DSO1000 Educator's Training Resources
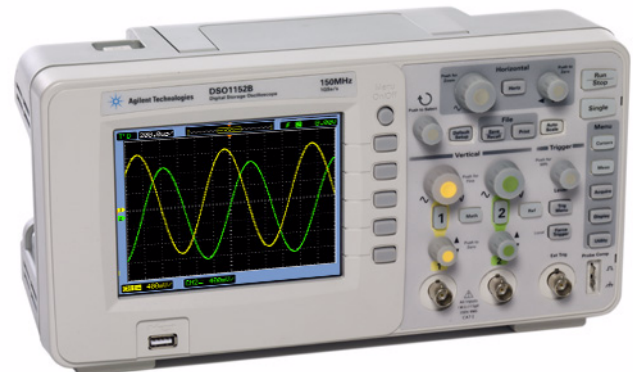
Lab Guide and Tutorial for Undergraduate Electrical Engineering and Physics Students

Notices

Trademarks

Manual Part Number

54136-97000

Edition

Warranty

Technology Licenses

Restricted Rights Legend

Safety Notices

CAUTION

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

WARNING

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

Lab Guide and Tutorial—At a Glance

This EE/Physics Student's Oscilloscope Lab Guide and Tutorial is intended to be used with Agilent Technologies DSO1000 Series oscilloscopes.

A Note to the EE/Physics Professor

Dear University EE/Physics Professor and/or Lab Instructor,

This EE/Physics student oscilloscope lab guide and tutorial for the Agilent DSO1000 Series oscilloscope consists of seven individual hands-on labs that students can complete to become familiar with what an oscilloscope is and how to use one. An oscilloscope is the one measurement tool that your students will use more than any other instrument to test circuit experiments that you assign to them, as well as to test their senior design projects. They will also use oscilloscopes extensively after they graduate and enter the world of today's electronics industry. So it is extremely important that they become proficient in the use of this vital tool.

Each of the labs requires about 15 to 20 minutes to complete. These labs are intended to be used with Agilent's 1000 Series oscilloscopes. Before your students begin testing any of their assigned experiments in their first circuits lab, it is suggested that they read Section 1, Appendix A and Appendix B of this document as pre-study (homework). Section 1 provides an introduction to the oscilloscope, as well as some fundamentals on probing. Appendix A and Appendix B are short tutorials on oscilloscope theory of operation and bandwidth.

Students should then complete the hands-on labs in Section 2 of this document during their first lab session. Your students should have a basic knowledge of how to use a scope after completing just Lab #1 and Lab #2. But if they have time to complete all seven labs, they will become even more proficient at using the scope, including learning how to document and save results for your required circuit experiment lab reports. Note that Lab #3 (Capturing Single-shot Events) instructs students to "tap" their oscilloscope probe on the table/bench to create an electrostatic discharge. If you don't want your students banging their probes on the bench, then you may want to instruct them to skip this particular lab. This oscilloscope lab guide was structured for flexibility in its use.

Sincerely,

Johnnie Hancock
Oscilloscope Education Program Manager
Agilent Technologies

Contents

B Oscilloscope Bandwidth Tutorial

1

Getting Started

Oscilloscopes are a critical tool for making voltage and timing measurements on today's analog and digital electrical circuits. When you finally graduate from Electrical Engineering school and enter the electronics industry, you will probably discover that an oscilloscope is the one measurement tool that you will use more than any other instrument to test, verify, and debug your designs. Even while in your EE or Physics program at your particular university, an oscilloscope is the measurement tool that you will use most often in your various circuits labs to test and verify your lab assignments and designs. Unfortunately many students never fully grasp how to use an oscilloscope. Their use-model is often one of randomly twisting knobs and pressing buttons until a picture close to what they are looking for magically appears on the scope's display. Hopefully after completing this series of short labs you will have a better understanding of what a scope is and how to use one more efficiently.

So, what is an oscilloscope? An oscilloscope is an electronic measurement instrument that unintrusively monitors input signals and then graphically displays these signals in a simple voltage versus time format. The type of oscilloscope that your professor used during his or her undergraduate studies was probably one that was based entirely on analog technology. These older technology oscilloscopes, typically called analog scopes, had limited bandwidth (which is discussed in Appendix B), did not perform any kind of automatic measurements, and also required that the input signal be repetitive (continuously occurring and repeating input signal).

The type of oscilloscope that you will be using in this series of labs, and probably throughout the rest of your university studies, is called a digital storage oscilloscope; sometimes simply referred to as a DSO. Today's DSOs can capture and display either repetitive or single-shot signals, and they often include an array of automatic measurements and analysis capabilities that should enable you to characterize your designs and student experiments faster and more accurately than your professor was able to do back in his or her undergraduate days.

Agilent Technologies

If you are interested in learning about the theory of operation of an oscilloscope, refer to Appendix A in this document. But the best way to quickly learn how to use an oscilloscope, and understand what it can do for you, is to first get acquainted with some of the most important controls on an oscilloscope, then simply begin using one to measure some basic signals.
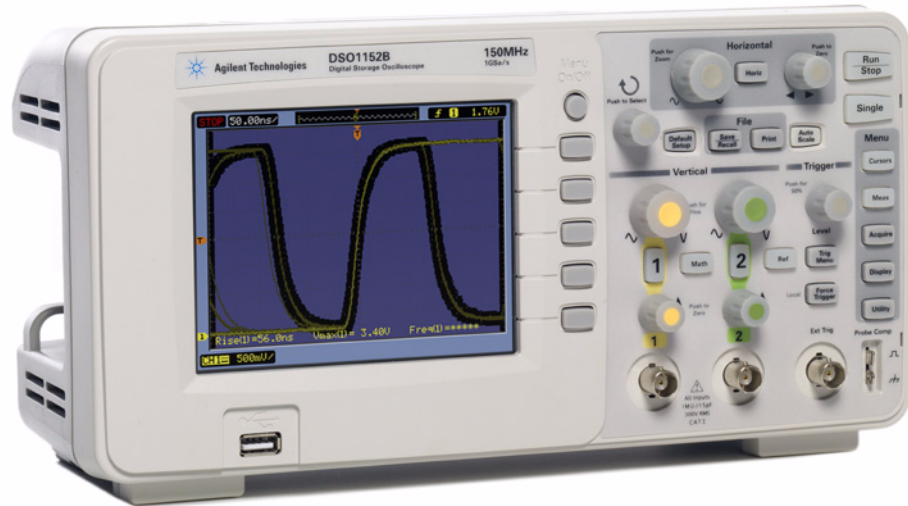
Figure 1   Agilent 1000B Series oscilloscope

Oscilloscope Probing

The first task when making oscilloscope measurements is typically to connect the oscilloscope probes between the device-under-test and the scope's input BNCs. Oscilloscope probes provide a relatively high input impedance termination (high resistance with low capacitance) at the test point. A high impedance connection is important in order to isolate the measuring instrument from the circuit being tested since we don't want the oscilloscope and its probe to change the characteristics of the signals under test.

There are a variety of different kinds of oscilloscope probes that are used for specific types of measurements, but the probes that you will be using today are the most commonly used type of probe and are called passive 10:1 voltage probes as shown in Figure 2. "Passive" simply means that this type of probe does not include any "active" components such as transistors or amplifiers. "10:1" means that this probe will attenuate the input signal received at the scope's input by a factor of 10.
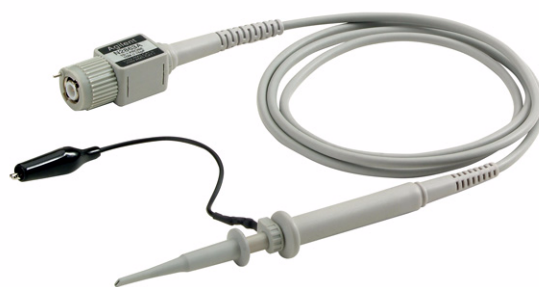
Figure 2   Passive 10:1 voltage probe

When using a standard 10:1 passive probe, all scope measurements should be performed between the signal test point and ground. It other words, you *must* connect the probe's ground clip to ground. You *cannot* measure voltages across a mid-circuit component using this type of probe. If you need to measure the voltage across a component that is not grounded, you could either use the scope's subtraction math function while measuring the signals at both ends of the component relative to ground using two channels of the scope, or you could use a special differential active probe. Also note that a circuit should never be completed using the oscilloscope.

Figure 3 shows an electrical model of a 10:1 passive probe when connected to an oscilloscope using the scope's default 1 MΩ input selection, which is required when using this type of probe. Note that many higher bandwidth scopes also have a user-selectable 50 Ω input termination selection often used for active probe terminations and/or used when inputting a signal directly from a 50 Ω source using a 50 Ω BNC coaxial cable.
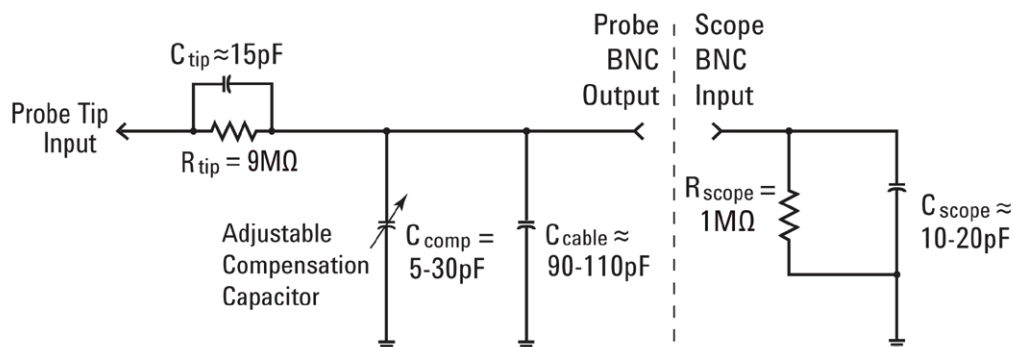
Figure 3   Simplified schematic of a passive 10:1 probe connected to the scope's 1 MΩ input impedance

Although the electrical model of the passive probe and scope includes both inherent/parasitic capacitance (not designed-in) as well as intentionally designed-in compensation capacitance networks, let's ignore these capacitive elements for now and analyze the ideal signal behavior of this probe/scope system under low-frequency or dc input conditions.

After we remove all of the capacitive components from our probe/scope electrical model, what remains is just a 9 MΩ probe tip resistor in series with the scope's 1 MΩ input impedance. The net input resistance at the probe tip is then 10 MΩ. Using Ohm's law, you can see that the voltage level received at the scope's input is then 1/10th the voltage level that is at the probe tip (V_{scope} = V_{probe} x (1 MΩ/10 MΩ).

This means that with a 10:1 passive probe the dynamic range of the oscilloscope measurement system has been extended. In other words, you can measure signals with 10X higher amplitude as compared to signals that you could measure using a 1:1 probe. In addition, the input impedance of your oscilloscope measurement system (probe + scope) is increased from 1 MΩ to 10 MΩ. This is a good thing since a lower input impedance could load your device-under-test (DUT) and possibly change actual voltage levels within your DUT, which is not a good thing. And although a net input impedance of 10 MΩ is indeed large, you must remember that this amount of loading impedance must be considered in relationship to the impedance of the device you are probing. For example, a simple op amp circuit with a 100 MΩ feedback resistor may give some false readings on a scope.

Although most higher performance oscilloscopes can detect when a 10:1 probe is connected, and then automatically set the scope's probe attenuation factor to 10:1, you must manually either enter the probe attenuation factor (10:1) when using an Agilent 1000B Series oscilloscope, or if you press the **[Default Setup]** front panel key to set the probe attenuation to 10:1. Once the scope knows what the probe attenuation factor is (either automatically-detected or manually-entered), the scope provides compensated readouts of all vertical settings so that all voltage measurements will be referenced to the unattenuated input signal at the probe tip. For example, if you probe a 10 Vpp signal, the signal received at the scope's input will actually be just 1 Vpp. But since the scope knows that you are using a 10:1 divider probe, the scope will report that it is seeing a 10 Vpp signal when performing voltage measurements.

When we get to Lab #4 (Compensating your Passive 10:1 Probes), we will come back to this passive probe model and address the capacitive components. These elements in the probe/scope electrical model will affect the dynamic/ac performance of the combined oscilloscope and probing system.

Getting Acquainted with the Front Panel

Let's begin by first getting acquainted with the most important controls/knobs on your oscilloscope. Near the top of your oscilloscope are the "Horizontal" controls shown in Figure 4. The larger knob sets the horizontal scaling in seconds/division. This control sets the X-axis scaling of the displayed waveform. One horizontal "division" is the Δ-time between each vertical grid line. If you want to view faster waveforms (higher frequency signals), then you will set the horizontal scaling to a smaller sec/div value. If you want to view slower waveforms (slower frequency signals), then you typically set the horizontal scaling to a higher sec/div setting. The smaller knob in the Horizontal section sets the horizontal position of the waveform. In other words, with this control you can move the horizontal placement of the waveform left and right. The scope's horizontal controls (s/div and position) are often called the scope's main "timebase" controls.

Figure 4   Oscilloscope horizontal (X-axis) controls

The controls/knobs closer the bottom of the oscilloscope (refer to Figure 5) in the Vertical section (just above the input BNCs) sets the vertical scaling of the oscilloscope. If you are using a 2-channel oscilloscope, then there will be two pairs of vertical scaling controls. If you are using a 4-channel oscilloscope, then there will be four pairs of vertical scaling controls. The larger knob for each input channel in the Vertical section sets the vertical scaling factor in Volts/division. This is the Y-axis graphical scaling of your waveforms. One vertical "division" is the ΔV between each horizontal grid line. If you want to view relatively large signals (high peak-to-peak voltages), then you would typically set the Volts/div setting to a relatively high value. If viewing small input signal levels, then you would set the Volts/div setting to a relatively low value. The smaller controls/knobs for each channel in the Vertical section are the position/offset controls. You use this knob to move the waveform up and down on the screen.
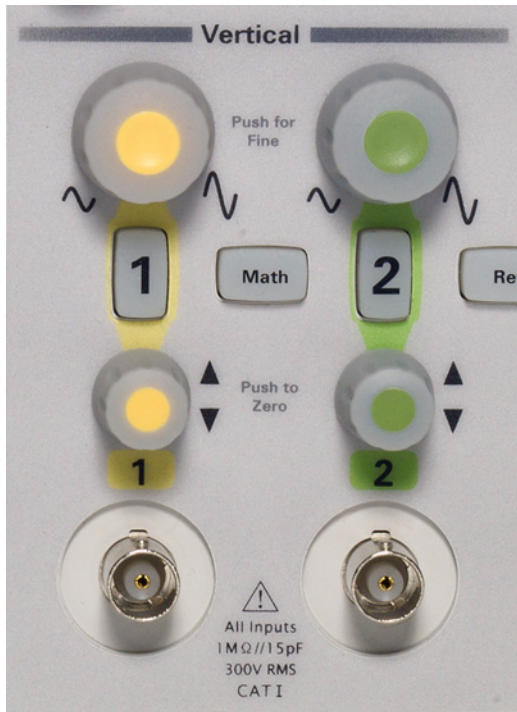
Figure 5   Oscilloscope vertical (Y-axis) controls

Another very important oscilloscope setup variable is the trigger level control/knob shown in Figure 6. This control knob is located near the right-hand side of your scope's front panel; just below the section labeled **Trigger**. Triggering is probably the least understood aspect of an oscilloscope, but it is one the most important capabilities of a scope that you should understand. We will cover oscilloscope triggering in more detail as we get into the hands-on labs.

Figure 6   Oscilloscope trigger level control

When reading the instructions in the following labs, anytime you see a bold word within brackets, such as **[Cursors]**, this is referring to a front panel key (or button) located on the righthand side of the oscilloscope. When the key is pressed, a unique menu with "softkey" selections associated with that particular front panel function will be activated.

"Softkeys" are the 5 keys/buttons located to right of the scope's display. The functions of these keys change depending upon which menu has been activated.

