Keysight InfiniiVision 2000 X-Series Oscilloscopes

User's Guide

KEYSIGHT TECHNOLOGIES

InfiniiVision 2000 X-Series Oscilloscopes—At a Glance

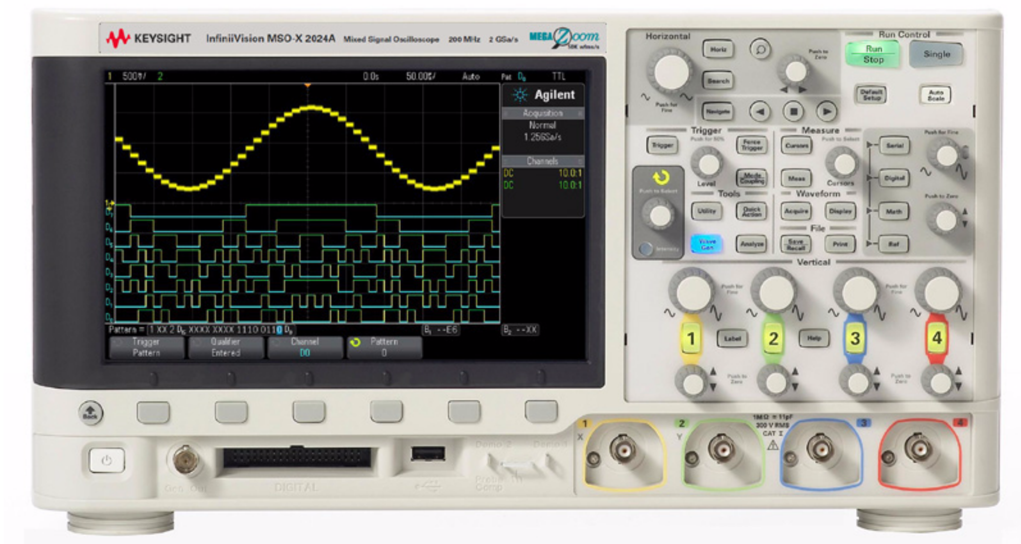
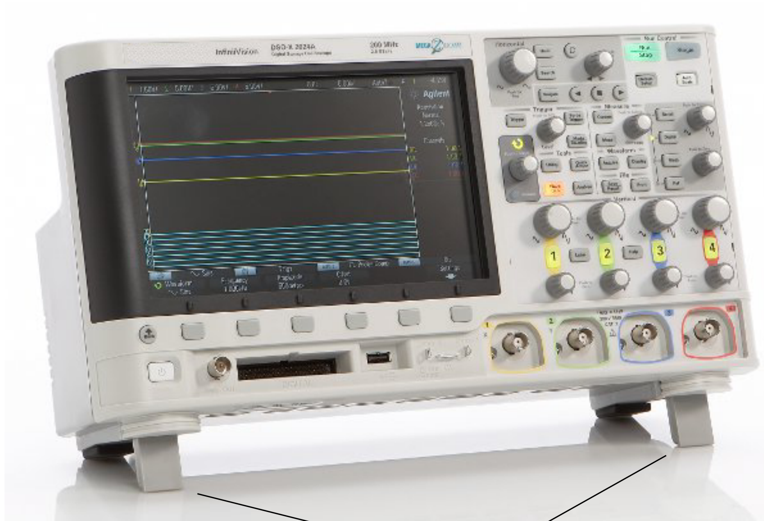


Table 1 2000 X-Series Model Numbers, Bandwidths

Bandwidth	70 MHz	100 MHz	200 MHz
2-Channel + 8 Logic Channels MSO	MSO-X 2002A	MSO-X 2012A	MSO-X 2022A
4-Channel + 8 Logic Channels MSO	MSO-X 2004A	MSO-X 2014A	MSO-X 2024A
2-Channel DSO	DSO-X 2002A	DSO-X 2012A	DSO-X 2022A
4-Channel DSO	DSO-X 2004A	DSO-X 2014A	DSO-X 2024A

The Keysight InfiniiVision 2000 X-Series oscilloscopes deliver these features:

- 70 MHz, 100 MHz, and 200 MHz bandwidth models.
- 2- and 4-channel digital storage oscilloscope (DSO) models.
- 2+8-channel and 4+8-channel mixed-signal oscilloscope (MSO) models.



Flip-Out Tabs

Power-On the Oscilloscope

Power Requirements

Line voltage, frequency, and power:

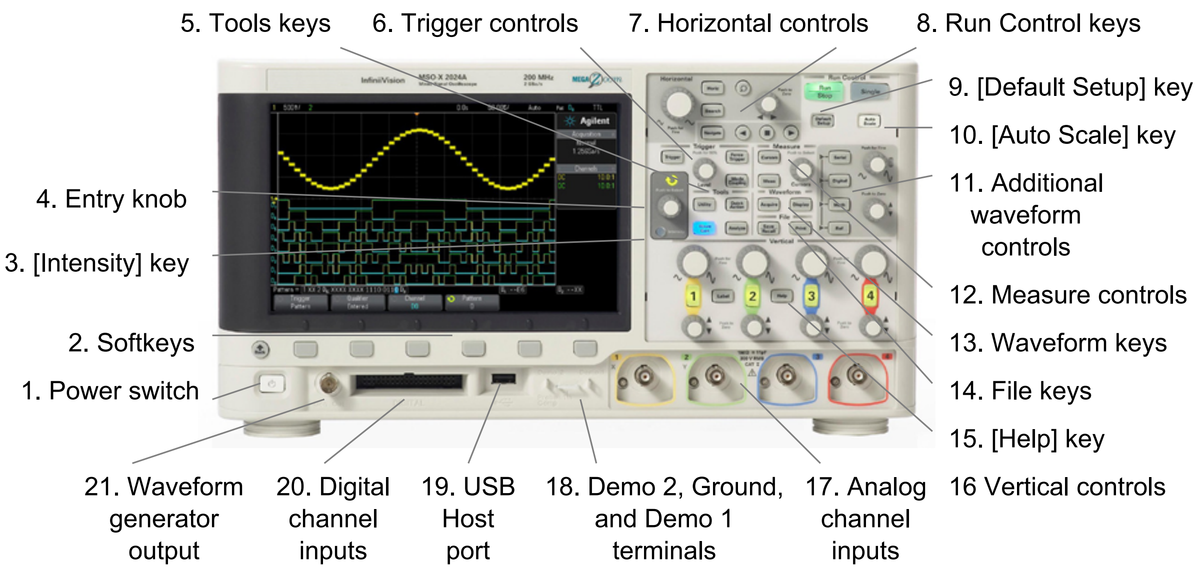
- ~Line 100-120 Vac, 50/60/400 Hz
- 100-240 Vac, 50/60 Hz
- 100 W max

CAUTION This instrument has auto-ranging line voltage input. Be sure the supply voltage is within the specified range and voltage fluctuations do not to exceed 10 percent of the nominal supply voltage.

Ventilation Requirements

The air intake and exhaust areas must be free from obstructions. Unrestricted air flow is required for proper cooling. Always ensure that the air intake and exhaust areas are free from obstructions.

The fan draws air in from the left side and bottom of the oscilloscope and pushes it out behind the oscilloscope.



1.	Power switch	Press once to switch power on; press again to switch power off. See "Power-On the Oscilloscope" on page 27.
2.	Softkeys	The functions of these keys change based upon the menus shown on the display directly above the keys. The Back/Up key moves up in the softkey menu hierarchy. At the top of the hierarchy, the Back/Up key turns the menus off, and oscilloscope information is shown instead.
3.	**[Intensity]** key	Press the key to illuminate it. When illuminated, turn the Entry knob to adjust waveform intensity. You can vary the intensity control to bring out signal detail, much like an analog oscilloscope. Digital channel waveform intensity is not adjustable. More details about using the Intensity control to view signal detail are on "To adjust waveform intensity" on page 113.

4.	Entry knob	The Entry knob is used to select items from menus and to change values. The function of the Entry knob changes based upon the current menu and softkey selections. Note that the curved arrow symbol above the entry knob illuminates whenever the entry knob can be used to select a value. Also, note that when the Entry knob symbol appears on a softkey, you can use the Entry knob, to select values. Often, rotating the Entry knob is enough to make a selection. Sometimes, you can push the Entry knob to enable or disable a selection. Pushing the Entry knob also makes popup menus disappear.
5.	Tools keys	The Tools keys consist of: ▪ **[Utility]** key — Press this key to access the Utility Menu, which lets you configure the oscilloscope's I/O settings, use the file explorer, set preferences, access the service menu, and choose other options. See Chapter 20, “Utility Settings,” starting on page 247. ▪ **[Quick Action]** key — Press this key to perform the selected quick action: measure all snapshot, print, save, recall, freeze display. and more. See “Configuring the [Quick Action] Key" on page 262. ▪ **[Analyze]** key — Press this key to access analysis features like mask testing (see Chapter 15, “Mask Testing,” starting on page 201), trigger level setting, measurement threshold setting, or Video trigger automatic set up and display. ▪ **[Wave Gen]** key — Press this key to access waveform generator functions. See Chapter 17, “Waveform Generator,” starting on page 217.
6.	Trigger controls	These controls determine how the oscilloscope triggers to capture data. See Chapter 10, “Triggers,” starting on page 125 and Chapter 11, “Trigger Mode/Coupling,” starting on page 147.

7.	Horizontal controls	The Horizontal controls consist of: ▪ Horizontal scale knob – Turn the knob in the Horizontal section that is marked to adjust the time/div (sweep speed) setting. The symbols under the knob indicate that this control has the effect of spreading out or zooming in on the waveform using the horizontal scale. ▪ Horizontal position knob – Turn the knob marked ◀▶ to pan through the waveform data horizontally. You can see the captured waveform before the trigger (turn the knob clockwise) or after the trigger (turn the knob counterclockwise). If you pan through the waveform when the oscilloscope is stopped (not in Run mode) then you are looking at the waveform data from the last acquisition taken. ▪ **[Horiz]** key – Press this key to open the Horizontal Menu where you can select XY and Roll modes, enable or disable Zoom, enable or disable horizontal time/division fine adjustment, and select the trigger time reference point. ▪ Zoom key – Press the zoom key to split the oscilloscope display into Normal and Zoom sections without opening the Horizontal Menu. ▪ **[Search]** key – Lets you search for events in the acquired data. ▪ **[Navigate]** keys – Press this key to navigate through captured data (Time), search events, or segmented memory acquisitions. See "Navigating the Time Base" on page 59. For more information see Chapter 2, "Horizontal Controls," starting on page 49.
8.	Run Control keys	When the **[Run/Stop]** key is green, the oscilloscope is running, that is, acquiring data when trigger conditions are met. To stop acquiring data, press **[Run/Stop]**. When the **[Run/Stop]** key is red, data acquisition is stopped. To start acquiring data, press **[Run/Stop]**. To capture and display a single acquisition (whether the oscilloscope is running or stopped), press **[Single]**. The **[Single]** key is yellow until the oscilloscope triggers. For more information, see "Running, Stopping, and Making Single Acquisitions (Run Control)" on page 155.
9.	**[Default Setup]** key	Press this key to restore the oscilloscope's default settings (details on "Recall the Default Oscilloscope Setup" on page 29).

10.	**[Auto Scale]** key	When you press the **[AutoScale]** key, the oscilloscope will quickly determine which channels have activity, and it will turn these channels on and scale them to display the input signals. See "Use Auto Scale" on page 30.

11.	Additional waveform controls	The additional waveform controls consist of: ▪ **[Math]** key – provides access to math (add, subtract, etc.) waveform functions. See Chapter 4, “Math Waveforms,” starting on page 71. ▪ **[Ref]** key – provides access to reference waveform functions. Reference waveforms are saved waveforms that can be displayed and compared against other analog channel or math waveforms. See Chapter 5, “Reference Waveforms,” starting on page 85. ▪ **[Digital]** key – Press this key to turn the digital channels on or off (the arrow to the left will illuminate). When the arrow to the left of the **[Digital]** key is illuminated, the upper multiplexed knob selects (and highlights in red) individual digital channels, and the lower multiplexed knob positions the selected digital channel. If a trace is repositioned over an existing trace the indicator at the left edge of the trace will change from **D**n designation (where n is a one digit channel number from 0 to 7) to **D***. The "*" indicates that two channels are overlaid. You can rotate the upper knob to select an overlaid channel, then rotate the lower knob to position it just as you would any other channel. For more information on digital channels see Chapter 6, “Digital Channels,” starting on page 89. ▪ **[Serial]** key – This key is used to enable serial decode. The multiplexed scale and position knobs are not used with serial decode. For more information on serial decode, see Chapter 7, “Serial Decode,” starting on page 107. Digital channels and serial decode cannot be on at the same time. The **[Serial]** key takes precedence over the **[Digital]** key. Serial triggers can be used when digital channels are on. ▪ Multiplexed scale knob – This scale knob is used with Math, Ref, or Digital waveforms, whichever has the illuminated arrow to the left. For math and reference waveforms, the scale knob acts like an analog channel vertical scale knob. ▪ Multiplexed position knob – This position knob is used with Math, Ref, or Digital waveforms, whichever has the illuminated arrow to the left. For math and reference waveforms, the position knob acts like an analog channel vertical position knob.

12.	Measure controls	The measure controls consist of: ▪ Cursors knob – Push this knob select cursors from a popup menu. Then, after the popup menu closes (either by timeout or by pushing the knob again), rotate the knob to adjust the selected cursor position. ▪ **[Cursors]** key – Press this key to open a menu that lets you select the cursors mode and source. ▪ **[Meas]** key – Press this key to access a set of predefined measurements. See Chapter 14, “Measurements,” starting on page 181.
13.	Waveform keys	The **[Acquire]** key lets you select Normal, Peak Detect, Averaging, or High Resolution acquisition modes (see “Selecting the Acquisition Mode" on page 161) and use segmented memory (see “Acquiring to Segmented Memory" on page 167). The **[Display]** key lets you access the menu where you can enable persistence (see “To set or clear persistence" on page 115), clear the display, and adjust the display grid (graticule) intensity (see “To adjust the grid intensity" on page 117).
14.	File keys	Press the **[Save/Recall]** key to save or recall a waveform or setup. See Chapter 18, “Save/Recall (Setups, Screens, Data),” starting on page 229. The **[Print]** key opens the Print Configuration Menu so you can print the displayed waveforms. See Chapter 19, “Print (Screens),” starting on page 241.
15.	**[Help]** key	Opens the Help Menu where you can display overview help topics and select the Language. See also “Access the Built-In Quick Help" on page 47.

16.	Vertical controls	The Vertical controls consist of: ▪ Analog channel on/off keys – Use these keys to switch a channel on or off, or to access a channel's menu in the softkeys. There is one channel on/off key for each analog channel. ▪ Vertical scale knob – There are knobs marked for each channel. Use these knobs to change the vertical sensitivity (gain) of each analog channel. ▪ Vertical position knobs – Use these knobs to change a channel's vertical position on the display. There is one Vertical Position control for each analog channel. ▪ **[Label]** key – Press this key to access the Label Menu, which lets you enter labels to identify each trace on the oscilloscope display. See Chapter 9, "Labels," starting on page 119. For more information, see Chapter 3, "Vertical Controls," starting on page 63.
17.	Analog channel inputs	Attach oscilloscope probes or BNC cables to these BNC connectors. In the InfiniiVision 2000 X-Series oscilloscopes, the analog channel inputs have 1 MΩ impedance. Also, there is no automatic probe detection, so you must properly set the probe attenuation for accurate measurement results. See "To specify the probe attenuation" on page 68.
18.	Demo 2, Ground, and Demo 1 terminals	▪ Demo 2 terminal – This terminal outputs the Probe Comp signal which helps you match a probe's input capacitance to the oscilloscope channel to which it is connected. See "Compensate Passive Probes" on page 32. With certain licensed features, the oscilloscope can also output demo or training signals on this terminal. ▪ Ground terminal – Use the ground terminal for oscilloscope probes connected to the Demo 1 or Demo 2 terminals. ▪ Demo 1 terminal – With certain licensed features, the oscilloscope can output demo or training signals on this terminal.

19.	USB Host port	This port is for connecting USB mass storage devices or printers to the oscilloscope. Connect a USB compliant mass storage device (flash drive, disk drive, etc.) to save or recall oscilloscope setup files and reference waveforms or to save data and screen images. See Chapter 18, “Save/Recall (Setups, Screens, Data),” starting on page 229. To print, connect a USB compliant printer. For more information about printing see Chapter 19, “Print (Screens),” starting on page 241. You can also use the USB port to update the oscilloscope's system software when updates are available. You do not need to take special precautions before removing the USB mass storage device from the oscilloscope (you do not need to "eject" it). Simply unplug the USB mass storage device from the oscilloscope when the file operation is complete. **CAUTION: ⚠ Do not connect a host computer to the oscilloscope's USB host port.** Use the device port. A host computer sees the oscilloscope as a device, so connect the host computer to the oscilloscope's device port (on the rear panel). See “I/O Interface Settings" on page 247. There is a second USB host port on the back panel.
20.	Digital channel inputs	Connect the digital probe cable to this connector (MSO models only). See Chapter 6, “Digital Channels,” starting on page 89.
21.	Waveform generator output	Outputs sine, square, ramp, pulse, DC, or noise on the Gen Out BNC. Press the **[Wave Gen]** key to set up the waveform generator. See Chapter 17, “Waveform Generator,” starting on page 217.

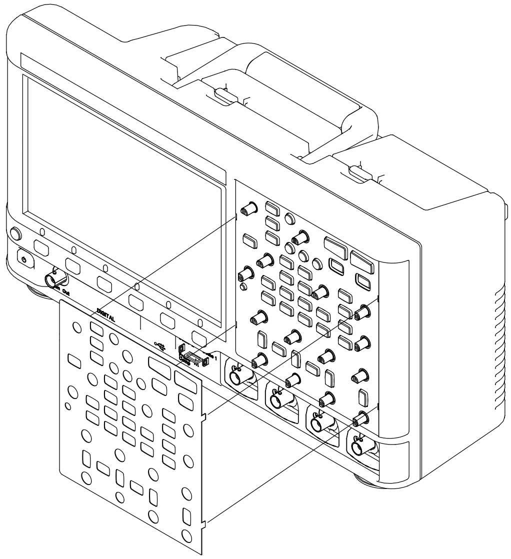
Front Panel Overlays for Different Languages

Front panel overlays, which have translations for the English front panel keys and label text, are available in 10 languages. The appropriate overlay is included when the localization option is chosen at time of purchase.

To install a front panel overlay:

1 Gently pull on the front panel knobs to remove them.

2 Insert the overlay's side tabs into the slots on the front panel.



3 Reinstall the front panel knobs.

Front panel overlays may be ordered from www.parts.keysight.com using the following part numbers:

2 Horizontal Controls

The horizontal controls include:

- The horizontal scale and position knobs.
- The **[Horiz]** key for accessing the Horizontal Menu.
- The zoom key for quickly enabling/disabling the split-screen zoom display.
- The **[Search]** key for finding events on analog channels or in serial decode.
- The **[Navigate]** keys for navigating time, search events, or segmented memory acquisitions.

The following figure shows the Horizontal Menu which appears after pressing the **[Horiz]** key.

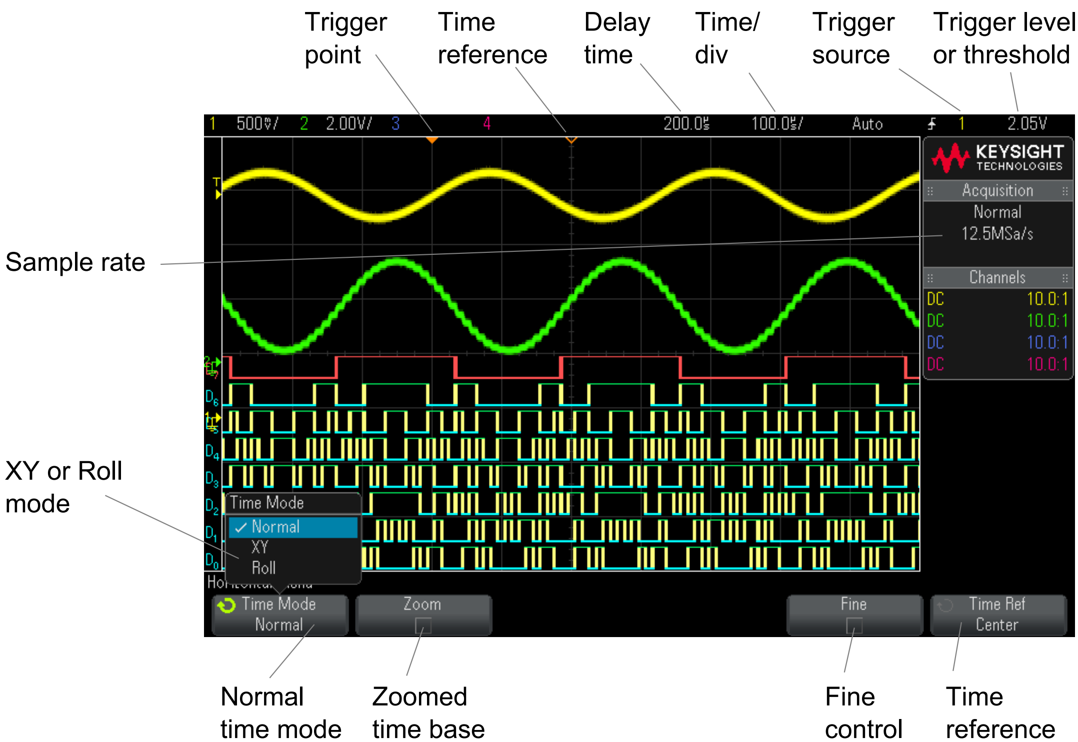


Figure 2 Horizontal Menu

The Horizontal Menu lets you select the time mode (Normal, XY, or Roll), enable Zoom, set the time base fine control (vernier), and specify the time reference.

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

To adjust the horizontal (time/div) scale

1 Turn the large horizontal scale (sweep speed) knob marked to change the horizontal time/div setting.

Notice how the time/div information in the status line changes.

The ∇ symbol at the top of the display indicates the time reference point.

The horizontal scale knob works (in the Normal time mode) while acquisitions are running or when they are stopped. When running, adjusting the horizontal scale knob changes the sample rate. When stopped, adjusting the horizontal scale knob lets you zoom into acquired data. See "Panning and Zooming Single or Stopped Acquisitions" on page 52.

Note that the horizontal scale knob has a different purpose in the Zoom display. See "To display the zoomed time base" on page 56.

To adjust the horizontal delay (position)

1 Turn the horizontal delay (position) knob (◀▶).

The trigger point moves horizontally, pausing at 0.00 s (mimicking a mechanical detent), and the delay value is displayed in the status line.

Changing the delay time moves the trigger point (solid inverted triangle) horizontally and indicates how far it is from the time reference point (hollow inverted triangle ∇). These reference points are indicated along the top of the display grid.

Figure 2 shows the trigger point with the delay time set to 200 µs. The delay time number tells you how far the time reference point is located from the trigger point. When delay time is set to zero, the delay time indicator overlays the time reference indicator.

All events displayed left of the trigger point happened before the trigger occurred. These events are called pre-trigger information, and they show events that led up to the trigger point.

Everything to the right of the trigger point is called post-trigger information. The amount of delay range (pre-trigger and post-trigger information) available depends on the time/div selected and memory depth.

The horizontal position knob works (in the Normal time mode) while acquisitions are running or when they are stopped. When running, adjusting the horizontal scale knob changes the sample rate. When stopped, adjusting the horizontal scale knob lets you zoom into acquired data. See "Panning and Zooming Single or Stopped Acquisitions" on page 52.

Note that the horizontal position knob has a different purpose in the Zoom display. See "To display the zoomed time base" on page 56.

Panning and Zooming Single or Stopped Acquisitions

When the oscilloscope is stopped, use the horizontal scale and position knobs to pan and zoom your waveform. The stopped display may contain several acquisitions worth of information, but only the last acquisition is available for pan and zoom.

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

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

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

NOTE

Zooming into stopped acquisitions

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

To change the horizontal time mode (Normal, XY, or Roll)

1 Press **[Horiz]**.
2 In the Horizontal Menu, press **Time Mode**; then, select:

- **Normal** — the normal viewing mode for the oscilloscope.

  In the Normal time mode, signal events occurring before the trigger are plotted to the left of the trigger point (▼) and signal events after the trigger plotted to the right of the trigger point.

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

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

  Use the cursors to make measurements on XY mode waveforms.

  For more information about using XY mode for measurements, refer to "XY Time Mode" on page 53.

- **Roll** — causes the waveform to move slowly across the screen from right to left. It only operates on time base settings of 50 ms/div and slower. If the current time base setting is faster than the 50 ms/div limit, it will be set to 50 ms/div when Roll mode is entered.

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

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

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

XY Time Mode

The XY time mode converts the oscilloscope from a volts-versus-time display to a volts-versus-volts display using two input channels. Channel 1 is the X-axis input, channel 2 is the Y-axis input. You can use various transducers so the display could show strain versus displacement, flow versus pressure, volts versus current, or voltage versus frequency.

Example This exercise shows a common use of the XY display mode by measuring the phase difference between two signals of the same frequency with the Lissajous method.

1. Connect a sine wave signal to channel 1, and a sine wave signal of the same frequency but out of phase to channel 2.
2. Press the **[AutoScale]** key, press the **[Horiz]** key; then, press **Time Mode** and select "XY".
3. Center the signal on the display with the channel 1 and 2 position (⬍) knobs. Use the channel 1 and 2 volts/div knobs and the channel 1 and 2 **Fine** softkeys to expand the signal for convenient viewing.

