Boost at fc

The pulse shape produced by a transition going under a head is nearly Lorentzian, but that is not the correct shape for PRML. Boosting the gain at high frequencies makes the pulses slimmer and eliminates the long tails they would otherwise have. The perfect (mathematically created) shape that an isolated pulse should have is illustrated below, for PR4, EPR4 and E2PR4. Note that the vertical lines show when the channel samples the waveform.

The boost



accuracy is within 0.5 dB of the setting.

Group Delay

The delay through the filter of the lowest frequencies can be adjusted. The normal setting is 0.0% adjustment; that is, flat response. This is used to compensate for group delay distortion before the filter. Group delay setting is unaffected by **Train Filter**.

The digital implementation of our equalizing filter does not have perfectly flat group delay. The non-flatness increases with the ratio of -3 dB frequency to sampling rate and with boost. The DDA requires 5 samples per bit cell, which means that the -3 dB frequency will be </= 10% of sampling rate. If this requirement is met, group delay should be sufficiently flat. E2PR4 is especially sensitive to non-flat group delay. For E2PR4, especially if the "5 samples per bit cell" requirement is not met, it will probably be beneficial to set the "Group Delay" field to a small positive number, perhaps 6% or so. This helps flatten the group delay of our equalizer filter.

Resampling ADC

Because the DDA data is already digital, this simply interpolates between DDA samples to produce a digital value at the channel sample time.

Finite Impulse Response (FIR)

In addition to the continuous time analog filter (CTAF), there is normally an FIR filter following the analog-to-digital converter at the PRML channel's sample rate. Its purpose is to 'adapt' and fine-tune equalization. The DDA's 21-tap FIR has coefficients that can be set using remote commands. The tap weights can be asymmetric to minimize delay through the filter. Extra delay will reduce the stability of our control loops for sampling phase and automatic gain control.

If the coefficients as entered sum to > 1 they are renormalized to sum to 1.0; if they sum to < 0.1 as entered they are rejected. Other than that, there are no restrictions on the tap weights.

In many channel chips, the FIR equalization filter is adaptive. However, the DDA does not change the values set by the user.

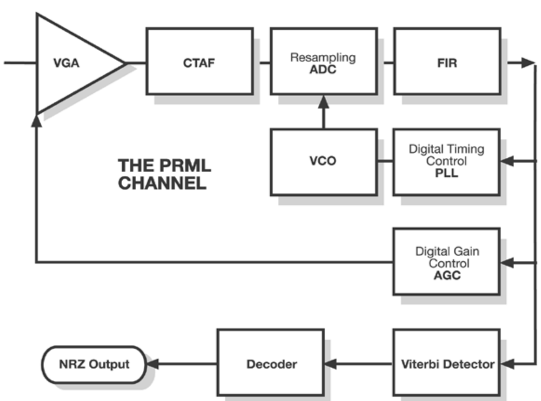
Phase Locked Loop (PLL)

Correct operation of any PRML system also depends on the taking of readback signal samples at exact "focus" positions. Shifting the clock slightly from the correct position is enough to distort sample values.

A clock recovery circuit, Phase Locked Loop (PLL), adjusts the phase of the oscillator, based on the value of the phase error. This is usually done in a feedback circuit.

The phase-error function is calculated in the phase detector circuit and is equal to zero in the correct clock position. When the signal is correct, the error signal is equal to zero, and oscillator frequency and phase remain in exactly the correct position. If for some reason the phase of the input signal and that of the oscillator diverge (owing to instability of disk rotation or noise, for example) the phase-error signal deviates from zero, and the frequency of the oscillator shifts.

Two main problems have to be resolved in the clock recovery circuit. One is the initial fast-phase acquisition: prior to a reading of the pattern, it is necessary to align the clock to the correct position of the pattern. The other is tracking: the following of relatively slow instabilities of the disk rotational speed. In order to avoid fast, noisy phase shifts, and to provide system stability, the phase-error signal is integrated by an integrator.



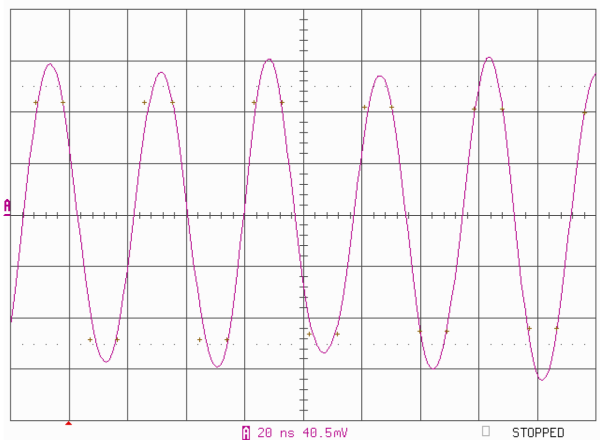
Automatic Gain Control (AGC)

The phase and gain steering algorithm in the DDA's channel emulation (AGC) will adjust for changes in the signal after VCO Synch. Read gate tells us where to start looking at the VCO Synch.

Phase is essentially the timing of when a sample is taken, while gain is the levels that are being searched for as +1 and -1. Thus, when a sample larger than the ideal value is taken, the level looked for by the emulation is increased. The screen below shows an example of phase and gain steering.

To start the AGC, we must find the maximum and minimum levels in the VCO Synch, and then initialize the levels to be searched. Thus, if the pulses are exactly the desired shape, then 1 and -1 will be where expected, with the other levels symmetrically spaced in between.

Note that the VCO Synch must exist in the acquired head signal, or the channel emulation cannot be run. The phase and gain steering algorithm in the emulation will adjust for changes in the signal after VCO Synch, assuming it starts correctly.



PLL and AGC

The sample, phase-steering (PLL) and gain adjust (AGC, level computation) work together, one sample at a time. They are used to compute sampled data for input to the Viterbi detector.

ML Detector

Samples on the output of the ADC ideally have a small number of levels: {-1,0,+1} for the PR4 system, for example. A threshold detector could be used to classify a current sample value comparing it to an amplitude threshold. For example, if sample > 0.5, sample = 1; if sample < -0.5, sample = -1; if sample ≤ 0.5, sample = 0.

For the sequence of samples 0.8 0.3 -0.7 -0.2 0.6 0.9 1.1 0.2, the threshold detector output would be: 1 0 -1 0 1 1 1 0.

The difficulty here is that the sequence of three ones in a row ("111") is impossible: the pulse is too wide. The only possible combinations are 011, 110, -111, and so on.

A threshold detector, such as the peak detector on a Peak-Detect drive, does not use the previous and subsequent samples. But the maximum likelihood (ML) detector "knows" that "111" is a forbidden sequence of samples and tries to determine the most probable data pattern for this sequence of samples (21 samples used for PR4).

Proposing several close allowable sequences -- {1 0 -1 0 1 1 0 0} or {1 0 -1 0 0 1 1 0} or {1 0 -1 0 0 0 1 1} -- is easy. But which is the most probable?

The decision is made based on a sequence of samples, instead of only a single, current sample, and the sequence with the minimum distance (maximum likelihood) is selected as the detection result.

Viterbi Detector & Trellis

The Viterbi detector is a state machine consisting of two distinct parts: states and transitions. While state is the current magnetization of the disk and some history (memorization of the latest states), transition relates the current state to the next state. For the detector, only two possibilities exist: either the state (medium magnetization) is the same between the current and the next bit periods or it is not.

The detector's trellis works according to this dichotomy: "0" or "1" is followed by either "0" or "1", and so on. The trellis is a mechanism that keeps track of a sequence of magnetization states. When the ML detector makes decisions, it keeps the states of several consecutive time instants and estimates the likelihood of possible "histories" (higher probability). The higher the order of the PRML system, the larger and more complex the trellis. However, some trellises do not allow certain transitions (d=1 constraint), thereby limiting the extent to which the pulses can overlap.

SAM

Sequenced Amplitude Margin (SAM) measures the error margin of every sample taken by a PRML channel chip. Determining that a written bit is either a "0" or a "1" is the disk drive's most basic decision. SAM measures the margin by which the Viterbi detector has made this decision, the margin or distance being a function of the path metrics and current sample taken together.

SAM can provide a prediction of the error rate, and can be used both for characterization and for optimizing equalization. Lower SAM values mean worse error rates.

The range of SAM values depends on the PRML order, as does the path metrics, memory, or sequence. The range is 0 to 2.0 for a PR4 channel. The "0" in this case signifies that the drive had no margin to make the decision, that it could have read every single bit wrongly, and that at 2.0 the drive had as much margin as it could and will never make an error.

The distribution of SAM values will always center on the square of the minimum distance to an error, i.e., 2.0 for PR4; 1.0 for EPR4 (these numbers are correct when the signal amplitude is normalized, so sample values are -1 to +1). The width of the distribution is narrow for clean signals, broader for noisy signals. Misequalized signals have broad, multi-peaked distributions. When the tail of this distribution crosses zero, it means an error has been made. This can only be determined with certainty if the correct decode (the correct path through the trellis) is known.

Encoding

Both Peak-Detect and PRML read channels use Run-Length Limited (RLL) coding. This corresponds to the only part of the channel chip that the DDA does not simulate: that between the Viterbi detector and the NRZ lines at the output of the chip.

RLL codes impose constraints on the data written to the disk by limiting the minimum and maximum number of 0's that must come between each pair of 1's in the encoded pattern written to the disk (head/analog signal). The limitation on the minimum number of "0"s provides transition separation to avoid pulse crowding. RLL codes are characterized by four parameters, referenced as (m/n)(d/k):

- modulation code maps *m* user bits (NRZ data) into *n* encoded bits (head/analog signal)
- *n* is always bigger than *m*, because *n* smaller than *m* would mean data is compressed on the disk; Code Rate = m/n
- *d* equals the minimum number of 0's between two consecutive 1's
- *k* equals the maximum number of 0's between two 1's

The DDA does not implement interleaved ML detection, therefore it does not check even and odd sample streams separately for this limit. However, when the constraints are specified as (d, k1, k2), one can sum k1 and k2 and use that where *k* is needed. This allows a series of non-transitions long enough to unquestionably be an error to be reported as an error. However, it will not catch all the sequences that would be an RLL violation for interleaved detection.

Error Correction

Prior to the RLL encoding, the user data is normally encoded inside the drive's microcontroller, using special error-correction codes (ECC). Thus, there are two levels of encoding.

User Defined Trellis

File Format and Language (version 1)

This feature allows you to define the trellis used by the Viterbi detector in the LeCroy DDA's channel emulation. The file specifies the target levels and all possible transitions. You can also specify many significant aspects of our emulated channel, including the proportional (phase) and integral (period) gain on the clock steering control loop, the AGC gain, adjustment limits, etc. This permits you to define how the signal will be handled by the emulated channel, as well as the trellis used in the Viterbi detector.

The User Definition file is a plain ASCII text file. Such a file can be made with Windows Notepad or any text editor, or by any word processor with a "Save As Text" capability. The file may have any name in 8.3 format (i.e., DOS file name). Our example uses the extension ".UDF", but it is not required. A User Definition file can be loaded from floppy disk or hard disk (the hard disk is optional).

The first characters in the file must be **USER DEFINED VITERBI TRELLIS FOR LECROY DDA**. If the string **USER DEFINED VITERBI TRELLIS FOR LECROY DDA** is not seen at the start of the file, the file is not parsed, and any previously loaded user definition is left undisturbed. The message "Header not found at start of file - read aborted" is displayed.

The rest of this Help file defines the keywords that can appear and the arguments they take. An example file follows. In all cases, if an argument is outside limits, or a stated order dependency is violated, an informative error message is displayed and the parse is canceled. The error message will persist on the screen until replaced. If a user definition is successfully loaded, a confirming message is displayed. The message disappears after approximately 10 seconds.

Loading Your UDT File Remotely

To call the UDT file when operating the instrument remotely, use remote command

DD_LOAD_UDF "filename"

where:

filename = full qualified path to the UDT file, for example, "C:\Lecroy\myudf"

General Rules and Error Messages

All lexical elements should be separated by white space. Lexical elements include keywords and their arguments. White space includes space, tab, carriage return, and line feed. One exception: the double slash //, which starts a comment, need not be followed by white space.

As the file is parsed, if the expected number of arguments for a keyword is not available, the error message "Could not read args for '<keyword>..! abort" will be displayed and the parse is canceled. If the word where we expect a keyword is not recognized as a keyword, the error message "Word starting '<text>': not a keyword! Abort" is displayed and the parse is canceled. Possible causes for this are comment text without a preceding //, or too many arguments for the preceding keyword.

When the end of the file is reached, the DDA verifies that all required keywords were seen in the file. If not, the message "Missing <number> required keywords!" is displayed and the parse is considered canceled. Not all keywords are required; however, we strongly encourage you to use them all, in every file. The result of omitting most optional keywords is to leave previously set values unchanged.