Now locate the **Entry** control knob shown in Figure 7. This is the knob just to the right of the scope's display. We will be using this knob quite frequently to change an array of setup variables and selections that don't have dedicated front panel controls. Any time you see the curled arrow (⟲) on a softkey selection; this is an indication that the **Entry** knob controls this variable. Let's begin making measurements on your oscilloscope!
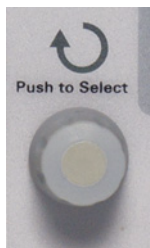
Figure 7   Oscilloscope general-entry control

2
Oscilloscope Familiarization Labs

Agilent Technologies

Lab #1: Making Basic Measurements

In this 1st lab you will learn how to use the scope's horizontal and vertical scaling controls in order to properly setup the scope to display a repetitive square wave. In addition, you will learn how to make some simple voltage and timing measurements on this signal.

1   Plug-in the power cord and turn the scope on.

2   Connect one oscilloscope probe between the channel-1 input BNC and the terminal labeled **Probe Comp** that has the "pulse" shape as shown in Figure 8. Connect the probe's ground clip to the terminal with the ground symbol. Note that on 4-channel models, the **Probe Comp** terminals are located below the display.
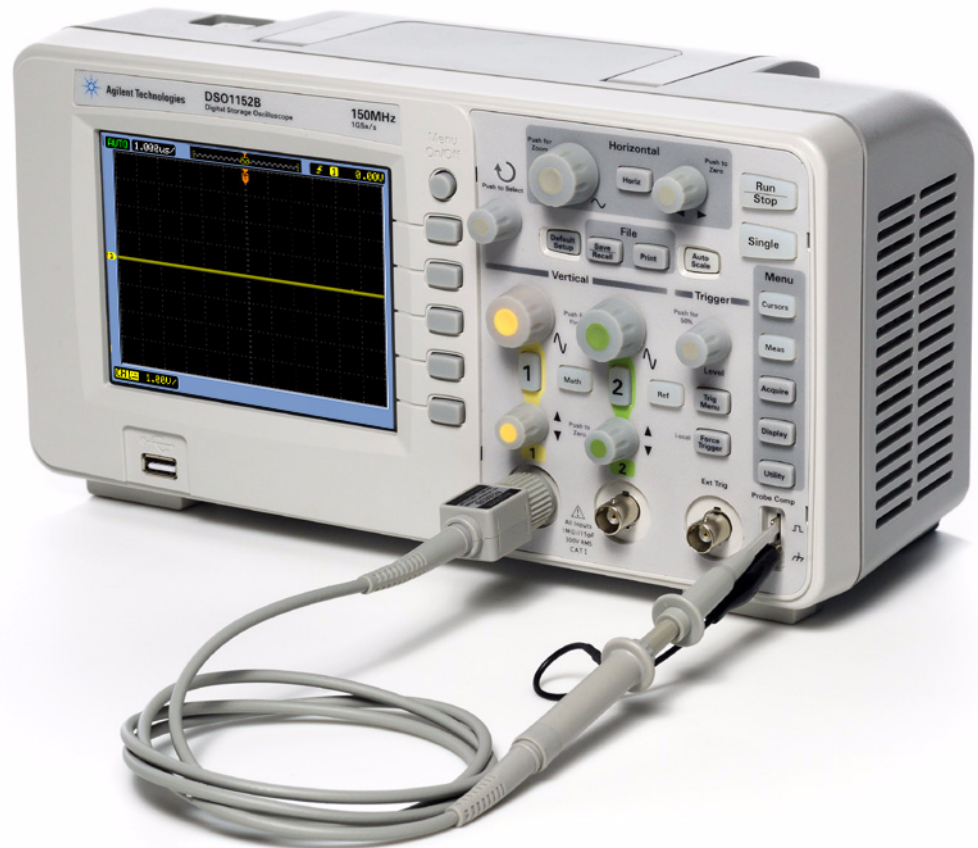
Figure 8   Connect a probe between the channel-1 input and the probe compensation terminal

There is always a 1 kHz square wave present on the "probe comp" terminal. Although the primary purpose of this signal is for calibrating/compensating the scope's probes − which we will do in a later lab − we will be using this signal for oscilloscope training purposes.

3 Press the **[Default Setup]** front panel key.

Default Setup will put the oscilloscope into a factory preset configuration. Not only will this set the scope's X and Y scaling factors to preset values, but it will also turn off any special modes of operation that one of your fellow students may have used. Performing a default setup will also set the scope's probe attenuation factors to 10X so that all amplitude measurements are referenced to the level of the signal present at the probe's tip. When beginning new measurements with the scope, it is always good practice to begin with a default setup. Let's now adjust the scope's vertical, horizontal, and trigger settings to properly scale and display the 1 kHz square wave on the scope's display.

4 Press the **[Menu On/Off]** front panel key (near upper right-hand corner of the display) to expand the scope's waveform viewing area.

5 Turn the **Trigger Level** knob clockwise until the trigger level setting indicates approximately **1.5 Volts** (readout near upper right-hand side of display), and the temporary orange horizontal trigger level line intersects with the middle of the waveform. Note that you will learn more about oscilloscope triggering in the next lab.

**6** Turn the large Horizontal knob (near top of the scope's front panel) counter-clockwise until you observe more than two periods of the square wave. The correct setting should be **200 us/** (readout near top left-hand side of display). This setting is an abbreviation for 200 ì sec/division. From this point forward we will simply refer to this as the scope's "timebase" setting.

7 Turn the Horizontal position knob (smaller knob near top of scope front panel) to move the waveform left and right. Press this knob to set it back to zero (0.0 seconds at center-screen).

8 Turn the channel-1 vertical position knob (smaller yellow knob in the vertical section of the front panel − just above the input BNC) until the top of the waveform is near the center of the display. The correct setting should be approximately **-3.0 V** (temporary readout near bottom left-hand side of display).

9 Turn the channel-1 V/div knob (larger yellow knob in the vertical section) clockwise until the readout in the bottom left-hand corner of the display says "**500mV/**". This means 500 mV/div. If you press the channel-1 V/div knob and then make adjustments, you will see that you will then have "fine" adjustment capability. Your professor may refer this as the scope's "vernier" adjust. Press the knob again to return to "coarse" adjustment, and then set it back to **500 mV/**.

Adjusting the scope's vertical position and V/div setting (step #8 and #9 above) is typically an iterative process; adjust one, and then the other, and then back again until the waveform is displayed properly.

Your scope's display should now look similar to Figure 9. Let's now make some measurements on this square wave. Notice that the oscilloscope's display is basically an X versus Y graph. On our X‑axis (horizontal) we can measure time, and on our Y‑axis (vertical) we can measure voltage. In many of your EE or Physics class assignments you have probably computed and graphed electrical signals in a similar, but static, format on paper. Or perhaps you have used various PC software applications to automatically graph your waveforms. When a repetitive input signal is applied to an oscilloscope, we are able to observe dynamic (continuously updated) graphing of our waveforms.
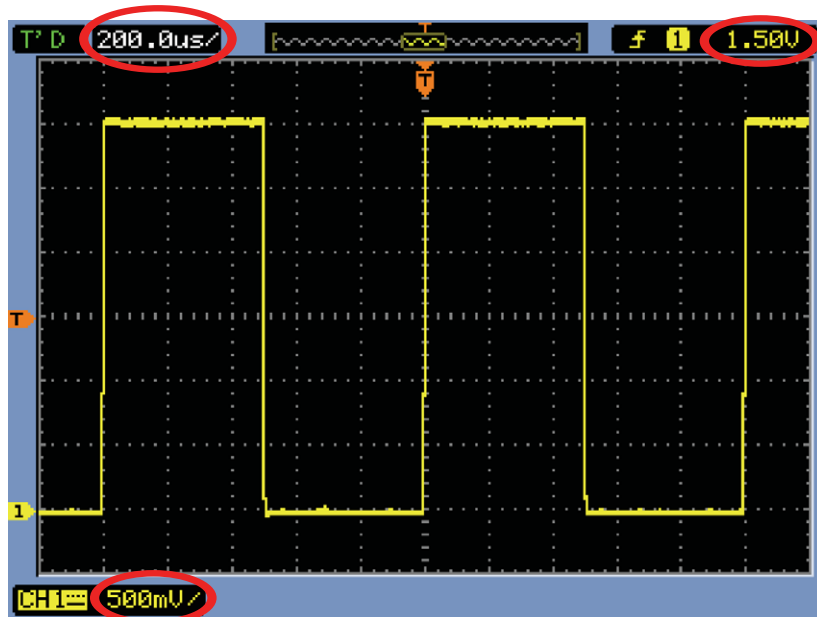
Figure 9 Setting up the scope's vertical, horizontal, and trigger level settings to display a waveform

Our X‑axis (horizontal) consists of 12 major divisions (if the menu display is off) across the screen with each major division being equal to the sec/div setting. In this case, each horizontal major division represents 200 microseconds of time; assuming that the scope's timebase is set to 200 ì s/div as instructed earlier. Since there are 12 divisions across the screen, then the scope is showing 2.4 ms of time (200 ì s/div x 12 divisions) from the left side of the display to the right side of the display. Notice that each major division is also divided into 5 minor divisions (0.2 divisions each), which are displayed as tick marks. Each minor division would then represent 1/5 div × 200 ì s/div = 40 ì s.

Our Y-axis (vertical) consists of 8 major divisions vertically with each major division being equal to the V/div setting, which should be set at 500 mV/div. At this setting the scope can measure signals as high as 4 Vp-p (500 mV/div x 8 divisions). Each major division is also divided into 5 minor divisions (0.2 divisions each). Each minor division, represented as tick marks, then represents 100 mV each.

10 Estimate the pulse width (PW) of one the positive pulses by counting the number of divisions (major and minor) from one rising edge to the next falling edge; then multiply by the s/div setting (should be 200 ì s/div).

PW   =   _____

11 Estimate the period (T) of one of these square waves by counting the number of divisions from one rising edge to the next rising edge; then multiple by the s/div setting.

T   =   _____

12 What is the frequency of this sine wave ($F = 1/T$).

F   =   _____

13 Determine the peak-to-peak voltage of this waveform by counting the number of divisions from the bottom of the waveform to the top of the waveform; then multiply by the V/div setting (should be 500 mV/div).

Vp-p   =   _____

Note that the yellow "1" label on the left side of the display indicates the ground level (0.0 V) of the channel-1 waveform. If channel-2 of this scope were also turned on, you would see a green "2" label indicating this channel's ground level (0.0 V). Let's now use the scope's "cursors" function to make these same voltage and timing measurements.

14 Press the **[Cursors]** front panel key; then press the **Mode** softkey. Rotate the **Entry** knob until **Manual** is highlighted; then push the **Entry** knob to select.

15 Press the **CurA---** softkey until this key is highlighted blue (note that it may already be highlighted blue).

16 Rotate the **Entry** knob to position the A timing cursor onto the 1st rising edge of the waveform near the left side of the display.

17 Press the **CurA---** softkey to disable this cursor, then press the **CurB---** softkey to enable the B timing cursor.

18 Rotate the **Entry** knob to position the B timing cursor onto the 2nd rising edge of the waveform at center-screen. Your scope's display should now look similar to Figure 10.
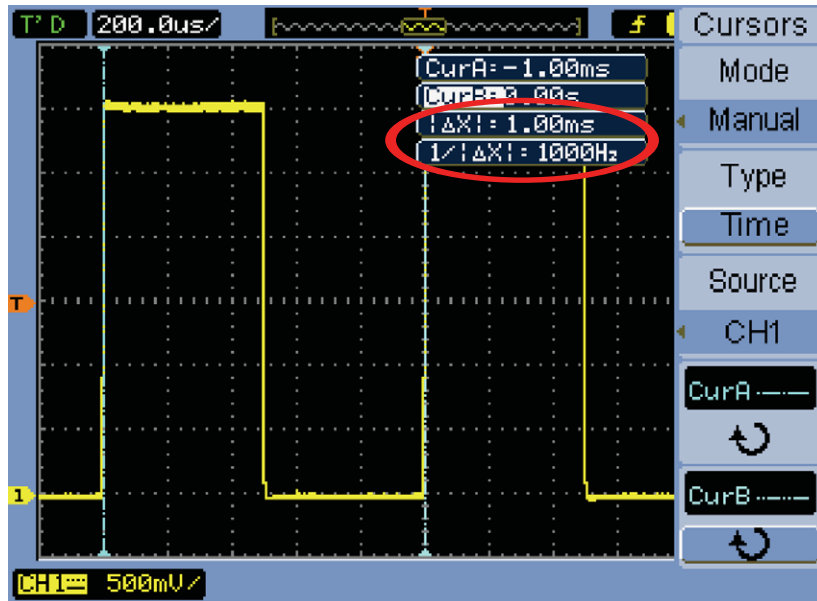
Figure 10 Using time cursors to measure the period and frequency of the square wave

19 What is the period and frequency of this waveform?

$$\Delta X \quad = \quad \underline{\hspace{3cm}}$$

$$1/\Delta X \quad = \quad \underline{\hspace{3cm}}$$

20 Press the **Type** softkey to change from **Time** cursors to **Amplitude** cursors.

21 Press the **CurB**--- softkey to disable this cursor, then press the **CurA**--- softkey to enable the A amplitude cursor.

22 Rotate the **Entry** knob to position the A amplitude cursor onto the bottom of the waveform.

23 Press the **CurA**--- softkey to disable this cursor, then press the **CurB**--- softkey to enable the B amplitude cursor.

24 Rotate the **Entry** knob to position the B amplitude cursor onto the top of the waveform. Your scope's display should now look similar to Figure 11.
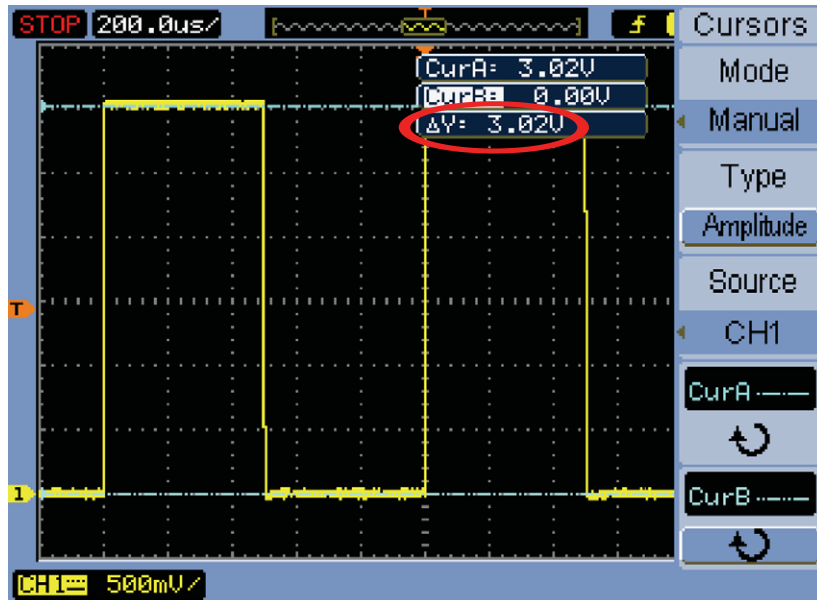
Figure 11   Using amplitude cursors to measurement the peak-to-peak voltage of the square wave

25 What is the top amplitude, bottom amplitude, and Vpp amplitude of this waveform?

Top (CurA) = _____

Bottom (CurB) = _____

Vpp (ΔY) = _____

Using cursors should provide a slightly more accurate measurement and take the guess work out of the measurement. Let's use an even easier and more accurate method to perform these measurements.

26 Press the **[Meas]** front panel key. Note that this front panel key is labeled **[Measure]** on 4-channel models of the Agilent DSO1000.

27 If the top softkey is labeled **2/2** (menu 2 of 2); press this softkey to toggle to **1/2** (menu 1 of 2).

28 Press the **Voltage** softkey; then rotate the **Entry** knob to highlight **Vpp**, and then push the **Entry** knob to select.

29 Press the **Time** softkey; then rotate the **Entry** knob to highlight **Freq**, and then push the **Entry** knob to select.