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

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

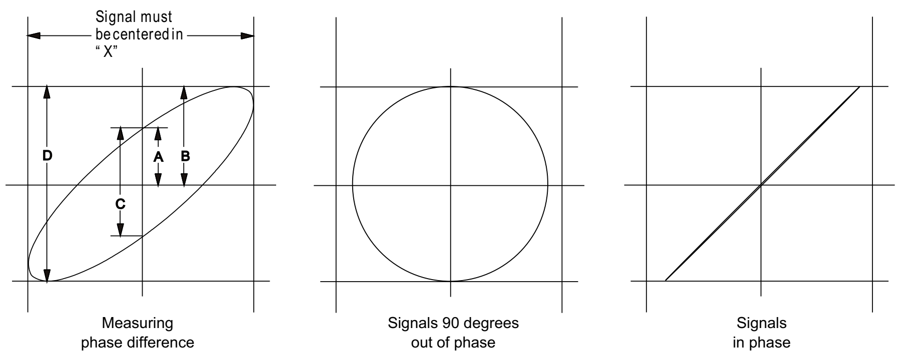


Figure 3 XY time mode signals, centered on display

4. Press the **[Cursors]** key.
5. Set the Y2 cursor to the top of the signal, and set Y1 to the bottom of the signal.

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

6 Move the Y1 and Y2 cursors to the intersection of the signal and the Y axis. Again, note the ΔY value.

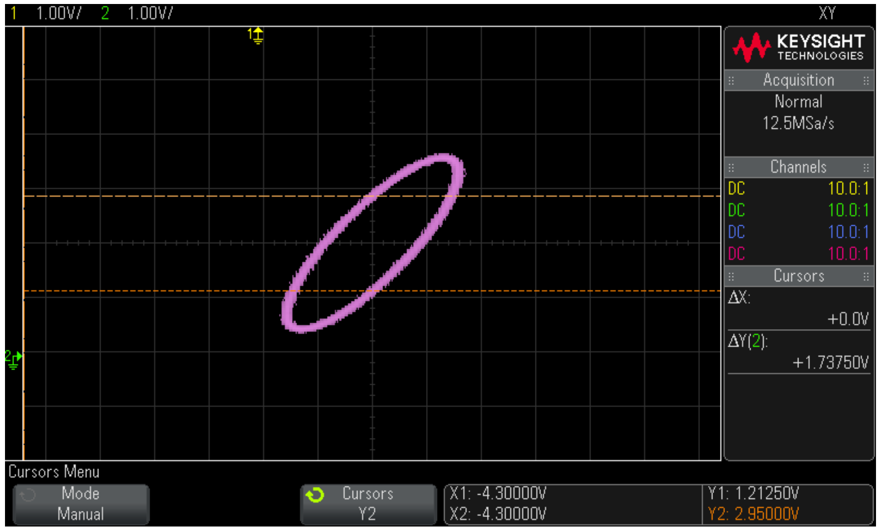


Figure 4 Phase difference measurements, automatic and using cursors

7 Calculate the phase difference using the formula below.

For example, if the first ΔY value is 1.688 and the second ΔY value is 1.031:

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

NOTE

Z-Axis Input in XY Display Mode (Blanking)

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

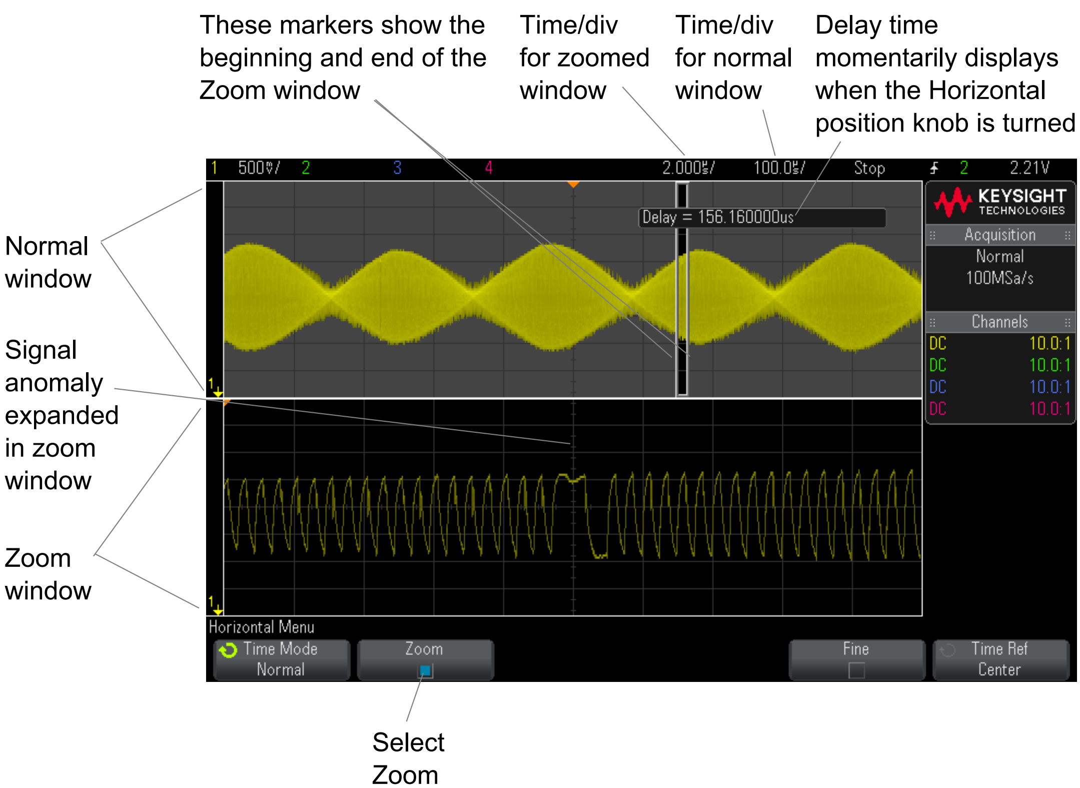
To display the zoomed time base

Zoom, formerly called Delayed sweep mode, is a horizontally expanded version of the normal display. When Zoom is selected, the display divides in half. The top half of the display shows the normal time/div window and the bottom half displays a faster Zoom time/div window.

The Zoom window is a magnified portion of the normal time/div window. You can use Zoom to locate and horizontally expand part of the normal window for a more detailed (higher-resolution) analysis of signals.

To turn on (or off) Zoom:

1 Press the ⌕ zoom key (or press the **[Horiz]** key and then the **Zoom** softkey).



The area of the normal display that is expanded is outlined with a box and the rest of the normal display is ghosted. The box shows the portion of the normal sweep that is expanded in the lower half.

To change the time/div for the Zoom window, turn the horizontal scale (sweep speed) knob. As you turn the knob, the zoomed window time/div is highlighted in the status line above the waveform display area. The Horizontal scale (sweep speed) knob controls the size of the box.

The Horizontal position (delay time) knob sets the left-to-right position of the zoom window. The delay value, which is the time displayed relative to the trigger point) is momentarily displayed in the upper-right portion of the display when the delay time (◀▶) knob is turned.

Negative delay values indicate you're looking at a portion of the waveform before the trigger event, and positive values indicate you're looking at the waveform after the trigger event.

To change the time/div of the normal window, turn off Zoom; then, turn the horizontal scale (sweep speed) knob.

For information about using zoom mode for measurements, refer to "To isolate a pulse for Top measurement" on page 187 and "To isolate an event for frequency measurement" on page 194.

To change the horizontal scale knob's coarse/fine adjustment setting

1 Push the horizontal scale knob (or press **[Horiz] > Fine**) to toggle between fine and coarse adjustment of the horizontal scale.

When **Fine** is enabled, turning the horizontal scale knob changes the time/div (displayed in the status line at the top of the display) in smaller increments. The time/div remains fully calibrated when **Fine** is on.

When **Fine** is turned off, the Horizontal scale knob changes the time/div setting in a 1-2-5 step sequence.

To position the time reference (left, center, right)

Time reference is the reference point on the display for delay time (horizontal position).

1 Press **[Horiz]**.

2 In the Horizontal Menu, press **Time Ref**; then, select:

- **Left** — the time reference is set to one major division from the left edge of the display.
- **Center** — the time reference is set to the center of the display.
- **Right** — the time reference is set to one major division from the right edge of the display.

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

The time reference position sets the initial position of the trigger event within acquisition memory and on the display, with delay set to 0.

Turning the Horizontal scale (sweep speed) knob expands or contracts the waveform about the time reference point (∇). See "To adjust the horizontal (time/div) scale" on page 50.

Turning the Horizontal position (◀▶) knob in Normal mode (not Zoom) moves the trigger point indicator (▼) to the left or right of the time reference point (∇). See "To adjust the horizontal delay (position)" on page 51.

Searching for Events

You can use the **[Search]** key and menu to search for Serial events on the analog channels.

Setting up searches (see "To set up searches" on page 59) is similar to setting up triggers.

Searches are different than triggers in that they use the measurement threshold settings instead of trigger levels.

Found search events are marked with white triangles at the top of the graticule, and the number of events found is displayed in the menu line just above the sofkey labels.

To set up searches

1 Press **[Search]**.
2 Setting up searches is similar to setting up triggers:
   - For setting up Serial searches, see Chapter 10, "Triggers," starting on page 125 and "Searching Lister Data" on page 110.

Remember that searches use the measurement threshold settings instead of trigger levels. Use the **Thresholds** softkey in the Search Menu to access the Measurement Threshold Menu. See "Measurement Thresholds" on page 198.

Navigating the Time Base

You can use the **[Navigate]** key and controls to navigate through:

- Captured data (see "To navigate time" on page 59).
- Search events (see "To navigate search events" on page 60).
- Segments, when segmented memory acquisitions are turned on (see "To navigate segments" on page 60).

To navigate time

When acquisitions are stopped, you can use the navigation controls to play through the captured data.

1 Press **[Navigate]**.
2 In the Navigate Menu, press **Navigate**; then, select **Time**.
3 Press the ◀ ■ ▶ navigation keys to play backward, stop, or play forward in time. You can press the ◀ or ▶ keys multiple times to speed up the playback. There are three speed levels.

To navigate search events

When acquisitions are stopped, you can use the navigation controls to go to found search events (set using the **[Search]** key and menu, see "Searching for Events" on page 58).

1 Press **[Navigate]**.
2 In the Navigate Menu, press **Navigate**; then, select **Search**.
3 Press the back and forward keys to go to the previous or next search event.

When searching Serial decode:

- You can press the stop key to set or clear a mark.
- The **Auto zoom** softkey specifies whether the waveform display is automatically zoomed to fit the marked row as you navigate.
- Pressing the **Scroll Lister** softkey lets you use the Entry knob to scroll through data rows in the Lister display.

To navigate segments

When the segmented memory acquisition is enabled and acquisitions are stopped, you can use the navigation controls to play through the acquired segments.

1 Press **[Navigate]**.
2 In the Navigate Menu, press **Navigate**; then, select **Segments**.
3 Press **Play Mode**; then, select:
   - **Manual** — to play through segments manually.

     In the Manual play mode:

     - Press the back and forward keys to go to the previous or next segment.
     - Press the softkey to go to the first segment.
     - Press the softkey to go to the last segment.
   - **Auto** — to play through segments in an automated fashion.

     In the Auto play mode:

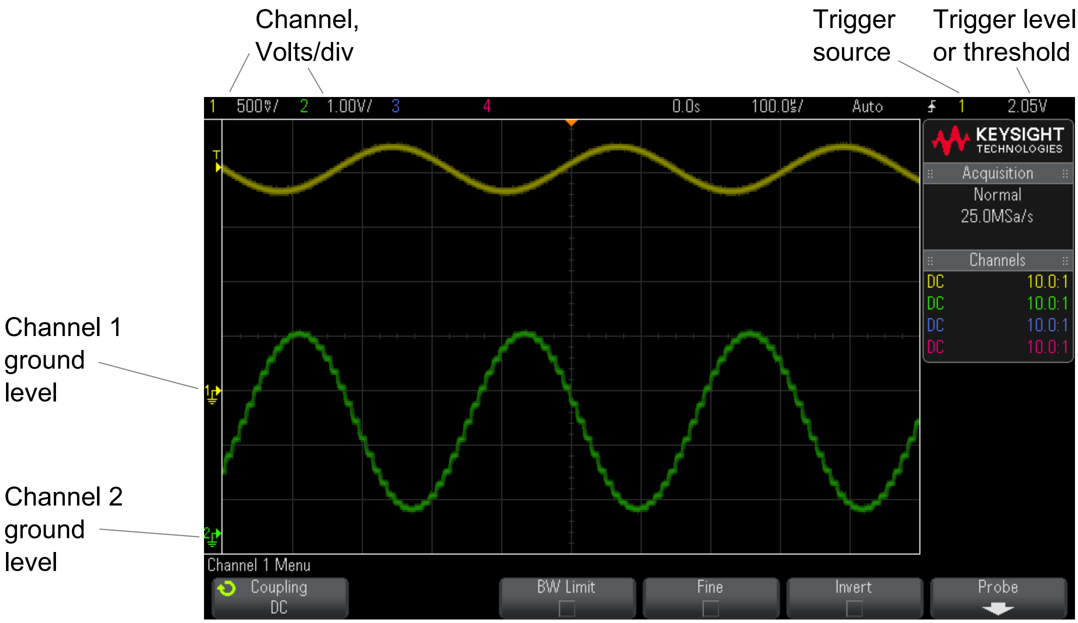
- Press the ◄■► navigation keys to play backward, stop, or play forward in time. You can press the ◄ or ► keys multiple times to speed up the playback. There are three speed levels.

3 Vertical Controls

The vertical controls include:

- The vertical scale and position knobs for each analog channel.
- The channel keys for turning a channel on or off and accessing the channel's softkey menu.

The following figure shows the Channel 1 Menu that appears after pressing the **[1]** channel key.



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

To turn waveforms on or off (channel or math)

1 Press an analog channel key turn the channel on or off (and to display the channel's menu).

When a channel is on, its key is illuminated.

NOTE

Turning channels off

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

To adjust the vertical scale

1 Turn the large knob above the channel key marked to set the vertical scale (volts/division) for the channel.

The vertical scale knob changes the analog channel scale in a 1-2-5 step sequence (with a 1:1 probe attached) unless fine adjustment is enabled (see "To change the vertical scale knob's coarse/fine adjustment setting" on page 66).

The analog channel Volts/Div value is displayed in the status line.

The default mode for expanding the signal when you turn the volts/division knob is vertical expansion about the ground level of the channel; however, you can change this to expand about the center of the display. See "To choose "expand about" center or ground" on page 253.

To adjust the vertical position

1 Turn the small vertical position knob (⬍) to move the channel's waveform up or down on the display.

The voltage value momentarily displayed in the upper right portion of the display represents the voltage difference between the vertical center of the display and the ground level () icon. It also represents the voltage at the vertical center of the display if vertical expansion is set to expand about ground (see "To choose "expand about" center or ground" on page 253).

To specify channel coupling

Coupling changes the channel's input coupling to either **AC** (alternating current) or **DC** (direct current).

TIP

If the channel is DC coupled, you can quickly measure the DC component of the signal by simply noting its distance from the ground symbol.

If the channel is AC coupled, the DC component of the signal is removed, allowing you to use greater sensitivity to display the AC component of the signal.

1 Press the desired channel key.
2 In the Channel Menu, press the **Coupling** softkey to select the input channel coupling:
   - **DC** — DC coupling is useful for viewing waveforms as low as 0 Hz that do not have large DC offsets.
   - **AC** — AC coupling is useful for viewing waveforms with large DC offsets.

     AC coupling places a 10 Hz high-pass filter in series with the input waveform that removes any DC offset voltage from the waveform.

Note that Channel Coupling is independent of Trigger Coupling. To change trigger coupling see "To select the trigger coupling" on page 149.

To specify bandwidth limiting

1 Press the desired channel key.
2 In the Channel Menu, press the **BW Limit** softkey to enable or disable bandwidth limiting.

When bandwidth limit is on, the maximum bandwidth for the channel is approximately 20 MHz. For waveforms with frequencies below this, turning bandwidth limit on removes unwanted high frequency noise from the waveform. The bandwidth limit also limits the trigger signal path of any channel that has **BW Limit** turned on.

To change the vertical scale knob's coarse/fine adjustment setting

1 Push the channel's vertical scale knob (or press the channel key and then the **Fine** softkey in the Channel Menu) to toggle between fine and coarse adjustment of the vertical scale.

When **Fine** adjustment is selected, you can change the channel's vertical sensitivity in smaller increments. The channel sensitivity remains fully calibrated when **Fine** is on.

The vertical scale value is displayed in the status line at the top of the display.

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

To invert a waveform

1 Press the desired channel key.
2 In the Channel Menu, press the **Invert** softkey to invert the selected channel.

When **Invert** is selected, the voltage values of the displayed waveform are inverted.

Invert affects how a channel is displayed. However, when using basic triggers, the oscilloscope attempts to maintain the same trigger point by changing trigger settings.

Inverting a channel also changes the result of any math function selected in the Waveform Math Menu or any measurement.

Setting Analog Channel Probe Options

1 Press the probe's associated channel key.
2 In the Channel Menu, press the **Probe** softkey to display the Channel Probe Menu.

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

   The **Probe Check** softkey guides you through the process of compensating passive probes (such as the N2841A, N2842A, N2843A, N2862A/B, N2863A/B, N2889A, N2890A, 10073C, 10074C, or 1165A probes).

See Also

- "To specify the channel units" on page 68
- "To specify the probe attenuation" on page 68
- "To specify the probe skew" on page 69

To specify the channel units

1 Press the probe's associated channel key.

2 In the Channel Menu, press **Probe**.

3 In the Channel Probe Menu, press **Units**; then, select:

   - **Volts** — for a voltage probe.
   - **Amps** — for a current probe.

Channel sensitivity, trigger level, measurement results, and math functions will reflect the measurement units you have selected.

To specify the probe attenuation

The probe attenuation factor must be set properly for accurate measurement results.

To set the probe attenuation factor:

1 Press the channel key.

2 Press the **Probe** softkey until you have selected how you want to specify the attenuation factor, choosing either **Ratio** or **Decibels**.

3 Turn the Entry knob to set the attenuation factor for the connected probe.

When measuring voltage values, the attenuation factor can be set from 0.001:1 to 10000:1 in a 1-2-5 sequence.

When measuring current values with a current probe, the attenuation factor can be set from 1000 V/A to 0.0001 V/A.

When specifying the attenuation factor in decibels, you can select values from -60 dB to 80 dB.

If Amps is chosen as the units and a manual attenuation factor is chosen, then the units as well as the attenuation factor are displayed above the **Probe** softkey.

To specify the probe skew

When measuring time intervals in the nanoseconds (ns) range, small differences in cable length can affect the measurement. Use **Skew** to remove cable-delay errors between any two channels.

1. Probe the same point with both probes.
2. Press one of the probes associated channel key.
3. In the Channel Menu, press **Probe**.
4. In the Channel Probe Menu, press **Skew**; then, select the desired skew value.

Each analog channel can be adjusted ±100 ns in 10 ps increments for a total of 200 ns difference.

The skew setting is not affected by pressing **[Default Setup]** or **[Auto Scale]**.

4 Math Waveforms

Math functions can be performed on analog channels. The resulting math waveform is displayed in light purple.

You can use a math function on a channel even if you choose not to display the channel on-screen.

You can:

- Perform an arithmetic operation (like add, subtract, or multiply) on analog input channels.
- Perform a transform function (like FFT) on an analog input channel.
- Perform a transform function on the result of an arithmetic operation.

To display math waveforms

1 Press the **[Math]** key on the front panel to display the Waveform Math Menu.

2 If **f(t)** is not already shown on the **Function** softkey, press the **Function** sofkey and select **f(t): Displayed**.

3 Use the **Operator** softkey to select an operator or transform.

For more information on the operators or transforms, see:

- "Math Operators" on page 74
- "Math Transforms" on page 76

4 Use the **Source 1** softkey to select the analog channel on which to perform math. You can rotate the Entry knob or repetitively press the **Source 1** softkey to make your selection. If you choose a transform function (FFT) the result is displayed.

5 If you select an arithmetic operator, use the **Source 2** softkey to select the second source for the arithmetic operation. The result is displayed.

6 To re-size and re-position the math waveform, see "To adjust the math waveform scale and offset" on page 73.

TIP

Math Operating Hints

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

Once the function is displayed, the analog channel(s) may be turned off for better viewing of the math waveform.

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

The math function waveform can be measured using **[Cursors]** and/or **[Meas]**.

To perform a transform function on an arithmetic operation

To perform a transform function (FFT) on an arithmetic operation (add, subtract, or multiply):

1 Press the **Function** softkey and select **g(t): Internal**.

2 Use the **Operator**, **Source 1**, and **Source 2** softkeys to set up an arithmetic operation.

3 Press the **Function** softkey and select **f(t): Displayed**.

4 Use the **Operator** softkey to select a transform function (FFT).

5 Press the **Source 1** softkey and select **g(t)** as the source. Note that **g(t)** is only available when you select a transform function in the previous step.

To adjust the math waveform scale and offset

1 Make sure the multiplexed scale and position knobs to the right of the **[Math]** key are selected for the math waveform.

   If the arrow to the left of the **[Math]** key is not illuminated, press the key.

2 Use the multiplexed scale and position knobs just to the right of the **[Math]** key to re-size and re-position the math waveform.

NOTE

Math Scale and Offset are Set Automatically

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

See Also

- "Units for Math Waveforms" on page 73

Units for Math Waveforms

Units for each input channel can be set to Volts or Amps using the **Units** softkey in the channel's Probe Menu. Units for math function waveforms are:

Math function	Units
add or subtract	V or A
multiply	V^2, A^2, or W (Volt-Amp)
FFT	dB* (decibels). See also "FFT Units" on page 81.

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

A scale unit of **U** (undefined) will be displayed for math functions when two source channels are used and they are set to dissimilar units and the combination of units cannot be resolved.

Math Operators

Math operators perform arithmetic operations (add, subtract, or multiply) on analog input channels.

- "Add or Subtract" on page 74
- "Multiply or Divide" on page 75

Add or Subtract

When you select add or subtract, the **Source 1** and **Source 2** values are added or subtracted point by point, and the result is displayed.

You can use subtract to make a differential measurement or to compare two waveforms.

If your waveforms' DC offsets are larger than the dynamic range of the oscilloscope's input channels you will need to use a differential probe instead.

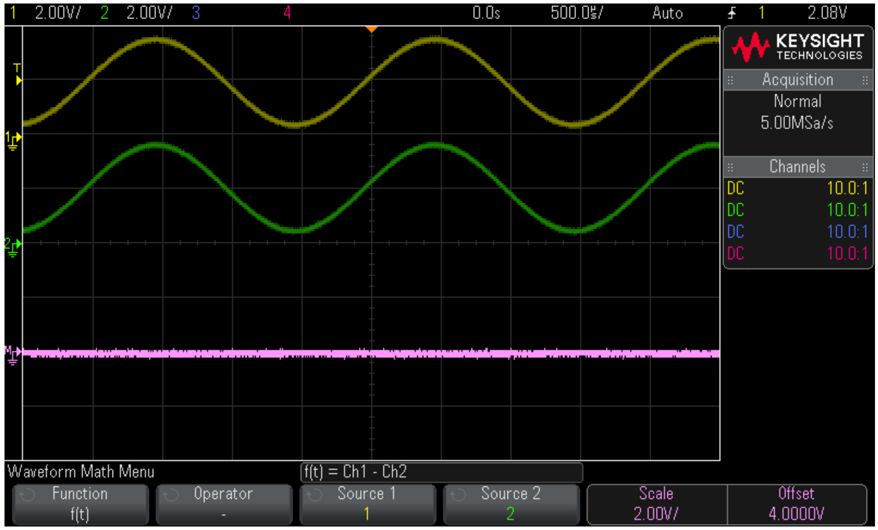


Figure 5 Example of Subtract Channel 2 from Channel 1

See Also

- "Units for Math Waveforms" on page 73

Multiply or Divide

When you select the multiply or divide math function, the **Source 1** and **Source 2** values are multiplied or divided point by point, and the result is displayed.

The divide by zero case places holes (that is, zero values) in the output waveform.

Multiply is useful for seeing power relationships when one of the channels is proportional to the current.

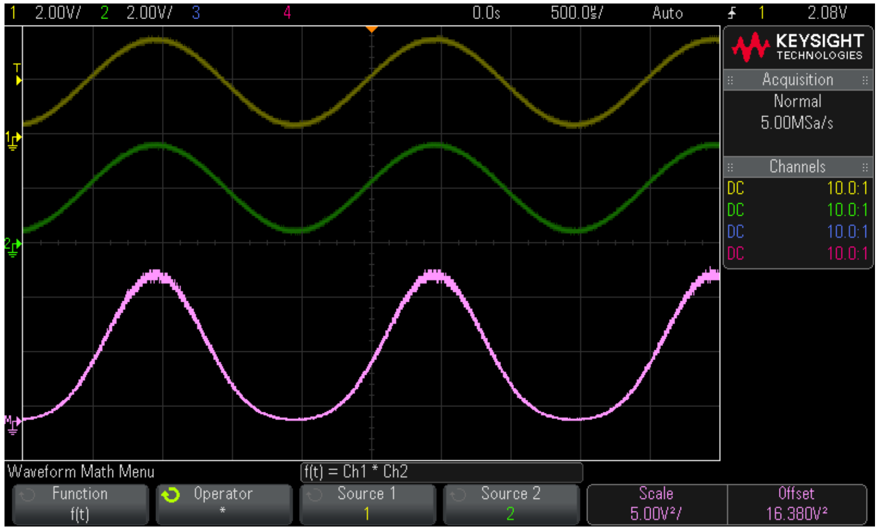


Figure 6 Example of Multiply Channel 1 by Channel 2

See Also · "Units for Math Waveforms" on page 73

Math Transforms

Math transforms perform a transform function (FFT) on an analog input channel or on the result of an arithmetic operation.

- "FFT Measurement" on page 76

FFT Measurement

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

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

To display a FFT waveform:

1 Press the **[Math]** key, press the **Function** softkey and select **f(t)**, press the **Operator** softkey and select **FFT**.

- **Source 1** — selects the source for the FFT. (See "To perform a transform function on an arithmetic operation" on page 72 for information about using **g(t)** as the source.)
- **Span** — sets the overall width of the FFT spectrum that you see on the display (left to right). Divide span by 10 to calculate the number of Hertz per division. It is possible to set Span above the maximum available frequency, in which case the displayed spectrum will not take up the whole screen. Press the **Span** softkey, then turn the Entry knob to set the desired frequency span of the display.
- **Center** — sets the FFT spectrum frequency represented at the center vertical grid line of the display. It is possible to set the Center to values below half the span or above the maximum available frequency, in which case the displayed spectrum will not take up the whole screen. Press the **Center** softkey, then turn the Entry knob to set the desired center frequency of the display.
- **Scale** — lets you set your own vertical scale factors for FFT expressed in dB/div (decibels/division). See "To adjust the math waveform scale and offset" on page 73.
- **Offset** — lets you set your own offset for the FFT. The offset value is in dB and is represented by the center horizontal grid line of the display. See "To adjust the math waveform scale and offset" on page 73.
- **More FFT** — displays the More FFT Settings Menu.