Keywords

keyword:	//
arguments:	none
required?:	no **Notes:** 1. // begins a comment. After // the rest of the line, up to line feed, is ignored. 2. // need not be followed by a space.

keyword:	AGC_GAIN
arguments:	<value, 0 to 0.125>
required?:	no **Notes:** The difference of the normalized (-1 to +1) sample value to the expected sample value is multiplied by this to determine the change in expected level. The DDA treats positive and negative levels separately, to handle asymmetric waveforms. The change in the 0 level is 1/4 of this, a correction to the 0 level shifts all the levels. The DDA also allows very slow changes of relative spacing; it starts analyzing each VCO sync field assuming symmetry and perfect spacing. In the DDA, this value is 0.12 for PR4, EPR4, and E2PR4. If this keyword is not seen, its value remains unchanged. If this value is never set, the default is 0.12.
possible error messages:	"AGC_GAIN must be >= 0 and <= 0.125"

keyword:	KEYWORDS_VERSION
arguments:	1
required?:	yes
possible error messages:	"Keywords version is newer than software rev!" When the keywords file is updated, this message will specify KEYWORDS_VERSION 2. New software will be able to handle versions 1 and 2. Old software will produce this error message if it encounters KEYWORDS_VERSION 2.

keyword:	LEVELS
arguments:	<number from 3 to 32>
required?:	yes **Note:** LEVELS must appear before NORMALIZED_LEVELS are specified
possible error messages:	"#levels must be >= 3 and <= 32!"

keyword:	MAX_LEVEL_ADJ
arguments:	<value from 0.0 to 1.0>
required?:	no **Notes:** This sets a hard limit on the maximum change in an expected level due to steering from one sample. Setting this value to 1.0, or near that value, makes the limit irrelevant. The DDA uses 0.0625 for PR4, 0.03125 for EPR4, and 0.020833 for E2PR4. If this keyword is not seen, its value remains unchanged. If this value is never set, the default is 0.020833.
possible error messages:	"MAX_LEVEL_ADJ must be >= 0 and <= 1.0"

keyword:	MAX_PHASE_ADJ
arguments:	<value 0 to 0.1>
required?:	no **Notes:** This sets a hard limit on the maximum phase change (and the maximum period change) of the emulated PLL due to steering from one sample. The value is a fraction of a bit cell time. The DDA uses 0.06 for PR4 and EPR4, and 0.04 for E2PR4. If this keyword is not seen, its value remains unchanged. If this value is never set, the default is 0.04.
possible error messages:	"MAX_PHASE_ADJ must be >=0 and <=0.1 (bit cells)"

keyword:	NORMALIZED_LEVELS
arguments:	(see note below)
required?:	no **Notes:** 1. This keyword takes LEVELS arguments, each a value from -1 to 1, in order. 2. If this keyword is not supplied, levels default to LEVELS: equally spaced values from -1 to 1. 3. LEVELS must be seen before this keyword.
possible error messages:	"Must set #levels before specifying levels!" "Must set TOP_LEVEL... before specifying levels!" "Normalized levels must be in order, -1 to 1!" "Normalized levels must be -1 to 1!"- arguments didn't include -1 or 1, or both

keyword:	PLL_INTEG_GAIN
arguments:	<value 0 to 1e-10>
required?:	no **Notes:** The value is multiplied by a code difference from the expected level times slope. The result is to produce a period correction as a fraction of bit cell time. The DDA uses 4e-11 for PR4, EPR4 and E2PR4. If this keyword is not seen its value remains unchanged. If it has never been set since power on the default is 4e-11. See notes for PLL_PROP_GAIN above.
possible error messages:	"PLL_INTEG_GAIN must be >= 0 and <= 1e-10"

keyword:	PLL_PROP_GAIN
arguments:	<value 0 to 1e-8>
required?:	no **Notes:** The value is multiplied by a code difference from the expected level times slope normalized to codes/bit cell near the sample. The result is to produce a phase correction as a fraction of a bit cell time. The phase error tends to have a significant random component so this value should be kept low. The DDA uses 5e-9 for PR4, 2e-9 for EPR4 and 1e-9 for E2PR4. If this keyword is not seen, its value remains unchanged. If it has never been set since power on, the default is 1e-9. **Notes about the PLL loop:** We steer our software PLLs phase (proportional) and period (integral) with coefficients phase_steering_gain and spacing_steering_gain. They are used like this: spacing correction = phase error * spacing_steering_gain phase correction = phase error * phase_steering_gain

	where: phase error = distance from sample to level * slope * DSO samples per bit cell **Notes:** 1. Distance is normalized (-7200 = -1 level, +7200 = +1 level) 2. Slope is (normalized codes/DSO sample) around the channel sample position 3. Multiplying by samples per bit cell normalizes for DSO sample rate: slope (codes/DSO sample) is cut in half when sample rate doubles codes/DSO sample * DSO sample/bit cell = codes/bit cell The gain terms must be small enough so that the correction is mostly uncompleted within the latency (STEERING_LATENCY + FIR delay) of the control loop. Otherwise the loop will be unstable. The spacing and phase corrections are in fractions of a bit cell.
possible error messages:	"PLL_PROP_GAIN must be >= 0 and <= 1e-8"

keyword:	STATE
arguments:	<which state> 0 to STATES minus 1 <previous state from 0> 0 to STATES minus 1 (or 100, see below) <expected sample level from 0> -1 to 1 <previous state from 1> 0 to STATES minus 1 (or 100, see below) <expected sample level from 1> -1 to 1
required?:	yes **Notes:** 1. The states are specified in NRZ (nonreturn to zero) format, that is 0 and 1 are the two polarities of magnetization. On the head signal (the first sample of) a negative going pulse begins a "0" polarity region on the media, and a positive going pulse begins a "1" polarity region. "from 0" means the bit cell which is no longer part of the state was 0 polarity. Consider the state 0011. If the preceding state was 0001 then the state 0011 was arrived at along the "from 0" transition. Similarly if the preceding state was 1001 then the state 0011 was arrived at along the "from 1" transition. 2. At least one STATE specification must be seen or the parse is considered failed. 3. In some cases one (or both) of the transitions to a state may not be valid, that is, they are disallowed due to encoding constraints. In such a case use the value 100 as the "previous state from" for that transition, this is recognized as disallowing the transition. Any other value outside of 0 to STATES minus one is declared a parse error. If both transitions to a state are disallowed, the state cannot be arrived at; it is invalid. You need not specify invalid states (states that cannot be arrived at). For proper operation you must specify every state that can be arrived at. 4. The STATES keyword, specifying the number of states, must be seen before a STATE

	can be specified.
possible error messages:	"Must set #states before STATE info!" - see note 4 above. "State <number>?? Expected states 0 to <STATES - 1>!" "State <number>: must be -1 < expected level < 1!"

keyword:	STATES
arguments:	<number from 4 to 16>
required?:	yes **Note:** STATES must appear before a STATE is specified
possible error messages:	"#states must be >= 4 and <= 16!"

keyword:	STEERING_LATENCY
arguments:	<value 0 to 10>
required?:	no **Notes:** The DDA's channel emulation uses decision directed steering. This means that the PLL and AGC are not determined based on the nearest level to each sample. Instead, we wait STEERING_LATENCY samples and then steer based on what the sample should have been according to the Viterbi detector. The DDA uses 6 for PR4, EPR4, and E2PR4. After 6 samples it is far more likely that the detector has already picked the correct path, and we will steer toward the correct target. If this keyword is not seen, its value remains unchanged. If it has never been set since power on, the default is 6. CAUTION: Increasing the steering latency increases the delay for the steering control loops and may make them unstable. For a 16 state signal (such as E2PR4), the detector uses the sample at STEERING_LATENCY into the trellis, and four more preceding it (that is, further into the trellis) to determine the expected value for the sample at STEERING_LATENCY. The difference between the actual sample value and the expected value determines steering.
possible error messages:	"STEERING_LATENCY must be >= 0 and <=10"

keyword:	TOP_LEVEL_AS_FRACTION_OF_PEAK
arguments:	<number, 0.5 to 1.5>
required?:	yes **Notes:** This is the ratio of the nominal "1" sample level to the peak height of the VCO sync pattern. This is used in our emulated channel's acquisition of the proper sampling phase and levels. For example, using a 2T sync pattern, PR4 has levels +1, +1, -1, -1 & and the peaks are significantly higher. For PR4, the DDA uses 0.7854 for this value. For EPR4, a 2T pattern is 0, +1, 0, -1& so the peaks are +1 and -1. For EPR4 the DDA uses 1.0000 for this value. For E2PR4 a 2T pattern is +0.6667, +0.6667, -0.6667, -0.6667, and the peaks are higher,

	slightly above the +1 level. For E2PR4 the DDA uses 1.06045 for this value.
possible error messages:	"top level must be > 0.5 and <= 1.50 x sync peak"

keyword:	TRELLIS_LENGTH
arguments:	<value 11 to 81>
required?:	no **Notes:** Because of internal implementation constraints, the trellis cannot be 28 to 32 long, or 60 to 64 long. An attempt to set it within those ranges will be accepted but will actually set the next larger permitted value, that is, 33 or 65. If this keyword is not seen its value remains unchanged. If this value is never set the default is 81. Longer trellis size does not noticeably slow down processing. Longer trellis size approaches true maximum likelihood to greater certainty. Real disk channels tend to have short trellises and rely on encoding to avoid sequences that would require a longer trellis to resolve. On a good signal all possible paths being maintained in the trellis are the same beyond a few bits in, the vast majority of the time.
possible error messages:	"TRELLIS_LENGTH must be >= 11 and <= 81"

keyword:	ZERO_CROSS_TO_SAMPLE
arguments:	<value 0 to 0.5>
required?:	yes **Notes:** This tells the DDA how the correct sampling times relate to zero crossings during the VCO synchronization preamble. The value is a fraction of the spacing between zero crossings. Example: The VCO synch is normally a 2T pattern. For PR4 the samples are 1, 1, -1, -1; there are two samples between zero crossings, and the first sample after a zero crossing is one quarter of the distance to the next zero crossing. Therefore ZERO_CROSS_TO_SAMPLE is 0.25 for PR4. For EPR4, a 2T pattern gives samples 0,1,0,-1; the first sample is at the zero crossing; therefore, ZERO_CROSS_TO_SAMPLE is 0 for EPR4. For E2PR4, a 2T pattern gives 2/3, 2/3, -2/3, -2/3; there are two samples between zero crossings, and the first sample after a zero crossing is one quarter of the distance to the next zero cross, just as for PR4. Therefore ZERO_CROSS_TO_SAMPLE is 0.25 for E2PR4.
possible error messages:	"ZERO_CROSS_TO_SAMPLE must be >= 0 and <= 0.5"

Example File

An example of a user definition equivalent to E2PR4 (d=0) follows. This file was named E2PR4.UDF:

```
USER DEFINED VITERBI TRELLIS FOR LECROY DDA // a comment can go here
// The above line is required exactly as shown or the file will be rejected!
KEYWORDS_VERSION 1
STATES 16
```

```
// State initializers - one for each valid state
// state prev from 0 level prev from 1 level
STATE 0 0 0 8 -0.33333
STATE 1 0  0.33333 8 0
STATE 2 1  0.66667 9 0.33333
STATE 3 1 1 9 0.66667
STATE 4 2 0 10 -0.33333
STATE 5 2  0.33333 10  0
STATE 6 3  0.66667 11  0.33333
STATE  7 3 1 11  0.66667
STATE  8 4  -0.66667 12  -1
STATE  9 4  -0.33333 12  -0.66667
STATE 10 5  0 13  -0.33333
STATE  11 5  0.33333 13  0
STATE 12 6  -0.66667 14  -1
STATE 13 6  -0.33333 14  -0.66667
STATE 14 7  0 15  -0.33333
STATE 15 7  0.33333 15  0
LEVELS 7
TOP_LEVEL_AS_FRACTION_OF_PEAK 1.06045 // nominal 1 / peak height in vco sync field.
For 2T
NORMALIZED_LEVELS -1 -.66667 -.33333 0 .33333 .66667 1
PLL_PROP_GAIN 1e-9
PLL_INTEG_GAIN 4e-11
MAX_PHASE_ADJ 0.04
STEERING_LATENCY 6
TRELLIS_LENGTH 81
AGC_GAIN 0.12
MAX_LEVEL_ADJ 20.833e-3
ZERO_CROSS_TO_SAMPLE 0.25  // Fraction of zero cross spacing
// End of file
```