Your scope's display should now look similar to Figure 12 showing the automatic Vpp and Frequency measurements near the bottom of the display.
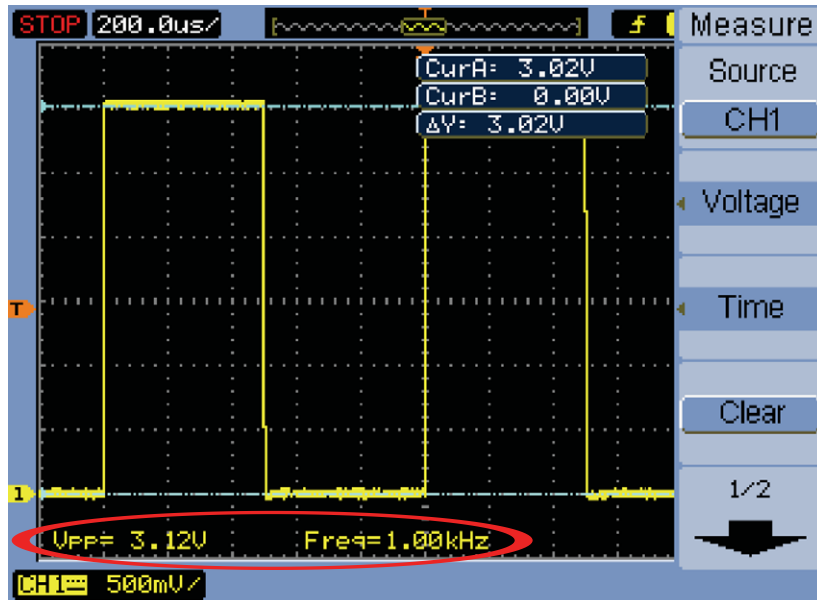
Figure 12   Using the scope's automatic parametric measurements

The most common method used to measure time and voltage on an oscilloscope is usually the "division counting" method we used first. Although divisions must be counted and then multiplied by scope settings, engineers that are familiar with their scopes can quickly estimate of the voltage and timing parameters of their signals... and sometimes a rough estimation is all that is required to know if a signal is valid or not.

Before we wrap-up this first lab, let's try one more interesting measurement.

30 Press the **1/2** softkey, which will take us to the 2nd page of this measurement menu.

31 Press the **Display All** softkey to toggle from **OFF** to **ON**.

Your scope's display should now look similar to Figure 13; showing a comprehensive list of multiple parametric measurements on our 1 kHz square wave.
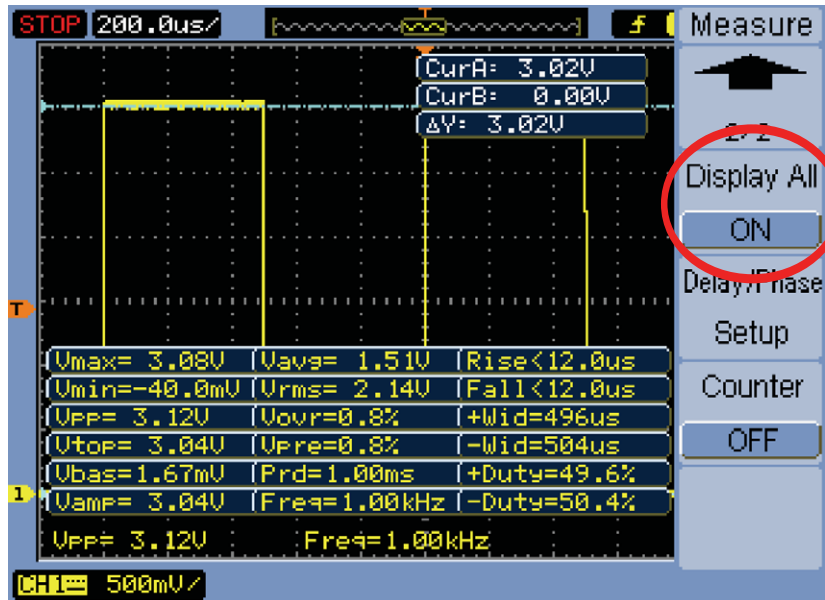
Figure 13   Performing "all" voltage and timing measurements on the square wave

One of the first steps in this oscilloscope familiarization lab was to set the scope's trigger level. But we haven't yet discussed what oscilloscope triggering is all about. Let's now learn more about oscilloscope triggering in this next hands-on lab.

Lab #2: Learning the Basics of Oscilloscope Triggering

As mentioned earlier, oscilloscope triggering is probably the most important capability of an oscilloscope that you should understand if you want to get the most out of your oscilloscope measurements. This is especially important when attempting to make measurements on many of today's more complex digital signals. Unfortunately, oscilloscope triggering is often the least understood aspect of oscilloscope operation.

You can think of oscilloscope "triggering" as "synchronized picture taking". When an oscilloscope is capturing and displaying a repetitive input signal, it may be taking thousands of pictures per second of the input signal. In order to view these waveforms (or pictures), the picture-taking must be synchronized to "something". That "something" is a unique point in time on the input signal.

An analogous situation to oscilloscope triggering is that of a photo finish of a horse race. Although not a repetitive event, the camera's shutter must be synchronized to the first horse's nose at the instant in time that it crosses the finish line. Randomly taking photos of the horse race sometime between the start and end of the race would be analogous to viewing untriggered waveforms on a scope.

To better understand oscilloscope triggering, let's make some more measurements on our familiar square wave that we used in Lab #1.

1 Ensure that your oscilloscope probe is still connected between the terminal labeled "**Probe Comp**" and the channel-1 input BNC.

2 Press **[Default Setup]** on the scope's front panel.

3 Press the **[Menu On/Off]** to turn off the menu and to expand the waveform viewing area.

4 Turn the trigger level knob clockwise in order to set the trigger level near the middle of the waveform (approximately **1.5 V**).

5 Adjust the channel-1 vertical position knob to center the waveform in the scope's display area.

You scope's display should now look similar to Figure 14. At this point we are viewing just a narrow portion of this square wave around its rising edge since the timebase is set to 1.0 ì s/div. Remember, the period of this signal is approximately 1.0 ms. Notice the "T" symbol near the top of the display. This marks the point in time where the scope is triggering on this waveform. Waveform data captured before the trigger point (left side of display) is considered negative time data, while waveform data captured after the trigger point (right side of display) is considered positive time data. Now take note of the "T" symbol near the left side of the display. This marks the voltage level where the scope is triggering on this waveform. Where the two "T" symbol points intersect on a rising edge of the waveform is the approximate trigger point — or synchronization point.
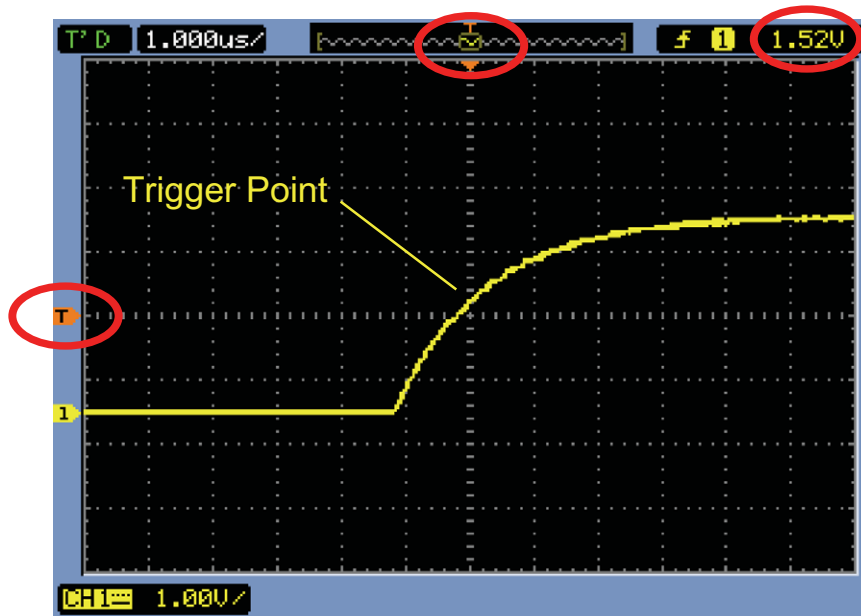
Figure 14   Triggering on a rising edge

6 Rotate the trigger level knob clockwise to increase the trigger level setting to be closer to the top of the waveform. You should notice the waveform slide to the left.

7 Rotate the trigger level knob counter-clockwise to decrease the trigger setting to be closer to the bottom of the waveform. You should notice the waveform slide to the right.

8 Set the trigger level back to approximately **1.5 V**, which will be near the middle of the waveform.

By default, the scope automatically selects to trigger on rising edges. Let's now setup the scope to trigger on a falling edge of this 1 kHz square wave.

9 Press the **[Trig Menu]** front panel key. Note that this front panel key is labeled **[Menu]** in the Trigger section on 4-channel models of the DSO1000.

10 Press the **Slope** softkey; then rotate the **Entry** knob to highlight the falling edge symbol.

Your scope's display should now be synchronized to and be displaying a falling edge of this waveform similar to Figure 15. Let's now learn about "Sweep" modes.
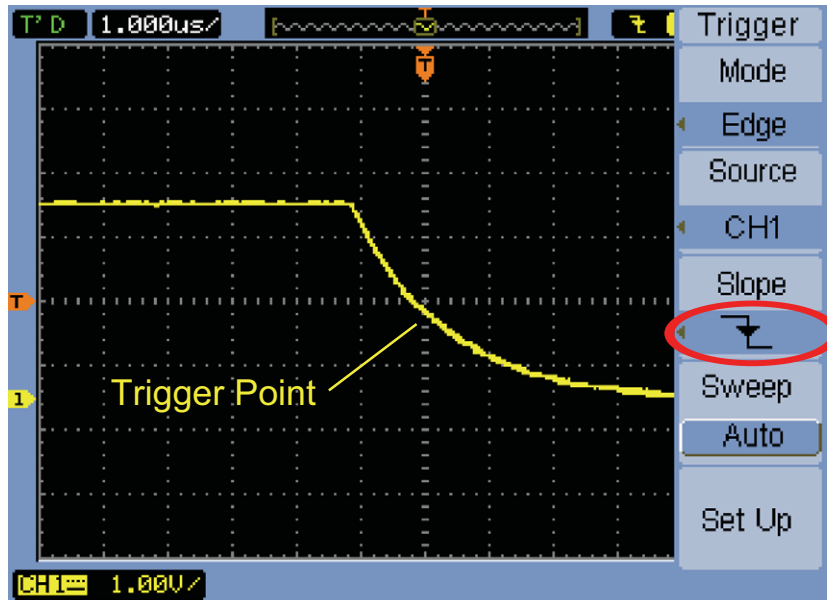
Figure 15   Triggering on a falling edge

11 Turn the large horizontal knob counterclockwise to set the timebase at **200 µs/div**. You should now observe a few periods of this waveform with a falling edge of the square wave still synchronized to the center-screen point.

12 Turn the trigger level knob clockwise until the trigger level indicator is above the waveform.

Since the trigger level is set above the waveform, the scope has nothing to synchronize its picture-taking on as shown in Figure 16. The scope is now "Auto" triggering and you should see a flashing green "AUTO" message near the top left-hand corner of the display. This means that the scope is generating automatic triggers, but these automatic triggers are not synchronized to the input signal. Note that you would observe the same thing if the trigger level were set below the waveform.
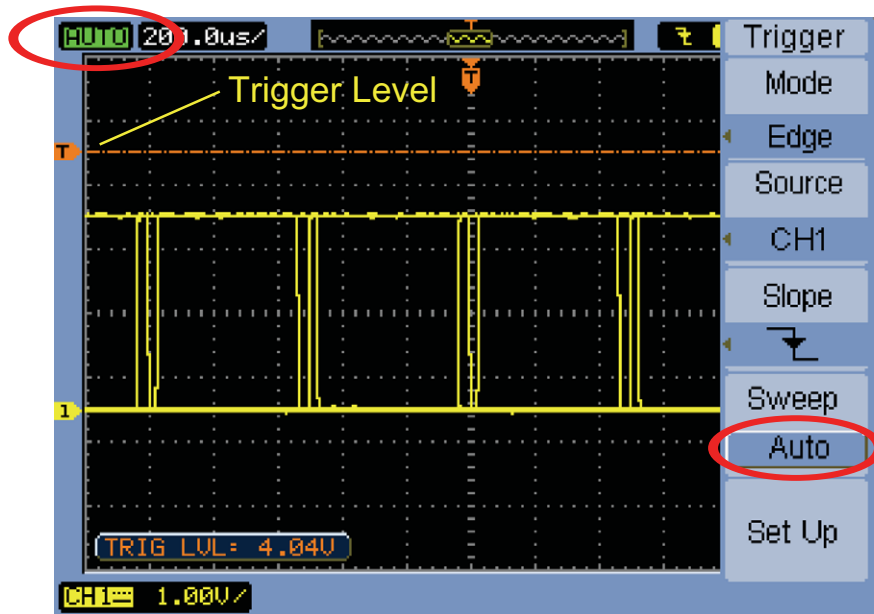
Figure 16   Auto triggering with the trigger level set above the input signal

Auto is the scope's default sweep mode (or trigger mode). When the scope is using the Auto sweep mode, if the scope doesn't find a valid trigger condition (falling edge crossing of the 1 kHz square wave in this case), then the scope will generate its own asynchronous trigger and begin taking pictures (acquisitions) of the input signal at random times. Since the "picture taking" is now random, rather than synchronized to the input signal, all we see is a "blur" or random display of waveforms across the screen. This random display of waveforms should be a clue to us that our scope is not triggering on the input signal.

Also note that the word "sweep" is a carry-over term from the old analog scope days when older technology scopes — like the ones your professor may have used in his/her university days — would "sweep" or deflect an electron beam across a vector cathode ray tube (CRT). Digital Storage Oscilloscopes (DSOs) no longer "sweep", but digitize input signals using an analog-to-digital converter (ADC), and then display the digitized points as a bit map on the scope's flat-panel display. But the term "sweep" is still in broad use today. However, be aware that some of today's scopes also call this mode of operation the "trigger mode".

13 Press the trigger level knob to automatically set the trigger level to the approximate **50%** level.

14 Disconnect the channel-1 probe from the "**Probe Comp**" terminal.

With the channel-1 probe disconnected from our signal source, we should now see a baseline 0.0 V DC signal. Since with this 0.0 V DC signal we no longer have any edge crossings, and hence the scope has nothing to trigger

on; the scope again "auto triggers" in order to show us this DC level signal.

In addition to the default Auto sweep mode, the oscilloscope also has another user-selectable sweep mode called the Normal sweep mode. Let's now see how the Normal sweep mode differs from the Auto sweep mode.

15 Re-connect the channel-1 probe to the "**Probe Comp**" terminal. You should see the triggered square wave again.

16 Press the **Sweep** softkey to toggle from the **Auto** sweep mode to the **Normal** sweep mode.

17 Disconnect the channel-1 probe from the "**Probe Comp**" terminal again.

You should now see the last acquisition (last picture) that occurred before the probe was disconnected, or perhaps you might see a transient. You should also see a flashing "Wait" message in the upper left-hand corner of the display. This is an indication that the scope is waiting for valid trigger events. Note that we do not see the 0.0 V DC level trace that the Auto sweep mode showed when the probe was disconnected. When the Normal sweep mode is selected, the scope will only display waveforms if and only if the scope detects valid trigger conditions (falling edge crossings in this case).

18 Rotate the trigger knob clockwise in order to set the trigger level at approximately **+4.00 V** (which will be above our square wave — if we had the probe connected).

19 Re-connect the channel-1 probe to the "**Probe Comp**" terminal.

The square wave is now connected and being input to the scope, but where is our repetitive display of this signal? Since we are using the Normal sweep mode, the scope still requires valid edge crossings, but since the trigger level is set above the waveform (@ +4.00 V), there are no valid edge crossings. So as you can see with the Normal sweep mode, we don't have any clue as to where our waveform is, and we can't measure DC either.