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

- **Window**– selects a window to apply to your FFT input signal:
  - **Hanning** — window for making accurate frequency measurements or for resolving two frequencies that are close together.
  - **Flat Top** — window for making accurate amplitude measurements of frequency peaks.
  - **Rectangular** — good frequency resolution and amplitude accuracy, but use only where there will be no leakage effects. Use on self-windowing waveforms such as pseudo-random noise, impulses, sine bursts, and decaying sinusoids.
  - **Blackman Harris** — window reduces time resolution compared to a rectangular window, but improves the capacity to detect smaller impulses due to lower secondary lobes.
- **Vertical Units** — lets you select Decibels or V RMS as the units for the FFT vertical scale.
- **Auto Setup** — sets the frequency Span and Center to values that will cause the entire available spectrum to be displayed. The maximum available frequency is half the FFT sample rate, which is a function of the time per division setting. The FFT resolution is the quotient of the sampling rate and the number of FFT points (f_S/N). The current FFT Resolution is displayed above the softkeys.

NOTE

Scale and offset considerations

If you do not manually change the FFT scale or offset settings, when you turn the horizontal scale knob, the span and center frequency settings will automatically change to allow optimum viewing of the full spectrum.

If you do manually set scale or offset, turning the horizontal scale knob will not change the span or center frequency settings, allowing you see better detail around a specific frequency.

Pressing the FFT **Auto Setup** softkey will automatically rescale the waveform and span and center will again automatically track the horizontal scale setting.

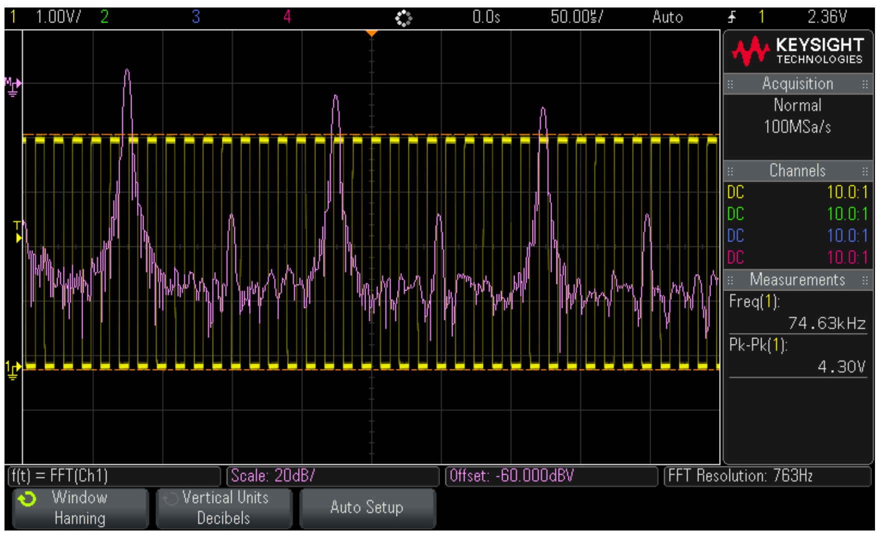
3 To make cursor measurements, press the **[Cursors]** key and set the **Source** softkey to **Math: f(t)**.

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

4 To make other measurements, press the **[Meas]** key and set the **Source** softkey to **Math: f(t)**.

You can make peak-to-peak, maximum, minimum, and average dB measurements on the FFT waveform.

The following FFT spectrum was obtained by connecting a 4 V, 75 kHz square wave to channel 1. Set the horizontal scale to 50 µs/div, vertical sensitivity to 1 V/div, Units/div to 20 dBV, Offset to -60.0 dBV, Center frequency to 250 kHz, frequency Span to 500 kHz, and window to Hanning.



See Also

- "To perform a transform function on an arithmetic operation" on page 72
- "FFT Measurement Hints" on page 80
- "FFT Units" on page 81
- "FFT DC Value" on page 81
- "FFT Aliasing" on page 81
- "FFT Spectral Leakage" on page 83
- "Units for Math Waveforms" on page 73

FFT Measurement Hints

The number of points acquired for the FFT record can be up to 65,536, and when frequency span is at maximum, all points are displayed. Once the FFT spectrum is displayed, the frequency span and center frequency controls are used much like the controls of a spectrum analyzer to examine the frequency of interest in greater detail. Place the desired part of the waveform at the center of the screen and decrease frequency span to increase the display resolution. As frequency span is decreased, the number of points shown is reduced, and the display is magnified.

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

NOTE

FFT Resolution

The FFT resolution is the quotient of the sampling rate and the number of FFT points (f_S/N). With a fixed number of FFT points (up to 65,536), the lower the sampling rate, the better the resolution.

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

For the best vertical accuracy on peak measurements:

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

For best frequency accuracy on peaks:

- Use the Hanning window.
- Use Cursors to place an X cursor on the frequency of interest.

- Adjust frequency span for better cursor placement.
- Return to the Cursors Menu to fine tune the X cursor.

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

FFT Units

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

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

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

FFT DC Value

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

FFT Aliasing

When using FFTs, it is important to be aware of frequency aliasing. This requires that the operator have some knowledge as to what the frequency domain should contain, and also consider the sampling rate, frequency span, and oscilloscope vertical bandwidth when making FFT measurements. The FFT resolution (the quotient of the sampling rate and the number of FFT points) is displayed directly above the softkeys when the FFT Menu is displayed.

NOTE

Nyquist Frequency and Aliasing in the Frequency Domain

The Nyquist frequency is the highest frequency that any real-time digitizing oscilloscope can acquire without aliasing. This frequency is half of the sample rate. Frequencies above the Nyquist frequency will be under sampled, which causes aliasing. The Nyquist frequency is also called the folding frequency because aliased frequency components fold back from that frequency when viewing the frequency domain.

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

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

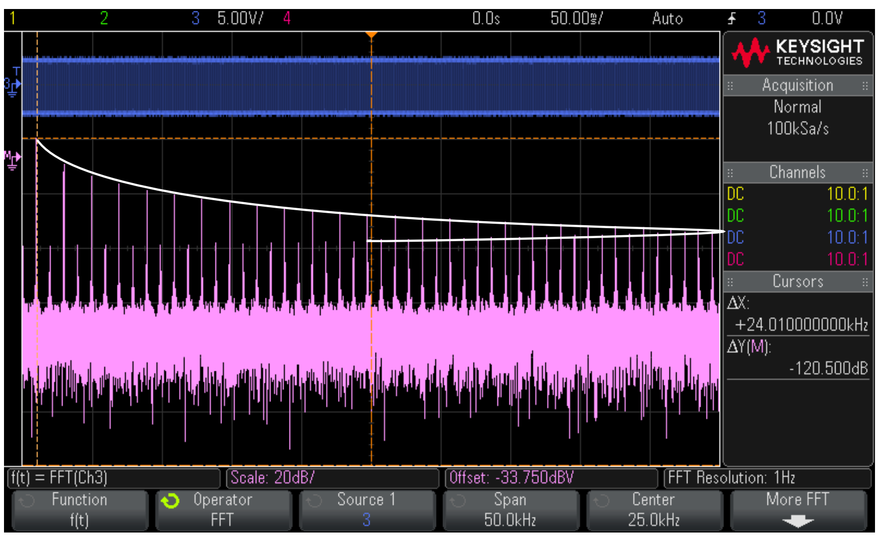


Figure 7 Aliasing

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

FFT Spectral Leakage

The FFT operation assumes that the time record repeats. Unless there is an integral number of cycles of the sampled waveform in the record, a discontinuity is created at the end of the record. This is referred to as leakage. In order to minimize spectral leakage, windows that approach zero smoothly at the beginning and end of the signal are employed as filters to the FFT. The FFT Menu provides four windows: Hanning, Flat Top, Rectangular, and Blackman-Harris. For more information on leakage, see Keysight Application Note 243, *The Fundamentals of Signal Analysis* at
http://literature.cdn.keysight.com/litweb/pdf/5952-8898E.pdf.

6 Digital Channels

This chapter describes how to use the digital channels of a Mixed-Signal Oscilloscope (MSO).

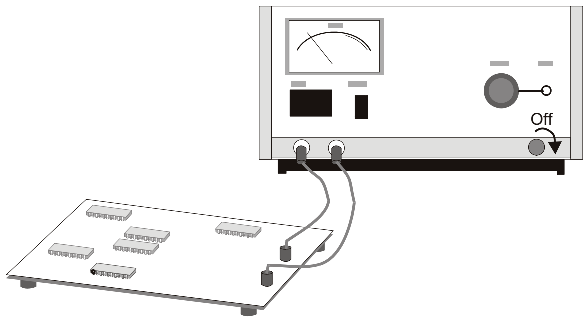
The digital channels are enabled on MSOX2000 X-Series models and DSOX2000 X-Series models that have the DSOX2MSO upgrade license installed.

Digital channels and serial decode cannot be on at the same time. The **[Serial]** key takes precedence over the **[Digital]** key. Serial triggers can be used when digital channels are on.

To connect the digital probes to the device under test

1 If necessary, turn off the power supply to the device under test.

Turning off power to the device under test would only prevent damage that might occur if you accidentally short two lines together while connecting probes. You can leave the oscilloscope powered on because no voltage appears at the probes.



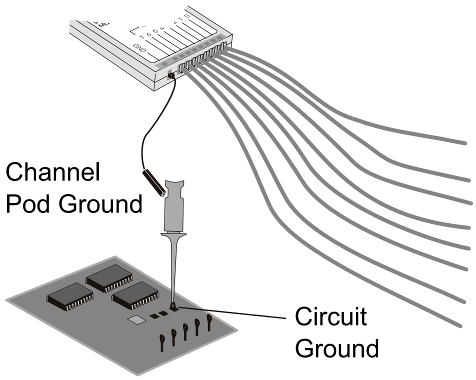
2 Connect the digital probe cable to the DIGITAL Dn - D0 connector on the front panel of the mixed-signal oscilloscope. The digital probe cable is keyed so you can connect it only one way. You do not need to power-off the oscilloscope.

CAUTION

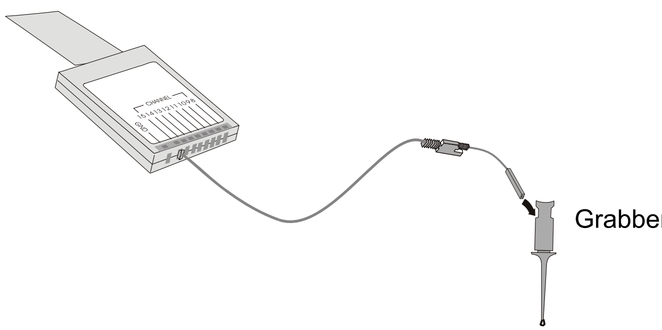
⚠ Probe cable for digital channels

Use only the Keysight logic probe and accessory kit supplied with the mixed-signal oscilloscope (see "Probes and Accessories" on page 285).

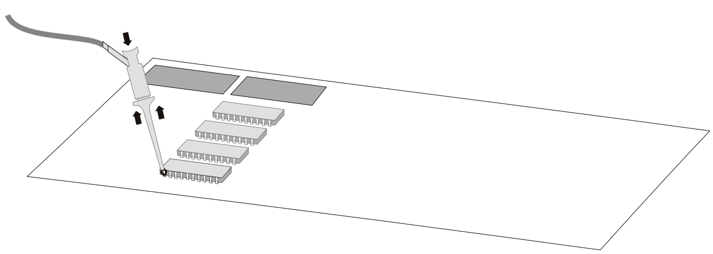
3 Connect the ground lead on each set of channels (each pod), using a probe grabber. The ground lead improves signal fidelity to the oscilloscope, ensuring accurate measurements.



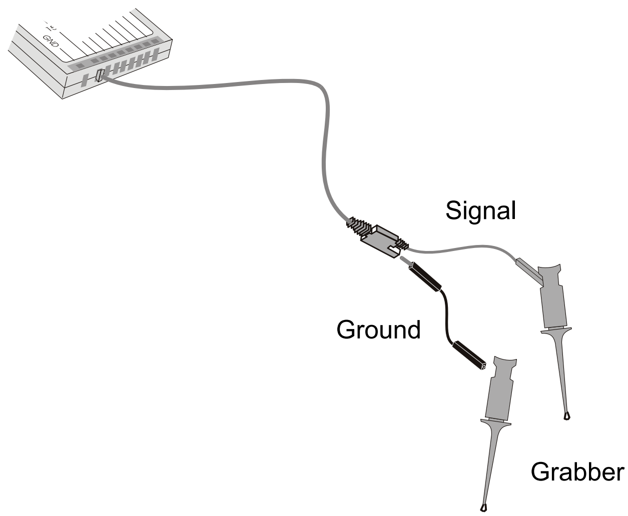
4 Connect a grabber to one of the probe leads. (Other probe leads are omitted from the figure for clarity.)



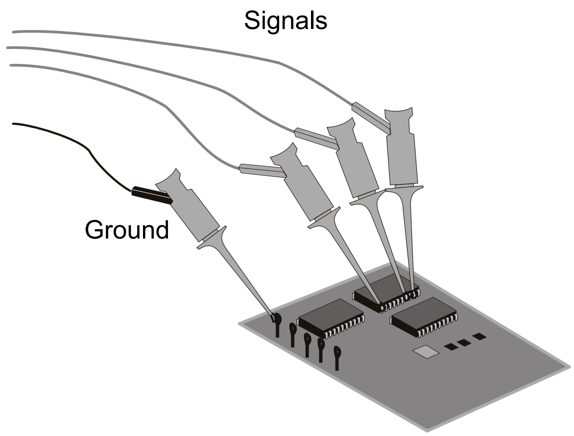
5 Connect the grabber to a node in the circuit you want to test.



6 For high-speed signals, connect a ground lead to the probe lead, connect a grabber to the ground lead, and attach the grabber to ground in the device under test.



7 Repeat these steps until you have connected all points of interest.



Acquiring waveforms using the digital channels

When you press **[Run/Stop]** or **[Single]** to run the oscilloscope, the oscilloscope examines the input voltage at each input probe. When the trigger conditions are met the oscilloscope triggers and displays the acquisition.

For digital channels, each time the oscilloscope takes a sample it compares the input voltage to the logic threshold. If the voltage is above the threshold, the oscilloscope stores a 1 in sample memory; otherwise, it stores a 0.

To display digital channels using AutoScale

When signals are connected to the digital channels — be sure to connect the ground leads — AutoScale quickly configures and displays the digital channels.

- To configure the instrument quickly, press the **[AutoScale]** key.

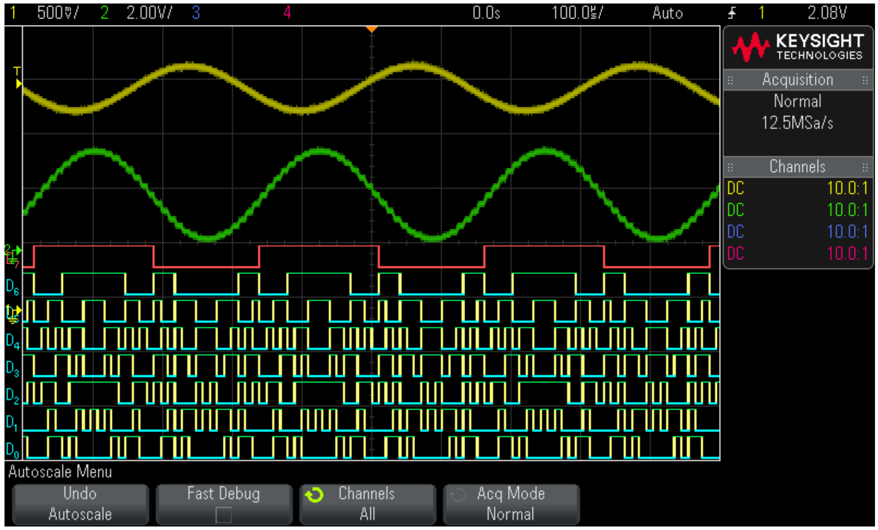


Figure 8 Example: AutoScale of digital channels (MSO models only)

Any digital channel with an active signal will be displayed. Any digital channels without active signals will be turned off.

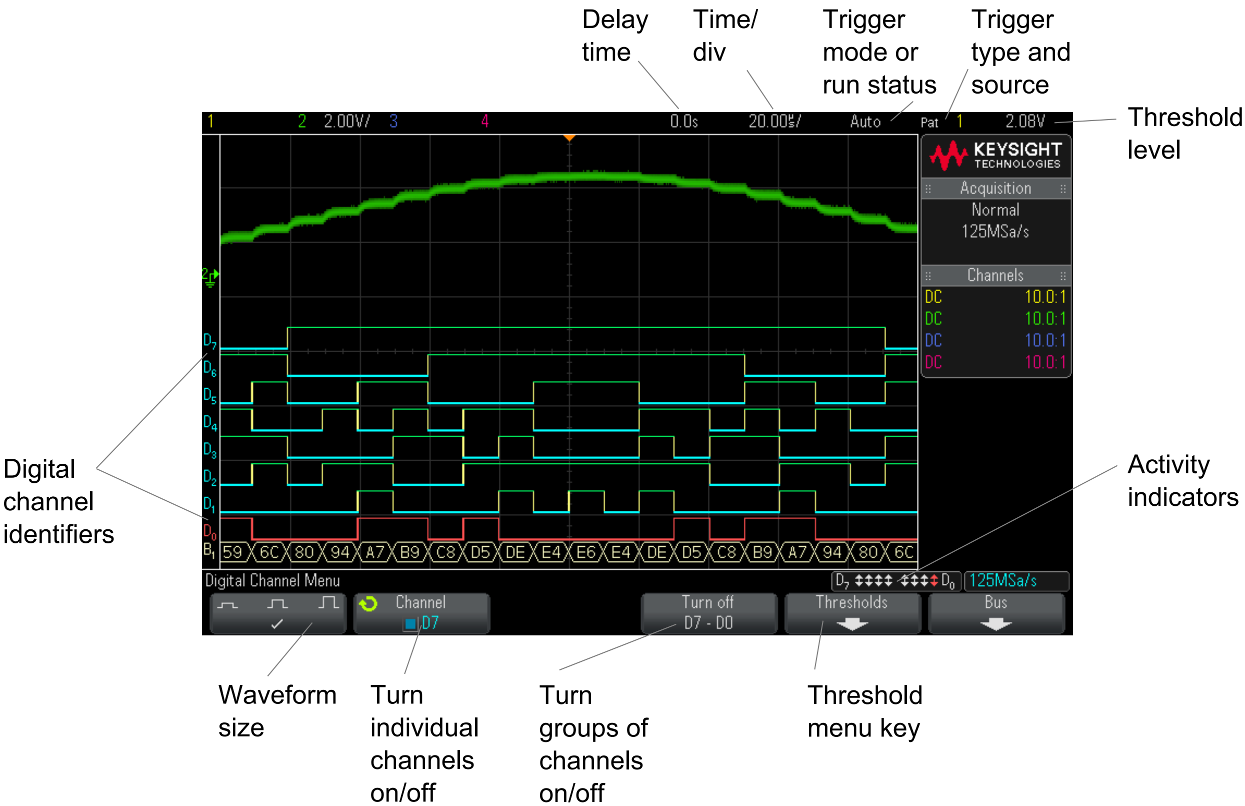
- To undo the effects of AutoScale, press the **Undo AutoScale** softkey before pressing any other key.

This is useful if you have unintentionally pressed the **[AutoScale]** key or do not like the settings AutoScale has selected. This will return the oscilloscope to its previous settings. See also: "How AutoScale Works" on page 31.

To set the instrument to the factory-default configuration, press the **[Default Setup]** key.

Interpreting the digital waveform display

The following figure shows a typical display with digital channels.



Activity indicator When any digital channels are turned on, an activity indicator is displayed in the status line at the bottom of the display. A digital channel can be always high (▀), always low (▄), or actively toggling logic states (↕). Any channel that is turned off will be grayed out in the activity indicator.

To change the displayed size of the digital channels

1 Press the **[Digital]** key.

2 Press the size (⎍ ⎍ ⎍) softkey to select how the digital channels are displayed.

The sizing control lets you spread out or compress the digital traces vertically on the display for more convenient viewing.

To switch a single channel on or off

1 With the Digital Channel Menu displayed, rotate the Entry knob to select the desired channel from the popup menu.
2 Push the Entry knob or press the softkey that is directly below the popup menu to switch the selected channel on or off.

To switch all digital channels on or off

1 Press the **[Digital]** key to toggle the display of digital channels. The Digital Channel Menu is displayed above the softkeys.

If you want to switch the digital channels off, and the Digital Channel Menu is not already displayed, you must push the **[Digital]** key twice to switch the digital channels off. The first push displays the Digital Channel Menu, and the second push switches the channels off.

To switch groups of channels on or off

1 Press the **[Digital]** key on the front panel if the Digital Channel Menu is not already displayed.
2 Press the **Turn off** (or **Turn on**) softkey for the **D7 - D0** group.

Each time you press the softkey, the softkey's mode toggles between **Turn on** and **Turn off**.

To change the logic threshold for digital channels

1 Press the **[Digital]** key so that the Digital Channel Menu is displayed.
2 Press the **Thresholds** softkey

3 Press the **D7 - D0** softkey, then select a logic family preset or select **User** to define your own threshold.

Logic family	Threshold Voltage
TTL	+1.4 V
CMOS	+2.5 V
ECL	–1.3 V
User	Variable from –8 V to +8 V

The threshold you set applies to all channels within the selected D7 - D0 group. Each of the two channel groups can be set to a different threshold if desired.

Values greater than the set threshold are high (1) and values less than the set threshold are low (0).

If the **Thresholds** softkey is set to **User**, press the **User** softkey for the channel group, then turn the Entry knob to set the logic threshold. There is one **User** softkey for each group of channels.

To reposition a digital channel

1 Make sure the multiplexed scale and position knobs to the right of the key are selected for digital channels.

   If the arrow to the left of the **[Digital]** key is not illuminated, press the key.

2 Use the multiplexed Select knob to select the channel.

   The selected waveform is highlighted in red.

3 Use the multiplexed Position knob to move the selected channel waveform.

   If a channel waveform is repositioned over another channel waveform, the indicator at the left edge of the trace will change from **D**nn designation (where nn is a one or two digit channel number) to **D***. The "*" indicates that two channels are overlaid.

To display digital channels as a bus

Digital channels may be grouped and displayed as a bus, with each bus value displayed at the bottom of the display in hex or binary. You can create up to two buses. To configure and display each bus, press the **[Digital]** key on the front panel. Then press the **Bus** softkey.

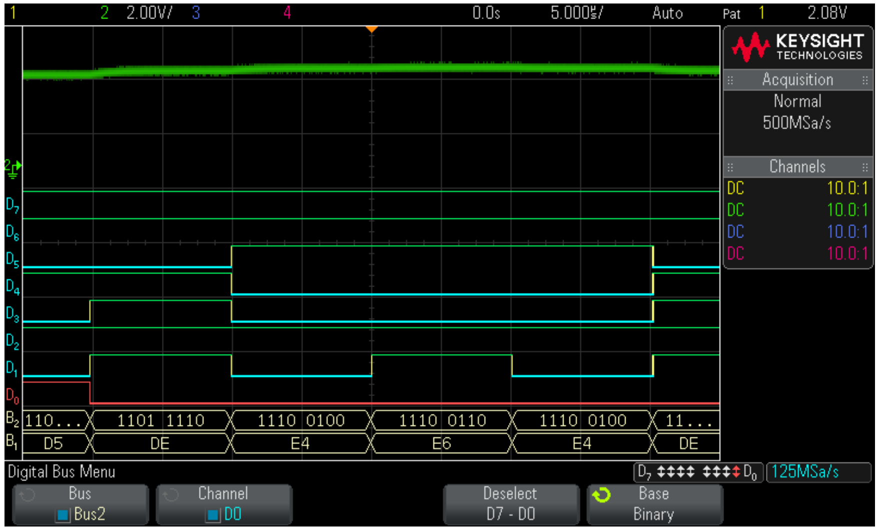
Next, select a bus. Rotate the Entry knob, then press the Entry knob or the **Bus1/Bus2** softkey to switch it on.

Use the **Channel** softkey and the Entry knob to select individual channels to be included in the bus. You can rotate the Entry knob and push it or push the softkey to select channels. You can also press the **Select/Deselect D7-D0** softkey to include or exclude groups of eight channels in each bus.

If the bus display is blank, completely white, or if the display includes "...", you need to expand the horizontal scale to allow space for the data to be shown, or use the cursors to display the values (see "Using cursors to read bus values" on page 99).

The **Base** softkey lets you choose to display the bus values in hex or binary.

The buses are shown at the bottom of the display.

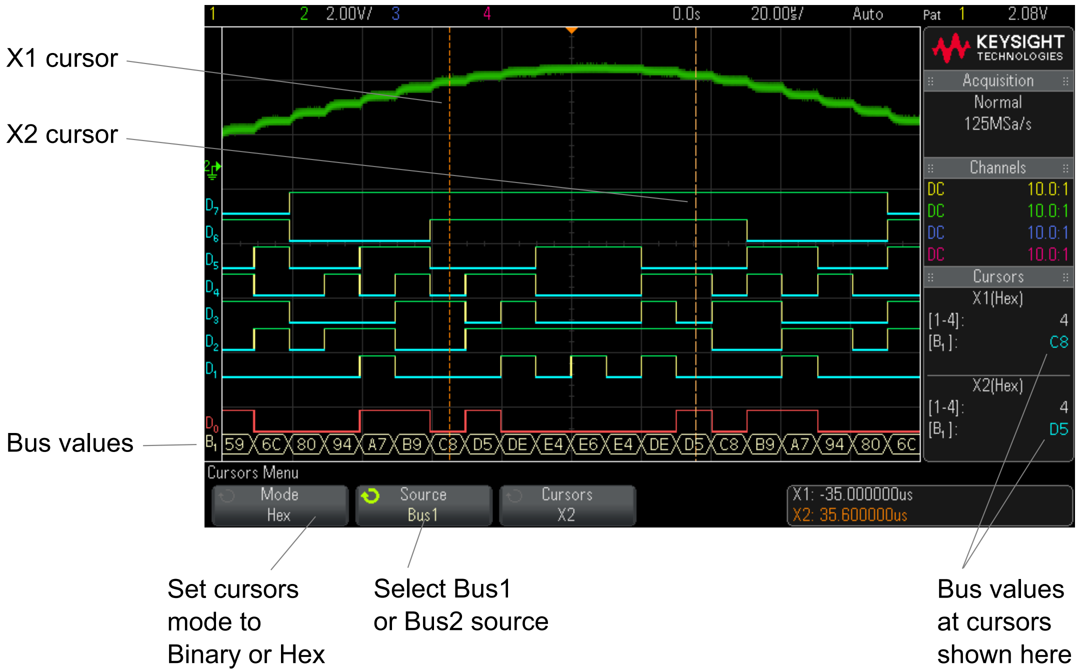


Bus values can be shown in hex or binary.