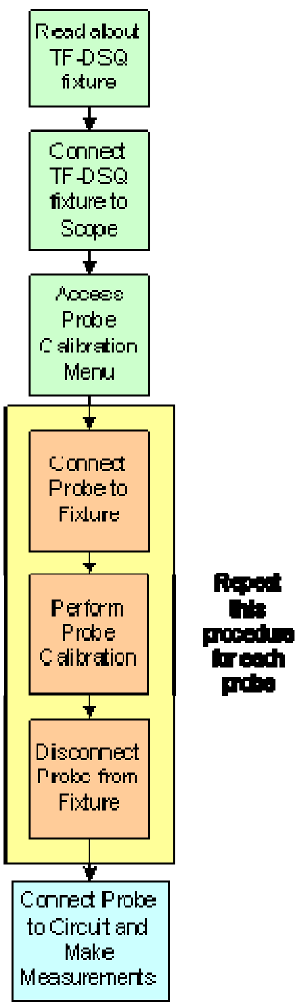
TF-DSQ Overview

Probe Calibration with the TF-DSQ Fixture

The TF-DSQ fixture is used in conjunction with the scope software to perform probe deskew and DC calibration. It has the following leading features and specifications:

- Deskew to +/- 20 ps typical accuracy
- Differential and single-ended drive
- 75 ps edge for precise deskewing
- Calibration of DC gain and offset and skew at same probing point
- Accounts for risetime variations
- Accounts for common-mode voltage
- DC gain calibration accounts for probe loading effects
- Integrated operation with scope for fully automatic calibration

	WavePro Zi	WaveMaster	SDA	DDA
Oscilloscope Compatibility	715, 725, 735, 740, and 760 models.	8300, 8300A, 8500, 8500A, 8600A, and 8620A including all WaveMaster XXL models.	SDA 3000, 3000A, 5000, 5000A, 6000, 6000A, and 6020 including all SDA XXL and 700Zi models.	DDA 5005A, 5005A XXL, and DDA 700Zi
Oscilloscope Connection	ProLink			
Probes Supported	D610,D620, D600A-AT, D310, D320, and D300A-ATwith all probing accessories AP020, AP033, AP034 HFP1000, 1500, 2500, and 3500 PP005 and PP005A			
DC Range	±5 V single-ended, ±10 V differential			
DC Accuracy	±(1% + 600 μV)			
Edge Risetime	75 ps (typical) < 95 ps (guaranteed)			
Edge Amplitude and Rep Rate	Approximately 800 mV @ 10 MHz			
Deskew Accuracy	±20 ps (typical)			



Probe calibration is accomplished with the TF-DSQ fixture by following the basic steps in the flowchart at left. It is recommended that you read the instructions presented here in their entirety to familiarize yourself with the advanced features of the TF-DSQ fixture.

The combination of TF-DSQ fixture and scope software is designed with an important use model. Connecting probes to the circuit under test can be a difficult procedure. LeCroy's system, therefore, is designed in a manner that allows you to set up the probe calibration fixture, calibrate each individual probe once, connect your probes to the circuit, and disconnect the fixture. Once your probes are in the circuit, there is no need to revisit the fixture until the next calibration interval.

You should familiarize yourself with the following topics:

TF-DSQ fixture overview

TF-DSQ Scope connection

Probe Calibration Menu access

Probe connection to fixture

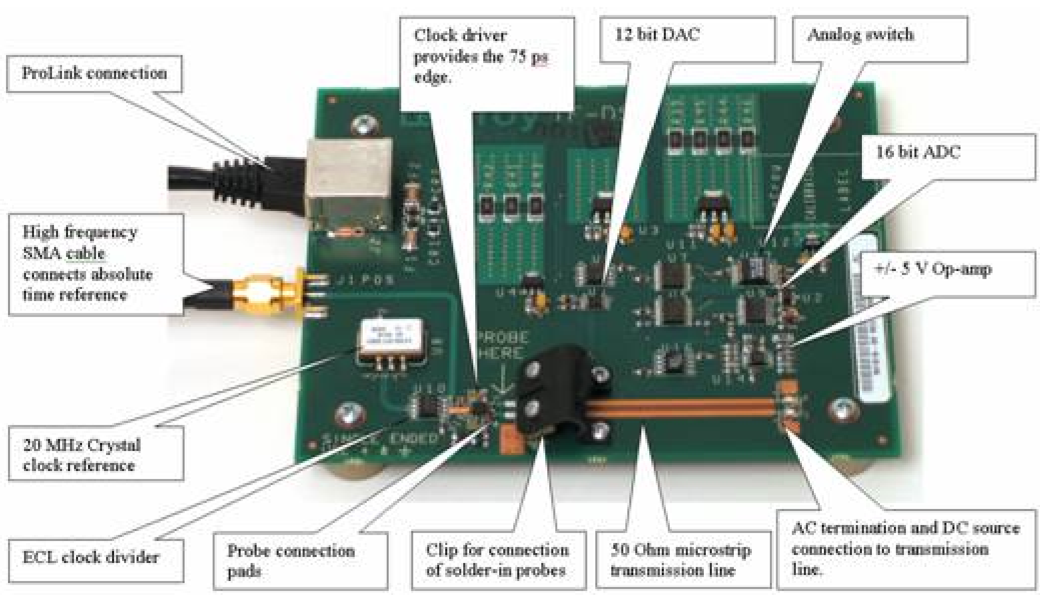
Calibrating probes

TF-DSQ Fixture Overview

The TF-DSQ fixture comes in a soft case containing the following components:

- The TF-DSQ fixture
- A 50-ohm cable
- A ProLink extender
- A CD containing PDF files of manuals for several scope options, including the TF-DSQ fixture

TF-DSQ Components



TF-DSQ Fixture Elements

Assembling the TF-DSQ fixture

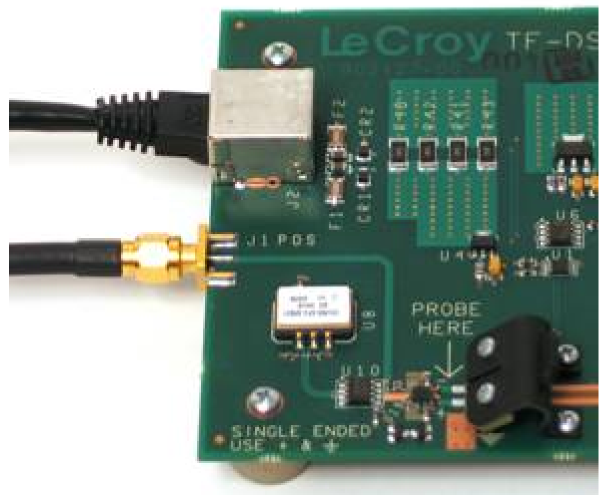
See the fixture overview to identify individual components.

System assembly is accomplished in the following steps:

1. Connect one end of the 50-ohm cable to the ProLink extender (this comes normally connected)
2. Connect the other end of the 50-ohm cable to the SMA connector on the TF-DSQ fixture.
3. Connect the ProLink cable from the extender to the TF-DSQ fixture.

The 50-ohm connection should be torqued with an RF torque wrench and must be properly tightened.

ProLink Extender with ProLink Cable and SMA



TF-DSQ Scope Connection

The TF-DSQ fixture is connected to either an unused scope channel or the external ("Aux In" on WaveMaster scopes) input, if one exists. In other words, any scope channel or scope input with a ProLink connector. The TF-DSQ fixture can be used only on scopes with blind mating adaptor (BMA) inputs.

Scope Auxiliary Input with ProLink and BMA Connector

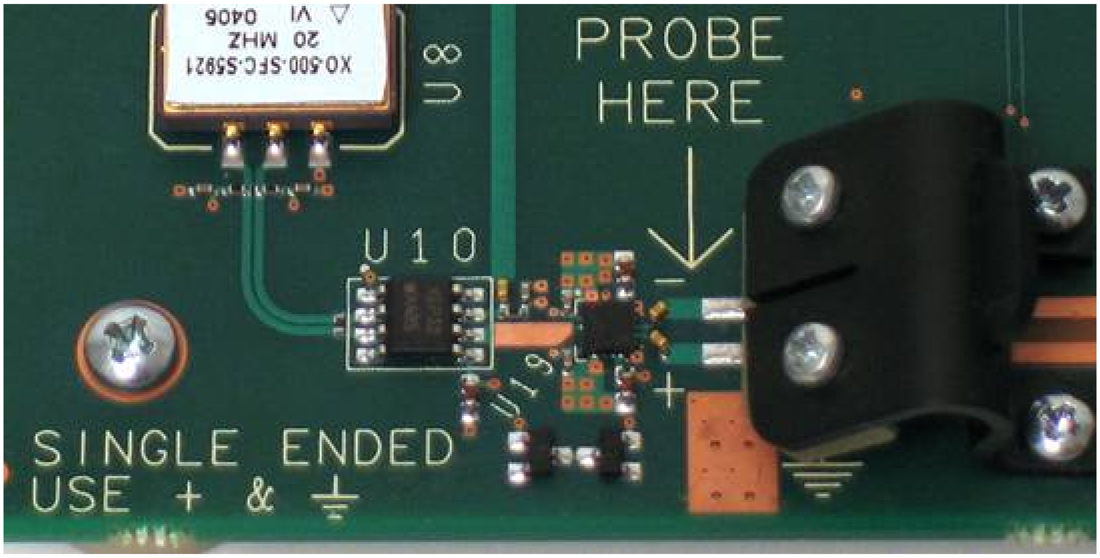
Connection of the TF-DSQ Fixture to the Scope Input

We recommend connecting to the external trigger input on a WaveMaster scope, or to an unused scope channel in the case of an Serial Data Analyzer with the "A" model suffix (these scopes replace the external trigger input with a clock recovery module).

To verify proper operation, you will see the TF-DSQ fixture called out as the probe calibration source in the probe calibration menu.

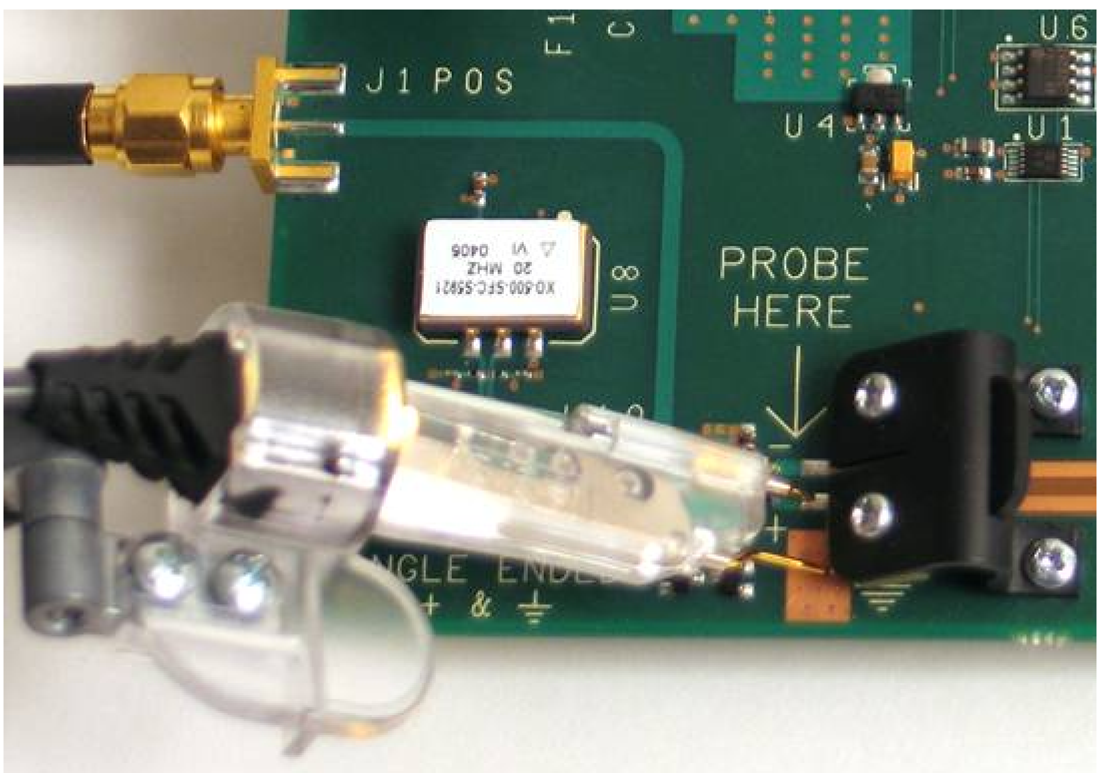
Probe Connection to TF-DSQ

The TF-DSQ provides multiple probe connectors for various kinds of probes. Probes are connected electrically in either a single-ended or differential arrangement, depending on the type of probe. Probes are connected mechanically using either the probing pads, or a probing clip provided for solder-in probing solutions.