20 Press the trigger level knob to automatically set the trigger level to the approximate **50%** level. Your scope should begin to show repetitive waveforms again.

Some older scopes use to refer to what we today call the Normal sweep mode as the Triggered sweep mode, which may actually be a more descriptive term for this triggering mode since in this mode the scope only triggers when it finds a valid trigger condition, and will not generate an automatic trigger (asynchronous trigger to generate asynchronous picture taking). And it is a bit of an oxymoron that the Normal sweep mode is not the "normally" used sweep mode, and it is not the scope's default sweep mode. The trigger mode that is normally used is the auto sweep mode, which is the scope's default trigger mode.

As this point you may be wondering when to use the Normal sweep mode. The Normal sweep mode should be used when the trigger event occurs very infrequently (including single-shot events). For example, if you had the scope setup to capture and display a signal that occurs at just a ½ Hz rate (one cycle every 2 seconds), if the scope's trigger mode was set to the Auto sweep mode, then the scope would generate lots of asynchronously generated automatic triggers and would not be able to show this slowly changing signal. In this case you would need to select the Normal sweep mode so that the scope would wait until obtaining valid trigger events before displaying waveforms. If you have a function generator in your lab, you can try setting the frequency to 0.5 Hz and then observe the difference between using the scope's Auto sweep mode and its Normal sweep mode.

Lab #3: Capturing Single-Shot Events

Up until now we have been capturing repetitive waveforms — in other words; signals that repeat. In the case of our 1 kHz square wave, this signal repeats 1,000 times a second. But sometimes engineers need to capture events (signals) that are single-shot, which means that they only occur once. Digital Storage Oscilloscopes (DSOs) are really good at capturing single-shot events because of their very fast sample rate. However, setting up the scope to capture a single-shot event is a bit more complicated than setting up a scope to capture a repeating waveform. You can no longer "tune" setup conditions such as V/div and sec/div while viewing the waveform. The signal is not there to do this with. You must have some knowledge of the approximate characteristics of the single-shot event — such as amplitude and event width — before setting up the scope to capture it. So, let's assume that you know that the single-shot event has a peak-to-peak amplitude of approximately 2 Vpp and an event width of approximately 10 milliseconds. Let's now setup the scope to capture a single-shot event with these characteristics.

1 Press the **[Default Setup]** front panel key.

2 Disconnect the channel-1 probe (probe tip grabber and ground clip) from the terminal labeled "**Probe Comp**".

3 Set the scope's timebase to **1.000 ms/div**.

4 Set the channel-1 V/div setting to **200 mV/div**.

5 Set the trigger level to approximately **300 mV**.

6 Press the **[Trig]** front panel. Note that this front panel key is labeled **[Menu]** in the Trigger section of 4-channel models.

7 Press the **Sweep** softkey to toggle from the Auto sweep mode to the Normal sweep mode.

8 Press the **[Menu On/Off]** front panel key.

9 Press the **[Single]** front panel key.

10 Pick up your probe and lightly tap it on your bench or table to create a capture of a small electrostatic discharge (ESD).

You may now see a stored waveform similar to Figure 17 on your scope's display. But it's likely to look a bit different. If your scope didn't capture anything, try reducing the trigger level, press the **[Single]** front panel again, and then tap your probe on the your bench again. To capture multiple single-shot events, press **[Single]** again before the event occurs the next time.
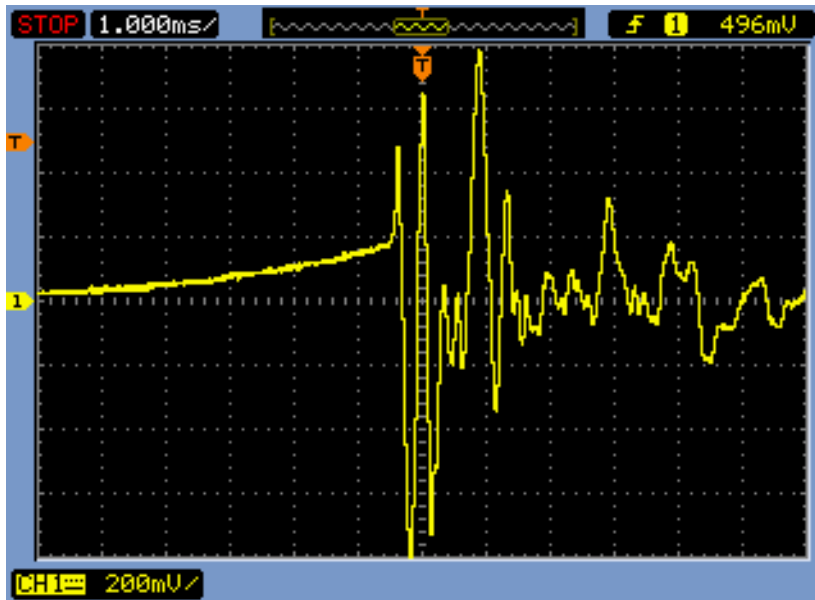
Figure 17   Setting up the scope to capture a single-shot event

To get a better understanding of the use of the Normal sweep mode — which we just covered during the previous lab — let's try capturing multiple infrequent events.

11 Press the **[Run/Stop]** front panel key (key should light up green).

12 Begin tapping your probe on your bench/table at a slow rate (perhaps once every second).

You should notice that the scope captures each single-shot event. Let's now change the sweep mode to Auto and see what happens.

13 Press the **[Trig Menu]** front panel key. Note that this front panel key is labeled **[Menu]** in the Trigger section of 4-channel models.

14 Press the **Sweep** softkey to toggle from **Normal** to **Auto**.

15 Begin tapping your probe on your bench/table again at a slow rate.

You should notice that the scope rarely captures the single-shot event and when it does, it appears randomly somewhere on the scope's display —not pegged to the center-screen trigger location. This is because the scope is Auto triggering. Remember, Auto sweep (trigger) works great when capturing repetitive waveforms, but it should not be used to capture either low repetition rate or single-shot events.

Lab #4: Compensating your Passive 10:1 Probes

Now that you've completed the first three labs in this oscilloscope training guide and should be somewhat familiar with how to use an oscilloscope to make basic voltage and timing measurements, let's take a step back and talk about probing again. In the Getting Started section of this guide we briefly discussed probing and showed the electrical input model of the combination of a passive 10:1 probe and the oscilloscope's input. This electrical model of the probe and oscilloscope is shown again here in Figure 18.
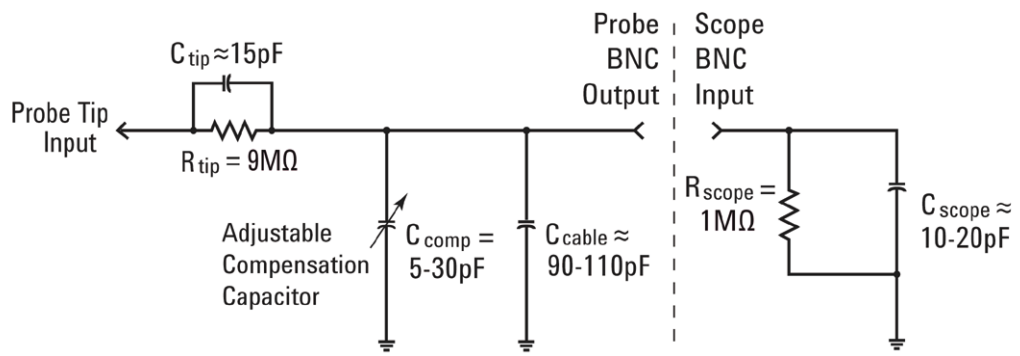

Figure 18 Simplified schematic of a passive 10:1 probe connected to the scope's 1 MΩ input impedance

If you remember, you were instructed to ignore the capacitive components in this electrical model and consider just the resistive components. When we looked at just the resistive components, we determined that the combination of the probe's 9 MΩ probe tip resistor plus the scope's 1 MΩ input impedance established a 10-to-1 voltage-divider ratio. For low-frequency or dc applications, ignoring the capacitive elements is appropriate. But if you need to measure dynamic signals, which is the primary measurement application for oscilloscopes, the capacitive elements of this electrical model can't be ignored.

Inherent in all oscilloscope probes and scope inputs are parasitic capacitances. These include the probe cable capacitance (C_{cable}), as well as the scope's input capacitance (C_{scope}). "Inherent/parasitic" simply means that these elements of the electrical model are not intentionally designed-in; but are just an unfortunate fact of life in the real world of electronics. And the amount of inherent/parasitic capacitance will vary from scope-to-scope and probe-to-probe. But without additional designed-in capacitive components to compensate for the inherent capacitive elements in the system, the reactance of the system under dynamic signal conditions (non-dc) can change the overall dynamic

attenuation of the probing system to something different than the desired 10:1 ratio. The purpose of the additional/designed-in probe tip capacitor (C $_{tip}$) along the adjustable compensation capacitor (C $_{comp}$) is to establish a capacitive reactance attenuation that matches the resistive attenuation of 10:1. When the compensation capacitor is properly adjusted, this also ensures that the time constant of the probe tip capacitance in parallel with the 9 MΩ resistor matches the time constant of the inherent and compensation capacitances in parallel with the scope's 1 MΩ input resistor.

Rather than spending a lot more time talking about the theory of this, let's just connect to a signal and see the affect of under-compensation, over-compensation, and proper-compensation.

1 Connect one oscilloscope probe between the channel-1 input BNC and the terminal labeled **Probe Comp**. Connect the probe's ground clip to the lower terminal (ground).

2 Connect another oscilloscope probe between the channel-2 input BNC and the terminal labeled **Probe Comp**. Connect the probe's ground clip to the lower terminal (ground).

3 Press **[Default Setup]** on the scope's front panel.

4 Press the **[2]** front panel key (between the two green knobs) to turn on channel-2.

5 Press the **[Menu On/Off]** front panel key to expand the waveform viewing area.

6 Set the horizontal timebase to **200.0 µs/div**.

7 Press the trigger level knob to automatically set the trigger level to approximately 50%.

8 Adjust the channel-2 vertical position (smaller green knob) in order to position the channel-2 waveform (green trace) below the channel-1 waveform (yellow).

If your probes are properly compensated, you should see two 1 kHz square waves with a very flat response on your scope's display similar to Figure 19. Let's now adjust the probe compensation on each probe.
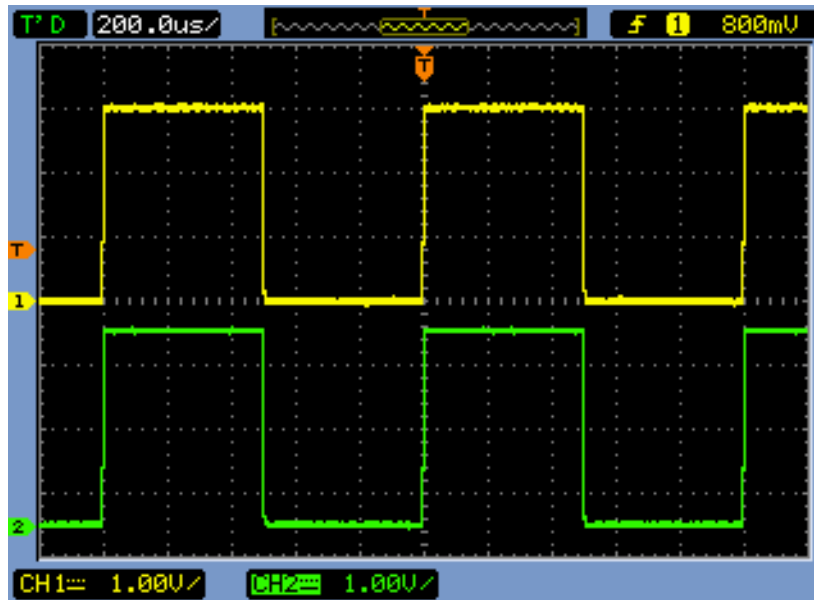
Figure 19 Using the scope's 1-kHz probe compensation signal to compensate 10:1 passive probes

9 Using a small slotted screw driver, adjust the variable capacitor located on the body of each probe. Note that this adjustment is sometimes located near the BNC connection end of some probes.

Figure 20 shows an example of the channel-1 probe (yellow waveform) over-compensated (too much capacitance), and an example of the channel-2 probe (green waveform) under-compensated (too little capacitance). If you don't observe a near-perfect square wave, then re-adjust the probe compensation on your probes until the waveforms on your scope are similar to Figure 19.
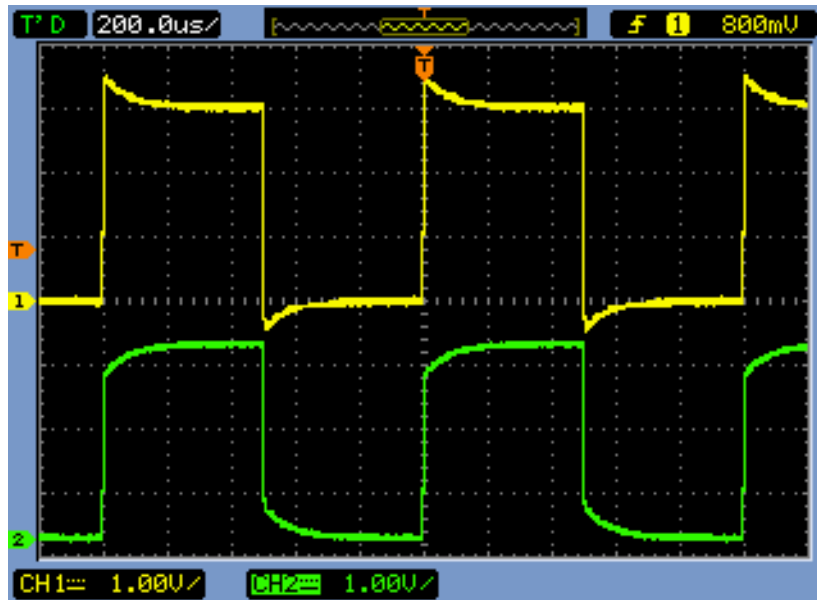
Figure 20   Improperly compensated probes

After your probes have been properly adjusted, so long as you continue to use these probes on this scope, you shouldn't need to re-adjust them the next time you use the scope.

At this point you have completed the hands-on portion of this lab. The next part of this lab is a detailed discussion of the theory behind probe compensation and probe loading. Your professor may assign this as your homework. If so, you may want move on to the next lab and then study this section later.

Computing the Proper Amount of Capacitive Compensation

If you are up for a challenge, compute the amount of compensation capacitance (C_{comp}) required for proper compensation using the following assumptions:

R_{tip} = 9 MΩ

R_{scope} = 1 MΩ

C_{scope} = 15 pF

C_{cable} = 100 pF

C_{tip} = 15 pF

$C_{parallel}$ = C_{scope} + C_{cable} + C_{comp}

C_{comp} = ?

To compute the required amount of compensation capacitance (C_{comp}), the easiest method is to equate the time constant (1/RC) of the parallel combination of R_{tip} and C_{tip} with the time constant of the parallel combination of R_{scope} and $C_{parallel}$:

$$\frac{1}{R_{tip} \times C_{tip}} = \frac{1}{R_{scope} \times C_{parallel}}$$

Remember that $C_{parallel}$ is the combination of three capacitive elements in the probe/scope model.