Using cursors to read bus values

To read the digital bus value at any point using the cursors:

1. Turn on Cursors (by pressing the **[Cursors]** key on the front panel)
2. Press the cursor **Mode** softkey and change the mode to **Hex** or **Binary**.
3. Press the **Source** softkey and select **Bus1** or **Bus2**.
4. Use the Entry knob and the **X1** and **X2** softkeys to position the cursors where you want to read the bus values.

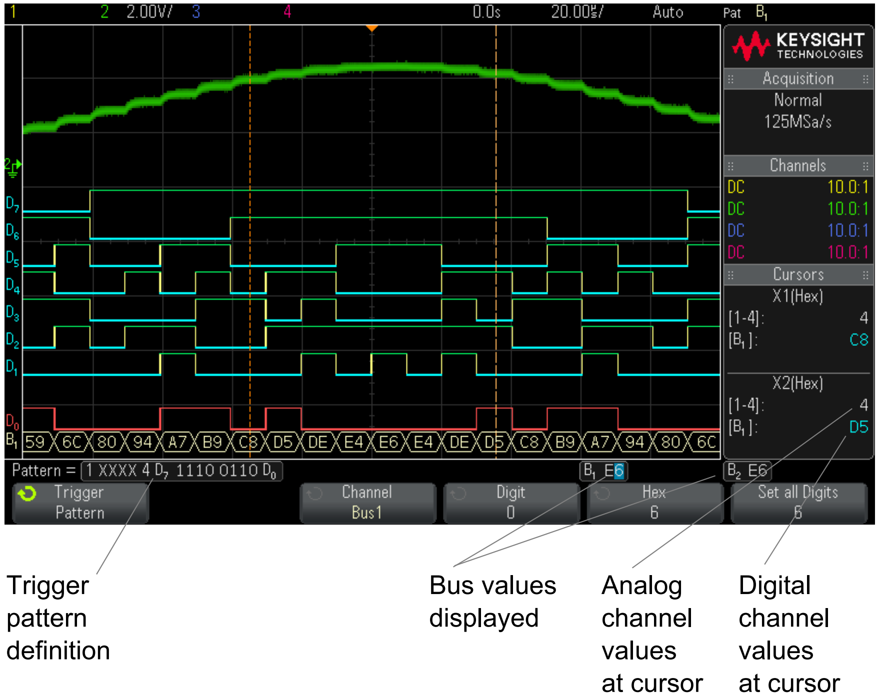


When you press the **[Digital]** key to display the Digital Channel Menu, the digital activity indicator is shown where the cursor values were and the bus values at the cursors are displayed in the graticule.

Bus values are displayed when using Pattern trigger

The bus values are also displayed when using the Pattern trigger function. Press the **[Pattern]** key on the front panel to display the Pattern Trigger Menu and the bus values will be displayed on the right, above the softkeys.

The dollar sign ($) will be displayed in the bus value when the bus value cannot be displayed as a hex value. This occurs when one or more "don't cares" (X) are combined with low (0) and high (1) logic levels in the pattern specification, or when a transition indicator — rising edge (↑) or falling edge (↓) — are included in the pattern specification. A byte that consists of all don't cares (X) will be displayed in the bus as a don't care (X).



See "Pattern Trigger" on page 130 for more information on Pattern triggering.

Digital channel signal fidelity: Probe impedance and grounding

When using the mixed-signal oscilloscope you may encounter problems that are related to probing. These problems fall into two categories: probe loading and probe grounding. Probe loading problems generally affect the device under test, while probe grounding problems affect the accuracy of the data to the measurement instrument. The design of the probes minimizes the first problem, while the second is easily addressed by good probing practices.

Input Impedance

The logic probes are passive probes, which offer high input impedance and high bandwidths. They usually provide some attenuation of the signal to the oscilloscope, typically 20 dB.

Passive probe input impedance is generally specified in terms of a parallel capacitance and resistance. The resistance is the sum of the tip resistor value and the input resistance of the test instrument (see the following figure). The capacitance is the series combination of the tip compensating capacitor and the cable, plus instrument capacitance in parallel with the stray tip capacitance to ground. While this results in an input impedance specification that is an accurate model for DC and low frequencies, the high-frequency model of the probe input is more useful (see the following figure). This high-frequency model takes into account pure tip capacitance to ground as well as series tip resistance, and the cable's characteristic impedance (Z_o).

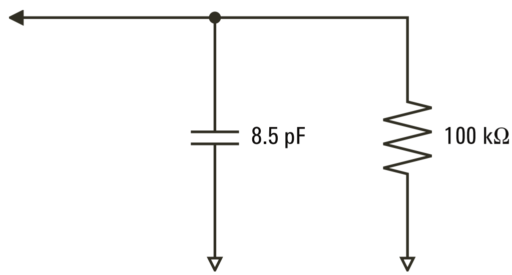


Figure 9 DC and Low-Frequency Probe Equivalent Circuit

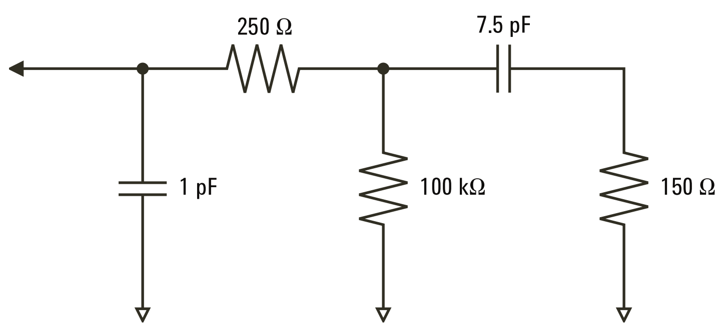


Figure 10 High-Frequency Probe Equivalent Circuit

The impedance plots for the two models are shown in these figures. By comparing the two plots, you can see that both the series tip resistor and the cable's characteristic impedance extend the input impedance significantly. The stray tip capacitance, which is generally small (1 pF), sets the final break point on the impedance chart.

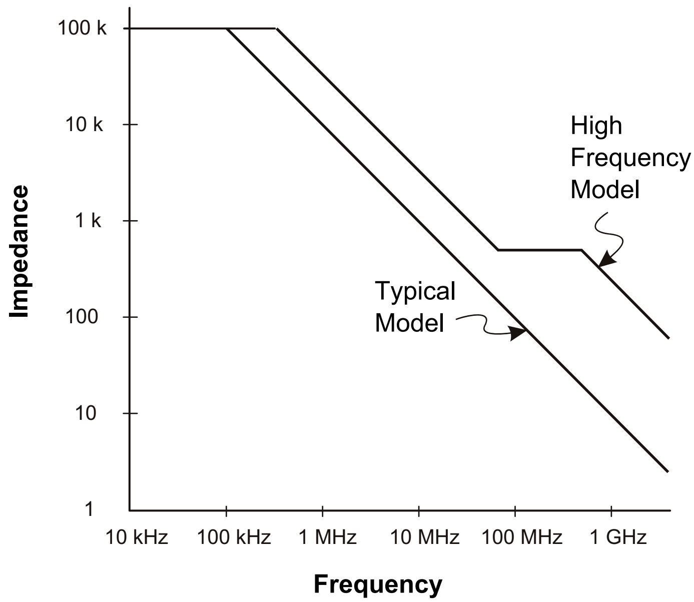


Figure 11 Impedance versus Frequency for Both Probe Circuit Models

The logic probes are represented by the high-frequency circuit model shown above. They are designed to provide as much series tip resistance as possible. Stray tip capacitance to ground is minimized by the proper mechanical design of the probe tip assembly. This provides the maximum input impedance at high frequencies.

Probe Grounding

A probe ground is the low-impedance path for current to return to the source from the probe. Increased length in this path will, at high frequencies, create large common mode voltages at the probe input. The voltage generated behaves as if this path were an inductor according to the equation:

$$V = L\frac{di}{dt}$$

Increasing the ground inductance (L), increasing the current (di) or decreasing the transition time (dt), will all result in increasing the voltage (V). When this voltage exceeds the threshold voltage defined in the oscilloscope, a false data measurement will occur.

Sharing one probe ground with many probes forces all the current that flows into each probe to return through the same common ground inductance of the probe whose ground return is used. The result is increased current (di) in the above equation, and, depending on the transition time (dt), the common mode voltage may increase to a level that causes false data generation.

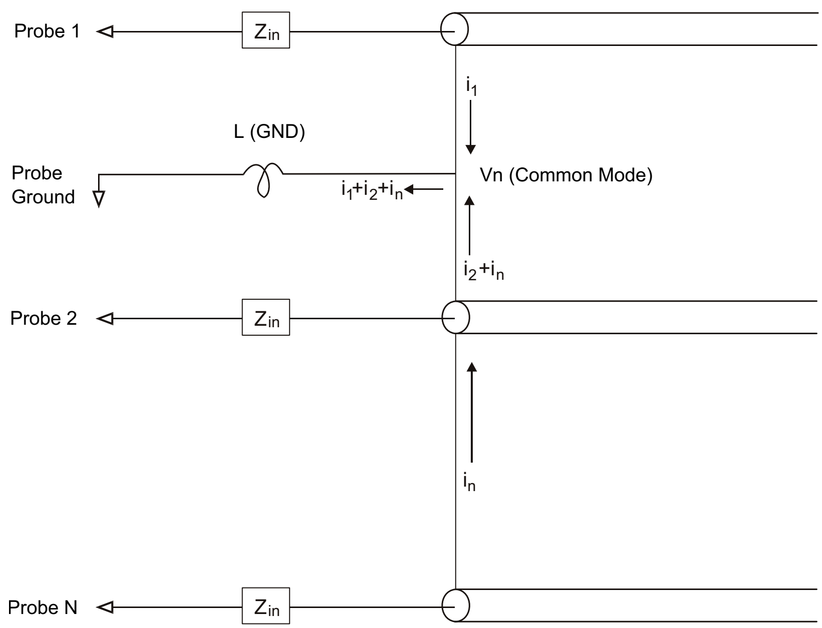


Figure 12 Common Mode Input Voltage Model

In addition to the common mode voltage,longer ground returns also degrade the pulse fidelity of the probe system. Rise time is increased, and ringing, due to the undamped LC circuit at the input of the probe, is also increased. Because the digital channels display reconstructed waveforms, they do not show ringing and perturbations. You will not find ground problems through examination of the waveform display. In fact, it is likely you will discover the problem through random glitches or inconsistent data measurements. Use the analog channels to view ringing and perturbations.

Best Probing Practices

Because of the variables L, di, and dt, you may be unsure how much margin is available in your measurement setup. The following are guidelines for good probing practices:

- The ground lead from each digital channel group (D15–D8 and D7–D0) should be attached to the ground of the device under test if any channel within the group is being used for data capture.
- When capturing data in a noisy environment, every third digital channel probe's ground should be used in addition to the channel group's ground.
- High-speed timing measurements (rise time < 3 ns) should make use of each digital channel probe's own ground.

When designing a high-speed digital system, you should consider designing dedicated test ports that interface directly to the instrument's probe system. This will ease measurement setup and ensure a repeatable method for obtaining test data. The 01650-61607 16-channel logic probe cable and the 01650-63203 termination adapter are designed to make it easy to connect to industry-standard, 20-pin board connectors. The cable is a 2 m logic analyzer probe cable, and the termination adapter provides the proper RC networks in a very convenient package. These parts, as well as the 1251-8106 20-pin, low-profile, straight board connector, can be ordered from Keysight Technologies.

8 Display Settings

To adjust waveform intensity

You can adjust the intensity of displayed waveforms to account for various signal characteristics, such as fast time/div settings and low trigger rates.

Increasing the intensity lets you see the maximum amount of noise and infrequently occurring events.

Reducing the intensity can expose more detail in complex signals as shown in the following figures.

1 Press the **[Intensity]** key to illuminate it.

   This key is located just below the Entry knob.

2 Turn the Entry knob to adjust the waveform instensity.

Waveform intensity adjustment affects analog channel waveforms only (not math waveforms, referencewaveforms, digital waveforms, etc.).

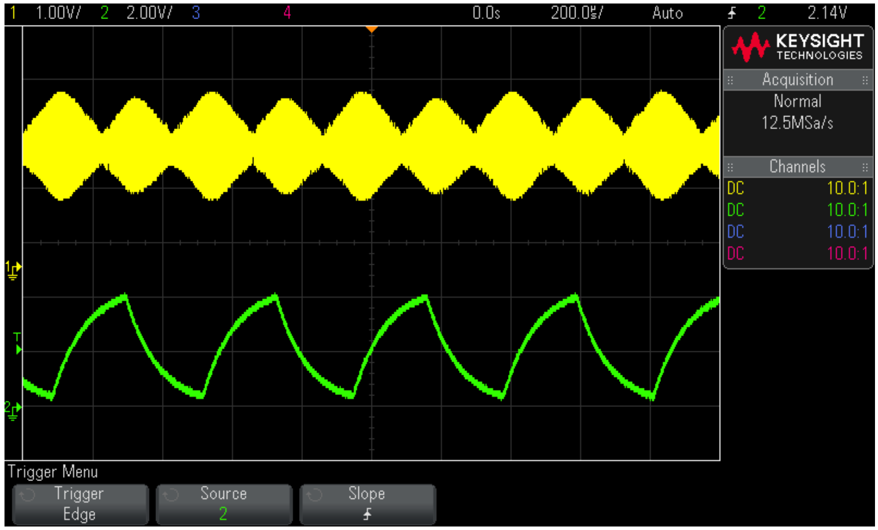


Figure 13 Amplitude Modulation Shown at 100% Intensity

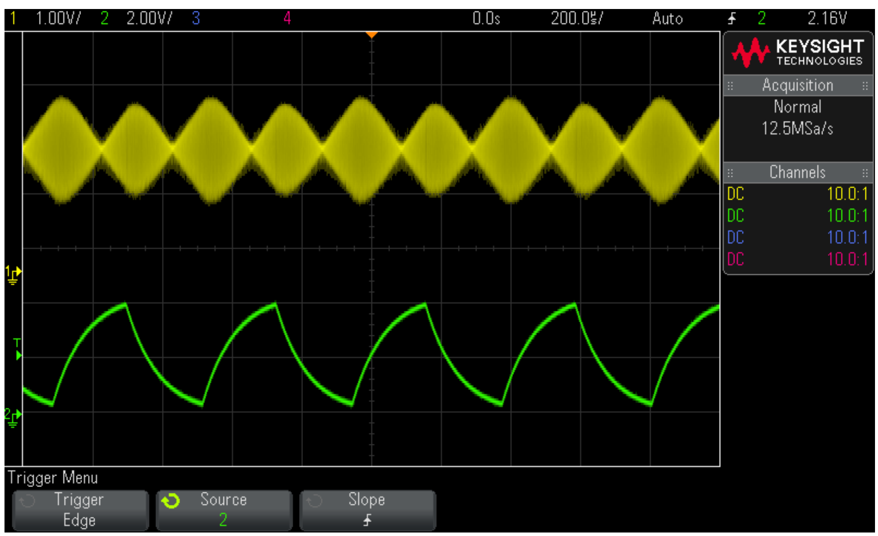


Figure 14 Amplitude Modulation Shown at 40% Intensity

To set or clear persistence

With persistence, the oscilloscope updates the display with new acquisitions, but does not immediately erase the results of previous acquisitions. All previous acquisitions are displayed with reduced intensity. New acquisitions are shown in their normal color with normal intensity.

Waveform persistence is kept only for the current display area; you cannot pan and zoom the persistence display.

To use persistence:

1 Press the **[Display]** key.

2 Press **Persistence**; then, turn the Entry knob to select between:

- **Off** — turns off persistence.

  When persistence is off, you can press the **Capture Waveforms** softkey to perform a single-shot infinite persistence. A single acquisition's data is displayed with reduced intensity, and it remains on the display until you clear persistence or clear the display.

- ∞ Persistence — (infinite presistence) Results of previous acquisitions are never erased.

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

- **Variable Persistence** — Results of previous acquisitions are erased after a certain amount of time.

  Variable persistence gives you a view of acquired data that is similar to analog oscilloscopes.

  When variable persistence is selected, press the **Time** softkey and use the Entry knob to specify the amount of time that previous acquisitions are to be displayed.

The display will begin accumulating multiple acquisitions.

3 To erase the results of previous acquisitions from the display, press the **Clear Persistence** softkey.

   The oscilloscope will start to accumulate acquisitions again.

4 To return the oscilloscope to the normal display mode, turn off persistence; then, press the **Clear Persistence** softkey.

   Turning off persistence does not clear the display. The display is cleared if you press the **Clear Display** softkey or if you press the **[AutoScale]** key (which also turns off persistence).

For another method of seeing worst-case extremes of varying waveforms, see "Glitch or Narrow Pulse Capture" on page 162.

To clear the display

1 Press **[Display] > Clear Display**.

You can also configure the **[Quick Action]** key to clear the display. See "Configuring the [Quick Action] Key" on page 262.

To select the grid type

When the **Video** trigger type is selected (see "Video Trigger" on page 135), and the vertical scaling of at least one displayed channel is 140 mV/div, the **Grid** softkey lets you select from these grid types:

- **Full** — the normal oscilloscope grid.
- **mV** — shows vertical grids, labeled on the left, from -0.3 V to 0.8 V.
- **IRE** — (Institute of Radio Engineers) shows vertical grids in IRE units, labeled on the left, from -40 to 100 IRE. The 0.35 V and 0.7 V levels from the **mV** grid are also shown and labeled at the right. When the **IRE** grid is selected, cursor values are also shown in IRE units. (Cursor values via the remote interface are not in IRE units.)

The **mV** and **IRE** grid values are accurate (and match Y cursor values) when the vertical scaling is 140 mV/division and the vertical offset is 245 mV.

To select the grid type:

1 Press **[Display]**.

2 Press the **Grid** softkey; then, turn the Entry knob ↻ to select the grid type.

To adjust the grid intensity

To adjust the display grid (graticule) intensity:

1 Press **[Display]**.

2 Press the **Intensity** softkey; then, turn the Entry knob ↻ to change the intensity of the displayed grid.

The intensity level is shown in the **Intensity** softkey and is adjustable from 0 to 100%.

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

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

To freeze the display

To freeze the display without stopping running acquisitions, you must configure the **[Quick Action]** key. See "Configuring the [Quick Action] Key" on page 262.

1 Once the **[Quick Action]** key has been configured, press it to freeze the display.

2 To un-freeze the display, press **[Quick Action]** again.

Manual cursors can be used on the frozen display.

Many activities, such as adjusting the trigger level, adjusting vertical or horizontal settings, or saving data will un-freeze the display.

10 Triggers

A trigger setup tells the oscilloscope when to acquire and display data. For example, you can set up to trigger on the rising edge of the analog channel 1 input signal.

You can adjust the vertical level used for analog channel edge detection by turning the Trigger Level knob.

In addition to the edge trigger type, you can also set up triggers on pulse widths, patterns, and video signals.

You can use any input channel or the "External Trigger Input" on page 152 BNC as the source for most trigger types.

Changes to the trigger setup are applied immediately. If the oscilloscope is stopped when you change a trigger setup, the oscilloscope uses the new specification when you press **[Run/Stop]** or **[Single]**. If the oscilloscope is running when you change a trigger setup, it uses the new trigger definition when it starts the next acquisition.

You can use the **[Force Trigger]** key to acquire and display data when triggers are not occurring.

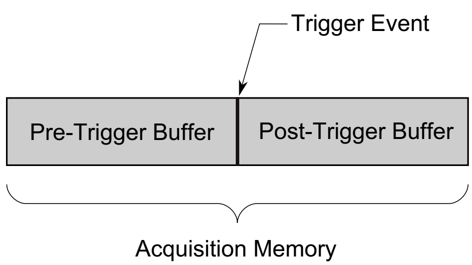
You can use the **[Mode/Coupling]** key to set options that affect all trigger types (see Chapter 11, "Trigger Mode/Coupling," starting on page 147).

You can save trigger setups along with the oscilloscope setup (see Chapter 18, "Save/Recall (Setups, Screens, Data)," starting on page 229).

Triggers - General Information

A triggered waveform is one in which the oscilloscope begins tracing (displaying) the waveform, from the left side of the display to the right, each time a particular trigger condition is met. This provides stable display of periodic signals such as sine waves and square waves, as well as nonperiodic signals such as serial data streams.

The figure below shows the conceptual representation of acquisition memory. You can think of the trigger event as dividing acquisition memory into a pre-trigger and post-trigger buffer. The position of the trigger event in acquisition memory is defined by the time reference point and the delay (horizontal position) setting (see "To adjust the horizontal delay (position)" on page 51).



Adjusting the Trigger Level

You can adjust the trigger level for a selected analog channel by turning the Trigger Level knob.

You can push the Trigger Level knob to set the level to the waveform's 50% value. If AC coupling is used, pushing the Trigger Level knob sets the trigger level to about 0 V.

The position of the trigger level for the analog channel is indicated by the trigger level icon T▸ (if the analog channel is on) at the far left side of the display. The value of the analog channel trigger level is displayed in the upper-right corner of the display.

The trigger level for a selected digital channel is set using the threshold menu in the Digital Channel Menu. Press the **[Digital]** key on the front panel, then press the **Thresholds** softkey to set the threshold level (TTL, CMOS, ECL, or user defined) for the selected digital channel group. The threshold value is displayed in the upper-right corner of the display.

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

NOTE You can also change the trigger level of all channels by pressing **[Analyze] > Features** and then selecting **Trigger Levels**.

Forcing a Trigger

The **[Force Trigger]** key causes a trigger (on anything) and displays the acquisition.

This key is useful in the Normal trigger mode where acquisitions are made only when the trigger condition is met. In this mode, if no triggers are occurring (that is, the "Trig'd?" indicator is displayed), you can press **[Force Trigger]** to force a trigger and see what the input signals look like.

In the Auto trigger mode, when the trigger condition is not met, triggers are forced and the "Auto?" indicator is displayed.

Edge Trigger

The Edge trigger type identifies a trigger by looking for a specified edge (slope) and voltage level on a waveform. You can define the trigger source and slope in this menu. The trigger type, source, and level (if applicable) are displayed in the upper-right corner of the display.

1 On the front panel, in the Trigger section, press the **[Trigger]** key.
2 In the Trigger Menu, press the **Trigger** softkey, and use the Entry knob to select **Edge**.
3 Select the trigger source:
   - Analog channel, **1** to the number of channels

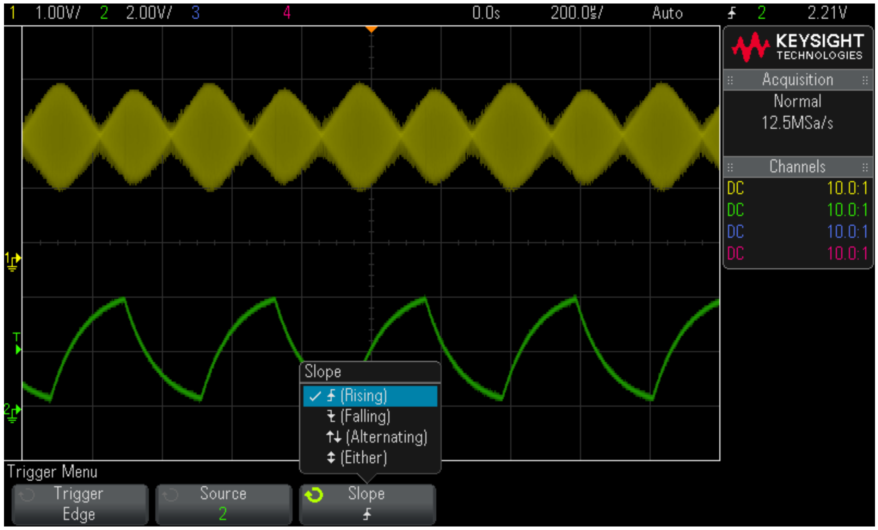
- Digital channel (on mixed-signal oscilloscopes), **D0** to the number of digital channels minus one.
- **External** — triggers on the rear panel EXT TRIG IN signal.
- **Line** — triggers at the 50% level of the rising or falling edge of the AC power source signal.
- **WaveGen** — triggers at the 50% level of the rising edge of the waveform generator output signal. (Not available when the DC or Noise waveforms are selected.)

You can choose a channel that is turned off (not displayed) as the source for the edge trigger.

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

- **1** through **4** = analog channels.
- **D0** through **Dn** = digital channels.
- **E** = External trigger input.
- **L** = Line trigger.
- **W** = Waveform generator.

4 Press the **Slope** softkey and select rising edge, falling edge, alternating edges, or either edge (depending on the selected source). The selected slope is displayed in the upper-right corner of the display.



NOTE

Alternating edge mode is useful when you want to trigger on both edges of a clock (for example, DDR signals).

Either edge mode is useful when you want to trigger on any activity of a selected source.

All modes operate up to the bandwidth of the oscilloscope except Either edge mode, which has a limitation. Either edge mode will trigger on Constant Wave signals up to 100 MHz, but can trigger on isolated pulses down to 1/(2*oscilloscope's bandwidth).

Using AutoScale to Set Up Edge Triggers

The easiest way to set up an Edge trigger on a waveform is to use AutoScale. Simply press the **[AutoScale]** key and the oscilloscope will attempt to trigger on the waveform using a simple Edge trigger type. See "Use Auto Scale" on page 30.