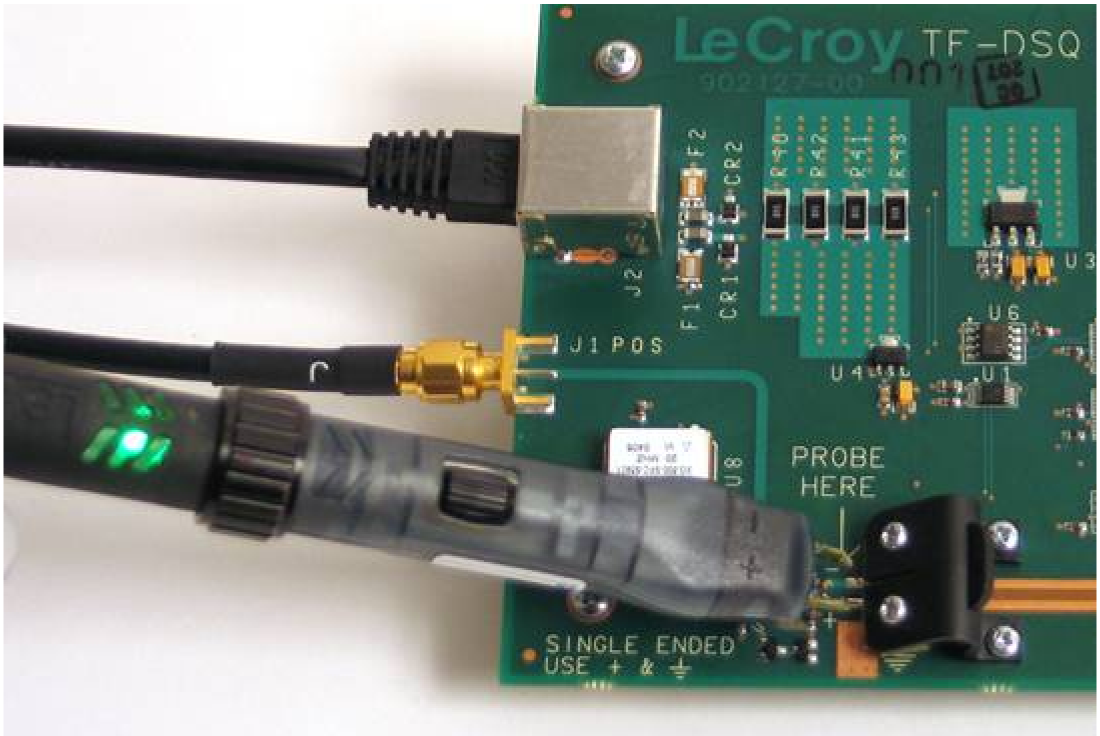


Probe Connection Points and the Clip

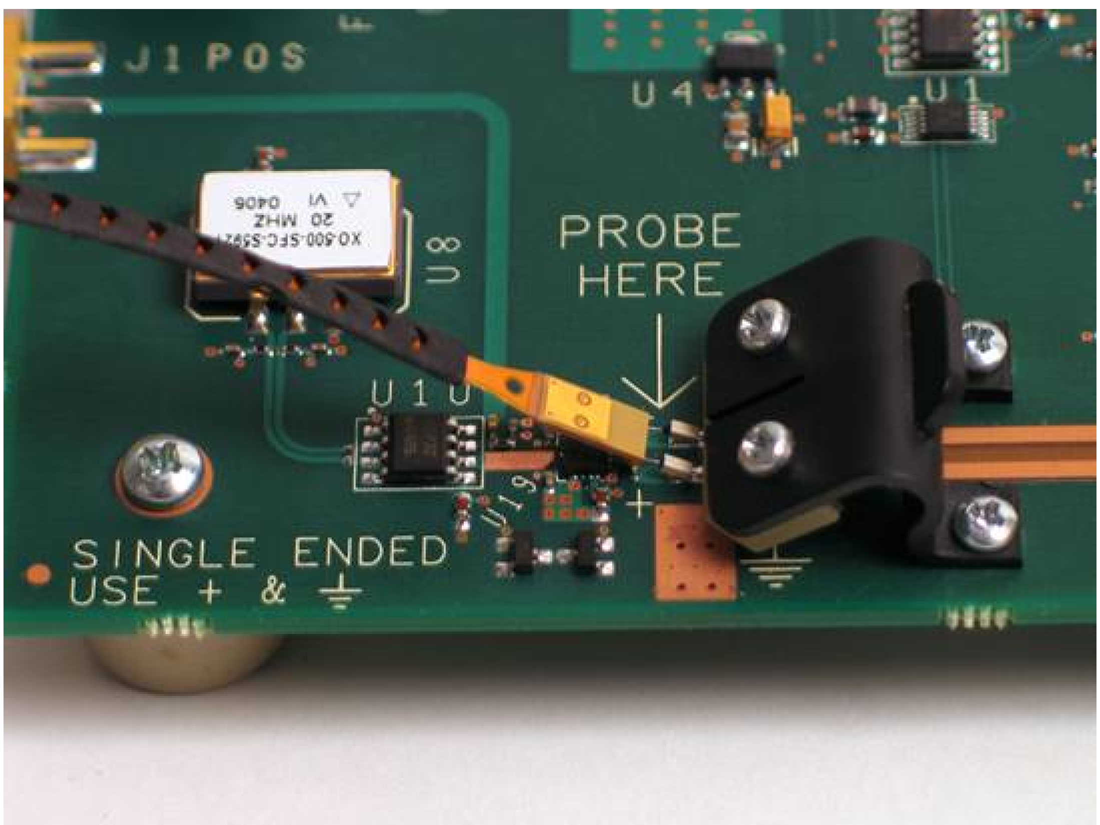
Differential probes are connected with the tip designated V+ mated to the "+" pad on the fixture, and the tip designated V- to the "-" pad on the fixture. Single-ended probes are connected with the probe tip connected to the V+ pad only, and the ground lead optionally connected to ground. Solder-in probes have their appropriate tips held down to the microstrip transmission line by the clip. Simply push down on the clip with your thumb, insert the probe connection leads under the clip and release. Make sure that V+ and V- are connected properly.



Single-ended Probe Properly Connected to the Fixture



Differential Probe Properly Connected to the Fixture (Browsing Configuration)

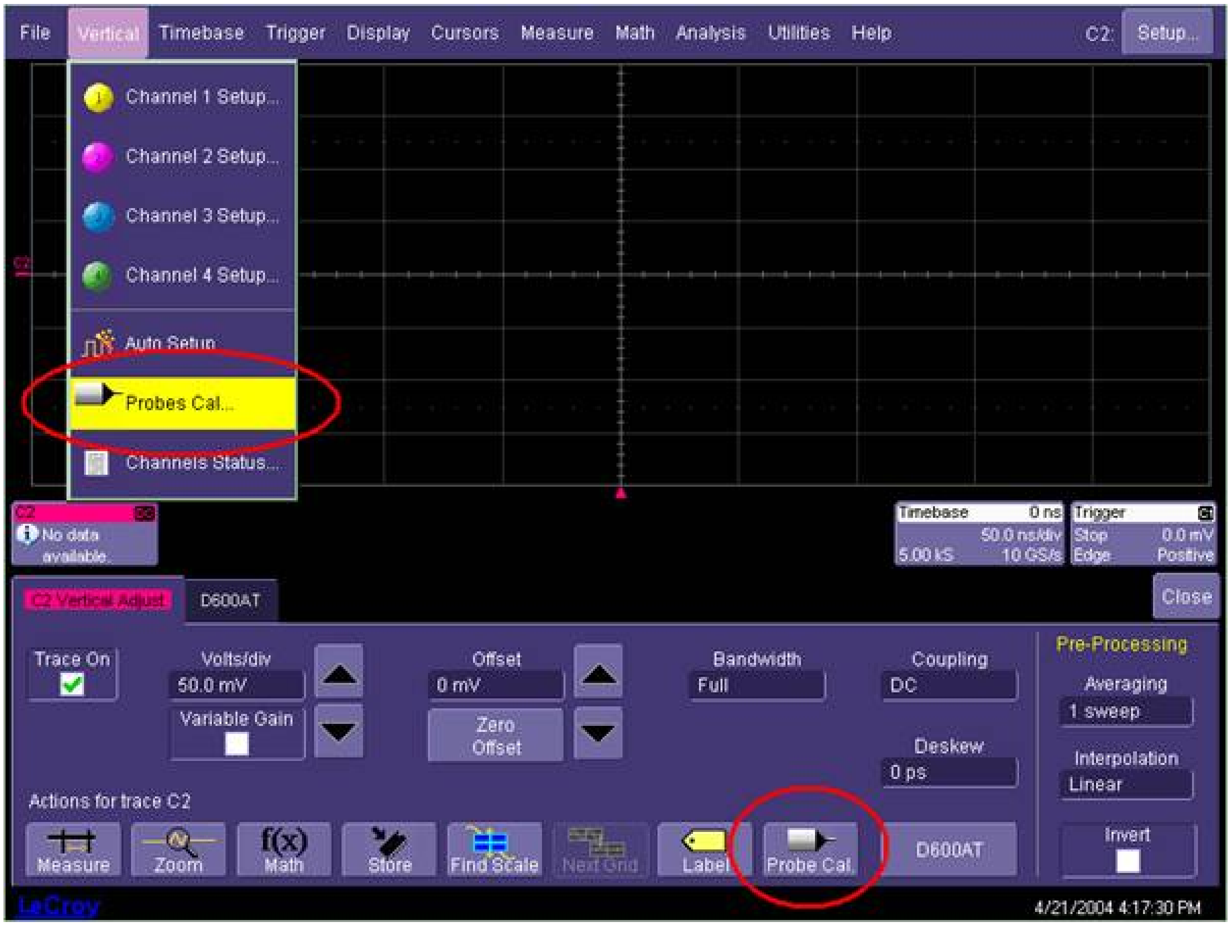


Differential Probe Properly Connected to the Fixture (Solder-in Configuration)

Probe Calibration Menu

Accessing the Probe Calibration Menu

The probe calibration menu can be accessed from the **Vertical** drop-down menu or from the channel "Vertical Adjust" dialog:



Probe Calibration Menu Description

Probe Calibration Information and Controls

The information in the probe calibration menu is organized such that each row represents the information for a given channel, and each column represents the calibration information or control for that channel. For each channel, the information and control provided includes:

- The channel number in the colored button icon and the probe type that is installed
- A **Full Calibration** button, which starts the calibration
- DC correction information including both gain and offset correction
- The skew correction
- A Clear button

Probe Channel and Type Identification

This area shows the type of probe connected to the channel. All other information shown in a given row is associated with that probe.

Full Calibration Button

This button causes the scope to automatically perform a full DC and deskew calibration. See details of probe DC calibration or probe deskew calibration.

Probe DC Information

Shows the gain and offset applied to the probe. If the probe measures a voltage of V, the new, calibrated voltage is:

$$V_{calibrated} = V \cdot Gain + Offset$$

Note that the offset is in Volts, and the gain is unitless.

The probe DC calibration information can be entered either manually or as the result of an automatic calibration. In the case of automatic calibration, it can be part of the full calibration or it can be a standalone DC calibration executed in advanced mode. When the DC calibration information is a result of an automatic calibration utilizing the TF-DSQ fixture, the information shown is the gain and offset utilized for the currently configured channel sensitivity (volt/division setting; see details of DC calibration). In this case, when the channel sensitivity is altered, you will notice that these values will change. **When the DC calibration information is entered manually, it will clear any automatic results and replace them globally with the newly entered values. This means that if new gain and offset numbers are entered manually, these values will apply across all sensitivity settings of the scope.**

The gain is limited to between 0.8 and 1.2, but the offset is not limited.

> **Note:** It is important to note that some passive probes, and any user-designed probes, do not provide proper probe identification information to the scope. In these cases, the scope may not be able to determine the proper attenuation values. In this situation, you should make sure that the proper attenuation is entered in the channel "Vertical" setup dialog. Furthermore, the gain entered should be the gain correction applied to the system with the correctly entered attenuation.

Probe Deskew Information

The probe deskew information contains the measured skew between the probe in the specified channel and the reference channel. It can be entered manually or as the result of an automatic calibration. In the case of automatic calibration, it can be the result of a portion of the **Error! Hyperlink reference not valid.** or it can be the result of a standalone deskew calibration. Even after the deskew has been performed automatically, the deskew correction can be tweaked manually.

Clear

All probe calibrations can be cleared by pressing this button corresponding to a probe.

Probe Calibration Source

This specifies the signal source used for DC and skew calibrations. When the TF-DSQ fixture is plugged into a scope input, the scope will automatically specify this fixture as the calibration source.

Skew Reference

This specifies the channel or external input where the skew reference is supplied. The skew reference is the absolute time reference to which all deskew measurements are made. When the TF-DSQ fixture is plugged into a scope input, the scope will detect this and automatically show the front panel connection of the fixture.

Recall Calibration

Whenever a probe calibration is applied, the scope saves the information in a file on the disk. If the scope must be rebooted for any reason, the probe calibration information is always cleared, but can be manually recalled by pressing this button.