Another computational method would be to equate 9X the capacitive reactance of $C_{parallel}$ with 1X the capacitance reactance of C_{tip}. This will establish the same attenuation factor contributed by capacitive reactances as the attenuation factor contributed by the resistive-only network (10:1):

$$\frac{1}{2\pi f C_{tip}} = 9 \times \frac{1}{2\pi f C_{parallel}}$$

$$C_{comp} = \underline{\quad\quad\quad}$$

Probe Loading

Besides properly compensating your 10:1 passive probes in order to achieve the most accurate oscilloscope measurements, another issue that must be considered is probe loading. In other words, will connecting the probe and scope to your device-under-test (DUT) change your circuit's behavior? When you connect any instrument to your circuit, the instrument itself becomes a part of your DUT and can "load" or change the behavior of your signals to some degree. If we use the given values of resistances and capacitances listed above (along with the value of C_{comp} that you computed), we can model the loading affect of the probe and scope together as the parallel combination of a single resistor and capacitor as shown in Figure 21.
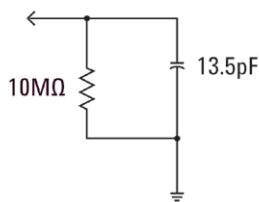

10MΩ   13.5pF

Figure 21   10:1 passive probe and scope loading model

For low frequency or dc applications, loading is dominated by the 10 MΩ resistance, which in most cases should not be a problem. But what if you are probing a 100 MHz digital clock signal? The 5th harmonic of this

digital clock, which is a significant component in creating the shape of this signal, would be 500 MHz. Now compute the reactance contributed by the 13.5 pF capacitance of this loading model shown in Figure 21:

$$X_c = \frac{1}{2\pi fC} = \frac{1}{2\pi \times 500 \times 10^6 \times 13.5 \times 10^{-12}} = 23.6\Omega$$

Although 13.5 pF may not sound like much, at higher frequencies this amount of loading capacitance can be significant. For higher frequency applications such as this, most scope vendors provide optional active probing solutions that have significantly lower input capacitances (sub pF). But these types of special probes cost significantly more than the typical 10:1 passive probe.

Lastly, be aware that the probe + scope models presented in this lab are very simplified. More accurate models would include inductive elements as well. Wire, especially your ground lead, should be considered as an inductive element, especially for high frequency applications.

To conduct your own probe loading experiment, download the application note titled, *Oscilloscope Probe Loading Experiment* listed in the "Related Agilent Literature" section of this document.

Lab #5: Documenting and Saving Oscilloscope Test Results

When you complete your various circuits lab assignments, your professor may require you to write up a test report. Including images (pictures) of your measurements in your lab report may be required. In addition, if you can't complete a lab assignment during one session, you may want to continue testing at a later time. But it would really be great if you could pick up where you left off; without having to re-setup your scope or possibly re-acquire waveforms. In this lab you will learn how to save and recall various types of oscilloscope files including image, reference waveform, and setup files. For this lab you must have access to a personal USB memory device.

1 Ensure that your oscilloscope probe is still connected between the terminal labeled **Probe Comp** and the channel-1 input BNC.

2 Press **[Default Setup]** on the scope's front panel.

3 Press the **[Menu On/Off]** front panel key to expand the waveform viewing area.

4 Set the scope's timebase to **200 µs/div**.

5 Press the trigger level knob to establish triggering at the approximate **50%** level.

At this point you should see a few cycles of the 1 kHz square wave as shown in Figure 22. Let's now save this image (picture).
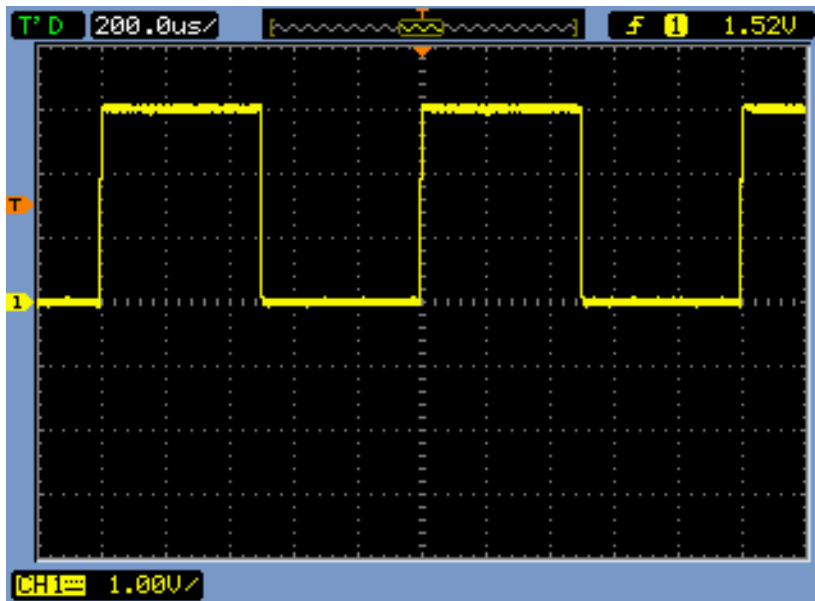

Figure 22   Two cycles of a 1 kHz square wave that we want to save for documentation and later analysis

6 Insert your personal USB memory device into the scope's front panel USB port.

7 Press the **[Save/Recall]** key in the File section of the front panel.

8 Press the **Storage** softkey; then rotate the **Entry** knob to highlight **PNG**, and then push the **Entry** knob to select this as the type of storage operation/file type that you want to save or recall.

9 Press the **External** softkey, which means that you want to either save or recall data to/from your external USB drive. Note that although you can also save certain types of data to the scope's internal memory, these limited memory locations can be overwritten by your fellow students.

10 Press the **New File** softkey to create a new file name. Although you can create a customized file name, let's just use the default name for now.

11 Press the **Save** softkey to save this image file.

This saves the image (picture) that was on the display of your scope prior to entering this menu. Although you cannot recall this type of file back into your scope, you can open a .png file on your PC, and you can insert this image into various types of documentation such as Microsoft Word. This is something that your professor will probably soon require in order to document your various lab experiments. In addition to saving images in a .png format, you can also save images as 8-bit or 24-bit .bmp images. Note that all of the oscilloscope screen images shown in this *Lab Guide* were originally saved in a .png format. Let's now save a setup file.

12 Press the **[Menu On/Off]** front panel key.

13 If you are using a 2-channel model, press the **[Save/Recall]** front panel key. If you are using a 4-channel model, the Save/Recall menu will already be displayed.

14 Press the **Storage** softkey; then rotate the **Entry** knob to highlight **Setups**, and then push the **Entry** knob to select.

15 Press the **External** softkey.

16 Press **New File** softkey; then press **Save**.

With a "setup" file, you can save the scope's setup conditions (V/div, sec/div, trigger level , etc.) so that you can later recall these same setup conditions to continue making measurement where you left off possible the previous day. Let's now try recalling the setup condition you just saved. But first, we'll destroy the scope's current setup condition.

17 Press the **[Menu On/Off]** front panel key.

18 Press the **[Default Setup]** front panel key. This destroy's the scope's current setup condition.

19 Press the **[Save/Recall]** front panel key.

20 Press the **Storage** softkey; then rotate the **Entry** knob to highlight **Setup**, and then push the **Entry** knob to select.

21 Press the **External** softkey.

22 Rotate the **Entry** knob to highlight the setup file that you just saved. Note that this file will have a .stp extension. If this is the only .stp file stored on your USB drive, then it will be highlighted automatically.

23 Press the **Recall** softkey to recall this setup.

If your probe in still connected to the **Probe Comp** terminal, your scope should be correctly setup to display the 1 kHz square wave again. Note that you can also save setups using the Waveform storage type. If saved and then recalled as a "Waveform", then the scope will recall the setup condition as well as the waveform(s). But if you then press the **[Run]** key after recalling setups and waveforms, the recalled waveforms will be replaced with actively digitized waveforms. There is a better way to permanently save waveforms for later analysis using "reference" waveforms. Let's try it.

24 Press the **[Ref]** front panel key (next to the channel-2 on/off key).

25 Press the **Location** softkey to toggle from **Internal** to **External** memory.

26 Press the **Save** softkey.

27 Press the **New File** softkey; then press **Save**.

Let's now import (recall) this reference waveform that you just saved back into the scope for post analysis. But first, let's clear the currently displayed channel-1 waveform (yellow trace).

28 Press the **[Menu On/Off]** front panel key.

29 Press the **[Display]** front panel key.

30 If the top selection says **2/2** (menu 2 of 2), then press this softkey to toggle to **1/2** (menu 1 of 2).

31 Press the **Clear** softkey to clear the channel-1 waveform.

32 Press the **[Ref]** front panel key.

33 Press the **Location** softkey until it indicates **External**.

34 Press the **Import** softkey.

35 Rotate the **Entry** knob to highlight the reference waveform file that you just saved. Note that this file will have a .ref extension.

36 Press **Import**.

You should now see a white waveform on the scope's display similar to Figure 23. You can now use this waveform to compare it against actively digitized waveforms.
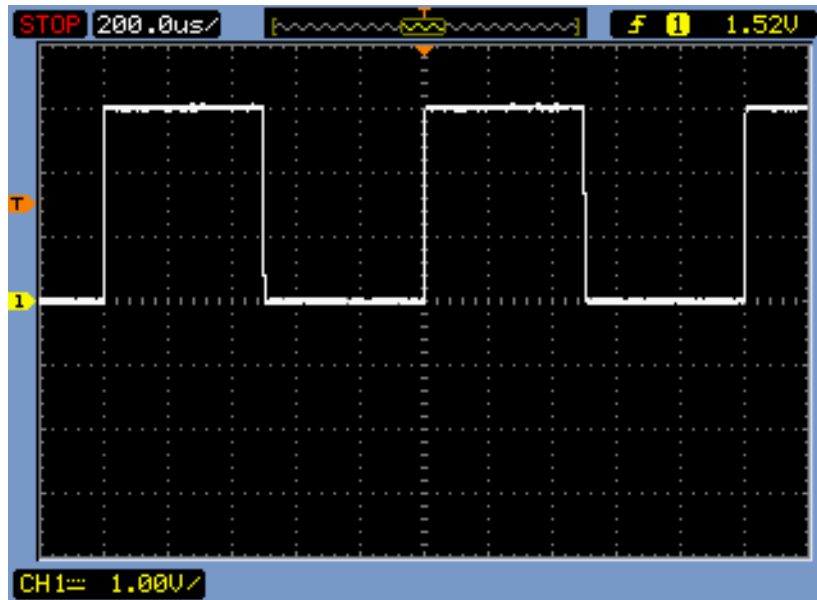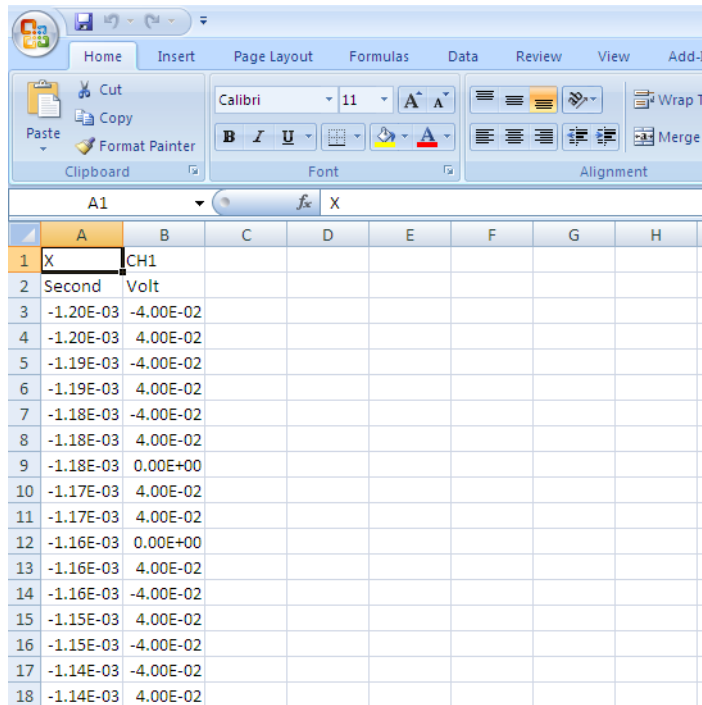
Figure 23   Importing a stored reference waveform (.ref) into the scope's internal reference memory

In addition to saving images (.png or .bmp), setups (.stp), and reference waveforms (.ref), you can also save waveforms in a .csv (comma separated values) format. This will consist of an array of XY pairs (time and voltage) that represents the individual digitize points in the waveform. Although you cannot recall this type of file back into the scope, you can open this type of file in a spreadsheet application such as Microsoft Excel. You can also import this type of data into various waveform/data analysis software packages such as LabView and MatLab for more advanced waveform analysis that may not be available on the scope.

37 Press the **[Ref]** front panel turn off the display of the reference waveform (Press twice on 2-channel models. Press once on 4-channel models.)

38 Press the **[Run/Stop]** key to begin acquisitions again (**[Run/Stop]** should turn green).

39 Press the **[Save/Recall]** front panel key.

40 Press the **Storage** softkey; then rotate the **Entry** knob to highlight **CSV**, and then push the **Entry** knob to select.

41 Press the **External** softkey.

42 Press the **New File** softkey.

43 Press the **Save** softkey.

44 Press the **[Menu On/Off]** front panel key.

If you open this file on your laptop using Microsoft Excel, you should see a list of time and voltage pairs of data similar to Figure 24. Although Excel provides limited analysis on waveform data, you can also open this file in applications such as LabView or MatLab for extensive and advanced waveform analysis.

Figure 24   Opening a saved waveform in .csv format using Microsoft Excel

Lab #6: Using Oscilloscope Waveform Math

Oscilloscopes can perform math operations on an entire waveform or pair waveforms. One very common waveform math function that you may want the scope to perform is to subtract one waveform from another. Figure 25 shows an example of a simple two-resistor voltage divider network. What if you wanted to see what a waveform looks like across just R1? With standard passive 10:1 probes, such as the ones that you are using today, you can measure Vin relative to ground using one channel of the scope, and also measure Vout relative ground using another channel of the scope. But you cannot probe across R1 since neither end of R1 is relative to ground. One option would be to use a special and expensive differential active probe. Another option would be to use the oscilloscope's "A-B" waveform math function to create a special math waveform that is the difference between the channel-1 and channel-2 waveforms. Your professor/lab instructor may assign such an experiment in the future using an external function generator along with passive and/or active components. But for now, let's use the scope's built-in probe comp signal to learn how to use waveform math functions on the scope.
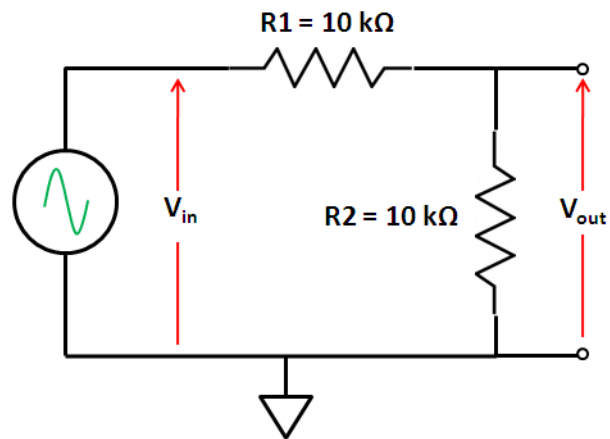
Figure 25   A 2-resistor voltage divider network

1   Connect one oscilloscope probe between the channel-1 input BNC and the terminal labeled "**Probe Comp**". Connect the probe's ground clip to the lower terminal (ground).

2   Connect another oscilloscope probe between the channel-2 input BNC and the terminal labeled "**Probe Comp**". Connect the probe's ground clip to the lower terminal (ground).

3   Press **[Default Setup]** on the scope's front panel.

4   Press the **[2]** front panel key (between the two green knobs) to turn on channel-2.

5 Set the horizontal timebase to **200.0 µs/div**.

6 Press the trigger level knob to automatically set the trigger level to approximately **50%**.

7 Adjust the channel-2 vertical position (smaller green knob) to position the channel-2 waveform (green trace) into the lower half of the display. Make sure that the top of the green waveform is slightly below centerscreen.

8 Adjust the channel-1 vertical position (smaller yellow knob) to re-position the channel-1 waveform such the bottom of the channel-1 waveform is slightly above center-screen.

9 Press the **[Math]** front panel key (between the channel-1 and channel-2 on/off keys).

10 If the top softkey selection says **2/2** (menu 2 of 2), then press this softkey to toggle to **1/2** (menu 1 of 2).

11 Press the **Operator** softkey; then rotate the **Entry** knob to highlight **A-B**, and then push the **Entry** knob to select.