NOTE

MegaZoom Technology Simplifies Triggering

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

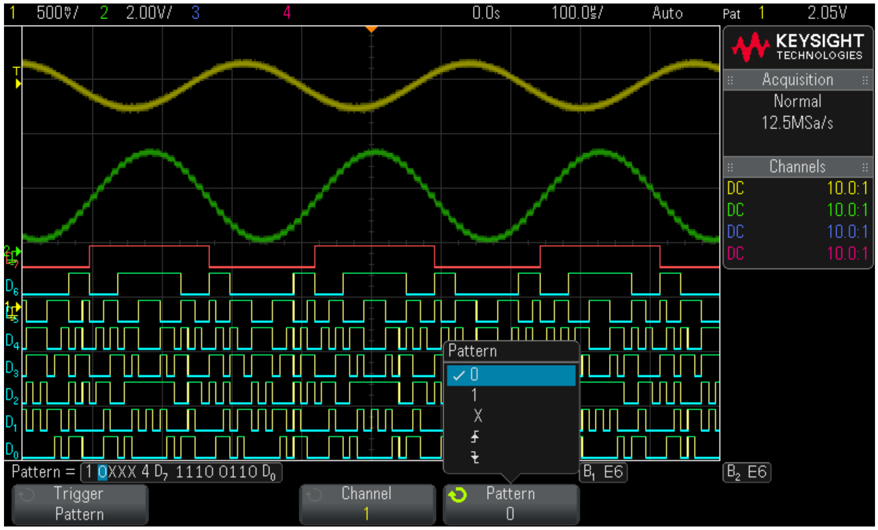
Pattern Trigger

The Pattern trigger identifies a trigger condition by looking for a specified pattern. This pattern is a logical AND combination of the channels. Each channel can have a value of 0 (low), 1 (high), or don't care (X). A rising or falling edge can be specified for one channel included in the pattern. You can also trigger on a hex bus value as described on "Hex Bus Pattern Trigger" on page 132.

1 Press the **[Trigger]** key.
2 In the Trigger Menu, press the **Trigger** softkey; then, turn the Entry knob to select **Pattern**.
3 For each analog or digital channel you want to include in the desired pattern, press the **Channel** softkey to select the channel.

   This is the channel source for the 0, 1, X, or edge condition. As you press the **Channel** softkey (or rotate the Entry knob), the channel you select is highlighted in the Pattern = line directly above the softkeys and in the upper-right corner of the display next to "Pat".

   Adjust the trigger level for the selected analog channel by turning the Trigger Level knob. Press the **[Digital]** key and select **Thresholds** to set the threshold level for digital channels. The value of the trigger level or digital threshold is displayed in the upper-right corner of the display.
4 For each channel you select, press the **Pattern** softkey; then, turn the Entry knob to set the condition for that channel in the pattern.



- **0** sets the pattern to zero (low) on the selected channel. A low is a voltage level that is less than the channel's trigger level or threshold level.
- **1** sets the pattern to 1 (high) on the selected channel. A high is a voltage level that is greater than the channel's trigger level or threshold level.
- **X** sets the pattern to don't care on the selected channel. Any channel set to don't care is ignored and is not used as part of the pattern. However, if all channels in the pattern are set to don't care, the oscilloscope will not trigger.
- The rising edge () or falling edge () softkey sets the pattern to an edge on the selected channel. Only one rising or falling edge can be specified in the pattern. When an edge is specified, the oscilloscope will trigger at the edge specified if the pattern set for the other channels is true.

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

NOTE

Specifying an Edge in a Pattern

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

Hex Bus Pattern Trigger

You can specify a bus value on which to trigger. To do this, first define the bus. See "To display digital channels as a bus" on page 98 for details. You can trigger on a bus value whether you are displaying the bus or not.

To trigger on a bus value:

1 Press the **[Pattern]** key on the front panel.
2 Press the **Channel** softkey and rotate the Entry knob to select **Bus1** or **Bus2**.
3 Press the **Digit** softkey and rotate the Entry knob to select a digit of the selected bus.
4 Press the **Hex** softkey and rotate the Entry knob to select a value for the digit.

NOTE If a digit is made up of less than four bits, then the value of the digit will be limited to the value that can be created by the selected bits.

5 You can use the **Set all Digits** softkey to set all digits to a particular value.

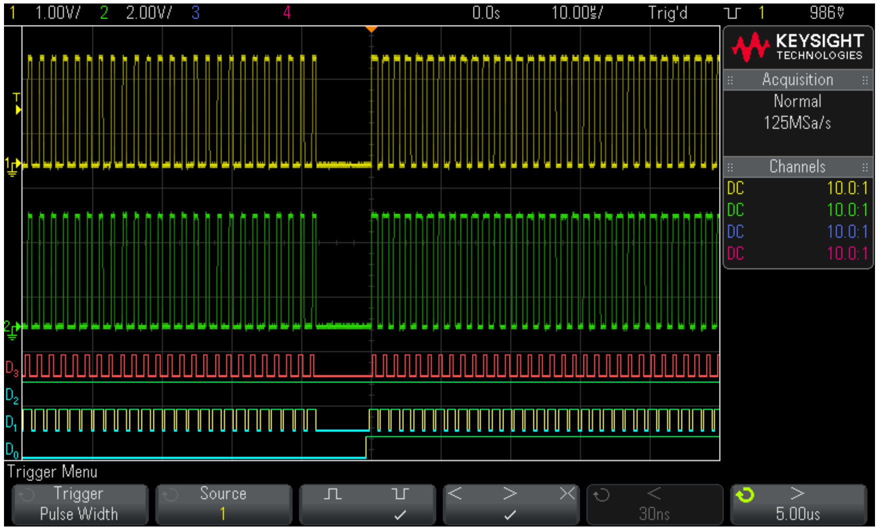
When a hex bus digit contains one or more don't care (X) bits and one or more bit with a value or 0 or 1, the "$" sign will be displayed for the digit.

For information regarding digital bus display when Pattern triggering see "Bus values are displayed when using Pattern trigger" on page 100.

Pulse Width Trigger

Pulse Width (glitch) triggering sets the oscilloscope to trigger on a positive or negative pulse of a specified width. If you want to trigger on a specific timeout value, use **Pattern** trigger in the Trigger Menu (see "Pattern Trigger" on page 130).

1 Press the **[Trigger]** key.
2 In the Trigger Menu, press the **Trigger** softkey; then, turn the Entry knob to select **Pulse Width**.



3 Press the **Source** softkey; then, rotate the Entry knob to select a channel source for the trigger.

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

The source can be any analog or digital channel available on your oscilloscope.

4 Adjust the trigger level:

- For analog channels, turn the Trigger Level knob.
- For digital channels, press the **[Digital]** key and select **Thresholds** to set the threshold level.

The value of the trigger level or digital threshold is displayed in the upper-right corner of the display.

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

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

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

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

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

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

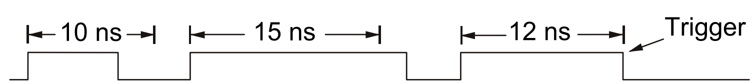
  For example, for a positive pulse, if you set t<10 ns:

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

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

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

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



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

The qualifiers can be set as follows:

- 2 ns to 10 s for > or < qualifier (5 ns to 10 s for 350 MHz bandwidth models).
- 10 ns to 10 s for >< qualifier, with minimum difference of 5 ns between upper and lower settings.

Pulse width trigger < qualifier time set softkey

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

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

Pulse width trigger > qualifier time set softkey

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

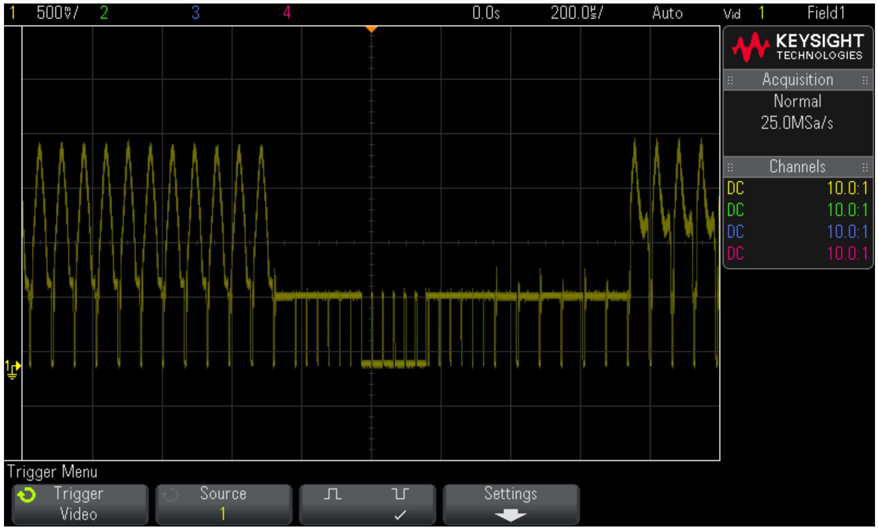
Video Trigger

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

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

NOTE **It is important, when using a 10:1 passive probe, that the probe is correctly compensated. The oscilloscope is sensitive to this and will not trigger if the probe is not properly compensated, especially for progressive formats.**

1 Press the **[Trigger]** key.
2 In the Trigger Menu, press the **Trigger** softkey; then, turn the Entry knob to select **Video**.



3 Press the **Source** softkey and select any analog channel as the video trigger source.

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

NOTE

Provide Correct Matching

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

4 Press the sync polarity softkey to set the Video trigger to either positive (⎍) or negative (⎎) sync polarity.

5 Press the **Settings** softkey.

6 In the Video Trigger Menu, press the **Standard** softkey to set the video standard.

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

Standard	Type	Sync Pulse
NTSC	Interlaced	Bi-level
PAL	Interlaced	Bi-level
PAL-M	Interlaced	Bi-level
SECAM	Interlaced	Bi-level

7 Press the **Auto Setup** softkey to automatically set up the oscilloscope for the selected **Source** and **Standard**:

- Source channel vertical scaling is set to 140 mV/div.
- Source channel offset is set to 245 mV.
- Source channel is turned on.
- Trigger type is set to **Video**.
- Video trigger mode is set to **All Lines**.
- Display **Grid** type is set to **IRE** (when **Standard** is **NTSC**) or **mV** (see "To select the grid type" on page 116).
- Horizontal time/division is set to 10 µs/div for NTSC/PAL/SECAM standards.
- Horizontal delay is set so that trigger is at first horizontal division from the left.

You can also press **[Analyze]> Features** and then select **Video** to quickly access the video triggering automatic set up and display options.

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

The Video trigger modes available are:

- **Field1** and **Field2** — Trigger on the rising edge of the first serration pulse of field 1 or field 2 (interlaced standards only).
- **All Fields** — Trigger on the rising edge of the first pulse in the vertical sync interval.
- **All Lines** — Trigger on all horizontal sync pulses.
- **Line: Field1** and **Line:Field2** — Trigger on the selected line # in field 1 or field 2 (interlaced standards only).
- **Line: Alternate** — Alternately trigger on the selected line # in field 1 and field 2 (NTSC, PAL, PAL-M, and SECAM only).

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

The following table lists the line (or count) numbers per field for each video standard.

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

Video Triggering Examples

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

- "To trigger on a specific line of video" on page 138
- "To trigger on all sync pulses" on page 140
- "To trigger on a specific field of the video signal" on page 140
- "To trigger on all fields of the video signal" on page 141
- "To trigger on odd or even fields" on page 142

To trigger on a specific line of video

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

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

1 Press the **[Trigger]** key.
2 In the Trigger Menu, press the **Trigger** softkey; then, turn the Entry knob to select **Video**.
3 Press the **Settings** softkey, then press the **Standard** softkey to select the appropriate TV standard (NTSC).
4 Press the **Mode** softkey and select the TV field of the line you want to trigger on. You can choose **Line:Field1**, **Line:Field2**, or **Line:Alternate**.
5 Press the **Line #** softkey and select the number of the line you want to examine.

NOTE

Alternate Triggering

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

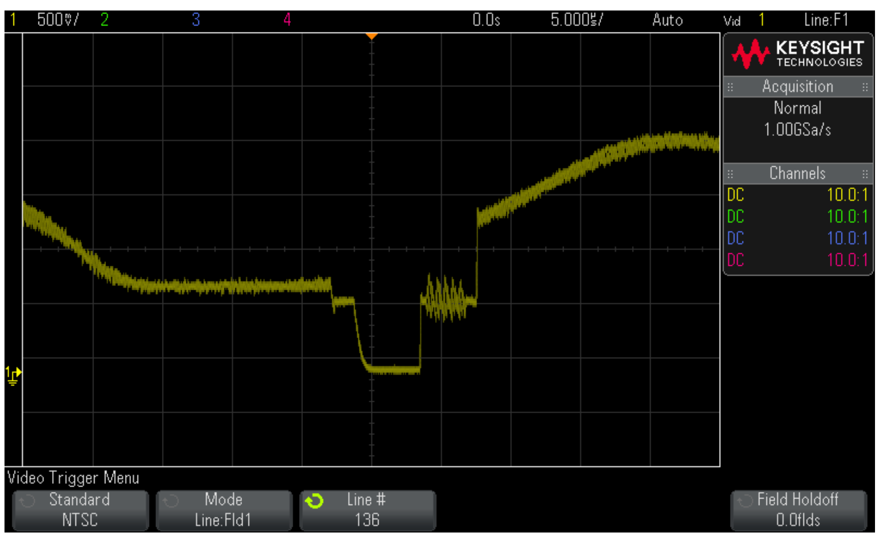


Figure 15 Example: Triggering on Line 136

To trigger on all sync pulses

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

1 Press the **[Trigger]** key.
2 In the Trigger Menu, press the **Trigger** softkey; then, turn the Entry knob to select **Video**.
3 Press the **Settings** softkey, then press the **Standard** softkey to select the appropriate TV standard.
4 Press the **Mode** softkey and select **All Lines**.

Figure 16 Triggering on All Lines

To trigger on a specific field of the video signal

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

1 Press the **[Trigger]** key.
2 In the Trigger Menu, press the **Trigger** softkey; then, turn the Entry knob to select **Video**.
3 Press the **Settings** softkey, then press the **Standard** softkey to select the appropriate TV standard.
4 Press the **Mode** softkey and select **Field1** or **Field2**.

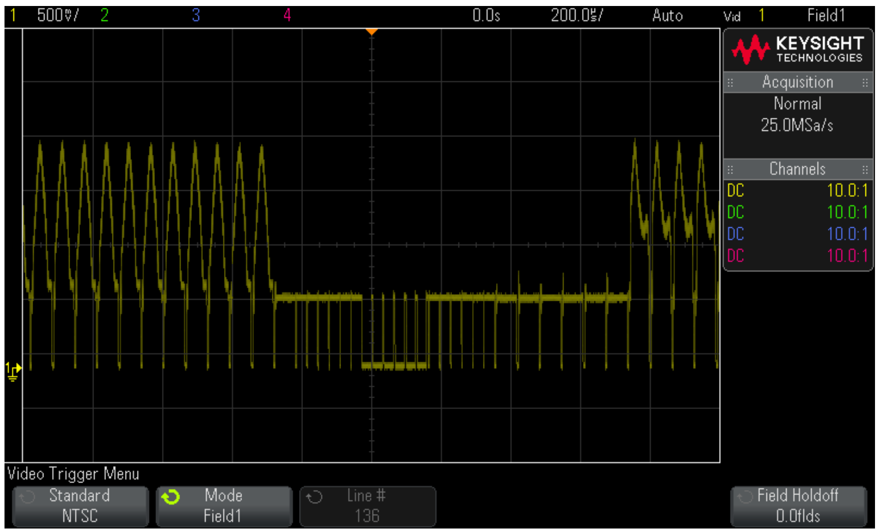


Figure 17 Triggering on Field 1

To trigger on all fields of the video signal

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

1 Press the **[Trigger]** key.
2 In the Trigger Menu, press the **Trigger** softkey; then, turn the Entry knob to select **Video**.
3 Press the **Settings** softkey, then press the **Standard** softkey to select the appropriate TV standard.
4 Press the **Mode** softkey and select **All Fields**.

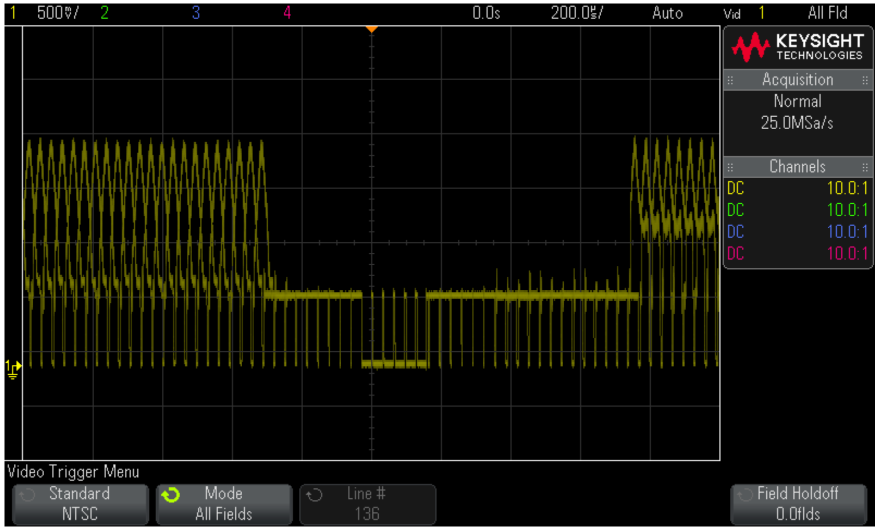


Figure 18 Triggering on All Fields

To trigger on odd or even fields

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

1. Press the **[Trigger]** key.
2. In the Trigger Menu, press the **Trigger** softkey; then, turn the Entry knob to select **Video**.
3. Press the **Settings** softkey, then press the **Standard** softkey to select the appropriate TV standard.
4. Press the **Mode** softkey and select **Field1** or **Field2**.

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

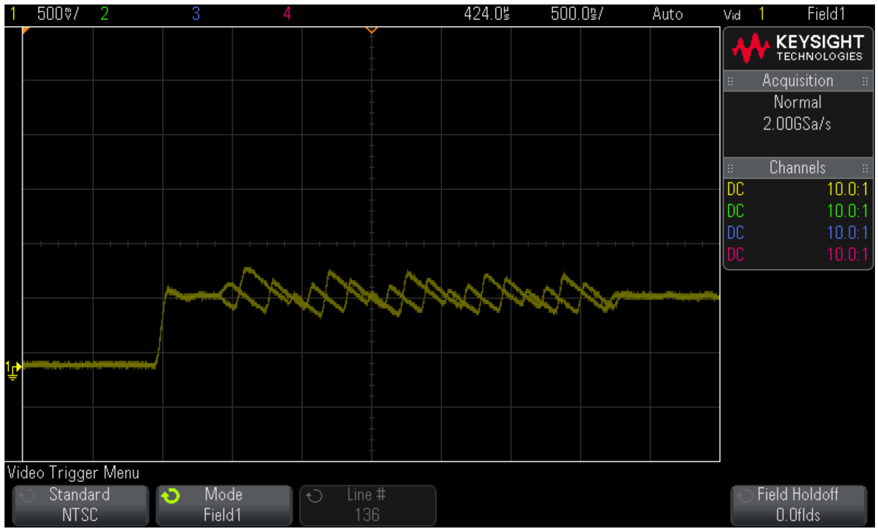


Figure 19 Triggering on Color Field 1 Alternating with Color Field 3

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

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

When holdoff is adjusted using the **Field Holdoff** softkey and the Entry knob, the corresponding holdoff time will be displayed in the Trigger Mode and Coupling Menu.

Table 3 Half-field holdoff time

Standard	Time
NTSC	8.35 ms
PAL	10 ms
PAL-M	10 ms
SECAM	10 ms

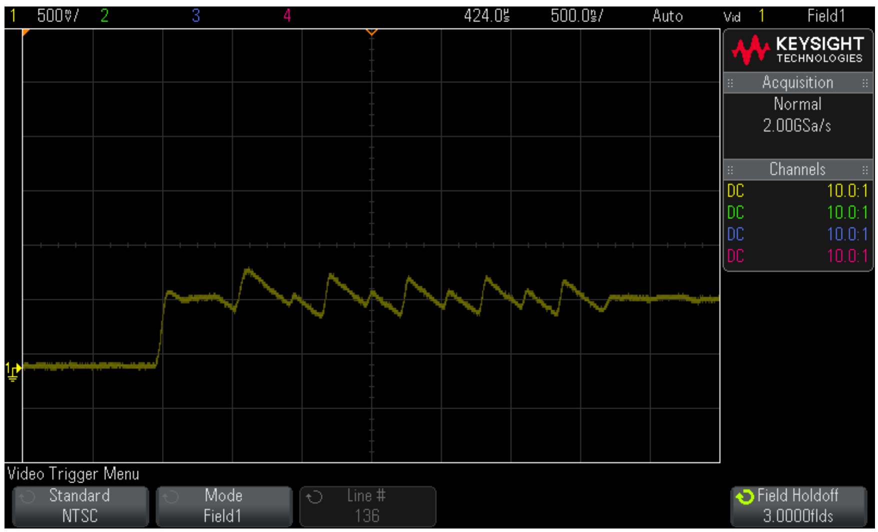


Figure 20 Using Field Holdoff to Synchronize to Color Field 1 or 3 (Field 1 mode)

Serial Trigger

With serial decode option licenses (see "Serial Decode Options" on page 107), you can enable serial trigger types. To set up these triggers, see:

- "CAN Triggering" on page 303
- "I2C Triggering" on page 318
- "LIN Triggering" on page 311

12 Acquisition Control

This chapter shows how to use the oscilloscope's acquisition and run controls.

Running, Stopping, and Making Single Acquisitions (Run Control)

There are two front panel keys for starting and stopping the oscilloscope's acquisition system: **[Run/Stop]** and **[Single]**.

- When the **[Run/Stop]** key is green, the oscilloscope is running, that is, acquiring data when trigger conditions are met.

  To stop acquiring data, press **[Run/Stop]**. When stopped, the last acquired waveform is displayed.

- When the **[Run/Stop]** key is red, data acquisition is stopped.

  "Stop" is displayed next to the trigger type in the status line at the top of the display.

  To start acquiring data, press **[Run/Stop]**.

- To capture and display a single acquisition (whether the oscilloscope is running or stopped), press **[Single]**.

  The **[Single]** run control lets you view single-shot events without subsequent waveform data overwriting the display. Use **[Single]** when you want maximum memory depth for pan and zoom.

When you press **[Single]**, the display is cleared, the trigger mode is temporarily set to Normal (to keep the oscilloscope from auto-triggering immediately), the trigger circuitry is armed, the **[Single]** key is illuminated, and the oscilloscope waits until a trigger condition occurs before it displays a waveform.

When the oscilloscope triggers, the single acquisition is displayed and the oscilloscope is stopped (the **[Run/Stop]** key is illuminated in red). Press **[Single]** again to acquire another waveform.

If the oscilloscope doesn't trigger, you can press the **[Force Trigger]** key to trigger on anything and make a single acquisition.

To display the results of multiple acquisitions, use persistence. See "To set or clear persistence" on page 115.

Single vs. Running and Record Length

The maximum data record length is greater for a single acquisition than when the oscilloscope is running (or when the oscilloscope is stopped after running):

- **Single** — Single acquisitions always use the maximum memory available — at least twice as much memory as acquisitions captured when running — and the oscilloscope stores at least twice as many samples. At slower time/div settings, because there is more memory available for a single acquisition, the acquisition has a higher effective sample rate.
- **Running** — When running (versus taking a single acquisition), the memory is divided in half. This lets the acquisition system acquire one record while processing the previous acquisition, dramatically improving the number of waveforms per second processed by the oscilloscope. When running, a high waveform update rate provides the best representation of your input signal.

To acquire data with the longest possible record length, press the **[Single]** key.

For more information on settings that affect record length, see "Length Control" on page 233.

Overview of Sampling

To understand the oscilloscope's sampling and acquisition modes, it is helpful to understand sampling theory, aliasing, oscilloscope bandwidth and sample rate, oscilloscope rise time, oscilloscope bandwidth required, and how memory depth affects sample rate.

Sampling Theory

The Nyquist sampling theorem states that for a limited bandwidth (band-limited) signal with maximum frequency f_{MAX}, the equally spaced sampling frequency f_S must be greater than twice the maximum frequency f_{MAX}, in order to have the signal be uniquely reconstructed without aliasing.

$f_{MAX} = f_S/2$ = Nyquist frequency (f_N) = folding frequency

Aliasing

Aliasing occurs when signals are under-sampled ($f_S < 2f_{MAX}$). Aliasing is the signal distortion caused by low frequencies falsely reconstructed from an insufficient number of sample points.

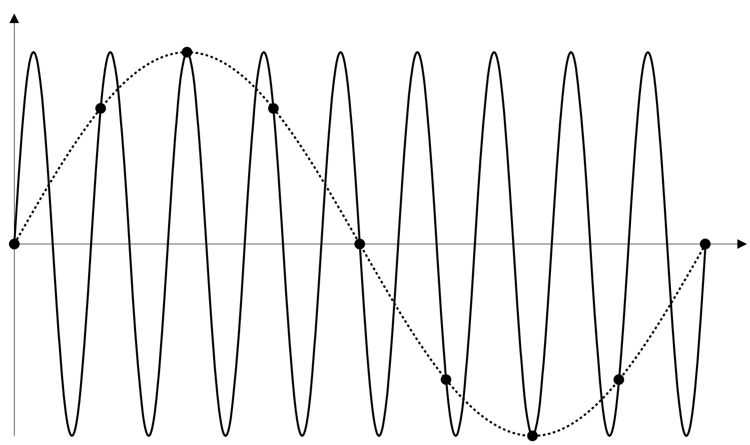


Figure 21 Aliasing

Oscilloscope Bandwidth and Sample Rate

An oscilloscope's bandwidth is typically described as the lowest frequency at which input signal sine waves are attenuated by 3 dB (-30% amplitude error).

At the oscilloscope bandwidth, sampling theory says the required sample rate is $f_S = 2f_{BW}$. However, the theory assumes there are no frequency components above f_{MAX} (f_{BW} in this case) and it requires a system with an ideal brick-wall frequency response.