Advanced Mode Checkbox

When the **Advanced Mode** checkbox is unchecked, you have access to the basic probe calibration menu. The basic probe calibration menu shows you only what is absolutely needed to perform a simple calibration of the probes. In other words, it shows you the calibration information and provides the capability to calibrate the probe with a single button press, clear the calibration information, and manually reload the calibration information following a scope reboot. When the advanced mode button is checked, you have access to the advanced mode probe calibration menu.

Basic Probe Calibration

The TF-DSQ fixture is used to calibrate probes. Prior to beginning your measurements

- Assemble the TF-DSQ fixture
- Attach the fixture, ideally to the Auxiliary Input or an unused scope channel
- Access the Probe Calibration Menu

Follow the instructions for each probe used:

- Connect the probe to the scope channel
- Attach the probe to the TF-DSQ fixture
- Press the **Full Calibration** button in the Probe Calibration Menu.
- Wait a few seconds as the probe is calibrated (calibration wizard closes at end of calibration).
- When the calibration completes, remove the probe from the TF-DSQ fixture

Now you are ready to probe the circuit and perform your measurements. If power is interrupted during your measurements, reboot the scope and manually recall your settings.

Advanced Mode Probe Calibration Menu

The advanced mode is entered by checking the advanced mode box at bottom-right of the basic probe calibration dialog:

Checking this box allows:

- Calibration of gain/offset only
- Calibration of deskew only
- Access to the advanced menu (shown as a tab behind the "Probes Cal" dialog)

Gain/Offset Only Calibration

Pressing this button performs only the DC calibration of the probe on the specified channel. See details of Probe DC Calibration.

Deskew Only

Pressing this button performs only the deskew calibration of the probe on the specified channel. See details Probe Deskew Calibration.

The Advanced Menu

The Advanced Menu contains information and functionality useful to the advanced user of the TF-DSQ fixture. These include:

- Risetime Skew Correction
- Deskew All (or common deskew capability)
- Common mode voltage settings for DC calibration

Rise Time Skew Correction

This field shows the signal risetime and the corresponding skew correction based on the signal risetime.

When probes are deskewed, the risetime measurement of the edge used for deskewing is displayed in the **Rise Time** field corresponding to the probe and probe channel, and an additional skew correction of zero is applied.

The measured risetime of the signals encountered can be entered into the **Rise Time** field, and the scope will automatically calculate and apply a new skew correction value to be utilized in addition to the deskew amount calculated during the deskew calibration procedure. With this use, a finer deskew calibration is performed because the risetimes of the signals measured are accounted for. See details of risetime correction.

Deskew All (or Common Skew)

This is the deskew amount applied to all channels. The time entered in this field is the absolute time by which all waveforms displayed by the scope are delayed in time. This value effectively adjusts the zero time reference of the system. See Probe Deskew Calibration for details.

Common Mode Voltage Selection

The TF-DSQ fixture will calibrate probes differentially or in single-ended mode depending on the type of probe. Differential probes allow the common mode voltage component to be applied during the DC calibration for improved calibration accuracy in situations where probe gain or offset correction depends on common mode components. See Probe DC Calibration or Differential and Single-ended Probe Basics for details.

Advanced Probe Calibration

When the **Advanced Mode** checkbox is checked, you can perform the DC calibration and the deskew calibration separately by pressing **Gain/Offset Only** or **Deskew Only**.

When performing DC calibration, you have the option to apply a common mode component to the differential DC levels applied to the probe during calibration. See Probe DC Calibration or Differential and Single-ended Probe Basics for details..

After performing the deskew calibration, you have the option to apply a common skew value to all channels to adjust the zero time reference of the system.

If you know the risetime of the signals being measured, you can enter the measured risetime of the signals in the **Rise Time** field to obtain a further skew correction that accounts for the risetime. If the risetime entered is less than the risetime measured during the calibration, no correction is applied; otherwise, the system will calculate a correction to account for the signal risetime. It is important to enter the measured risetime. That is the risetime of the signal that the scope measures (or will measure). See details of risetime correction.

Deskew Theory of Operation

Deskewing is an adjustment of the times of waveform data points on the screen. Deskewing is an operation to correct the times that waveforms are displayed on the screen, mainly to account for propagation delays through probes and cables.

When considering skew, there are two important things to consider:

- The relative skew between two channels
- The absolute skew from the zero time reference (i.e., the trigger point)

Two channels are properly deskewed relative to each other when the difference between the deskew values entered for each channel aligns an edge occurring at the same time and applied to both channels. A channel is properly deskewed in an absolute sense when an edge is applied to that channel, the scope is triggered on that edge, and that edge appears such that the trigger threshold crossing appears at the trigger delay, which is the zero time reference on the scope screen.

Probes are deskewed one at a time relative to a reference such that the resulting calibration will deskew each probe relative to every other probe.

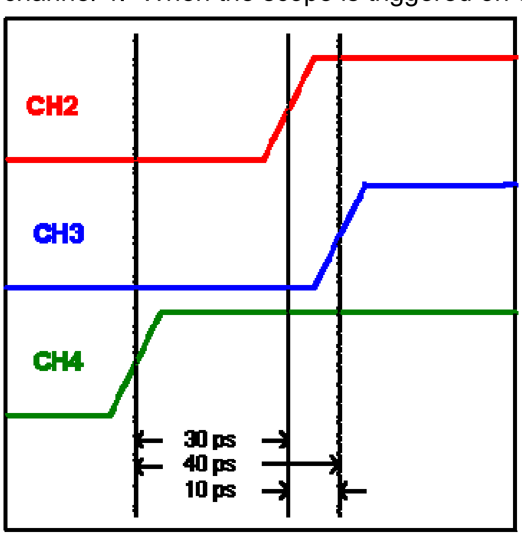
The TF-DSQ fixture has a built-in step generator. The edge generated is driven out of a cable into a scope channel or external input where the fixture is connected. The edge is simultaneously driven onto a transmission line containing the probing pads. When a probe is connected to the probing pads, the edge appears at the fixture connection to the scope and on the channel where the probe is connected. Generally, there is a difference in time between the edges due to propagation delay variations between the fixture cabling and the probe.

During the deskew procedure, the scope triggers on the channel or external input used for the fixture connection and the time from this trigger to the edge on the probe channel being deskewed is calculated. This time becomes the skew time for the probe. This process is repeated for all of the other probes being utilized.

During scope operation, there is a small dilemma. Each probe that is deskewed causes the waveform to be delayed or advanced due to the deskew time calculated for the channel. This causes the trigger to be misaligned because the trigger point is where the edge reaches the internal scope trigger circuitry, not when the edge appears at the probe tips. The scope accounts for this by subtracting the deskew correction for the channel used as the trigger source from the deskew correction on all channels. In essence, the channel used as the trigger source has zero deskew correction applied, while the relative deskew difference between all channels is maintained. This trigger compensation is hidden from the user. If better trigger alignment is desired or if there is some need to shift all of the waveforms, the **Deskew All** value in the "Advanced" menu is utilized to shift all traces together.

A simple example illustrates this:

Two probes are used in a system. They are connected to channels 2 and 3. The TF-DSQ fixture is connected to channel 4. When the scope is triggered on channel 4, you observe the following edges:



When the probes are deskewed, the relative time between channel 4 (where the TF-DSQ fixture is connected) and channel 2 is calculated as 30 ps, and **-30** ps is entered in channel two's skew entry. Similarly, the time between channel 4 and channel 3 is calculated as 40 ps and **-40** ps is entered in channel 4's skew entry. When triggering on channel 4, channel 2 is advanced 30 ps in time and channel 3 is advanced 40 ps in time such that all of the edges are aligned with each other and with the trigger point.

When triggering on channel 2, these times are adjusted by a common 30 ps. So, 30 ps is added to channel 2 to cause 0 deskew amount, and 30 ps is added to channel 4 to cause -10 ps deskew. Without this common addition (or delay) of all waveforms by 30 ps, all edges would remain aligned, but channel 2's trigger position would be off by 30 ps. So, you can see that this 30 ps commonly, added to all waveforms, keeps the trigger points aligned.

Deskew Risetime Adjustment Theory

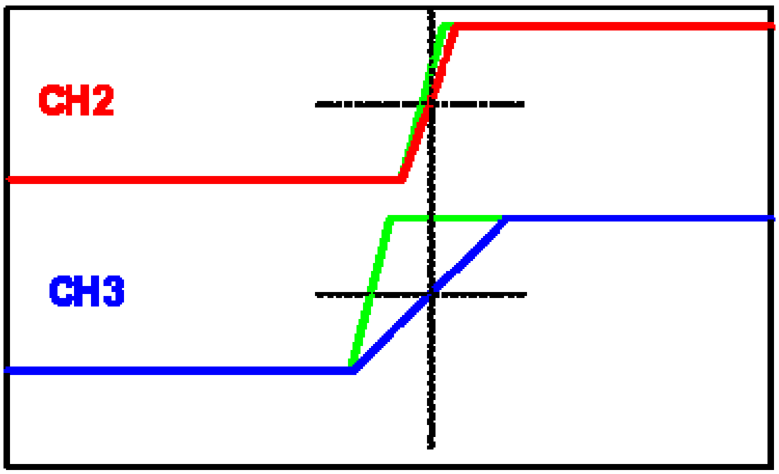
There are two situations that require adjustment of the deskew values to account for risetime:

- Two probes are used for relative time measurements, but each *probe* has a different risetime.
- Two probes are used for relative time measurements, but each *signal* has a different risetime.

The first case will be explained with obvious analogy to the second case.

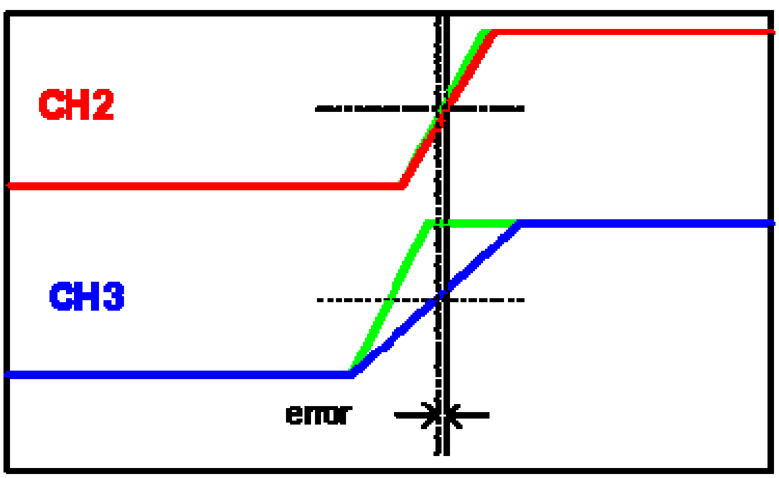
Often, two probes are utilized for measurements where each probe has a different risetime. This might occur because you are making a measurement where a reasonably low bandwidth probe can be used, but you do not have two of them, so you substitute a higher speed probe.

When the probes are deskewed, the high-speed edge from the TF-DSQ fixture is utilized. The deskewed probes might show signals that look as follows when the high-speed edge supplied by the TF-DSQ fixture is observed:



In the above picture, the green trace edge is the edge actually supplied by the TF-DSQ fixture, but because of risetime limitations of the probe and channel, the edges appear as the red and blue traces. These traces are shown aligned at the 50% crossing point at the end of the deskew calibration.

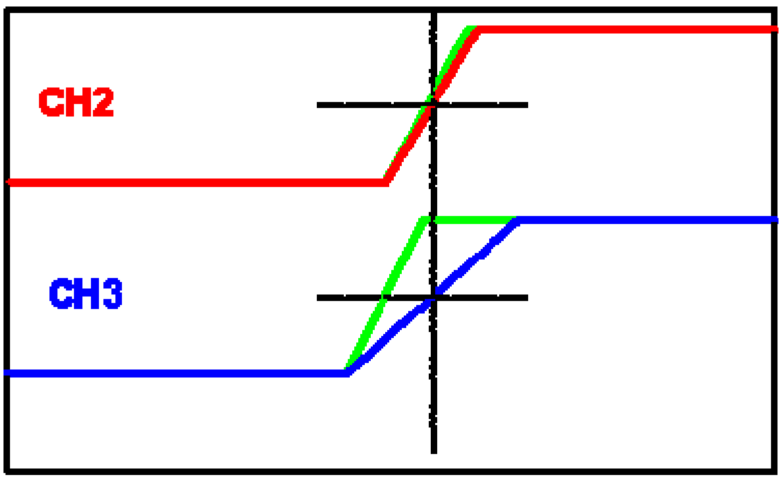
Now, lets assume that these probes are applied to signals with the same risetime, but different from the risetime applied by the deskew fixture:



In this example, you can see that the application of signals with the same risetime, but different from that applied by the deskew fixture produces a deskew error, unless compensated. In the above figure, the green trace is the actual signal applied to the probes that has a risetime of twice that employed by the fixture. The red and blue traces represent the trace acquired through the probes and channel due to this edge. Because of the different risetime applied, there is a small error.