Your scope's display should now look similar to Figure 26. If your probes have been properly compensated, the Math waveform (purple trace), which is the difference between channel-1 and channel-2, should show a 0 V "flat-line" trace. Since both channels of the scope are capturing the same signal, the result should be zero.
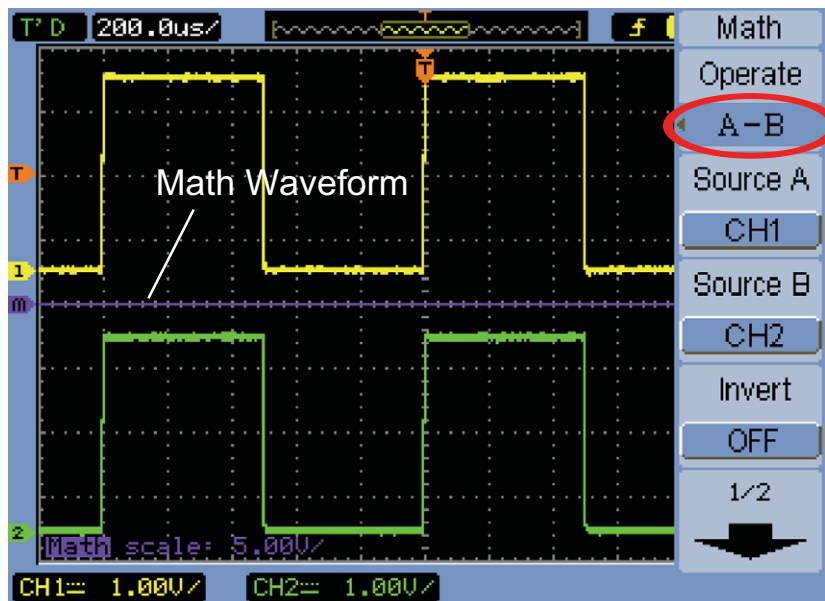

Figure 26   Using waveform math to subtract channel-1 from channel-2

To make this differential measurement a little more interesting, try mis-adjusting the probe compensation on just one of the probes (either channel-1 probe or channel-2 probe). This will create a difference between these two input signals. Your scope's display should now look something like Figure 27.
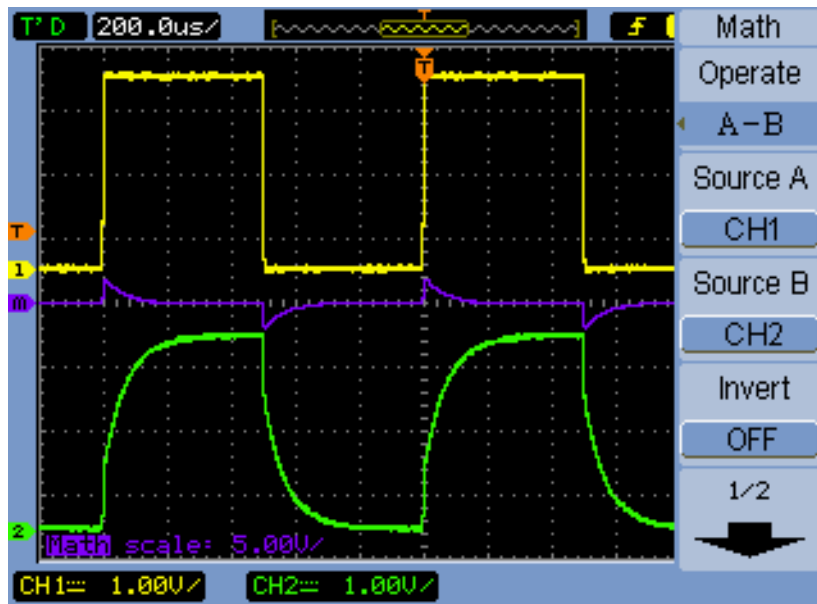

Figure 27   Creating a difference between the channel-1 and channel-2 waveforms

Note that the default scaling of the math waveform (**5.00 V/div**) is different than the scaling of the channel-1 and channel-2 waveforms (**1.00 V/div**). You can change the math waveform scaling and offset/position in the **2/2** (menu 2 of 2) level menu.

Now properly adjust the probe compensation before proceeding the next measurement.

Let's now perform a more complex math function on just the channel-1 waveform. We will convert the channel-1 waveform from a time-domain waveform into a frequency-domain waveform using a Fast Fourier Transform (FFT) math function.

12 Press the **[2]** ON/OFF front panel key until the channel-2 waveform (green trace) is turned off.

13 Set channel-1 vertical scaling (larger yellow knob) to **500 mV/div**.

14 Re-position the channel-1 waveform to be in the center of the display (smaller yellow knob).

15 Change the timebase setting (large horizontal knob) to **1.000 ms/div**.

16 Press the **[Math]** front panel key.

17 Press the **Operator** softkey; rotate the **Entry** knob to highlight **FFT**, and then push the **Entry** knob to select.

You should now see a display similar to Figure 28. The scope is now displaying both a time domain waveform (Voltage versus Time) as well as a frequency domain waveform (Amplitude in units of Vrms versus Frequency). Note that horizontal frequency scaling will be different on 4-channel models.
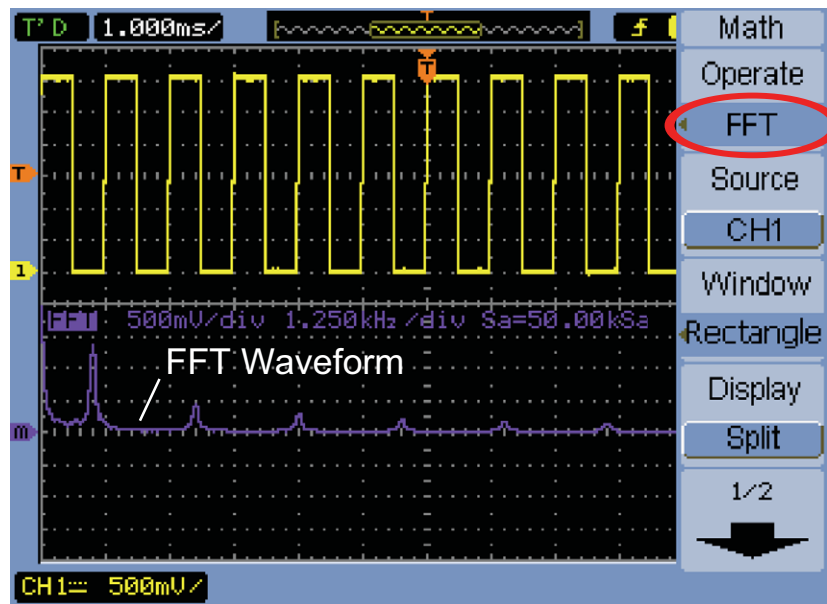

Figure 28   Using waveform math to subtract channel-2 from channel-1

An FFT math function breaks signals down into their individual sine wave frequency components. And if you remember from some of your EE circuits or Physics classes, all electrical signals, including digital signals, are composed of multiple sine waves of different frequencies. An ideal clock signal that has a 50% duty cycle should consist of a fundamental sine wave frequency component (signal's repetitive frequency), plus its odd harmonics (3rd, 5th, 7th, etc.). Note that non-ideal square waves will also include lower-level even harmonics. Let's now verify the frequencies of the fundamental and odd harmonics of this input signal.

18 Press the **[Cursors]** front panel key.

19 Press the **Mode** softkey; rotate the **Entry** knob to highlight **Track**, and then push the **Entry** knob to select.

Note that during Lab #1 we used cursors to make voltage and timing measurements using the default "manual" cursor setting. In the manual cursors mode, you can control the vertical (Y) and horizontal (X) cursor settings independently. When the "track" cursor mode is selected, you can

only control the horizontal setting of the cursors. The scope will then automatically slave the position of the amplitude cursors to the location on the waveform where the timing cursors intersects with the waveform. Let's now assign the cursors to "track" and make measurements on the FFT waveform.

20 Press the **Cursor A** softkey; then rotate the **Entry** knob to highlight **Math**, and then push the **Entry** knob to select.

21 Press the **Cursor B** softkey; rotate the **Entry** knob to highlight **Math**, and then push the **Entry** knob to select.

22 Press the **CurA—** softkey; rotate the **Entry** knob until the tracking "cross-hair" cursor lands on top of the highest frequency peak (near left side of screen).

23 Press the **CurB—** softkey; then rotate the **Entry** knob until the tracking "cross-hair" cursor lands on top of the second highest frequency peak.

Your scope's display should now look similar to Figure 29.

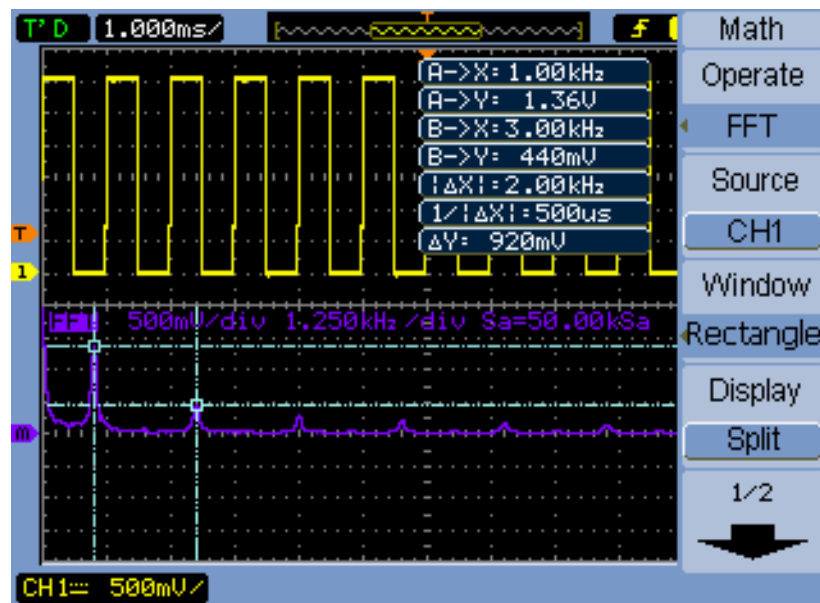**Figure 29**   Using tracking cursors to measure FFT waveform

24 What is the "A -> X" frequency, which is the fundamental component?

$$F1 \ = \ _____$$

25 What is the "B -> X" frequency, which should be the 3rd harmonic?

$$F3 \ = \ _____$$

Lab #7: Using the Scope's Zoom Mode

Most scopes have two timebases. This means that they can display waveforms based on two different horizontal settings (sec/division). Up until now, we have only been using the scope's "main" timebase. The 2nd timebase is usually called the horizontal "zoom" display. If your professor/lab instructor used an older analog oscilloscope during his/her university days, this 2nd timebase was often called the "delayed" timebase. When the zoom display is turned on, not only can you view waveforms at two different settings, but you can also perform "gated" or selective measurements.

1 Connect an oscilloscope probe between the channel-1 input BNC and the terminal labeled "**Probe Comp**". Connect the probe's ground clip to the lower terminal (ground).

2 Press the **[Default Setup]** front panel key.

3 Set the scope's timebase (large horizontal knob) to **200.0 µs/div**.

4 Press the trigger knob to automatically set the trigger level at approximately **50%**.

5 Set channel-1's vertical scaling to **500 mV/div** using the larger yellow knob.

6 Re-position the waveform to center-screen using the smaller yellow knob.

7 Press the **[Meas]** front panel key. This front panel key is labeled **[Measure]** on 4-channel models.

8 Press the **Time** softkey; then rotate the **Entry** knob to highlight **Rise Time**, and then push the **Entry** knob to select.

Your scope's display should now look similar to Figure 30. Note the Rise Time displayed near the bottom of the screen probably says something like "<40 us". Or it might say "20 us" on 4-channel models). At this timebase setting (200.0 ì s/div), the scope does not have enough horizontal resolution to perform an accurate measurement on this fast rising edge. If you can't visually see details of this fast rising edge, then the scope probably can't either. Let's now turn on the scope's horizontal zoom mode in order to make a more accurate "gated" measurement on this particular edge.
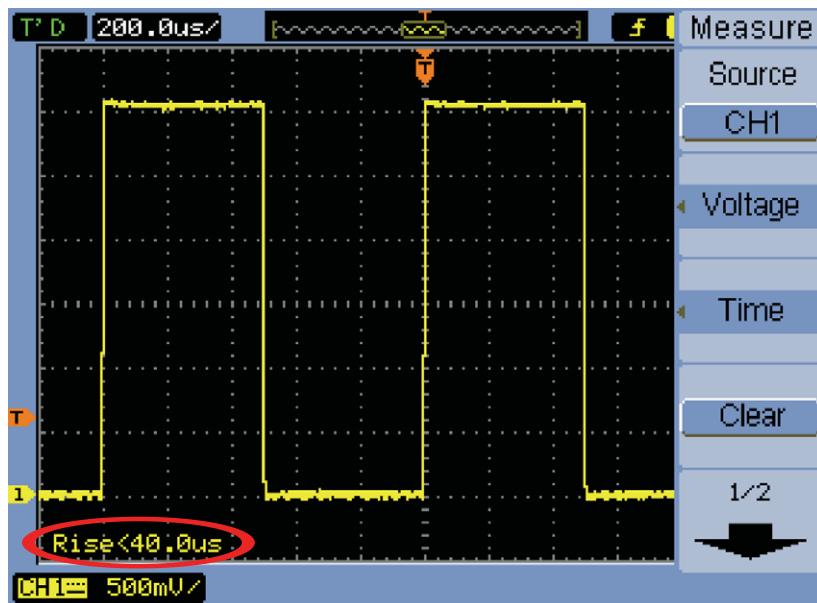
**Figure 30**   Making a poor resolution (inaccurate) rise time measurement

9 Press the large horizontal knob to turn the scope's zoom mode.

10 Rotate the large horizontal knob to set the zoom timebase to **5.000 µs/div** (displayed in white characters near the bottom of the display.

Your scope's display should now look similar to Figure 31, and the scope should now be performing a much more accurate timing measurement on this fast rising edge. When the scope's zoom mode is turned on, we can see the "big picture" and the "detailed picture" on the same scope display. We can also perform more accurate timing measurements while also viewing the "big picture". The zoom display mode also allows to select which edge or pulse to perform measurements on.

Figure 31   Making an accurate "gated" rise time measurement using the scope's zoom mode

3
Summary
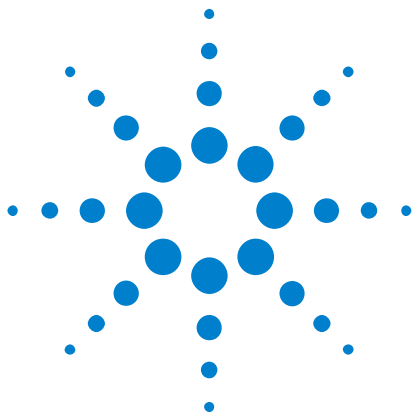
If you have been able to complete all the labs in this oscilloscope lab guide and tutorial, then you should now have a very good understanding of what an oscilloscope is and how to use one effectively. Not only will this help you to complete your assigned circuits experiments more efficiently and with greater understanding of theoretical EE and Physics concepts, but when you graduate and begin using scopes to verify and test your designs in industry, you will be able to debug your designs faster and get your products to market faster. If you would like to learn more about oscilloscopes and oscilloscope measurements, Agilent provides a broad range of application notes on this topic listed on the next page.