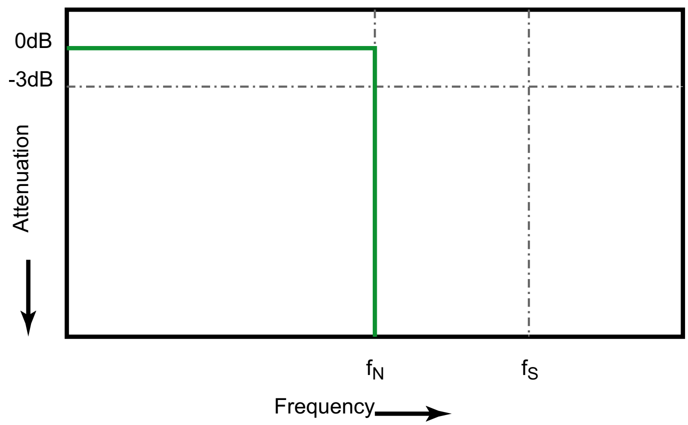
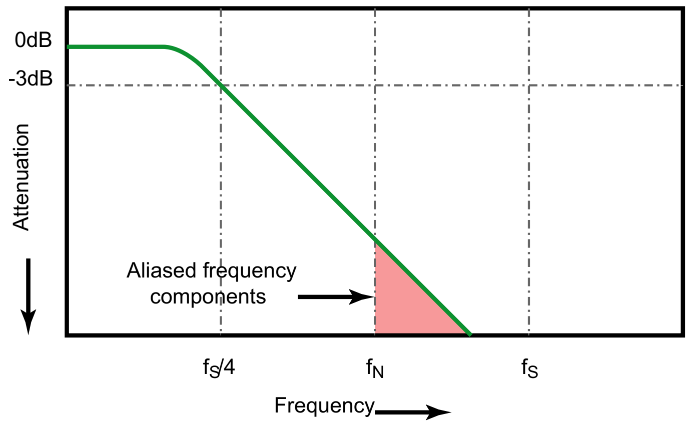


Figure 22 Theoretical Brick-Wall Frequency Response

However, digital signals have frequency components above the fundamental frequency (square waves are made up of sine waves at the fundamental frequency and an infinite number of odd harmonics), and typically, for 500 MHz bandwidths and below, oscilloscopes have a Gaussian frequency response.



Limiting oscilloscope bandwidth (f_{BW}) to 1/4 the sample rate ($f_S/4$) reduces frequency components above the Nyquist frequency (f_N).

Figure 23 Sample Rate and Oscilloscope Bandwidth

So, in practice, an oscilloscope's sample rate should be four or more times its bandwidth: $f_S = 4f_{BW}$. This way, there is less aliasing, and aliased frequency components have a greater amount of attenuation.

See Also *Evaluating Oscilloscope Sample Rates vs. Sampling Fidelity: How to Make the Most Accurate Digital Measurements*, Keysight Application Note 1587 (http://literature.cdn.keysight.com/litweb/pdf/5989-5732EN.pdf)

Oscilloscope Rise Time

Closely related to an oscilloscope's bandwidth specification is its rise time specification. Oscilloscopes with a Gaussian-type frequency response have an approximate rise time of $0.35/f_{BW}$ based on a 10% to 90% criterion.

An oscilloscope's rise time is not the fastest edge speed that the oscilloscope can accurately measure. It is the fastest edge speed the oscilloscope can possibly produce.

Oscilloscope Bandwidth Required

The oscilloscope bandwidth required to accurately measure a signal is primarily determined by the signal's rise time, not the signal's frequency. You can use these steps to calculate the oscilloscope bandwidth required:

1 Determine the fastest edge speeds.

   You can usually obtain rise time information from published specifications for devices used in your designs.

2 Compute the maximum "practical" frequency component.

   From Dr. Howard W. Johnson's book, *High-Speed Digital Design – A Handbook of Black Magic*, all fast edges have an infinite spectrum of frequency components. However, there is an inflection (or "knee") in the frequency spectrum of fast edges where frequency components higher than f_{knee} are insignificant in determining the shape of the signal.

   f_{knee} = 0.5 / signal rise time (based on 10% - 90% thresholds)

   f_{knee} = 0.4 / signal rise time (based on 20% - 80% thresholds)

3 Use a multiplication factor for the required accuracy to determine the oscilloscope bandwidth required.

Required accuracy	Oscilloscope bandwidth required
20%	$f_{BW} = 1.0 \times f_{knee}$
10%	$f_{BW} = 1.3 \times f_{knee}$
3%	$f_{BW} = 1.9 \times f_{knee}$

See Also *Choosing an Oscilloscope with the Right Bandwidth for your Application*, Keysight Application Note 1588 (http://literature.cdn.keysight.com/litweb/pdf/5989-5733EN.pdf)

Memory Depth and Sample Rate

The number of points of oscilloscope memory is fixed, and there is a maximum sample rate associated with oscilloscope's analog-to-digital converter; however, the actual sample rate is determined by the time of the acquisition (which is set according to the oscilloscope's horizontal time/div scale).

sample rate = number of samples / time of acquisition

For example, when storing 50 µs of data in 50,000 points of memory, the actual sample rate is 1 GSa/s.

Likewise, when storing 50 ms of data in 50,000 points of memory, the actual sample rate is 1 MSa/s.

The actual sample rate is displayed in the right-side information area.

The oscilloscope achieves the actual sample rate by throwing away (decimating) unneeded samples.

Selecting the Acquisition Mode

When selecting the oscilloscope acquisition mode, keep in mind that samples are normally decimated at slower time/div settings.

At slower time/div settings, the effective sample rate drops (and the effective sample period increases) because the acquisition time increases and the oscilloscope's digitizer is sampling faster than is required to fill memory.

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

To select the acquisition mode:

1 Press the **[Acquire]** key on the front panel.
2 In the Acquire Menu, press the **Acq Mode** softkey; then, turn the Entry knob to select the acquisition mode.

   The InfiniiVision oscilloscopes have the following acquisition modes:

   - **Normal** — at slower time/div settings, normal decimation occurs, and there is no averaging. Use this mode for most waveforms. See "Normal Acquisition Mode" on page 162.
   - **Peak Detect** — at slower time/div settings, the maximum and minimum samples in the effective sample period are stored. Use this mode for displaying narrow pulses that occur infrequently. See "Peak Detect Acquisition Mode" on page 162.

- **Averaging** — at all time/div settings, the specified number of triggers are averaged together. Use this mode for reducing noise and increasing resolution of periodic signals without bandwidth or rise time degradation. See "Averaging Acquisition Mode" on page 164.
- **High Resolution** — at slower time/div settings, all samples in the effective sample period are averaged and the average value is stored. Use this mode for reducing random noise. See "High Resolution Acquisition Mode" on page 166.

Normal Acquisition Mode

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

Peak Detect Acquisition Mode

In Peak Detect mode, at slower time/div settings when decimation would normally occur, minimum and maximum value samples are kept in order to capture infrequent and narrow events (at the expense of exaggerating any noise). This mode displays all pulses that are at least as wide as the sample period.

For InfiniiVision 2000 X-Series oscilloscopes, which have a maximum sample rate of 2 GSa/s, a sample is taken every 500 ps (sample period).

See Also

- "Glitch or Narrow Pulse Capture" on page 162
- "Using Peak Detect Mode to Find a Glitch" on page 164

Glitch or Narrow Pulse Capture

A glitch is a rapid change in the waveform that is usually narrow as compared to the waveform. Peak detect mode can be used to more easily view glitches or narrow pulses. In peak detect mode, narrow glitches and sharp edges are displayed more brightly than when in Normal acquire mode, making them easier to see.

To characterize the glitch, use the cursors or the automatic measurement capabilities of the oscilloscope.

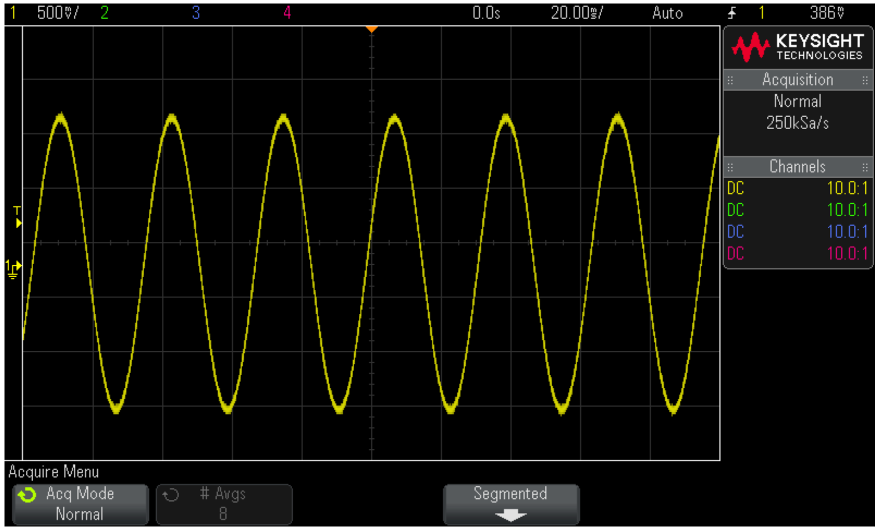


Figure 24 Sine With Glitch, Normal Mode

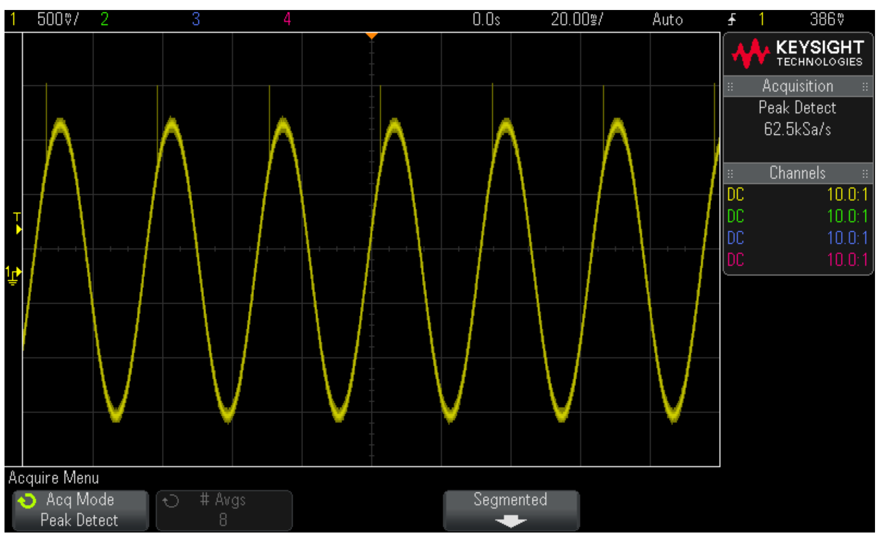


Figure 25 Sine With Glitch, Peak Detect Mode

Using Peak Detect Mode to Find a Glitch

1 Connect a signal to the oscilloscope and obtain a stable display.
2 To find the glitch, press the **[Acquire]** key; then, press the **Acq Mode** softkey until **Peak Detect** is selected.
3 Press the **[Display]** key then press the ∞ **Persistence** (infinite persistence) softkey.

   Infinite persistence updates the display with new acquisitions but does not erase previous acquisitions. New sample points are shown at normal intensity while previous acquisitions are displayed at reduced intensity. Waveform persistence is not kept beyond the display area boundary.

   Press the **Clear Display** softkey to erase previously acquired points. The display will accumulate points until ∞ **Persistence** is turned off.
4 Characterize the glitch with Zoom mode:
   a Press the zoom key (or press the **[Horiz]** key and then the **Zoom** softkey).
   b To obtain a better resolution of the glitch, expand the time base.

Use the horizontal position knob (◀▶) to pan through the waveform to set the expanded portion of the normal window around the glitch.

Averaging Acquisition Mode

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

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

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

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

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

To use the Averaging mode:

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

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

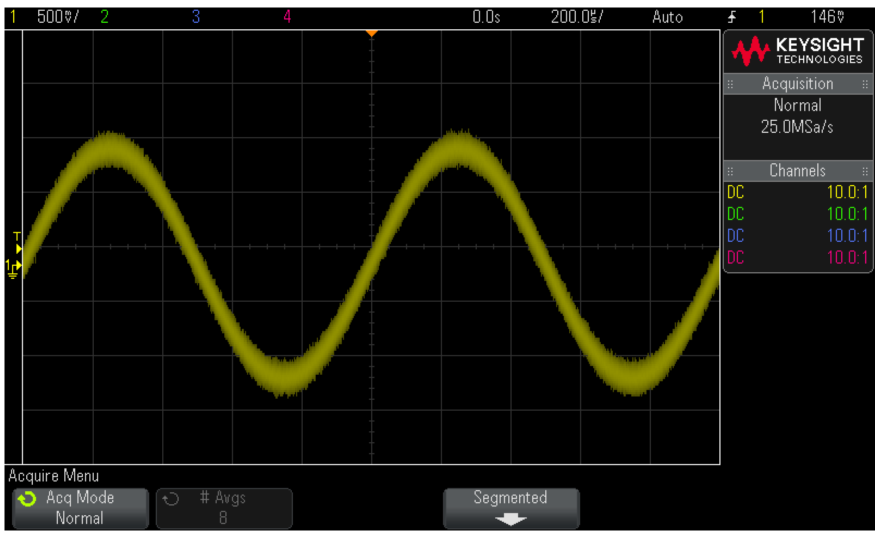


Figure 26 Random noise on the displayed waveform

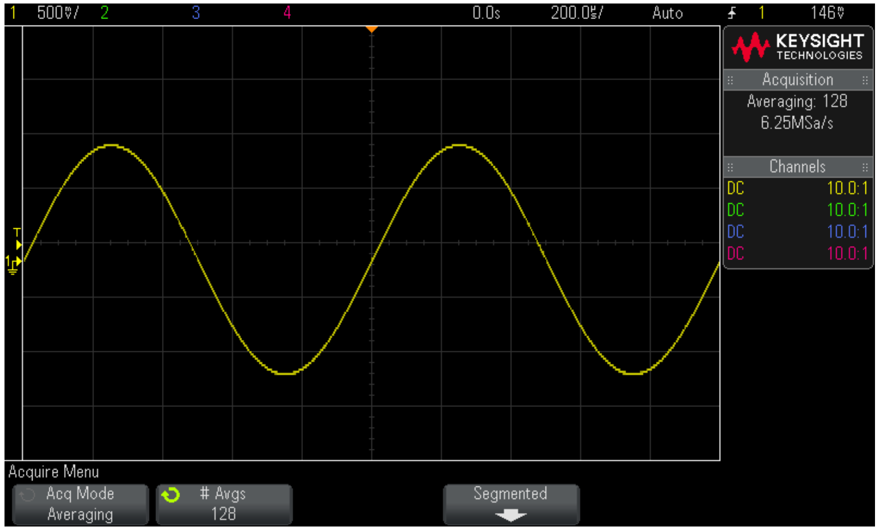


Figure 27 128 Averages used to reduce random noise

See Also · Chapter 11, “Trigger Mode/Coupling,” starting on page 147

High Resolution Acquisition Mode

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

High Resolution mode averages sequential sample points within the same acquisition. An extra bit of vertical resolution is produced for every factor of 2 averages. Random noise is reduced by ½ for every factor of 4 averages. The number of extra bits of vertical resolution is dependent on the time per division setting (sweep speed) of the oscilloscope.

The slower the time/div setting, the greater the number of samples that are averaged together for each display point.

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

Sweep speed	Bits of resolution
≤ 1 µs/div	8
2 µs/div	9
5 µs/div	10
10 µs/div	11
≥ 20 µs/div	12

Acquiring to Segmented Memory

You can purchase the oscilloscope with the segmented memory option factory-installed (Option SGM) or you can install a license to enable it (order model number DSOX2SGM "Segmented Memory").

When capturing multiple infrequent trigger events it is advantageous to divide the oscilloscope's memory into segments. This lets you capture signal activity without capturing long periods of signal inactivity.

Each segment is complete with all analog channel, digital channel (on MSO models), and serial decode data.

When using segmented memory, use the Analyze Segments feature (see "Infinite Persistence with Segmented Memory" on page 169) to show infinite persistence across all acquired segments. See also "To set or clear persistence" on page 115 for details.

To acquire to segmented memory

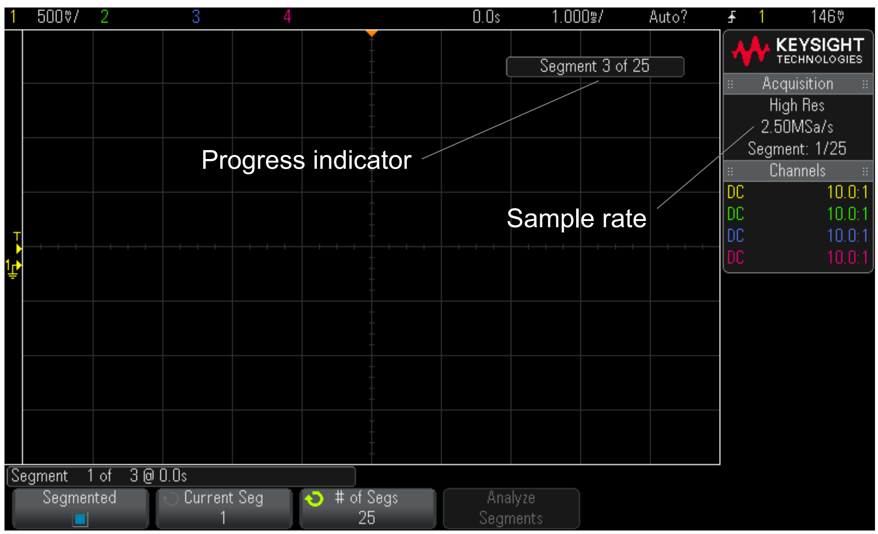
1 Set up a trigger condition. (See Chapter 10, "Triggers," starting on page 125 for details.)
2 Press the **[Acquire]** key in the Waveform section of the front panel.
3 Press the **Segmented** softkey.
4 In the Segmented Memory Menu, press the **Segmented** softkey to enable segmented memory acquisitions.
5 Press the **# of Segs** softkey and turn the Entry knob to select the number of segments into which you would like to divide the oscilloscope's memory.

   Memory can be divided into as few as two segments and as many as 25 segments.

6 Press the **[Run]** or **[Single]** key.

The oscilloscope runs and fills a memory segment for each trigger event. When the oscilloscope is busy acquiring multiple segments, the progress is displayed in the upper right area of the display. The oscilloscope continues to trigger until memory is filled, then the oscilloscope stops.

If the signal you are measuring has more than about 1 s of inactivity, consider selecting the **Normal** trigger mode to prevent AutoTriggering. See "To select the Auto or Normal trigger mode" on page 148.



See Also

- "Navigating Segments" on page 168
- "Infinite Persistence with Segmented Memory" on page 169
- "Segmented Memory Re-Arm Time" on page 169
- "Saving Data from Segmented Memory" on page 169

Navigating Segments

1 Press the **Current Seg** softkey and turn the Entry knob to display the desired segment along with a time tag indicating the time from the first trigger event.

You can also navigate segments using the **[Navigate]** key and controls. See "To navigate segments" on page 60.

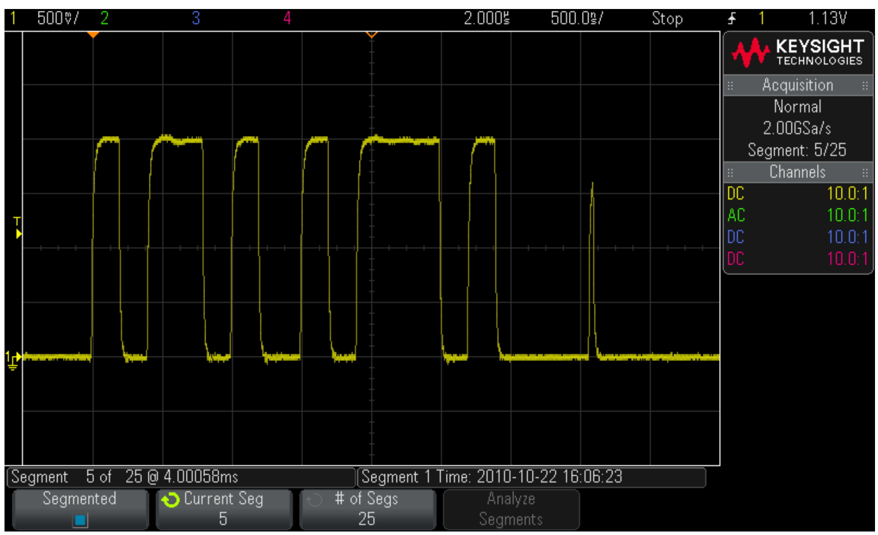
Infinite Persistence with Segmented Memory

When data has been acquired to segmented memory, you can also turn on infinite persistence (in the Display Menu) and press the **Analyze Segments** softkey to create an infinite persistence display. The **Analyze Segments** softkey appears when the acquisition is stopped and the segmented memory feature is on.

Segmented Memory Re-Arm Time

After each segment fills, the oscilloscope re-arms and is ready to trigger in about 8 µs.

Remember though, for example: if the horizontal time per division control is set to 5 µs/div, and the Time Reference is set to **Center**, it will take at least 50 µs to fill all ten divisions and re-arm. (That is 25 µs to capture pre-trigger data and 25 µs to capture post-trigger data.)



Saving Data from Segmented Memory

You can save either the currently displayed segment (**Save Segment - Current**), or all segments (**Save Segment - All**) in the following data formats: CSV, ASCII XY, and BIN.

Be sure to set the Length control to capture enough points to accurately represent the captured data. When the oscilloscope is busy saving multiple segments, progress is displayed in the upper right area of the display.

For more information, see “To save CSV, ASCII XY, or BIN data files" on page 232.

14 Measurements

The **[Meas]** key lets you make automatic measurements on waveforms. Some measurements can only be made on analog input channels.

The results of the last four selected measurements are displayed in the Measurements information area on the right-hand side of the screen.

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

NOTE

Post Acquisition Processing

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

To make automatic measurements

1 Press the **[Meas]** key to display the Measurement Menu.

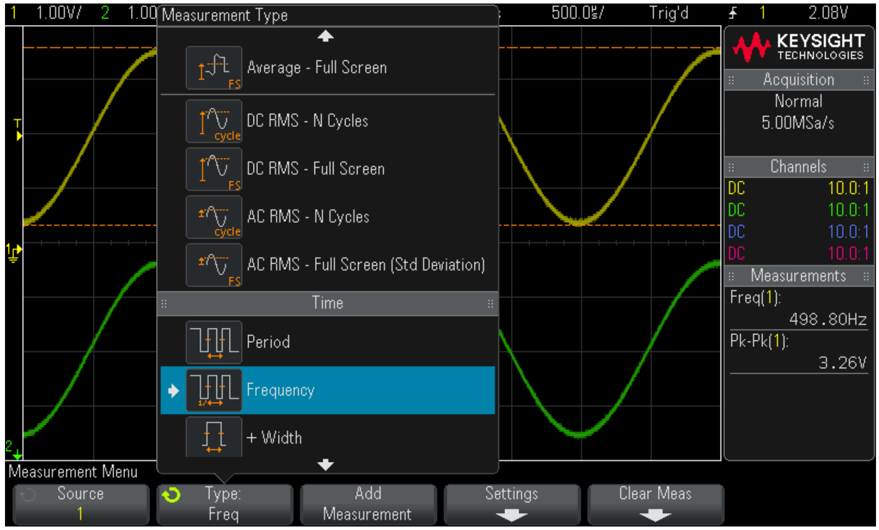
2 Press the **Source** softkey to select the channel, running math function, or reference waveform to be measured.

Only channels, math functions, or reference waveforms that are displayed are available for measurements.

NOTE

If a portion of the waveform required for a measurement is not displayed or does not display enough resolution to make the measurement (approximately 4% of full scale), the result will display "No Edges", "Clipped", "Low Signal" (not enough amplitude), "< value", or "> value", or a similar message to indicate that the measurement may not be reliable.

3 Press the **Type** softkey then rotate the Entry knob to select a measurement to be made.



For more information on the types of measurements, see "Measurements Summary" on page 184.

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

5 Press the **Add Measurement** softkey or push the Entry knob to display the measurement.

6 To turn off measurements, press the **[Meas]** key again.

Measurements are erased from the display.

7 To stop making one or more measurements, press the **Clear Meas** softkey and choose the measurement to clear, or press **Clear All**.

After all measurements have been cleared, when **[Meas]** is pressed again, the default measurements will be Frequency and Peak-Peak.

Measurements Summary

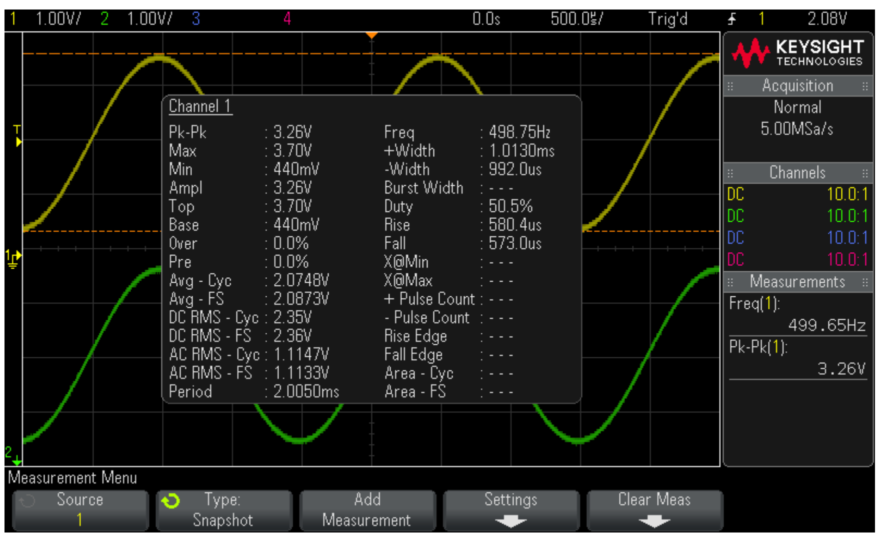
The automatic measurements provided by the oscilloscope are listed in the following table. All measurements are available for analog channel waveforms. All measurements except Counter are available for math waveforms other than FFT. A limited set of measurements is available for math FFT waveforms and for digital channel waveforms (as described in the following table).

Measurement	Valid for Math FFT*	Valid for Digital Channels	Notes
"Snapshot All" on page 185			
"Amplitude" on page 187			
"Average" on page 190	Yes, Full Screen		
"Base" on page 188			
"Delay" on page 196			Measures between two sources. Press **Settings** to specify the second source.
"Duty Cycle" on page 195		Yes	
"Fall Time" on page 195			
"Frequency" on page 194		Yes	
"Maximum" on page 187	Yes		
"Minimum" on page 187	Yes		
"Overshoot" on page 188			
"Peak-Peak" on page 186	Yes		
"Period" on page 193		Yes	
"Phase" on page 197			Measures between two sources. Press **Settings** to specify the second source.
"Preshoot" on page 189			

Measurement	Valid for Math FFT*	Valid for Digital Channels	Notes
"Rise Time" on page 195			
"DC RMS" on page 190			
"AC RMS" on page 191			
"Top" on page 187			
"+ Width" on page 195		Yes	
"– Width" on page 195		Yes	
* Use the cursors to make other measurements on FFT.			