A possible solution to this would be to vary the risetime of the edge applied to the probe by the fixture, a very difficult design. But even despite this difficulty, with a variable risetime solution, you would be required to know ahead of time what the measured risetimes would be, or to measure the risetimes first.

The TF-DSQ fixture, in accordance with the philosophy of requiring only one calibration in the fixture, handles this in a special manner. The user simply enters the measured risetime of the signals after the probe is connected to the circuit. Since the scope software saves the edge acquired during the deskew calibration process, it applies this saved edge to a variable filter using digital signal processing until the measured risetime is arrived at. At that point, the software calculates the difference in the time of the 50% crossing and calculates an additional skew correction to be applied. In this manner, the risetime is compensated for in the deskew calibration without the requirement of recalibration.



The above figure shows the traces realigned as a result of the skew correction applied after the measured risetime has been entered.

DC Calibration Theory

DC calibration involves the calculation of two constants to be applied to waveforms to correct for voltage measurement inaccuracy. The two constants are the gain, applied multiplicatively, and the offset, applied additively. It is important to distinguish the gain and offset correction from the channel gain determined by the sensitivity control (volts/division selection) or the offset control. The sensitivity and offset controls change the absolute gain and offset of the front end amplifier, but cannot correct for inaccuracy. This is true also with the vertical gain and offset controls in zooms. The way to visualize this is to place a cursor at a point on a waveform and read the voltage. Adjusting volts/div or offset, or adjusting the gain or offset of a zoom, will affect the size of the waveform on the screen, but will not affect the voltage measured at the cursor position. The gain and offset correction applied during DC calibration will affect the voltage measured according to the following formula:

$$V_{calibrated} = V \cdot Gain + Offset$$

where V is the voltage measured prior to calibration.

Probes are calibrated for each fixed gain setting of the scope, meaning they are calibrated at 10, 20, 50, 100, 200, 500 mV and 1V per division. A unique gain and offset calculation is made for each range.

The calibration of the probe is performed utilizing 5 DC levels. The DC levels are applied such that the voltages ideally appear on the scope screen at -3, -1.5, 0, 1.5 and 3 vertical divisions. The best fit line is calculated, and the appropriate gain and offset that would make the line fit the actual voltages applied is also calculated. The gain and offset for each range is the gain and offset correction displayed in the gain and offset fields.

In all cases, the DC levels applied to the probe are measured by an ADC on the fixture placed near the probing points. In this way, the absolute voltage at the probe tips are known precisely and any DC probe loading effects are accounted for.

In the case of single-ended probes, the DC levels applied to the V+ probing pad are the same as the voltages that appear at the appropriate grid locations on the scope screen. Differential probes are handled slightly differently. In their case, the voltage applied to the V+ tip is the voltage specified in the Common mode voltage field, plus half the voltage desired on the scope screen. The voltage applied to the V- tip is the common mode voltage minus half the voltage desired. In this way, the probe experiences the specified common mode voltage, and the differential voltage measured by the probe is calibrated for common mode voltage effects.

Differential and Single-ended Probe Basics

Differential and single-ended probe discussions are a sometimes confusing subject. There are aspects of operation that must be known in order to understand their calibration.

A single-ended probe exposes a probing tip and a ground connection lead. Typically, the ground lead is connected to the outer conductor in a coaxial cable, which is connected directly to the scope's ground. The probing point is typically connected to the center conductor. Typically, the ground connection represents an essentially zero-ohm connection to scope ground, and the probe tip is a specified impedance to that ground. Other than probes, a 50-ohm cable is often used for single-ended measurements. In this case, the outer conductor connects the scope's ground to the circuit's ground, and the cable is terminated with 50 ohms at the scope, such that the conductor looks like 50 ohms looking into it at all frequencies.

In the case of single-ended measurements, the scope is measuring the difference between the probe tip and ground. Since ground is considered to be zero volts, one can say that the voltage measured at the probe tip is the absolute voltage.

A differential probe, in contrast, exposes two probing tips with an optional ground connection lead. Often, the ground connection lead is left unconnected (more on this later). Each of the probe tips are connected to two different signals in a circuit. The probe measures only the difference between the two probing points with no actual notion of the absolute voltages present. In practice, differential probes have limitations not only on the difference allowed between the two probing tips, but also the absolute voltage allowed. This absolute voltage is referenced to the scope's ground. For this reason, the probe ground lead is sometimes connected to ground in the circuit to make scope ground and circuit ground the same. This is often done only when the circuit is floating, which means that the circuit's ground is free to move to any voltage, depending on its ground connections.

Despite the fact that the differential probe only measures the difference between the voltages at its probe tips, its accuracy is sometimes affected by the absolute voltages present.

Differential signaling is used commonly for high-speed signals. In a differential system, two wires are used to transmit the signal. Often the signals are the direct opposite of each other, with both swinging across zero volts. Frequently, these signals have a common offset or bias applied to them. In this case, the common offset applied to each signal is called the common mode signal component, and the difference between each signal is called the differential mode signal component. Typically, the differential signal is the actual information signal being transmitted, with the common mode signal being present for other physical reasons, such as the biasing of an ECL gate. An ideal differential probe receives only the differential mode signal. Practical probes reject the common mode signal to a large extent. The ability of a differential probe to reject the common mode signal is stipulated by the common mode rejection ratio (CMRR).

Despite the fact that differential probes measure differentially, they can be used to measure common mode signals. In this case, the V- tip is connected to ground, and the probe continues to measure the difference between the tips. This is a useful configuration for identifying ground problems.

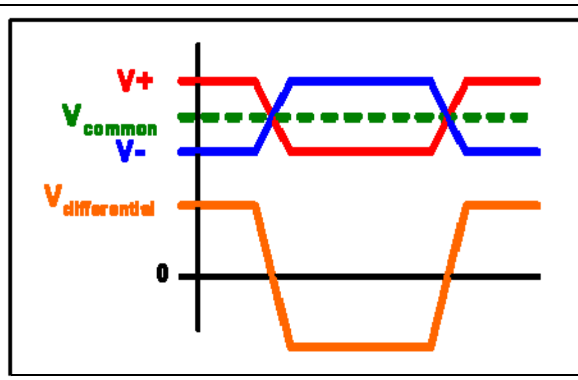
The voltages in differential systems can be described by the following diagram and equations:

$$V_{differential} = V_{+} - V_{-}$$

$$V_{common} = \frac{V_{+} + V_{-}}{2}$$

$$V_{+} = V_{common} + \frac{V_{differential}}{2}$$

$$V_{-} = V_{common} - \frac{V_{differential}}{2}$$



WavePro Specifications

Note: Specifications are subject to change without notice.

Vertical System	WavePro 715Zi	SDA, WavePro 725Zi	SDA, DDA, WavePro 735Zi	SDA, WavePro 740Zi	SDA, DDA, WavePro 760Zi
Analog (ProLink Input) Bandwidth @ 50Ω (-3dB) (≥ 10 mV/div)	NA	NA	NA	4 GHz (≥ 10mV/div)	6 GHz (≥ 10mV/div)
Analog (ProBus Input) Bandwidth @ 50Ω (-3dB)	1.5 GHz (≥ 10mV/div)	2.5 GHz (≥ 10mV/div)	3.5 GHz (≥ 10mV/div)	3.5 GHz (≥ 10mV/div)	3.5 GHz (≥ 10mV/div)
Analog (ProBus Input) Bandwidth @ 1 MΩ (-3dB)	500 MHz (typical)	500 MHz (typical)	500 MHz (typical)	500 MHz (typical)	500 MHz (typical)
Rise Time (Typical)	235 ps	150 ps	120 ps	105 ps	70 ps
Input Channels	4				
Bandwidth Limiters	20 MHz, 200 MHz, 1 GHz		20 MHz, 200 MHz, 1 GHz, 3 GHz		20 MHz, 200 MHz, 1 GHz, 3 GHz, 4 GHz
Input Impedance	50 Ω ±2% or 1 MΩ || 16 pF, 10 MΩ || 11 pF with supplied probe				
Input Cooling	1 MΩ: AC, DC, GND; 50 Ω: DC, GND				
Maximum Input Voltage	50 Ω: ±5 V_{rms} 1 MΩ: 250 V max, (peak AC: ≤ 10 kHz + DC)			50 Ω (ProBus): ±5 Vrms 50 Ω (ProLink): ±4 Vpeak 1 MΩ (ProBus): 250 V max (peak AC: ≤ 10 kHz + DC)	
Vertical Resolution	8 bits; up to 11 bits with enhanced resolution (ERES)				
Sensitivity	50 Ω: 2 mV-1 V/div, fully variable (2-9.99 mV/div via zoom); 1 MΩ: 2 mV-10 V/div, fully variable				
DC Gain Accuracy	±1.5% of full scale				
Offset Range	50 Ω (ProBus Input): ±750 mV @ 10–170 mV/div ±4 V @ 172 mV/div–1 V/div 1 MΩ: (ProBus Input): ±1 V @ 2–128 mV/div ±10 V @ 130 mV–1.28 V/div ±100 V @ 1.3 V–10 V/div			50 Ω (ProLink Input): ±750 mV @ 10–118 mV/div ±4 V @ 120 mV/div–1 V/div 50 Ω (ProBus Input): ±750 mV @ 10–170 mV/div ±4 V @ 172 mV/div–1 V/div 1 MΩ: (ProBus Input): ±1 V @ 2–128 mV/div ±10 V @ 130 mV–1.28 V/div ±100 V @ 1.3 V–10 V/div	
Offset Accuracy	±(1.5% of full scale +1.0% of offset value +1 mV)				

Horizontal System	WavePro 715Zi	SDA, WavePro 725Zi	SDA, DDA, WavePro 735Zi	SDA, WavePro 740Zi	SDA, DDA, WavePro 760Zi
Timebases	Internal timebase common to 4 input channels; an external clock may be applied at the auxiliary input				
Time/Division Range	Real time: 20 ps/div-1000 s/div (RIS mode: 20 ps/div-10 ns/div; Roll mode: up to 1000 s/div)				
Clock Accuracy	≤ 1 ppm+ (aging of 0.5 ppm/yr from last calibration)				
Time Interval Accuracy	< 0.06 / SR + (clock accuracy* Reading) (rms)				
Jitter Noise Floor	1.5 ps (Typical)	1 ps (Typical)	800 fs (Typical)	750 fs (Typical)	560 fs (Typical)
Trigger and Interpolator Jitter	3 ps_{rms}(Typical)	2 ps_{rms} (Typical)		1 ps_{rms} (Typical)	
Channel-Channel Deskew Range	±9 x time/div. setting, 100 ms max., each channel				

Horizontal System	WavePro 715Zi	SDA, WavePro 725Zi	SDA, DDA, WavePro 735Zi	SDA, WavePro 740Zi	SDA, DDA, WavePro 760Zi
External Timebaase Reference (Input)	10 MHz; 50 Ω impedance, applied at the rear input				
Extrenal Timebase Reference (Output)	10 MHz; 50 Ω impedance, applied at the rear output				
External Clock	0.1 Hz-100 MHz, 50 Ω or 1 MΩ impedance, applied at the auxiliary input				

Acquisition System	WP715Zi	WP725Zi	WP735Zi	WP740Zi	WP760Zi
Single-Shot Sample Rate/Ch	20 GS/s on 2 Ch 10 GS/s on 4 Ch (Option WPZi-1.5GHz-4X20GS doubles the sample rate)	40 GS/s on 2 Ch 20 GS/s on 4 Ch			
Random Interleaved Sampling (RIS)	200 GS/s for repetitive signals (20 ps/div, to 10 ns/div)				
Maximum Trigger Rate	1,000,000 waveforms/second (in Sequence Mode, up to 4 channels)				
Intersegment Time	1 µs				
Maximum Acquisition Memory Points/Ch	**(4 Ch / 2 Ch)**	**Number of Segments**			
Standard Memory	10 M / 20 M (Standard memory for SDA and DDA Oscilloscopes are 20 M / 40 M)	5000			
S-32 - Memory Option	32 M / 64 M	15,000			
M-64 - Memory Option	64 M / 128 M	15,000			
L-128 - Memory Option	128 M / 256 M	15,000			

Acquisition Processing	
Averaging	Summed averaging to 1 million sweeps; continuous averaging to 1 million sweeps
Enhanced Resolution (ERES)	From 8.5 to 11 bits vertical resolution
Envelope (Extrema)	Envelope, floor, or roof for up to 1 million sweeps
Interpolation	Linear or Sin x/x