Agilent Technologies

Related Agilent Literature

Table 1   Related Agilent literature

Publication title	Publication type	Publication number
Evaluating Oscilloscope Fundamentals	Application note	5989-8064EN
Evaluating Oscilloscope Bandwidths for your Application	Application note	5989-5733EN
Oscilloscope Probe Loading Experiment	Application note	5990-9175EN
Evaluating Oscilloscope Sample Rates vs. Sampling Fidelity	Application note	5989-5732EN
Evaluating Oscilloscopes for Best Waveform Update Rates	Application note	5989-7885EN
Evaluating Oscilloscope Vertical Noise Characteristics	Application note	5989-3020EN
Evaluating Oscilloscopes for Best Display Quality	Application note	5989-2003EN
Evaluating Oscilloscopes to Debug Mixed-signal Designs	Application note	5989-3702EN

To download these documents, insert the publication number in the URL: http://cp.literature.agilent.com/litweb/pdf/xxxx-xxxxEN.pdf
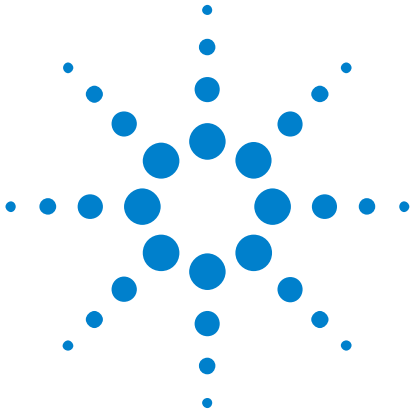
A
Oscilloscope Block Diagram and Theory of Operation

Agilent Technologies
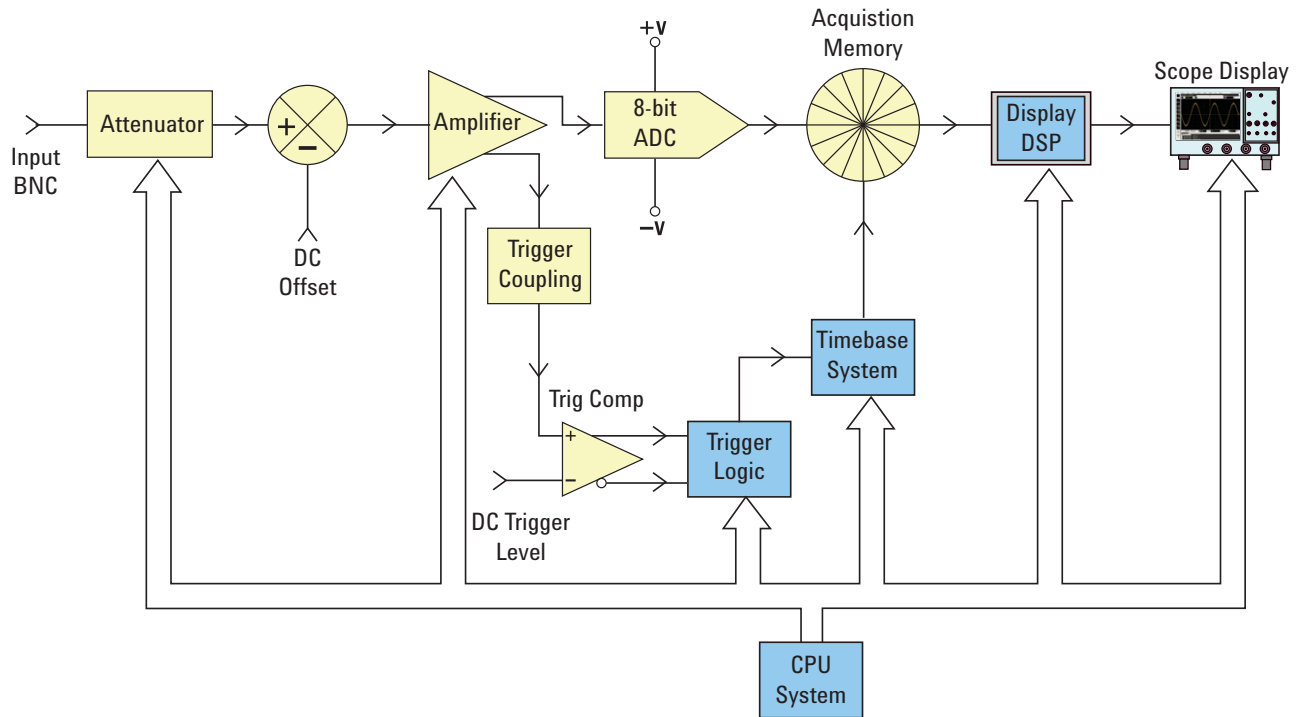
DSO Block Diagram

Figure 32   DSO block diagram

Figure 32 shows the block diagram of one channel of acquisition of a typical digital storage oscilloscope (DSO). The yellow-shaded blocks represent system components unique to a single channel of acquisition, such as Channel-1 or Channel-2. The blue-shaded blocks represent system components that are common among all channels of acquisition, such as the scope's common timebase and CPU system.

ADC Block

Shown near the middle of this block diagram is the analog-to-digital convertor (ADC). The ADC block is the core/central component of all DSOs. The function of this block is to convert the analog input into a series of digital words. Most of today's DSOs utilize 8-bit ADCs which will provide 256 unique digital output levels/codes. These digital binary codes are stored in the scope's acquisition memory, which will be discussed later. If the analog input level to the ADC is equal to or less than –V, then the output of the ADC will be 00000000 (0 decimal). If the analog input level to the ADC is equal to or greater than +V, then the output of the

ADC will be 11111111 (255 decimal). If the analog input level to the ADC is equal to 0.0 V, then the output of the ADC will be 10000000 (128 decimal).

To obtain the highest resolution and accurate measurements, the input to the ADC must be scaled within its dynamic range, which is ± V. Although the ADC has a limited and fixed input dynamic range based on the ADC's reference voltages (± V), scopes must be able to capture a broad dynamic range of signals including both high level and low level input signals. Scaling the ADCs analog input to be within the ADC's dynamic range is the combined function of attenuator, DC offset, and amplifier blocks, which are discussed next.

Attenuator Block

The attenuator block is basically a network of resistor dividers used to scale the input signal to be within the dynamic range of the scope's variable gain analog amplifier and ADC. When inputting a high level input signal such as 40 Vpp, the signal level must be reduced (attenuated). If inputting a low level input signal such as 10 mVpp, then the input signal is passed through to the amplifier with no attenuation (1:1). When you change the scope's V/div setting, you may hear audible clicking sounds. The clicking that you hear are mechanical relays switching-in different resistor divider networks. Note that the attenuator block also includes switching of the user-selectable input impedance (1 MΩ or 50 Ω), as well as AC or DC input coupling.

DC Offset Block

When inputting a signal with DC offset such as a digital signal that swings between 0V to 5V, if we want to display this signal centered on the scope's display, then an internal DC offset of the opposite polarity must to added to the signal in order to shift the input signal to be within the ADC's dynamic range. Alternatively, AC coupling can be selected to eliminate the DC component of the input signal.

Amplifier Block

The final analog processing stage to scale the input signal to be within the dynamic range of the ADC system is the scope's variable gain amplifier. If inputting a very low level input signal, you would typically set the V/div setting to a relatively low setting. With a low V/div setting the attenuator stage would then pass this signal straight through to the amplifier with no attenuation (gain =1), and then the amplifier would boost (gain >1) the amplitude the signal in order to take advantage of the full dynamic range

of the ADC. If inputting a very high level input signal, you would typically set the V/div setting to a relatively high setting. Using a high V/div setting, the attenuator stage would first attenuate the input signal (gain < 1) to get it within the dynamic range of the amplifier, and then the amplifier may further attenuate the signal (gain <1) to get it within the dynamic range of the ADC.

Note that when you select a specific V/div setting, the scope automatically determines the required amount of attenuation in the attenuator block and the required amount of gain (or possibly additional attenuation) in the amplifier block. You can think of the attenuator block, DC Offset block, and amplifier block as a single block of analog input signal conditioning that linearly conditions a signal that is representative of the input signal to be within the dynamic range of the ADC block based on the V/div and offset setting of that particular channel of the scope.

Trigger Comparator and Trigger Logic Blocks

The purpose of the trigger comparator and trigger logic blocks are to establish a unique point in time on the input signal (or combination of multiple input signals) upon which to establish a synchronized acquisition. After you've completed Lab #2 (Learning the Basics of Oscilloscope Triggering) in this document, you should have a better understanding of what triggering is all about.

Let's assume that your input signal is a sine wave and that you want to trigger acquisitions on the rising edge of the sine wave at the 50% level. In this case the non-inverted output of the trigger comparator would be a square wave with a 50% duty cycle. If you set the trigger level above the 50% level, then the non-inverted output of the trigger comparator would be less than 50%. Alternatively, if you set the trigger level below the 50% level, then the non-inverted output would be greater than 50%. Assuming that triggering is to be based on just a positive edge crossing of a single channel, the trigger logic block would pass the non-inverted output of the trigger comparator to the timebase block. If you have selected to trigger on a negative edge crossing of a single channel, then the trigger logic block would pass the inverted output of the trigger comparator to the timebase block. The timebase block then uses the rising edge of the trigger signal as the unique synchronization point in time. Also note that triggering can be based on many other variables including time-qualification, as well as a combination of input signals from multiple input channels.

Timebase and Acquisition Memory Blocks

The timebase block controls when ADC sampling is started and stopped relative to the trigger event. In addition, the timebase block controls the ADCs sample rate based on the scope's available acquisition memory depth and the timebase setting. For example, let's assume that the scope has been setup to trigger at exactly center screen (default setting) using a timebase setting of 1 ms/div. Let also assume for simplicity that the scope's acquisition memory depth is just 1000 points. Using these assumptions, the scope should acquire 500 points before the trigger event followed by acquiring 500 points after the trigger event. At this timebase setting, the scope will acquire 1000 points across a 10 ms time span (1 ms/div x 10 divisions). Even though the scope's maximum specified sample rate may be 2 GSa/s, at this timebase setting the timebase block will reduce the scope's continuous sample rate to 100 k samples/sec (Sample Rate = Memory/Time-span = 1000 samples/10ms = 100 kSa/s).

When the Run key is pressed, the timebase block enables continuous storing of digitized data into the scope's "circular" acquisition memory at the appropriate sample rate (100 kSa/s). While the timebase block increments addressing of the circular acquisition memory buffer after each sample, it also counts the number of samples taken up to 500 (assuming a memory depth of 1000 and also assuming triggering at center screen). After the timebase block determines that a minimum of 500 samples have been stored (meaning that at least half of the acquisition memory is full), the timebase block then enables triggering and begins to look for the first rising edge of the output trigger comparator (assuming simple edge triggering mode). While looking for the trigger event, acquisitions continue to be stored in the scope's circular acquisition memory buffer. If the trigger event is very infrequent, stored samples can actually be overwritten while waiting on the trigger event. But this is okay. Once the trigger event is detected, the timebase block then begins counting up to 500 again. When an additional 500 samples have been stored, the timebase block then disables (shuts down) sampling. This means that the last 500 stored samples represent sequential points on the waveform that occurred *after* the trigger event, while the previous 500 points represent sequential points on the waveform that occurred *before* the trigger event. At this point operation is turned over to the Display DSP block.

Although we used an example of triggering at center-screen, using the horizontal delay/position control you can position the trigger point at any point. For example, if you adjust the delay such that the trigger point should occur at the 75% point along the horizontal axis (relative to the left side of the screen), then the timebase block would set the counter to initially store 750 points (assuming a memory depth of 1000 points) before enabling the trigger, and then capture an additional 250 points after detection of the trigger event.

Display DSP Block

Once an acquisition is completed, the Display DSP block then backs out the stored data from the acquisition memory block in a last-in-first-out sequence. Not only can the Display DSP block quickly perform digital signal processing on the stored data such as running a Sin(x)/x digital reconstruction filter, but it can also "pipeline" the stored and/or processed data into the scope's pixel display memory. After the data has been "backed out" of acquisition memory, the DSP block then signals the timebase block that it can begin another acquisition.

Note that early generations of DSOs did not include an explicit Display DSP block. This function was traditionally handled by the scope's CPU system, but with much lower efficiency that produced much slower waveform update rates. With custom display DSP processing, some of today's DSOs can update waveforms as fast as 1,000,000 waveforms/second.

TIP	To learn more about the fundamentals of oscilloscopes, download Agilent's application note titled *Evaluating Oscilloscope Fundamentals*. This publication is listed in the "Related Agilent Literature" section of this document with instructions on how to download.

B
Oscilloscope Bandwidth Tutorial

Oscilloscopes have many different specifications that determine the accuracy that signals can be captured and measured. But the primary specification of an oscilloscope is its bandwidth. The oscilloscopes that you are using in your Electrical Engineering undergraduate student labs probably have sufficient bandwidth for most, if not all, of the experiments that your professor will assign. When you finally graduate from your EE program and get into the electronics industry, it is very possible that you will need to either select a scope from an instrument pool at your company to perform testing on your designs, or perhaps you will be tasked with evaluating various oscilloscopes to purchase. This tutorial on oscilloscope bandwidth will provide you some helpful hints on how to select an oscilloscope with the appropriate bandwidth for your digital and analog applications. But first, let's define oscilloscope bandwidth.

Defining Oscilloscope Bandwidth

All oscilloscopes exhibit a low-pass frequency response that rolls-off at higher frequencies as shown in Figure 33. Most scopes with bandwidth specifications of 1 GHz and below typically have what is called a Gaussian frequency response. An oscilloscope Gaussian frequency response approximates a single-pole low-pass filter, which is something you may have already studied about in some of your circuits classes and perhaps graphed as a Bode plot.
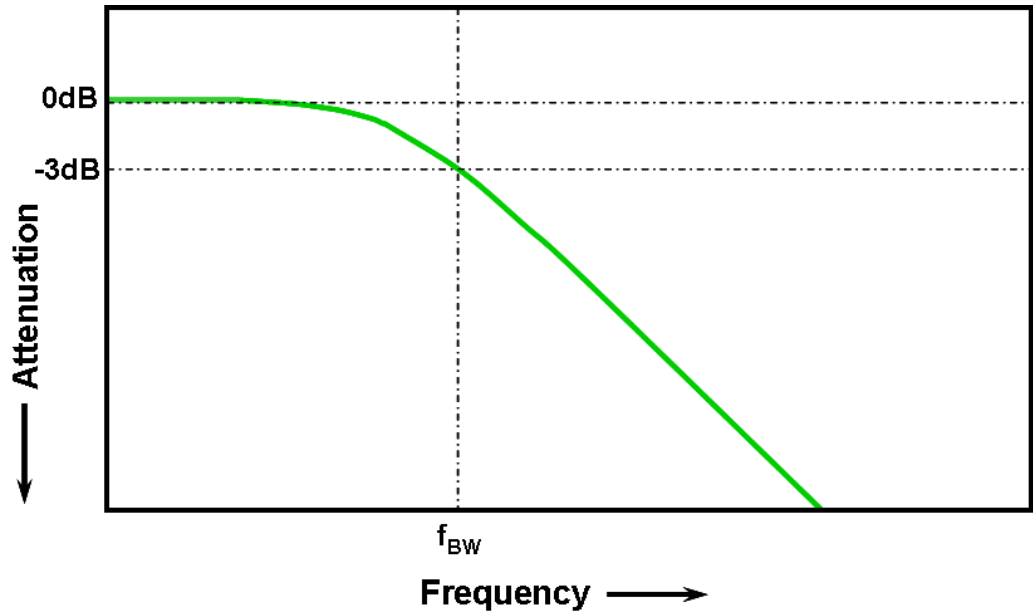
Agilent Technologies

Figure 33   Oscilloscope Gaussian frequency response

The lowest frequency at which the input signal is attenuated by 3 dB is considered the scope's bandwidth (f_{BW}). Signal attenuation at the -3 dB frequency translates into approximately -30% amplitude error. In other words, if you input a 1 Vp-p, 100 MHz sine wave into a 100 MHz bandwidth oscilloscope, the measured peak-to-peak voltage using this scope would be in the range of approximately 700 mVp-p (-3 dB = 20 Log [0.707/1.0]). So you can't expect to make accurate measurements on signals that have significant frequencies near your scope's bandwidth.