Snapshot All

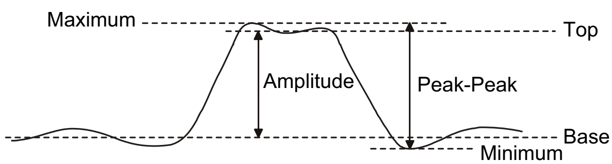
The Snapshot All measurement type displays a popup containing a snapshot of all the single waveform measurements.



You can also configure the **[Quick Action]** key to display the Snapshot All popup. See "Configuring the [Quick Action] Key" on page 262.

Voltage Measurements

The following figure shows the voltage measurement points.



Measurement units for each input channel can be set to Volts or Amps using the channel **Probe Units** softkey. See "To specify the channel units" on page 68.

The units of math waveforms are described in "Units for Math Waveforms" on page 73.

- "Peak-Peak" on page 186
- "Maximum" on page 187
- "Minimum" on page 187
- "Amplitude" on page 187
- "Top" on page 187
- "Base" on page 188
- "Overshoot" on page 188
- "Preshoot" on page 189
- "Average" on page 190
- "DC RMS" on page 190
- "AC RMS" on page 191

Peak-Peak

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

Maximum

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

Minimum

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

Amplitude

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

Top

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

See Also

- "To isolate a pulse for Top measurement" on page 187

To isolate a pulse for Top measurement

The following figure shows how to use Zoom mode to isolate a pulse for a **Top** measurement.

You may need to change the measurement window setting so that the measurement is made in the lower, Zoom window. See "Measurement Window with Zoom Display" on page 200.

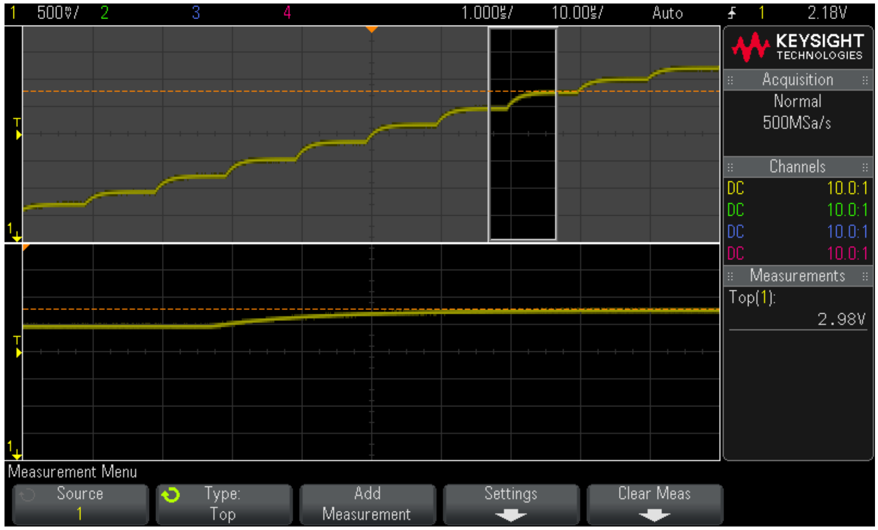


Figure 33 Isolating area for Top measurement

Base

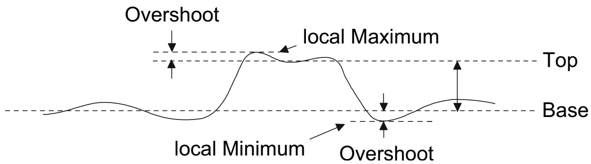
The Base of a waveform is the mode (most common value) of the lower part of the waveform, or if the mode is not well defined, the base is the same as Minimum. The Y cursor shows the value being measured.

Overshoot

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

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

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



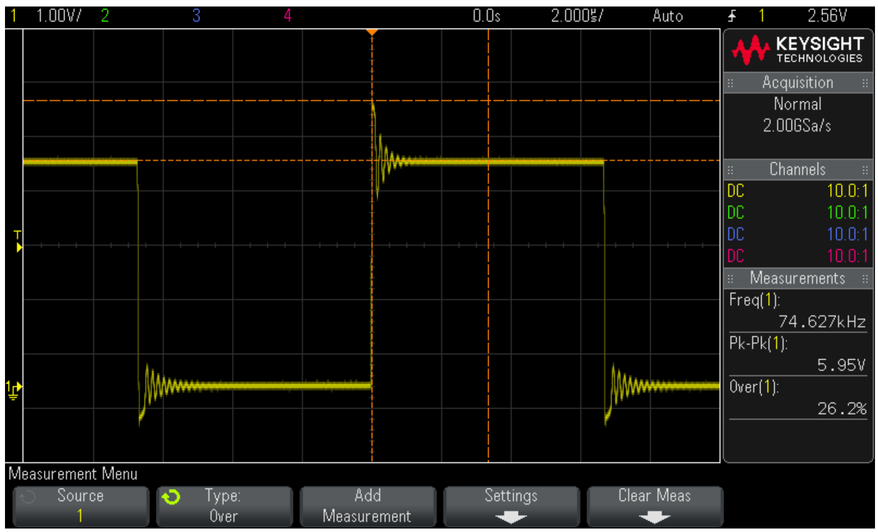


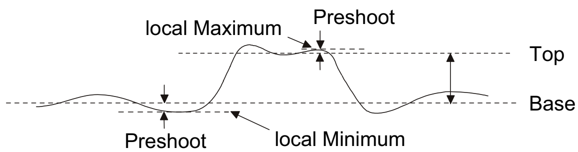
Figure 34 Automatic Overshoot measurement

Preshoot

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

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

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



Average

Average is the sum of the levels of the waveform samples divided by the number of samples.

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

Where x_i = value at *i*th point being measured, n = number of points in measurement interval.

The Full Screen measurement interval variation measures the value on all displayed data points.

The N Cycles measurement interval variation measures the value on an integral number of periods of the displayed signal. If less than three edges are present, the measurement shows "No edges".

The X cursors show what interval of the waveform is being measured.

DC RMS

DC RMS is the root-mean-square value of the waveform over one or more full periods.

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

Where x_i = value at *i*th point being measured, n = number of points in measurement interval.

The Full Screen measurement interval variation measures the value on all displayed data points.

The N Cycles measurement interval variation measures the value on an integral number of periods of the displayed signal. If less than three edges are present, the measurement shows "No edges".

The X cursors show the interval of the waveform being measured.

AC RMS

AC RMS is the root-mean-square value of the waveform, with the DC component removed. It is useful, for example, for measuring power supply noise.

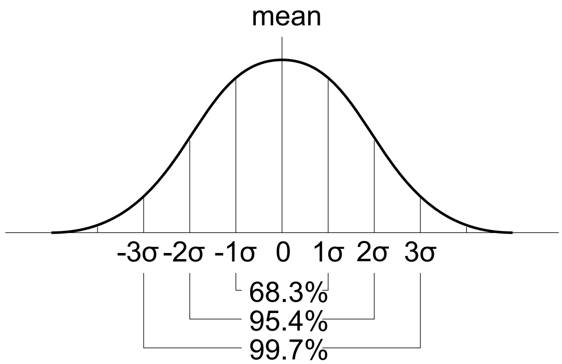
The N Cycles measurement interval measures the value on an integral number of periods of the displayed signal. If less than three edges are present, the measurement shows "No edges".

The X cursors show the interval of the waveform being measured.

The Full Screen (Std Deviation) measurement interval variation is an RMS measurement across the full screen with the DC component removed. It shows the standard deviation of the displayed voltage values.

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

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



The mean is calculated as follows:

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

where:

- x = the mean.
- N = the number of measurements taken.
- x_i = the ith measurement result.

The standard deviation is calculated as follows:

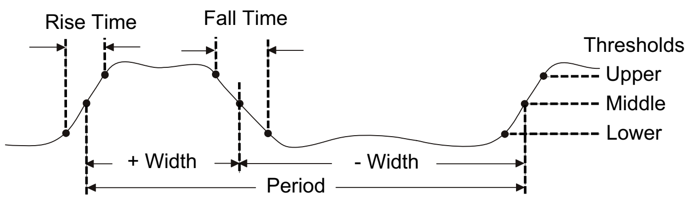
$$\sigma = \sqrt{\frac{\sum_{i=1}^{N} (x_i - \overline{x})^2}{N}}$$

where:

- σ = the standard deviation.
- N = the number of measurements taken.
- x_i = the ith measurement result.
- x = the mean.

Time Measurements

The following figure shows time measurement points.



The default lower, middle, and upper measurement thresholds are 10%, 50%, and 90% between Top and Base values. See "Measurement Thresholds" on page 198 for other percentage threshold and absolute value threshold settings.

- "Period" on page 193
- "Frequency" on page 194
- "+ Width" on page 195
- "– Width" on page 195
- "Duty Cycle" on page 195
- "Rise Time" on page 195
- "Fall Time" on page 195
- "Delay" on page 196
- "Phase" on page 197

Period

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

Frequency

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

See Also

- "To isolate an event for frequency measurement" on page 194

To isolate an event for frequency measurement

The following figure shows how to use Zoom mode to isolate an event for a frequency measurement.

You may need to change the measurement window setting so that the measurement is made in the lower, Zoom window. See "Measurement Window with Zoom Display" on page 200.

If the waveform is clipped, it may not be possible to make the measurement.

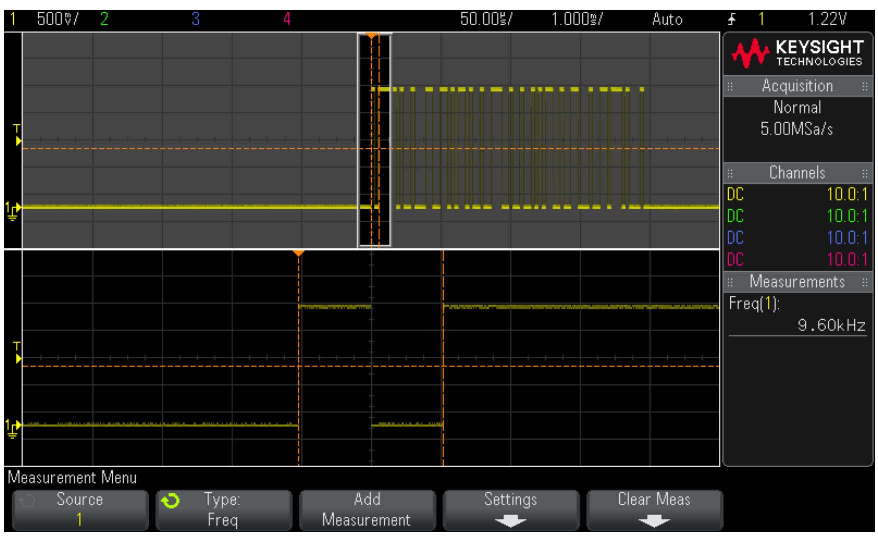


Figure 35 Isolating an event for Frequency measurement

+ Width

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

– Width

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

Duty Cycle

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

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

Rise Time

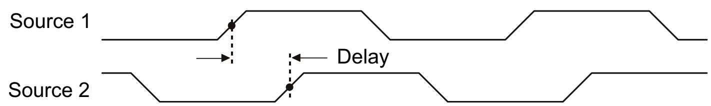
The rise time of a signal is the time difference between the crossing of the lower threshold and the crossing of the upper threshold for a positive-going edge. The X cursor shows the edge being measured. For maximum measurement accuracy, set the horizontal time/div as fast as possible while leaving the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

Fall Time

The fall time of a signal is the time difference between the crossing of the upper threshold and the crossing of the lower threshold for a negative-going edge. The X cursor shows the edge being measured. For maximum measurement accuracy, set the horizontal time/div as fast as possible while leaving the complete falling edge of the waveform on the display. The Y cursors show the lower and upper threshold points.

Delay

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

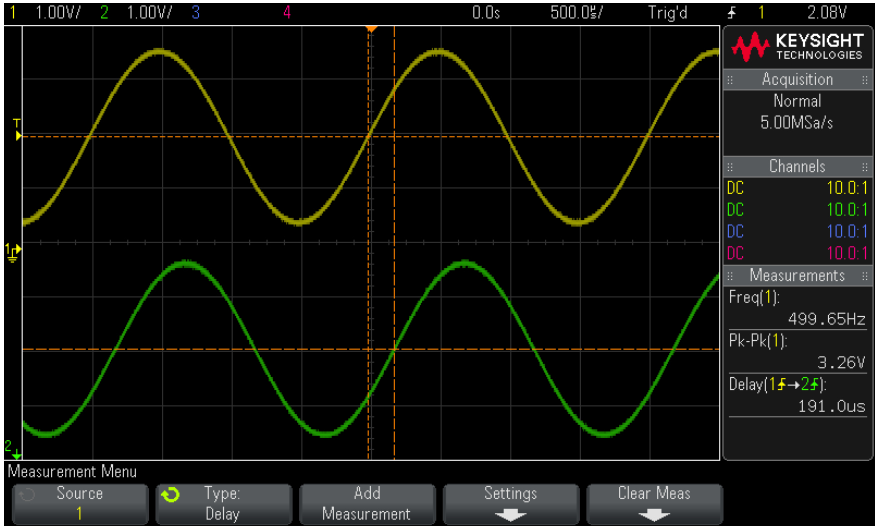


1 Press the **[Meas]** key to display the Measurement Menu.
2 Press the **Source** softkey; then turn the Entry knob to select the first analog channel source.
3 Press the **Type:** softkey; then, turn the Entry knob to select **Delay**.
4 Press the **Settings** softkey to select the second analog channel source and slope for the delay measurement.

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

5 Press the Back/Up key to return to the Measurement Menu.
6 Press the **Add Measurement** softkey to make the measurement.

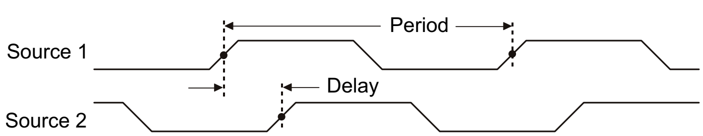
The example below shows a delay measurement between the rising edge of channel 1 and the rising edge of channel 2.



Phase

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

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



1 Press the **[Meas]** key to display the Measurement Menu.
2 Press the **Source** softkey; then turn the Entry knob to select the first analog channel source.
3 Press the **Type:** softkey; then, turn the Entry knob to select **Delay**.

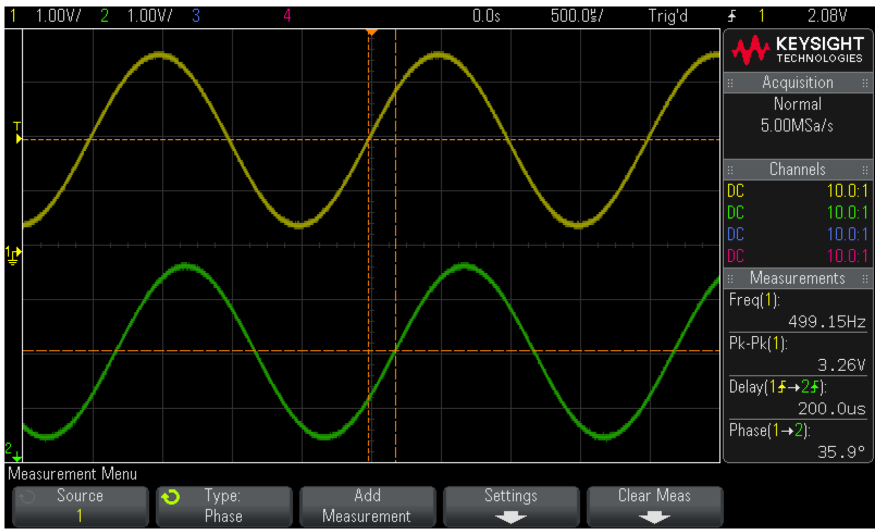
4 Press the **Settings** softkey to select the second analog channel source for the phase measurement.

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

5 Press the Back/Up key to return to the Measurement Menu.

6 Press the **Add Measurement** softkey to make the measurement.

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



Measurement Thresholds

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

NOTE

Changing default thresholds may change measurement results

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

1 From the Measurement Menu, press the **Settings** softkey; then, press the **Thresholds** softkey to set analog channel measurement thresholds.

   You can also open the Measurement Threshold Menu by pressing **[Analyze] > Features** and then selecting **Measurement Thresholds**.

2 Press the **Source** softkey to select the analog channel or math waveform source for which you want to change measurement thresholds.

   Each analog channel and the math waveform can be assigned unique threshold values.

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

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

TIP

Absolute threshold hints

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

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

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

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

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

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

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

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

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

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

Measurement Window with Zoom Display

When the zoomed time base is displayed, you can choose whether measurements are made in the Main window portion of the display or the Zoom window portion of the display.

1 Press the **[Meas]** key.

2 In the Measurement Menu, press the **Settings** softkey.

3 In the Measurement Settings Menu, press the **Meas Window** softkey; then, turn the Entry knob to select from:

   - **Auto Select** — The measurement is attempted in the lower, Zoom window; if it cannot be made there, the upper, Main window is used.
   - **Main** — The measurement window is the upper, Main window.
   - **Zoom** — The measurement window is the lower, Zoom window.

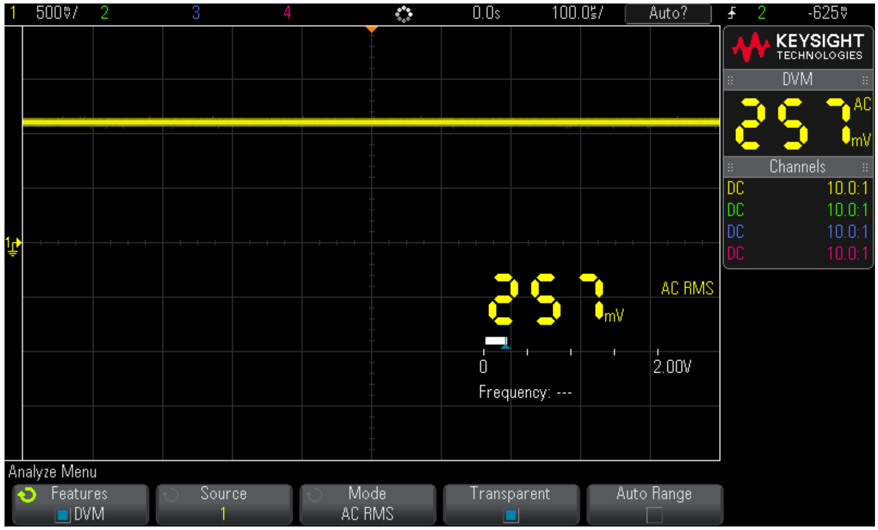
16 Digital Voltmeter

The Digital Voltmeter (DVM) analysis feature provides 3-digit voltage and 5-digit frequency measurements using any analog channel. DVM measurements are asynchronous from the oscilloscope's acquisition system and are always acquiring.

To enable the digital voltmeter analysis feature, order Option DVM at time of oscilloscope purchase, or order DSOXDVM as a stand-alone item after oscilloscope purchase.

The DVM display is a seven-segment readout like you would see on a digital voltmeter. It shows the selected mode as well as the units. Units are selected using the **Units** softkey in the channel's Probe Menu.

After pressing the **[Analyze]** key, the DVM display also appears in the graticule along with a scale and the frequency counter value. The DVM scale is determined by the channel's vertical scale and reference level. The scale's blue triangle pointer shows the most recent measurement. The white bar above that shows the measurement extrema over the last 3 seconds.



The DVM makes accurate RMS measurements when the signal frequency is between 20 Hz and 100 kHz. When the signal frequency is outside this range, "<BW Limit?" or ">BW Limit?" appears in the DVM display to caution you about inaccurate RMS measurement results.

To use the digital voltmeter:

1 Press the **[Analyze]** key.

2 Press **Features**; then, select **Digital Voltmeter**.

3 Press **Features** again to enable the DVM measurements.

4 Press the **Source** softkey and turn the Entry knob to select the analog channel on which digital voltmeter (DVM) measurements are made.

The selected channel does not have to be on (displaying a waveform) in order for DVM measurements to be made.

5 Press the **Mode** softkey and turn the Entry knob to select the digital voltmeter (DVM) mode:

- **AC RMS** — displays the root-mean-square value of the acquired data, with the DC component removed.
- **DC** — displays the DC value of the acquired data.
- **DC RMS** — displays the root-mean-square value of the acquired data.

- **Frequency** — displays the frequency counter measurement.

6 Press **Transparent** to toggle between a transparent and shaded background for the DVM display.

7 If the selected source channel is not used in oscilloscope triggering, press **Auto Range** to disable or enable automatic adjustment of the DVM channel's vertical scale, vertical (ground level) position, and trigger (threshold voltage) level (used for the counter frequency measurement).

   When enabled, **Auto Range** overrides attempted adjustments of the channel's vertical scale and position knobs.

   When disabled, you can use the channel's vertical scale and position knobs normally.

17 Waveform Generator

A waveform generator is built into the oscilloscope. It is enabled by Option WGN or the DSOX2WAVEGEN upgrade. The waveform generator gives you an easy way to provide input signals when testing circuitry with the oscilloscope.

Waveform generator settings can be saved and recalled with oscilloscope setups. See Chapter 18, “Save/Recall (Setups, Screens, Data),” starting on page 229.

To select generated waveform types and settings

1 To access the Waveform Generator Menu and enable or disable the waveform generator output on the front panel Gen Out BNC, press the **[Wave Gen]** key.

   When waveform generator output is enabled, the **[Wave Gen]** key is illuminated. When waveform generator output is disabled, the **[Wave Gen]** key is off.

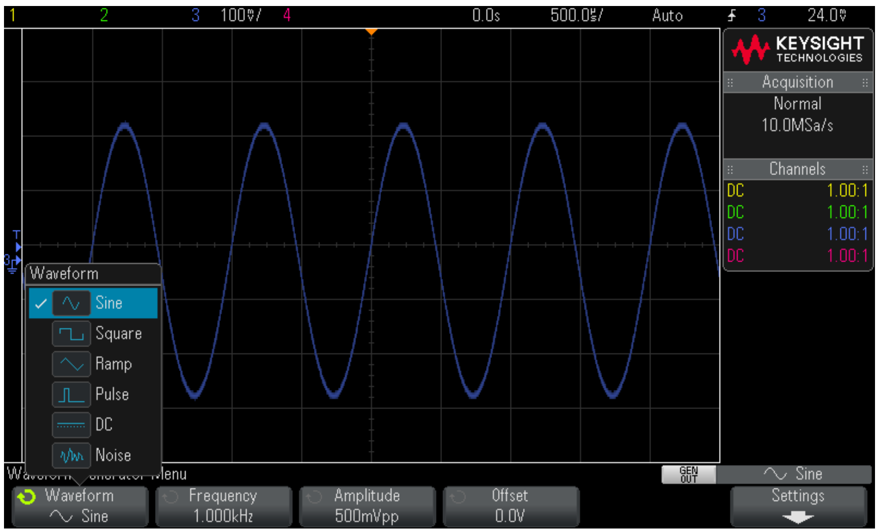
   The waveform generator output is always disabled when the instrument is first turned on.

   The waveform generator output is automatically disabled if excessive voltage is applied to the Gen Out BNC.

CAUTION

It takes the overload protection circuit about 10 ms to respond to an overload. If you instantly apply a voltage greater than ~40 V, you are likely to damage waveform generator circuitry before the protection circuit can respond.

2 In the Waveform Generator Menu, press the **Waveform** softkey and turn the Entry knob to select the waveform type.



3 Depending on the selected waveform type, use the remaining softkeys and the Entry knob to set the waveform's characteristics.

Waveform Type	Characteristics
Sine	Use the **Frequency/Frequency Fine/Period/Period Fine**, **Amplitude**/**High-Level**, and **Offset**/**Low-Level** softkeys to set the sine signal parameters. The frequency can be adjusted from 100 mHz to 20 MHz.
Square	Use the **Frequency/Frequency Fine/Period/Period Fine**, **Amplitude**/**High-Level**, **Offset**/**Low-Level**, and **Duty Cycle** softkeys to set the square wave signal parameters. The frequency can be adjusted from 100 mHz to 10 MHz. The duty cycle can be adjusted from 20% to 80%.
Ramp	Use the **Frequency/Frequency Fine/Period/Period Fine**, **Amplitude**/**High-Level**, **Offset**/**Low-Level**, and **Symmetry** softkeys to set the ramp signal parameters. The frequency can be adjusted from 100 mHz to 100 kHz. Symmetry represents the amount of time per cycle that the ramp waveform is rising and can be adjusted from 0% to 100%.
Pulse	Use the **Frequency/Frequency Fine/Period/Period Fine**, **Amplitude**/**High-Level**, **Offset**/**Low-Level**, and **Width/Width Fine** softkeys to set the pulse signal parameters. The frequency can be adjusted from 100 mHz to 10 MHz. The pulse width can be adjusted from 20 ns to the period minus 20 ns.
DC	Use the **Offset** softkey to set the DC level.
Noise	Use the **Amplitude**/**High-Level** and **Offset**/**Low-Level** to set the noise signal parameters.

For all waveform types, the output amplitude, into 50 Ω, can be adjusted from 10 mVpp to 2.5 Vpp (or from 20 mVpp to 5 Vpp into an open-circuit load).

Pressing a signal parameter softkey can open a menu for selecting the type of adjustment. For example, you can choose to enter amplitude and offset values, or you can choose to enter high-level and low-level values. Or, you can choose to enter frequency values or period values. Keep pressing the softkey to select the type of adjustment. Turn the Entry knob to adjust the value.

Notice that you can select between coarse and fine adjustments for frequency, period, and width. Also, pushing the Entry knob is a quick way to toggle between coarse and fine adjustments.

The **Settings** softkey opens the Waveform Generator Settings Menu which lets you make other settings related to the waveform generator.

See:

- "To output the waveform generator sync pulse" on page 220
- "To specify the expected output load" on page 220
- "To use waveform generator logic presets" on page 221
- "To restore waveform generator defaults" on page 227

To output the waveform generator sync pulse

1 If the Waveform Generator Menu is not currently displayed on the oscilloscope's softkeys, press the **[Wave Gen]** key.
2 In the Waveform Generator Menu, press the **Settings** softkey.
3 In the Waveform Generator Settings Menu, press the **Trig Out** softkey and turn the Entry knob to select **Waveform Generator Sync Pulse**.