Triggering System	WavePro 715Zi	SDA, WavePro 725Zi	SDA, DDA, WavePro 735Zi	SDA, WavePro 740Zi	SDA, DDA, WavePro 760Zi
Modes	Normal, Auto, Single, and Stop				
Sources Coupling Mode	Any input channel, Aux, Aux/10, or line; slope and level unique to each source (except line trigger)				
Coupling Mode	DC, AC, HFRej, LFRej				
Pre-Trigger Delay	0-100% of memory size (adjustable in 1% increments of 100 ns)				
Post-Trigger Delay	0-10,000 divisions in real time mode, limited at slower time/div settings or in roll mode				
Hold-off by Time or Events	From 2 ns up to 20 s or from 1 to 99, 999,999 events				
Internal Trigger Range	±4.1 div from center				
Trigger Sensitivity with Edge Trigger (Ch 1-4) ProBus Inputs	2 div @ < 1 GHz 1.5 div @ < 500 MHz 1.0 div @ < 200 MHz (for DC, AC, LFRej coupling, ≥ 10 mV/div, 50 Ω)	2 div @ < 2.5 GHz 1.5 div @ < 1.25 GHz 1.0 div @ < 200 MHz (for DC, AC, LFRej coupling, ≥ 10 mV/div, 50 Ω)	2 div @ < 3.5 GHz 1.5 div @ < 1.75 GHz 1.0 div @ < 200 MHz (for DC, AC, LFRej coupling, ≥ 10 mV/div, 50 Ω)		
Trigger Sensitivity with	Not Applicable			2 div @ < 4 GHz	2 div @ < 6 GHz

Triggering System	WavePro 715Zi	SDA, WavePro 725Zi	SDA, DDA, WavePro 735Zi	SDA, WavePro 740Zi	SDA, DDA, WavePro 760Zi
Edge Trigger (Ch 1-4) ProLink Inputs				1.5 div @ 2 GHz 1.0 div @ 200 MHz (for DC, AC, LFRej coupling, ≥ 10 mV/div, 50 Ω)	1.5 div @ 3 GHz 1.0 div @ 200 MHz (for DC, AC, LFRej coupling, ≥ 10 mV/div, 50 Ω)
External Trigger Sensitivity, (Edge Trigger)	2 div @ < 1 GHz 1.5 div @ < 500 MHz 1.0 div @ < 200 MHz (for DC, AC, LFRej coupling)				
Max, Trigger Frequency, SMART Trigger™	1.0 GHz @ ≥ 10 mV/div (minimum triggerable width 500 ps)	2.0 GHz @ ≥ 10 mV/div (minimum triggerable width 500 ps)	2.0 GHz @ ≥ 10 mV/div (minimum triggerable width 500 ps)	2.0 GHz @ ≥ 10 mV/div (minimum triggerable width 500 ps)	
External Trigger Input Range	Aux (±0.4 V); Aux/10 (±4 V)				

Basic Triggers	
Edge	Triggers when signal meets slope (positive, negative, or either) and level condition.
TV-Composite Video	Triggers NTSC or PAL with selectable line and field; HDTV (720p, 1080i, 1080p) with selectable frame rate (50 or 60 Hz) and Line; or CUSTOM with selectable Fields (1-8), Lines (up to 2000), Frame Rates (25, 30, 50, or 60 Hz), Interlacing (1:1, 2:1, 4:1, 8:1), or Synch Pulse Slope (Positive or Negative).
Window	Trigger when signal or exits a window defined by adjustable thresholds.

SMART Triggers	
State or Edge Qualified	Triggers on any input source only if a defined state or edge occurred on another input source. Delay between source is selectable by time or events.
Qualified First	In Sequence acquisition mode, triggers repeatably on event B only if a defined pattern, state, or edge (event A) is satisfied in the first segment of the acquisition. Delay between sources is selectable by time or events.
Dropout	Triggers if signal drops out for longer than selected time between 1 ns and 20 s.
Pattern	Logic combination (AND, NAND, OR, NOR) of 5 inputs (4 channels and external trigger input). Each source can be high, low, or don't care. The High and Low level can be selected independently. Triggers at start or end of the pattern.

SMART Triggers with Exclusion Technology	
Glitch	Triggers on positive or negative glitches with widths selectable as low as 200 ps (depending on oscilloscope bandwidth) to 20 s, or on intermittent faults.
Width (Signal or Pattern)	Triggers on positive, negative, or both widths with widths selectable as low as 200 ps (depending on oscilloscope bandwidth) to 20 s, or on intermittent faults.
Interval (Signal or Pattern)	Triggers on intervals selectable between 1 ns and 20 s.
Timeout (State/Edge Qualified)	Triggers on any source if a given state (or transition edge) has occurred on another source.

SMART Triggers with Exclusion Technology	
	Delay between sources is 1 ns, to 20 s, or 1 to 99,999,999 events.
Runt	Trigger on positive or negative runts defined by two voltage limits and two time limits. Select between 1 ns and 20 ns.
Slew Rate	Trigger on edge rates. Select limits for dV, dt, and slope. Select edge limits between 1 ns and 20 ns.
Exclusion Triggering	Trigger on intermittent faults by specifying the expected behavior and triggering when that condition is not met.

High-Speed Serial Protocol Triggering	WavePro715Zi	WavePro 725Zi	WavePro 735Zi	WavePro 740Zi	WavePro 760Zi
Data Rates	Not available	(Option WPZi-MSPT Standard with SDA) 50 Mb/s-1.25 Gb/s		(Option WPZi-HSPT Standard with SDA) 50 Mb/s-2.7 Gb/s	
Pattern Length	-	80 bits, NRZ or 8b10b			
Clock and Data Outputs	-	400 mV_{p-p} (typical), AC coupled			
Clock Recovery Jitter	-	1 ps_{rms} + 0.3% Unit Interval rms for PRBS data patterns with 50% transition density			
Hardware Clock Recovery Loop BW	-	PLL Loop BW = Fbaud/5500, 50 Mb/s to 1.25 Gb/s (Typical)			

Low-Speed Serial Protocol Triggering (Optional)	
Available	I^2C, SPI, (SPI, SSPI, SIOP), UART-RS232, CAN, LIN, FlexRay Reference individual datasheets for complete specifications.

Color Waveform Display	
Type	Color 15.3" flat panel TFT-Active Matrix LCD with high resolution touch screen
Resolution	WXGA; 1280 X 768 pixels
Number of Traces	Display a maximum of 8 traces. Simultaneously display channel zoom, memory, and math traces.
Grid Styles	Auto, Single, Dual, Quad, Octal, X-Y, Single+X-Y, Dual +X-Y
Waveform Representation	Sample dots joined, or sample dots only

Integrated Second Display	
Type	Color 15.3" flat panel TFT-Active Matrix LCD with high resolution touch screen
Resolution	WXGA; 1280 x 768 pixels

LeCroy WaveStream Fast Viewing Mode	
Intensity	256 Intensity Levels, 1-100% adjustable via front panel control
Number of Channels	Up to 4 simultaneously
Type	Select analog or color graded
Max. Sampling Rate	40 GS/s (20 GS/s for WavePro 715Zi without WPZi-1GHZ-4X20GS option)
Persistence Aging	Select from 500 ms to Infinite
Waveforms/Second (continuous)	Up to 2500 Waveforms/second

Analog Persistence Display	
Analog and Color-Graded	Variable saturation levels; stores each trace's persistence data in memory

WavePro 7Zi

Analog Persistence Display	
Persistence	
Persistence Types	Select analog, color, or three-dimensional
Trace Selection	Active persistence on all or any combination of traces
Persistence Aging	Select from 500 ms to infinity
Sweep Display Modes	All accumulated, or all accumulated with last trace highlighted

High-Speed Digitizer Output (Option)	
Type	LeCroy LSIB
Transfer Rate	Up to 250 Mpts/s (Maximum)
Output Protocol	PCI Express, Gen1 (4 lanes utilized for data transfer)
Control Protocol	TCP/IP
Command Set	Via Windows Automation or LeCroy Remote Command Set

Zoom Expansion Traces	
	Display up to 4 Zoom and 8 Math/Zoom traces

Processor/CPU	
Type	Intel® Core™ 2 Quad, 2.5 GHz (or better)
Processor Memory	2 GB standard, up to 8 GB optional (4 GB standard with S-32 memory, 8 GB standard with M-64 or L-128 memory)
Operating System	Microsoft Windows® Vista® Business Edition (64-bit) with SP1
Real Time Clock	Date and time displayed with waveform and in hardcopy files. SNTP support to synchronize to precision internal clocks.

Internal Waveform Memory	
	4 active waveform memory traces (M1-M4) store 16-bit/point full length waveforms. Waveforms can be stored to any number of files limited only by the data storage media capacity.

Setup Storage	
Front Panel and Instrument Status	Store to the internal hard drive or to a USB-connected peripheral device.

Interface	WavePro Z10Zi	WavePro Z25Zi	WavePro Z35Zi	WavePro Z40Zi	WavePro Z60Zi
Remote Control	Via Windows Automation, or LeCroy Remote Command Set				
Network Communication Standard	LXI Class C, VXI-11, VICP, DCOM				
GPIB Port (optional)	Supports IEEE - 488.2				
LSIB Port (optional)	Supports PCI Express Gen1 x4 protocol with LeCroy supplied API				
USB Device Port (optional)	Type-B				
Ethernet Port	Supports 10/100/1000BaseT Ethernet interface (RJ45 port)				
USB Ports	Minimum 6 total (including 3 front panel) USB 2.0 ports support Windows compatible devices				
External Monitor Port	15-pin D-Type WXGA compatible to support customer-supplied external monitor. DVI and power connector to support LeCroy WPZi-EXTDISP-15 additional touch screen display accessory. Includes support for extended desktop operation with optional LeCroy or other second monitor.				

Interface	WavePro Z10Zi	WavePro Z25Zi	WavePro Z35Zi	WavePro Z40Zi	WavePro Z60Zi
Peripheral Bus	LeCroy LBUS standard				

Auxiliary Input	
Signal Types	Select External Trigger or External Clock Input on the front panel
Coupling	50 Ω: DC; 1 MΩ: AC, DC, GND
Max. Input Voltage	50 Ω: 5 V_{rms}; 1 MΩ: 250 V (Peak AC < 10 kHz + DC)

Auxiliary Output	
Signal Types	Select from calibrator, control signals, or Off
Calibrator Signal	500 Hz-5 MHz square wave or DC level; 0.0 to 500 mV into 50 Ω (0-1 V into 1 MΩ)
Control Signals	Trigger enabled, trigger out, pass/fail status

Automatic Setup	
Auto Setup	Automatically sets timebase, trigger, and sensitivity to display a wide range of repetitive signals
Find Vertical Scale	Automatically sets the vertical sensitivity and offset for the selected channel to display a waveform with the maximum dynamic range

General	
Auto Calibration	Ensures specified DC and timing accuracy is maintained for 1 year minimum.

Probes	
Probes	Qty. (4) ÷ 10 Passive Probes
Probe System	ProBus (and ProLink on 4 and 6 GHz models). Automatically detects and supports a variety of compatible probes
Scale Factors	Automatically or manually selected depending on probe used
Calibration Output	1 kHz square wave, 1 V_{p-p} (typical), output to probe hook

Power Requirements	
Voltage	100-240 VAC ±10% at 50-60 Hz; 100-120 VAC ±10% at 400 Hz; Automatic AC Voltage Selection
Max. Power Consumption	800 W/800 VA

Environmental	
Temperature (Operating)	+5 °C to +40 °C including CD-RW/DVD-ROM drive
Temperature (Non-Operating)	-20 °C to + 60 °C
Humidity (Operating)	5% to 80% relative humidity (non-condensing) up to +31 °C. Upper limit derates to 50% relative humidity (non-condensing) at +40 °C.
Humidity (Non-Operating)	5% to 95% relative humidity (non-condensing) as tested per MIL-PRF-28800F
Altitude (Operating)	Up to 10,000 ft. (3048 m) at or below +25 °C
Altitude (Non-Operating)	Up to 40,000 ft. (12,192 m)
Random Vibration (Operating)	0.5 g_{rms} 5 Hz to 500 Hz, 15 minutes in each of three orthogonal axes
Random Vibration (Non-Operating)	2.4 g_{rms} 5 Hz to 500 Hz, 15 minutes in each of three orthogonal axes as tested per MIL-PRF-28800F
Functional Shock	20 g peak, half sine, 11 ms pulse, 3 shocks (positive and negative) in each of

Environmental	
	three orthogonal axes, 18 shocks total as tested per MIL-PRF-28800F

Certifications

CE Compliant, UL and cUL Listed

CE Declaration of Conformity	
The oscilloscope meets requirements of EMC Directive 89/336/EEC for Electromagnetic Compatibility and Low Voltage Directive 73/23/EEC for Product Safety.	
EMC Directive:	• EN 61326/A3:2003 • EMC requirements for electrical equipment for measurement, control, and laboratory use.
Electromagnetic Emissions:	• EN 55011/A2:2002, Radiated and conducted emissions (Class A)* • EN 61000-3-2/A2:2005 Harmonic Current Emissions (Class A) • EN 61000-3-3/A2:2005 Voltage Fluctuations and Flickers (Pst = 1)

* To conform to Radiated Emissions standard, use properly shielded cables on all I/O terminals.