Closely related to an oscilloscope's bandwidth specification is its rise time specification. Scopes with a Gaussian-type frequency response will have an approximate rise time of $0.35/f_{BW}$ based on a 10% to 90% criterion. But you need to remember that a scope's rise time is not the fastest edge speed that the oscilloscope can accurately measure. It is the fastest edge speed the scope can possibly produce if the input signal has a theoretical infinitely fast rise time (0 ps). Although this theoretical specification is impossible to test — since pulse generators don't have infinitely fast edges — from a practical perspective, you can test your oscilloscope's rise time by inputting a pulse that has edge speeds that are 5 to 10 times faster than the scope's rise time specification.

Required Bandwidth for Analog Applications

Years ago, most oscilloscope vendors recommended that your scope's bandwidth should be at least three times higher than the maximum input signal frequency. And this rule-of-thumb recommendation may be what

your professor remembers. Although this "3X" multiplying factor would not apply to digital applications based on clock rates or edge speeds, it still applies to analog applications, such as modulated RF. To understand where this 3-to-1 multiplying factor comes from, let's look at an actual frequency response of a 1 GHz bandwidth scope.

Figure 34 shows a measured frequency response test (1 MHz to 2 GHz) on an Agilent 1 GHz bandwidth oscilloscope. As you can see, at exactly 1 GHz the measured output (waveform on oscilloscope display) is attenuated by slightly less than 3 dB (Vo/Vi > 0.7). To make accurate measurements on analog signals, you would need to use the scope in the portion of the frequency band where it is still relatively flat with minimal attenuation. At approximately one-third the scope's 1 GHz bandwidth, this scope exhibits very little attenuation (-0.2 dB).
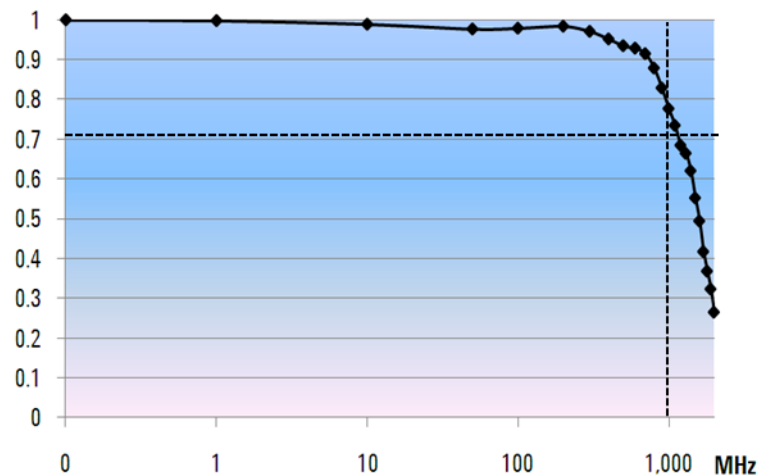

Figure 34   Actual frequency response of an Agilent 1 GHz bandwidth oscilloscope

Required Bandwidth for Digital Applications

The vast majority of today's EE graduates will be focused on digital design applications when they enter the electronics industry. And digital clock rates and serial data links in the multi-Giga bits/sec range are very common today.

Rule of Thumb

As a rule of thumb, your scope's bandwidth should be at least five times higher than the fastest digital clock rate in your system under test. If your scope meets this criterion, then it will be able to capture up to the fifth harmonic with minimum signal attenuation. This component of the signal is very important in determining the overall shape of your digital signals.

$$f_{BW} >= 5 \times f_{clk}$$

But if you need to make accurate measurements on high-speed edges, this simple formula does not take into account the actual highest-frequency components embedded in fast rising and falling edges.

Step 1: Determine fastest actual edge speeds

A more accurate method to determine required oscilloscope bandwidth is to ascertain the maximum frequency present in your digital signals, which is not the maximum clock rate. The maximum frequency will be based on the fastest edge speeds in your designs. So the first thing you need to do is determine the rise and fall times of your fastest signals. You can usually obtain this information from published specifications for devices used in your designs.

Step 2: Calculate f_{knee}

You can then use a simple formula to compute the maximum "practical" frequency component. Dr. Howard W. Johnson has written a book on this topic, "*High-speed Digital Design – A Handbook of Black Magic.*" [1] He refers to this frequency component as the "knee" frequency (f_{knee}). 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.

$$f_{knee} = 0.5/RT \ (10\% \ 90\%)$$
$$f_{knee} = 0.4/RT \ (20\% \ 80\%)$$

For signals with rise time characteristics based on 10% to 90% thresholds, f_{knee} is equal to 0.5 divided by the rise time of the signal. For signals with rise time characteristics based on 20% to 80% thresholds, which is very common in many of today's device specifications, f_{knee} is equal to 0.4

divided by the rise time of the signal. Now don't confuse these rise times with a scope's specified rise time. We are talking about actual signal edge speeds.

Step 3: Calculate scope bandwidth

The third step is to determine the oscilloscope bandwidth required to measure this signal, based on your desired degree of accuracy when measuring rise times and fall times. Table 2 shows multiplying factors for various degrees of accuracy for scopes with a Gaussian frequency response.

Table 2   Multiplying factors to calculate required scope bandwidth based on desired accuracy

Required Accuracy	Require Bandwidth
20%	$f_{BW} = 1.0 \times f_{knee}$
10%	$f_{BW} = 1.3 \times f_{knee}$
3%	$f_{BW} = 1.9 \times f_{knee}$

Example

Let's now walk through this simple example:

Determine the minimum required bandwidth of an oscilloscope with an approximate Gaussian frequency response to measure a 1 ns rise time (10-90%)

If the signal has an approximate rise/fall time of 1 ns (based on a 10% to 90% criteria), then the maximum practical frequency component (f_{knee}) in the signal would be approximately 500 MHz.

$$f_{knee} = 0.5/1 \text{ ns} = 500 \text{ MHz}$$

If you are able tolerate up to 20% timing errors when making parametric rise time and fall time measurements on your signals, then you could use a 500 MHz bandwidth oscilloscope for your digital measurement applications. But if you need timing accuracy in the range of 3%, then a scope with 1 GHz bandwidth would be the better choice.

> **20% timing accuracy:**
> **Scope bandwidth = 1.0 x 500 MHz = 500 MHz**
>
> **3% timing accuracy:**
> **Scope bandwidth = 1.9 x 500 MHz = 950 MHz**

Let's now make some measurements on a digital clock signal with characteristics similar to this example, using various bandwidth scopes....

Digital Clock Measurement Comparisons

Figure 35 shows the waveform results when measuring a 100 MHz digital clock signal with fast edge speeds using a 100 MHz bandwidth oscilloscope. As you can see, this scope primarily just passes through the 100 MHz fundamental of this clock signal, thus representing our clock signal as an approximate sine wave. A 100 MHz scope may be a good solution for many 8-bit, MCU-based designs with clock rates in the 10 MHz to 20 MHz range, but 100 MHz bandwidth is clearly insufficient for this 100 MHz digital clock signal.
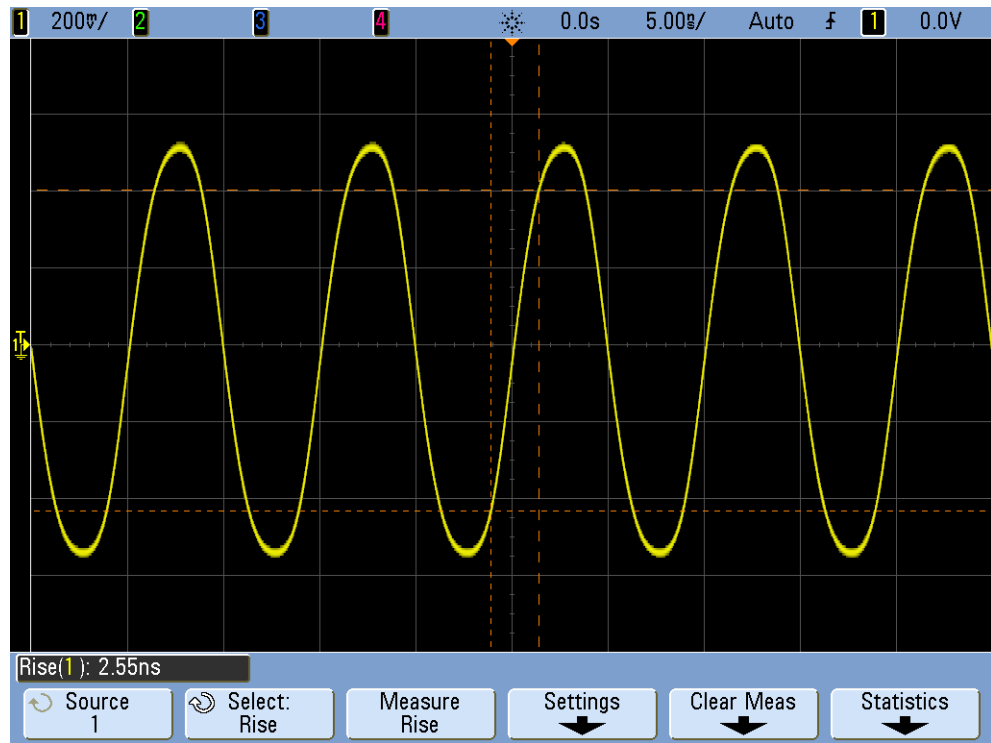
Figure 35   100 MHz digital clock signal captured on a 100 MHz bandwidth scope

Using a 500 MHz bandwidth oscilloscope, Figure 36 shows that this scope is able to capture up to the fifth harmonic, which was our first rule of thumb recommendation. But when we measure the rise time, we see that the scope measures approximately 750 ps. In this case, the scope is not making a very accurate measurement on the rise time of this signal. The scope is actually measuring something closer to its own rise time (700 ps), not the input signal's rise time, which is closer to 500 ps. We need a higher-bandwidth scope for this digital measurement application if timing measurements are important.
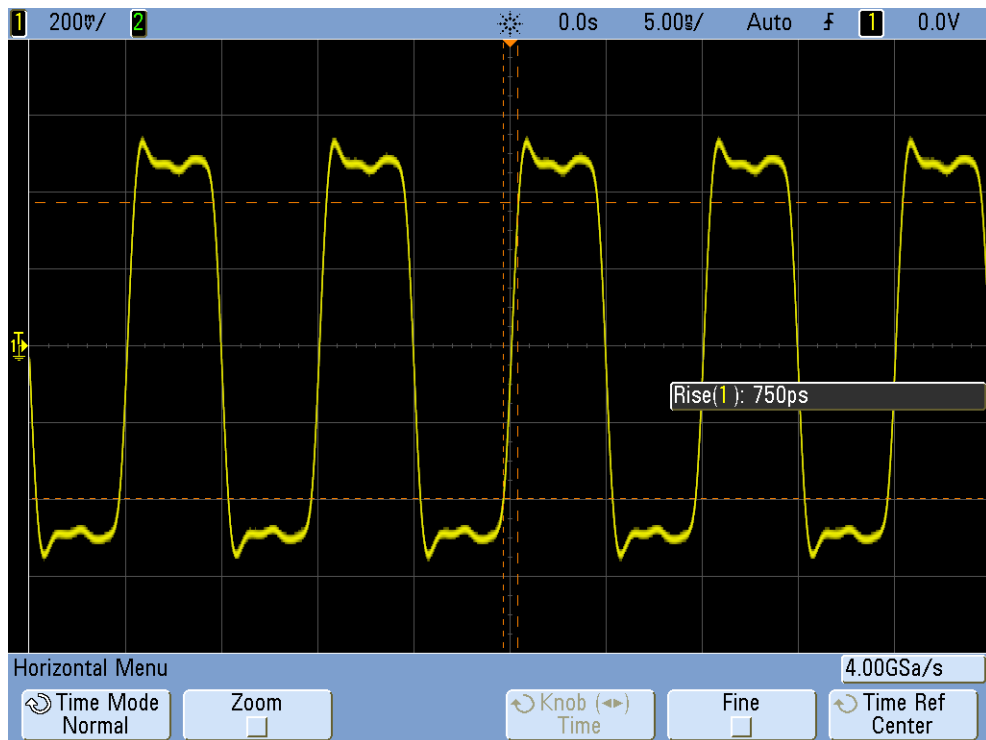
Figure 36   100 MHz digital clock signal captured on a 500 MHz bandwidth scope

When we use a 1 GHz bandwidth scope to capture this 100 MHz digital clock, the result is that we now have a much more accurate picture of this signal, as shown in Figure 37. We can measure faster rise and fall times, we observe less overshoot, and we can even observe subtle reflections that the lower bandwidth scope masked.
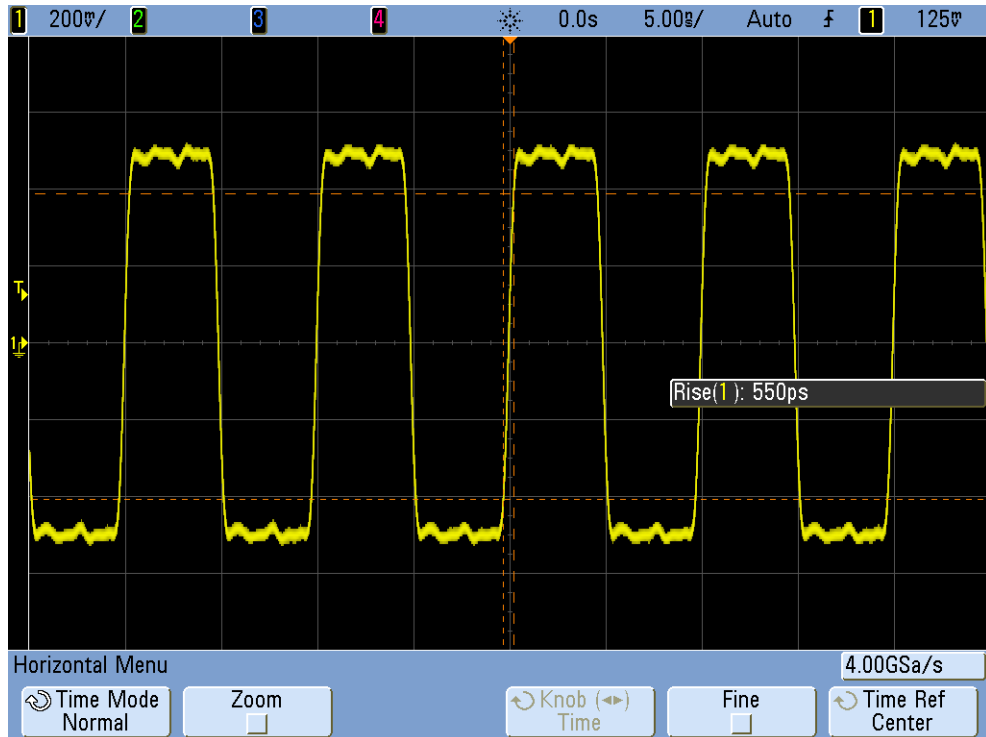
Figure 37   100 MHz digital clock signal captured on a 1 GHz bandwidth scope

This tutorial on oscilloscope bandwidth focused on oscilloscopes that exhibit a Gaussian frequency response, which is typical of scopes that have bandwidth specifications of 1 GHz and below. Many higher bandwidth oscilloscopes exhibit a frequency response that has a sharper roll-off characteristic. With this type of frequency response, in-band frequencies (frequencies below the -3 dB frequency) are attenuated less, while out-of-band frequencies (frequencies above the -3 dB frequency) are suppressed to a higher degree. This type of frequency response, which begins to approximate an ideal "brick-wall" filter, is sometimes called a "maximally-flat" frequency response. The formulas for computing required oscilloscope bandwidth on these higher bandwidth scopes (> 1 GHz) are different than what was presented in this tutorial guide. If you would like to learn more about oscilloscope bandwidth, you can download Agilent's application note title, "*Evaluating Oscilloscope Bandwidths for your Application*". This publication is listed in the "Related Literature" section of this document with instructions on how to download.

[1] High-Speed Digital Design, A Handbook of Black Magic, Howard Johnson, Martin Graham, 1993, Prentice Hall PTD, Prentice-Hall, Inc, Upper Saddle River, New Jersey 07458

Index