Waveform Type	Sync Signal Characteristics
All waveforms except DC and Noise	The Sync signal is a TTL positive pulse that occurs when the waveform rises above zero volts (or the DC offset value).
DC	N/A
Noise	N/A

To specify the expected output load

1 If the Waveform Generator Menu is not currently displayed on the oscilloscope's softkeys, press the **[Wave Gen]** key.
2 In the Waveform Generator Menu, press the **Settings** softkey.

3 In the Waveform Generator Settings Menu, press the **Out Load** softkey and turn the Entry knob to select:

- **50 Ω**
- **High-Z**

The output impedance of the Gen Out BNC is fixed at 50 ohms. However, the output load selection lets the waveform generator display the correct amplitude and offset levels for the expected output load.

If the actual load impedance is different than the selected value, the displayed amplitude and offset levels will be incorrect.

To use waveform generator logic presets

With logic level presets, you can easily set the output voltage to TTL, CMOS (5.0V), CMOS (3.3V), CMOS (2.5V), or ECL compatible Low and High levels.

1 If the Waveform Generator Menu is not currently displayed on the oscilloscope's softkeys, press the **[Wave Gen]** key.

2 In the Waveform Generator Menu, press the **Settings** softkey.

3 In the Waveform Generator Settings Menu, press the **Logic Presets** softkey.

4 In the Waveform Generator Logic Level Presets Menu, press one of the softkeys to set the generated signal's Low and High voltages to logic compatible levels:

Softkey (logic levels)	Low level	High level, 50 ohm expected output load	High level, high-Z expected output load
TTL	0 V	+2.5 V (TTL compatible)	+5 V
CMOS (5.0V)	0 V	Not Available	+5 V
CMOS (3.3V)	0 V	+2.5 V (CMOS compatible)	+3.3 V
CMOS (2.5V)	0 V	+2.5 V	+2.5 V
ECL	-1.7 V	-0.8 V (ECL compatible)	-0.9 V

To add noise to the waveform generator output

1 If the Waveform Generator Menu is not currently displayed on the oscilloscope's softkeys, press the **[Wave Gen]** key.

2 In the Waveform Generator Menu, press the **Settings** softkey.

3 In the Waveform Generator Settings Menu, press the **Add Noise** softkey and turn the Entry knob to select the amount of white noise to add to the waveform generator output.

Note that adding noise affects edge triggering on the waveform generator source (see “Edge Trigger" on page 127) as well as the waveform generator sync pulse output signal (which can be sent to TRIG OUT, see “Setting the Rear Panel TRIG OUT Source" on page 256). This is because the trigger comparator is located after the noise source.

To add modulation to the waveform generator output

Modulation is where an original carrier signal is modified according to the amplitude of a second modulating signal. The modulation type (AM, FM, or FSK) specifies how the carrier signal is modified.

To enable and set up modulation for the waveform generator output:

1 If the Waveform Generator Menu is not currently displayed on the oscilloscope's softkeys, press the **[Wave Gen]** key.

2 In the Waveform Generator Menu, press the **Settings** softkey.

3 In the Waveform Generator Settings Menu, press the **Modulation** softkey.

4 In the Waveform Generator Modulation Menu:

- Press the **Modulation** softkey to enable or disable modulated waveform generator output.

  You can enable modulation for all waveform generator function types except pulse, DC, and noise.

- Press the **Type** softkey and turn the Entry knob to select the modulation type:
  - **Amplitude Modulation (AM)** — the amplitude of the original carrier signal is modified according to the amplitude of the modulating signal. See "To set up Amplitude Modulation (AM)" on page 223.
  - **Frequency Modulation (FM)** — the frequency of the original carrier signal is modified according to the amplitude of the modulating signal. See "To set up Frequency Modulation (FM)" on page 224.
  - **Frequency-Shift Keying Modulation (FSK)** — the output frequency "shifts" between the original carrier frequency and a "hop frequency" at the specified FSK rate. The FSK rate specifies a digital square wave modulating signal. See "To set up Frequency-Shift Keying Modulation (FSK)" on page 226.

To set up Amplitude Modulation (AM)

In the Waveform Generator Modulation Menu (under **[Wave Gen] > Settings > Modulation**):

1 Press the **Type** softkey and turn the Entry knob to select **Amplitude Modulation (AM)**.

2 Press the **Waveform** softkey and turn the Entry knob to select the shape of the modulating signal:

   - **Sine**
   - **Square**
   - **Ramp**
   - **Sine Cardinal**
   - **Exponential Rise**
   - **Exponential Fall**

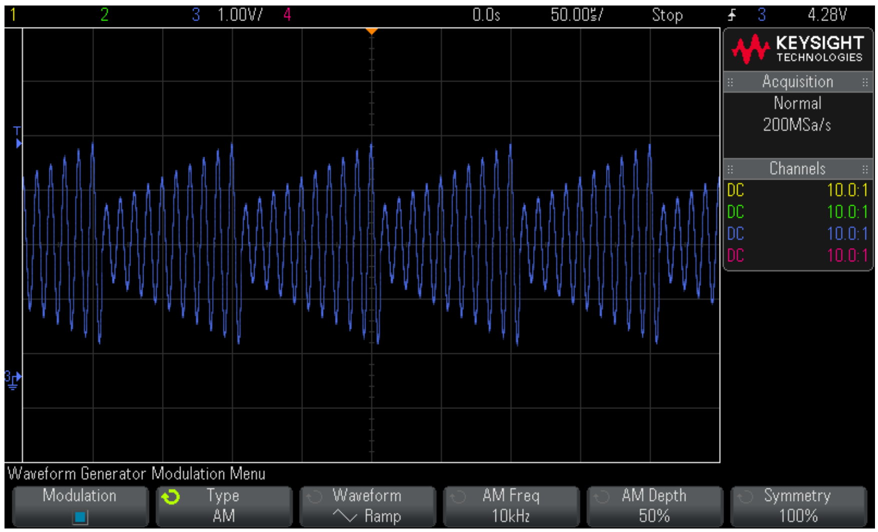
   When the **Ramp** shape is selected, a **Symmetry** softkey appears so that you can specify the amount of time per cycle that the ramp waveform is rising.

3 Press the **AM Freq** softkey and turn the Entry knob to specify the frequency of the modulating signal.

4 Press the **AM Depth** softkey and turn the Entry knob to specify the amount of amplitude modulation.

AM Depth refers to the portion of the amplitude range that will be used by the modulation. For example, a depth setting of 80% causes the output amplitude to vary from 10% to 90% (90% – 10% = 80%) of the original amplitude as the modulating signal goes from its minimum to maximum amplitude.

The following screen shows an AM modulation of a 100 kHz sine wave carrier signal.



To set up Frequency Modulation (FM)

In the Waveform Generator Modulation Menu (under **[Wave Gen] > Settings > Modulation**):

1 Press the **Type** softkey and turn the Entry knob to select **Frequency Modulation (FM)**.

2 Press the **Waveform** softkey and turn the Entry knob to select the shape of the modulating signal:

- **Sine**
- **Square**
- **Ramp**
- **Sine Cardinal**

- **Exponential Rise**
- **Exponential Fall**

When the **Ramp** shape is selected, a **Symmetry** softkey appears so that you can specify the amount of time per cycle that the ramp waveform is rising.

3 Press the **FM Freq** softkey and turn the Entry knob to specify the frequency of the modulating signal.

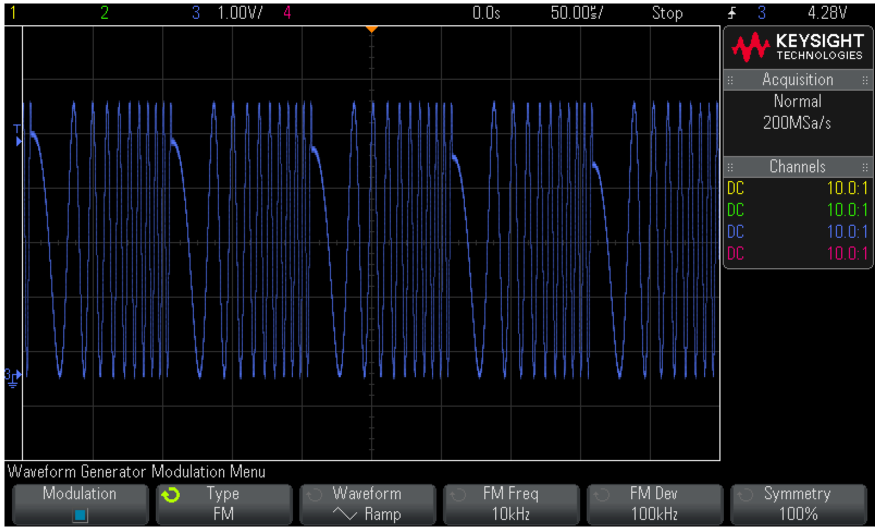
4 Press the **FM Dev** softkey and turn the Entry knob to specify the frequency deviation from the original carrier signal frequency.

When the modulating signal is at its maximum amplitude, the output frequency is the carrier signal frequency plus the deviation amount, and when the modulating signal is at its minimum amplitude, the output frequency is the carrier signal frequency minus the deviation amount.

The frequency deviation cannot be greater than the original carrier signal frequency.

Also, the sum of the original carrier signal frequency and the frequency deviation must be less than or equal to the maximum frequency for the selected waveform generator function plus 100 kHz.

The following screen shows an FM modulation of a 100 kHz sine wave carrier signal.



To set up Frequency-Shift Keying Modulation (FSK)

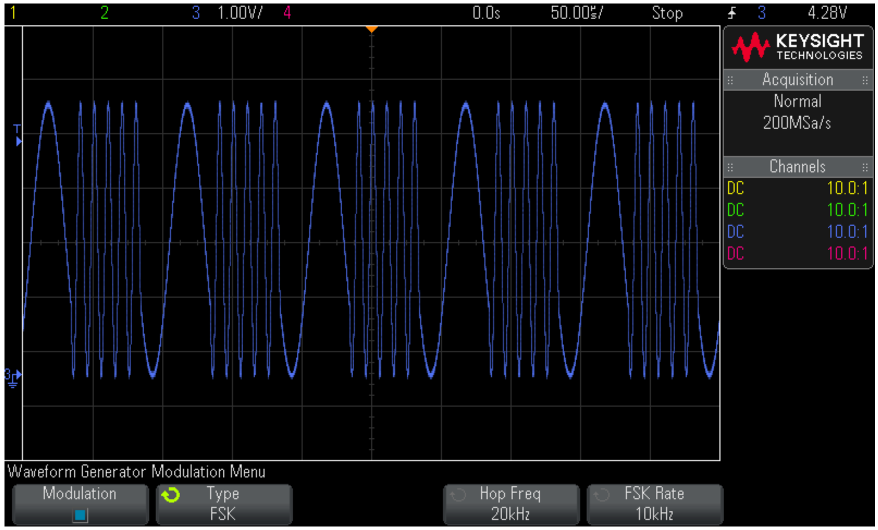
In the Waveform Generator Modulation Menu (under **[Wave Gen] > Settings > Modulation**):

1. Press the **Type** softkey and turn the Entry knob to select **Frequency-Shift Keying Modulation (FSK)**.
2. Press the **Hop Freq** softkey and turn the Entry knob to specify the "hop frequency".

   The output frequency "shifts" between the original carrier frequency and this "hop frequency".
3. Press the **FSK Rate** softkey and turn the Entry knob to specify the rate at which the output frequency "shifts".

   The FSK rate specifies a digital square wave modulating signal.

The following screen shows an FSK modulation of a 100 kHz sine wave carrier signal.



To restore waveform generator defaults

1 If the Waveform Generator Menu is not currently displayed on the oscilloscope's softkeys, press the **[Wave Gen]** key.

2 In the Waveform Generator Menu, press the **Settings** softkey.

3 In the Waveform Generator Settings Menu, press the **Default Wave Gen** softkey.

The waveform generator factory default settings (1 kHz sine wave, 500 mVpp, 0 V offset, High-Z output load) are restored.

18 Save/Recall (Setups, Screens, Data)

Oscilloscope setups, reference waveforms, and mask files can be saved to internal oscilloscope memory or to a USB storage device and recalled later. You can also recall default or factory default setups.

Oscilloscope screen images can be saved to a USB storage device in BMP or PNG formats.

Acquired waveform data can be saved to a USB storage device in comma-separated value (CSV), ASCII XY, and binary (BIN) formats.

There is also a command to securely erase all the oscilloscope's non-volatile internal memory.

Saving Setups, Screen Images, or Data

1 Press the **[Save/Recall]** key.
2 In the Save/Recall Menu, press **Save**.
3 In the Save Trace and Setup Menu, press **Format**, then, turn the Entry knob to select the type of file you want to save:

- **Setup (*.scp)** — The oscilloscope's horizontal timebase, vertical sensitivity, trigger mode, trigger level, measurements, cursors, and math function settings that tell the oscilloscope how to make a particular measurement. See "To save setup files" on page 231.
- **8-bit Bitmap image (*.bmp)** — The complete screen image in a reduced color (8-bit) bitmap format. See "To save BMP or PNG image files" on page 231.
- **24-bit Bitmap image (*.bmp)** — The complete screen image in a 24-bit color bitmap format. See "To save BMP or PNG image files" on page 231.
- **24-bit image (*.png)** — The complete screen image in a 24-bit color PNG format that uses lossless compression. Files are much smaller than the BMP format. See "To save BMP or PNG image files" on page 231.
- **CSV data (*.csv)** — This creates a file of comma-separated values of all displayed channels and math waveforms. This format is suitable for spreadsheet analysis. See "To save CSV, ASCII XY, or BIN data files" on page 232.
- **ASCII XY data (*.csv)** — This creates separate files of comma-separated values for each displayed channel. This format is also suitable for spreadsheets. See "To save CSV, ASCII XY, or BIN data files" on page 232.
- **Reference Waveform data (*.h5)** — Saves waveform data in a format that can be recalled to one of the oscilloscope's reference waveform locations. See "To save reference waveform files to a USB storage device" on page 235.
- **Multi Channel Waveform data (*.h5)** — Saves multiple channels of waveform data in a format that can be opened by the N8900A InfiniiView oscilloscope analysis software. You can recall the first Analog or Math channel from a multi channel waveform data file.
- **Binary data (*.bin)** — This creates a binary file, with a header, and data in the form of time and voltage pairs. This file is much smaller than the ASCII XY data file. See "To save CSV, ASCII XY, or BIN data files" on page 232.
- **Lister data (*.csv)** — This is a CSV format file containing serial decode row information with commas separating the columns. See "To save Lister data files" on page 234.
- **Mask (*.msk)** — This creates a mask file in a Keysight proprietary format that can be read by Keysight InfiniiVision oscilloscopes. A mask data file includes some oscilloscope setup information, but not all setup information. To save all setup information including the mask data file, choose "Setup (*.scp)" format instead. See "To save masks" on page 235.

You can also configure the **[Quick Action]** key to save setups, screen images, or data. See "Configuring the [Quick Action] Key" on page 262.

To save setup files

Setup files can be saved to one of 10 internal (\Keysight Flash) locations or to an external USB storage device.

1 Press **[Save/Recall] > Save > Format**; then, turn the Entry knob to select **Setup (*.scp)**.

2 Press the softkey in the second position and use the Entry knob to navigate to the save location. See "To navigate storage locations" on page 236.

3 Finally, press the **Press to Save** softkey.

   A message indicating whether the save was successful is displayed.

Setup files have the extension SCP. These extensions appear when using the File Explorer (see "File Explorer" on page 251), but they do not appear when using the Recall Menu.

To save BMP or PNG image files

Image files can be saved to an external USB storage device.

1 Press **[Save/Recall] > Save > Format**; then, turn the Entry knob to select **8-bit Bitmap image (*.bmp)**, **24-bit Bitmap image (*.bmp)**, or **24-bit image (*.png)**.

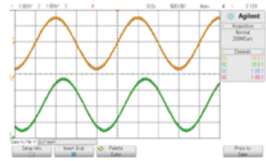
2 Press the softkey in the second position and use the Entry knob to navigate to the save location. See "To navigate storage locations" on page 236.

3 Press the **Settings** softkey.

   In the File Settings Menu, you have these softkeys and options:

   - **Setup Info** — setup information (vertical, horizontal, trigger, acquisition, math, and display settings) is also saved in a separate file with a TXT extension.
   - **Invert Grat** — the graticule in the image file has a white background instead of the black backgound that appears on-screen.

Graticule Not Inverted



Graticule Inverted

- **Palette** — lets you choose between **Color** or **Grayscale** images.

4 Finally, press the **Press to Save** softkey.

A message indicating whether the save was successful is displayed.

NOTE When saving screen images, the oscilloscope uses the last menu visited before pressing the **[Save/Recall]** key. This lets you save any relevant information within the softkey menu area.

To save a screen image showing the Save/Recall Menu at the bottom, press the **[Save/Recall]** key twice before saving the image.

NOTE You can also save the oscilloscope's display image using a web browser. See "Get Image" on page 277 for details.

See Also

- "Adding an Annotation" on page 263

To save CSV, ASCII XY, or BIN data files

Data files can be saved to an external USB storage device.

1 Press **[Save/Recall] > Save > Format**; then, turn the Entry knob to select **CSV data (*.csv)**, **ASCII XY data (*.csv)**, or **Binary data (*.bin)**.

2 Press the softkey in the second position and use the Entry knob to navigate to the save location. See "To navigate storage locations" on page 236.

3 Press the **Settings** softkey.

In the File Settings Menu, you have these softkeys and options:

- **Setup Info** — when enabled, setup information (vertical, horizontal, trigger, acquisition, math, and display settings) is also saved in a separate file with a TXT extension.

- **Length** — sets the number of data points that will be output to the file. For more information, see "Length Control" on page 233.
- **Save Seg** — when data is acquired to segmented memory, you can specify whether the currently displayed segment is saved or all acquired segments are saved. (See also "Saving Data from Segmented Memory" on page 169.)

4 Finally, press the **Press to Save** softkey.

A message indicating whether the save was successful is displayed.

See Also

- "Binary Data (.bin) Format" on page 288
- "CSV and ASCII XY files" on page 295
- "Minimum and Maximum Values in CSV Files" on page 296

Length Control

The **Length** control is available when saving data to CSV, ASCII XY, or BIN format files. It sets the number of data points that will be output to the file. Only displayed data points are saved.

The maximum number of data points depends on these things:

- Whether acquisitions are running. When stopped, data comes from the raw acquisition record. When running, data comes from the smaller measurement record.
- Whether the oscilloscope was stopped using **[Stop]** or **[Single]**. Running acquisitions split memory to provide fast waveform update rates. Single acquisitions use full memory.
- Whether only one channel of a pair is turned on. (Channels 1 and 2 are one pair, channels 3 and 4 are the other.) Acquisition memory is divided among the channels in a pair.
- Whether reference waveforms are on. Displayed reference waveforms consume acquisition memory.
- Whether digital channels are on. Displayed digital channels consume acquisition memory.
- Whether segmented memory is on. Acquisition memory is divided by the number of segments.
- The horizontal time/div (sweep speed) setting. At faster settings, fewer data points appear on the display.

- When saving to a CSV format file, the maximum number of data points is 50,000.

When necessary, the Length control performs a "1 of n" decimation of the data . For example: if the **Length** is set to 1000, and you are displaying a record that is 5000 data points in length, four of each five data points will be decimated, creating an output file 1000 data points in length.

When saving waveform data, the save times depend on the chosen format:

Data File Format	Save Times
BIN	fastest
ASCII XY	medium
CSV	slowest

See Also

- "Binary Data (.bin) Format" on page 288
- "CSV and ASCII XY files" on page 295
- "Minimum and Maximum Values in CSV Files" on page 296

To save Lister data files

Lister data files can be saved to an external USB storage device.

1 Press **[Save/Recall] > Save > Format**; then, turn the Entry knob to select **Lister data file**.

2 Press the softkey in the second position and use the Entry knob to navigate to the save location. See "To navigate storage locations" on page 236.

3 Press the **Settings** softkey.

   In the File Settings Menu, you have these softkeys and options:

   - **Setup Info** — when enabled, setup information (vertical, horizontal, trigger, acquisition, math, and display settings) is also saved in a separate file with a TXT extension.

4 Finally, press the **Press to Save** softkey.

   A message indicating whether the save was successful is displayed.

To save reference waveform files to a USB storage device

1. Press the **[Save/Recall]** key.
2. In the Save/Recall Menu, press the **Save** softkey.
3. In the Save Menu, press the **Format** softkey and turn the Entry knob to select **Reference Waveform data (*.h5)** .
4. Press the **Source** softkey and turn the Entry knob to select the source waveform.
5. Press the softkey in the second position and use the Entry knob to navigate to the save location. See "To navigate storage locations" on page 236.
6. Finally, press the **Press to Save** softkey.

   A message indicating whether the save was successful is displayed.

To save masks

Mask files can be saved to one of four internal (\Keysight Flash) locations or to an external USB storage device.

1. Press **[Save/Recall] > Save > Format**; then, turn the Entry knob to select **Mask (*.msk)**.
2. Press the softkey in the second position and use the Entry knob to navigate to the save location. See "To navigate storage locations" on page 236.
3. Finally, press the **Press to Save** softkey.

   A message indicating whether the save was successful is displayed.

Mask files have the extension MSK.

NOTE Masks are also saved as part of setup files. See "To save setup files" on page 231.

See Also · Chapter 15, "Mask Testing," starting on page 201

To navigate storage locations

When saving or recalling files, the softkey in the second position of the Save Menu or Recall Menu, along with the Entry knob, are used to navigate to storage locations. The storage locations can be internal oscilloscope storage locations (for setup files or mask files) or they can be external storage locations on a connected USB storage device.

The softkey in the second position can have these labels:

- **Press to go** — when you can push the Entry knob to navigate to a new folder or storage location.
- **Location** — when you have navigated to the current folder location (and are not saving files).
- **Save to** — when you can save to the selected location.
- **Load from** — when you can recall from the selected file.

When saving files:

- The proposed file name is shown in the **Save to file =** line above the softkeys.
- To overwrite an existing file, browse to that file and select it. To create a new file name, see "To enter file names" on page 236.

To enter file names

To create new file names when saving files to a USB storage device:

1 In the Save Menu, press the **File Name** softkey.

You must have a USB storage device connected to the oscilloscope for this softkey to be active.

2 In the File Name Menu, use the **Spell**, **Enter**, and **Delete Character** softkeys to enter the file name:

- **Spell** — press this softkey and turn the Entry knob to select the character at the current position.
- **Enter** — press this softkey to enter characters and move the cursor to the next character position. Pushing the Entry knob is the same as pressing the **Enter** softkey.
- **Delete Character** — press this softkey to delete the character at the current position.

NOTE

You can use a connected USB keyboard instead of using the **Spell** (and other) character editing softkeys.

When available, the **Increment** softkey can be used to enable or disable automatically incremented file names. Auto increment adds a numeric suffix to your file name and increments the number with each successive save. It will truncate characters as necessary when the file name length is at maximum and more digits are required for the numeric portion of the file name.

Recalling Setups, Masks, or Reference Waveforms

1 Press the **[Save/Recall]** key.
2 In the Save/Recall Menu, press **Recall**.
3 In the Recall Menu, press **Recall:**, then, turn the Entry knob to select the type of file you want to recall:
   - **Setup (*.scp)** — See "To recall setup files" on page 237.
   - **Mask (*.msk)** — See "To recall mask files" on page 238.
   - **Reference Waveform data (*.h5)** — See "To recall reference waveform files from a USB storage device" on page 238.

You can also recall setups and mask files by loading them using the File Explorer. See "File Explorer" on page 251.

You can also configure the **[Quick Action]** key to recall setups, masks, or reference waveforms. See "Configuring the [Quick Action] Key" on page 262.

To recall setup files

Setup files can be recalled from one of 10 internal (\Keysight Flash) locations or from an external USB storage device.

1 Press **[Save/Recall] > Recall > Recall:**; then, turn the Entry knob to select **Setup (*.scp)**.
2 Press the softkey in the second position and use the Entry knob to navigate to the file to recall. See "To navigate storage locations" on page 236.
3 Press the **Press to Recall** softkey.

A message indicating whether the recall was successful is displayed.

4 If you would like to clear the display, press **Clear Display**.

To recall mask files

Mask files can be recalled from one of four internal (\Keysight Flash) locations or from an external USB storage device.

1 Press **[Save/Recall] > Recall > Recall:**; then, turn the Entry knob to select **Mask (*.msk)**.

2 Press the softkey in the second position and use the Entry knob to navigate to the file to recall. See "To navigate storage locations" on page 236.

3 Press the **Press to Recall** softkey.

A message indicating whether the recall was successful is displayed.

4 If you would like to clear the display or clear the recalled mask, press **Clear Display** or **Clear Mask**.

To recall reference waveform files from a USB storage device

1 Press the **[Save/Recall]** key.

2 In the Save/Recall Menu, press the **Recall** softkey.

3 In the Recall Menu, press the **Recall** softkey and turn the Entry knob to select **Reference Waveform data (*.h5)** .

4 Press the **To Ref:** softkey and turn the Entry knob to select the desired reference waveform location.

5 Press the softkey in the second position and use the Entry knob to navigate to the file to recall. See "To navigate storage locations" on page 236.

6 Press the **Press to Recall** softkey.

A message indicating whether the recall was successful is displayed.

7 If you would like to clear the display of everything except the reference waveform, press **Clear Display**.

Recalling Default Setups

1 Press the **[Save/Recall]** key.

2 In the Save/Recall Menu, press **Default/Erase**.

3 In the Default Menu, press one of these softkeys:

- **Default Setup**— recalls the oscilloscope's default setup. This is the same as pressing the front panel **[Default Setup]** key. See "Recall the Default Oscilloscope Setup" on page 29.

  Some user settings are not changed when recalling the default setup.

- **Factory Default**— recalls the oscilloscope's factory default settings.

  You must confirm the recall because there are no user settings that are left unchanged.

Performing a Secure Erase

1 Press the **[Save/Recall]** key.

2 In the Save/Recall Menu, press **Default/Erase**.

3 In the Default menu, press **Secure Erase**.

   This performs a secure erase of all non-volatile memory in compliance with National Industrial Security Program Operation Manual (NISPOM) Chapter 8 requirements.

   You must confirm the secure erase, and the oscilloscope will reboot when finished.