Electromagnetic Immunity	
	• EN 61000-4-2/A2:2001* Electrostatic Discharge (4 kV contact, 8 kV air, 4 kV vertical/horizontal coupling planes) • EN 61000-4-3/A1:2003* RF Radiated Electromagnetic Field (3 V/m, 80-1000 MHz) • EN 61000-4-4:2004* Electrical Fast Transient/Burst (1 kV AC Mains, 0.5 kV I/O signal/control) • EN 61000-4-5/A1:2001* Surges (1 kV AC Mains, 0.5 kV I/O signal/control) • EN 61000-4-6/A1:2001* RF Conducted Electromagnetic Field (1 kV / 0.5 kV common mode / differential mode - AC Mains) • EN 61000-4-11:2004† Mains Dips and Interruptions (1 cycle voltage dip, 100% short interruption)

* Meets Performance Criteria "B" limits during the disturbance; product undergoes a temporary degradation or loss of function of performance which is self recoverable.

† Meets Performance Criteria "C" limits during the disturbance; product undergoes a temporary degradation or loss of function of performance which requires operator intervention or system reset.

Low Voltage Directive	
	• EN 61010-1:2001 • Safety requirements for electrical equipment for measurement, control, and laboratory use. **The oscilloscope has been qualified to the following EN 61010-1 limits:** • Installation Categories II (Mains Supply Connector) & I (Measuring Terminals) • Pollution Degree 2 (Normally only dry non-conductive pollution occurs. Occasionally a temporary conductivity caused by condensation must be expected.) • Protection Class I (Provided with terminal for protective ground)

UL and cUL Certifications	
	• UL Standard: UL 61010-1 2nd Edition • Canadian Standard: CSA-C22.2 No. 61010-1-04

Physical Dimensions	
Dimensions (HWD)	355 mm x 467 mm x 289 mm; 14' x 18.4" x 11.4" (height excludes feet)
Weight	18.4 kg; 40 lbs.
Shipping Weight	26.6 kg; 58 lbs.

Certifications	
	CE Compliant, UL and cUL listed; conforms to EN 61326, EN 61010-1, UL 61010-1, 2nd edition, and CSA C22.2 No. 61010-1-04

Warranty and Service	
	Three-year warranty; calibration recommended annually. Optional service programs include extended warranty, upgrades, and calibration services.

Technical Support

For assistance with installation, calibration, and a full range of software applications contact the customer service center nearest you. You can also find contact information for the following offices on the LeCroy Web site at http://www.lecroy.com/tm/ServiceSupport.

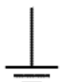
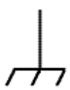
Contact Your Local LeCroy Technical Support Office for Help	
United States and Canada **Phone Sales**: 800-553-2769 or 845-425-2000 **Fax (Sales&Service)**: 845-578-5985 **Email Sales**: contact.corp@lecroy.com **Web Site**: http://www.lecroy.com/	***Europe*** **Phone Sales**: + 41 22 719 2228 **Fax Sales**: + 41 22 719 2230 **Email Sales**: contact.sa@lecroy.com **Web Site**: http://www.lecroy.com/europe
China **Phone Sales**: ++86 28- 86527180 / 7181 / 7182 **Fax Sales**: +86 28-8652 7183 **Email Sales**: george.ni@lecroy.com	***Singapore*** **Phone Sales**: ++ (65) 64424880 **Fax Sales**: ++ (65) 64427811 **Email Sales**: jimmy.ong@lecroy.com
Taiwan**Phone Sales**: (886) 2 8226 1366 **Fax Sales**: (886) 2 8226 1368 **Email Sales**: sales_twn@lecoln.com.tw **Web Site**: http://www.lecoln.com.tw	***Tokyo*** **Phone Sales**: ++ 81 3 3376 9400 **Fax Sales**: ++ 81 3 3376 9587 **Web Site**: http://www.lecroy.com/japan
Korea - Seoul **Phone Sales**: ++ 82 2 3452 0400 **Fax Sales**: ++ 82 2 3452 0490 **Web Site**: http://www.lecroy.co.kr	

Safety Requirements

This section contains information and warnings that must be observed to keep the instrument operating in a correct and safe condition. You are required to follow generally accepted safety procedures in addition to the safety precautions specified in this section.

Safety Symbols

Where the following symbols appear on the instrument's front or rear panels, or in this manual, they alert you to important safety considerations.

	This symbol is used where caution is required. Refer to the accompanying information or documents in order to protect against personal injury or damage to the instrument.
	This symbol warns of a potential risk of shock hazard.
	This symbol is used to denote the measurement ground connection.
	This symbol is used to denote a safety ground connection.
	This symbol is used to denote a grounded frame or chassis terminal.
	This symbol shows that the switch is a Standby (power) switch. When it is pressed, the oscilloscope's state toggles between operating and Standby mode. This switch is not a disconnect device. The instrument can only be placed in a complete Power Off state by flipping the main power switch to the off (Zero) position.
	This symbol is used to denote Alternating Current.
CAUTION	The CAUTION sign indicates a potential hazard. It calls attention to a procedure, practice or condition which, if not followed, could possibly cause damage to equipment. If a CAUTION is indicated, do not proceed until its conditions are fully understood and met.
WARNING	The WARNING sign indicates a potential hazard. It calls attention to a procedure, practice or condition which, if not followed, could possibly cause bodily injury or death. If a WARNING is indicated, do not proceed until its conditions are fully understood and met.
CAT I	Installation (Overvoltage) Category rating per EN 61010-1 safety standard and is applicable for the oscilloscope front panel measuring terminals. CAT I rated terminals must only be connected to source circuits in which measures are taken to limit transient voltages to an appropriately low level.

Operating Environment

The instrument is intended for indoor use and should be operated in a clean, dry environment with an ambient temperature within the range of 5 °C to 40 °C.

Note: Direct sunlight, radiators, and other heat sources should be taken into account when assessing the ambient temperature.

WARNING

The oscilloscope must not be operated in explosive, dusty, or wet atmospheres.

CAUTION

Protect the oscilloscope's display touch screen from excessive impacts with foreign objects.

CAUTION

Do not exceed the maximum specified front panel terminal (CH1, CH1, CH2, CH3, CH4, AUX IN, AUX OUT) voltage levels. Refer to Specifications for more details.

Installation (Overvoltage) Category II refers to local distribution level, which is applicable to equipment connected to the mains supply (AC power source).

Installation (Overvoltage) Category I refers to signal level, which is applicable to equipment measuring terminals that are connected to source circuits in which measures are taken to limit transient voltages to an appropriately low level.

Pollution Degree 2 refers to an operating environment where normally only dry non-conductive pollution occurs. Occasionally a temporary conductivity caused by condensation must be expected.

Protection Class 1 refers to a grounded equipment, in which protection against electric shock is achieved by Basic Insulation and by means of a connection to the protective ground conductor in the building wiring.

PLEASE NOTE THE FOLLOWING:

The design of the instrument has been verified to conform to EN 61010-1 safety standard per the following limits:

- Installation (Overvoltage) Categories II (Mains Supply Connector) & I (Measuring Terminals)
- Pollution Degree 2
- Protection Class I

Cooling

The instrument relies on forced air cooling with internal fans and ventilation openings. Care must be taken to avoid restricting the airflow around the apertures (fan holes) at the sides and rear of the oscilloscope. Ensure adequate ventilation by leaving the required 10 cm (4 inch) minimum gap around the sides and rear of the instrument.

CAUTION

Do not block the ventilation holes located on both sides and rear of the oscilloscope.

The instrument also has internal fan control circuitry that regulates the fan speed based on the ambient temperature. This is performed automatically after start-up with no manual intervention required.

CAUTION

Do not allow any foreign matter to enter the oscilloscope through the ventilation holes, etc.

AC Power Source

100 to 240 Vrms (+/-10%) AC at 50/60 Hz; 115 Vrms (+/-10%) AC at 400 Hz; Automatic AC voltage selection; Installation Category: 300V CAT II

No manual voltage selection is required because the instrument automatically adapts to line voltage.

WavePro 700Zi - </= 800 watts (800 VA) depending on accessories installed (probes, PC port plug-ins, etc.).

> **Note**: The instrument automatically adapts itself to the AC line input within the following ranges:

Voltage Range:	90 to 264 Vrms	90 to 132 Vrms
Frequency Range:	45 to 66 Hz	360 to 440 Hz

Power and Ground Connections

The instrument is provided with a grounded cord set containing a molded three-terminal polarized plug and a standard IEC320 (Type C13) connector for making line voltage and safety ground connection. The AC inlet ground terminal is connected directly to the frame of the instrument. For adequate protection against electrical shock hazard, the power cord plug must be inserted into a mating AC outlet containing a safety ground contact.

WARNING - Electrical Shock Hazard

Any interruption of the protective conductor inside or outside of the oscilloscope, or disconnection of the safety ground terminal creates a hazardous situation.

Intentional interruption is prohibited.

In Standby mode the oscilloscope is still connected to the AC supply. The instrument can only be placed in a complete Power Off state by physically disconnecting the power cord from the AC supply.

The oscilloscope should be positioned to allow easy access to the socket-outlet. Disconnect the oscilloscope from the AC supply by unplugging the instrument's power cord from the AC outlet after the oscilloscope is placed in Standby state.

CAUTION

The outer shells of the front panel terminals (CH1, CH2, CH3, CH4, AUX IN, AUX OUT) are connected to the instrument's chassis and therefore to the safety ground.

Standby (Power) Switch and Oscilloscope Operational States

The front Standby (Power) switch controls the operational state of the oscilloscope. This toggle switch is activated by momentarily pressing and releasing it. The color of the LED below the switch indicates the status of the oscilloscope as follows:

- **On (LED Green)*** – oscilloscope is fully powered and operational
- **Standby (LED off)*** – oscilloscope is powered off (except for some housekeeping circuits)
- **Standby (LED Blinks Green)** – oscilloscope's computer subsystems (hard drive, etc.) are in Standby (reduced Power mode). All other oscilloscope subsystems are fully powered.

* Factory Settings

The oscilloscope's factory settings result in only two basic oscilloscope states: On (LED Green) or Standby (LED Off). In this case of Standby (LED Off), the oscilloscope is powered off with the exception of some housekeeping circuitry (approximately 12 watts dissipation). The oscilloscope can only be placed in a complete power off state by unplugging the instrument's power cord from the primary power source (AC outlet). It is recommended that the power cord be unplugged from the AC outlet if the oscilloscope is not being used for an extended period of time.

You have the ability to change the oscilloscope's original factory settings via the **Power Options Properties** menu in Windows under **Control Panel → Power Options**. It is important to note that the Windows Power Option named **Standby** provides control of only the oscilloscope's computer subsystems (CPU, hard drive, etc.) and does not affect the other subsystems within the oscilloscope. In general, these other subsystems remain fully powered. For additional information on setting these Power Options, see the Windows Help menu or other related technical documentation. In terms of control buttons, this oscilloscope uses only a power button/switch and therefore references to a sleep button are not applicable.

The oscilloscope can always be placed in the Standby state (LED Off) – Power Off (except for some housekeeping circuits) by pressing and holding in the Standby toggle switch for approximately 5 seconds.

Calibration

The recommended calibration interval is one year. Calibration should be performed by qualified personnel only.

Cleaning

Clean only the exterior of the instrument, using a damp, soft cloth. Do not use chemicals or abrasive elements. Under no circumstances allow moisture to penetrate the instrument.

Avoid electrical shock hazard by unplugging the power cord from the AC outlet before cleaning.

WARNING - Electrical Shock Hazard

- No operator serviceable parts inside.
- Do not remove covers.
- Refer servicing to qualified personnel.

Abnormal Conditions

Operate the instrument only as intended by the manufacturer.

If you suspect the oscilloscope's protection has been impaired, disconnect the power cord and secure the instrument against any unintended operation.

The oscilloscope's protection is likely to be impaired if, for example, the instrument shows visible damage or has been subjected to severe transport stresses.

Proper use of the instrument depends on careful reading of all instructions and labels.

WARNING

Any use of the oscilloscope in a manner not specified by the manufacturer may impair the instrument's safety protection. The instrument and related accessories should not be directly connected to human subjects or used for patient monitoring.

Index

B

C

D

E

F

M

N

O

P

Q

R

S

T

U

V

W

X

Z

Thank you for purchasing a WavePro, SDA, or DDA 7 Zi Oscilloscope

LeCroy

Corporate Headquarters
700 Chestnut Ridge Road
Chestnut Ridge, NY 10977
USA
www.lecroy.com