

The Oscilloscope and the Function Generator:

Some introductory exercises for students in the advanced labs

Introduction

So many of the experiments in the advanced labs make use of oscilloscopes and function generators that it is useful to learn their general operation. Function generators are *signal sources* which provide a specifiable voltage applied over a specifiable time, such as a “sine wave” or “triangle wave” signal. These signals are used to control other apparatus to, for example, vary a magnetic field (superconductivity and NMR experiments) send a radioactive source back and forth (Mössbauer effect experiment), or act as a timing signal, i.e., “clock” (phase-sensitive detection experiment). Oscilloscopes are a type of *signal analyzer*—they show the experimenter a picture of the signal, usually in the form of a voltage versus time graph. The user can then study this picture to learn the amplitude, frequency, and overall shape of the signal which may depend on the physics being explored in the experiment.

Both function generators and oscilloscopes are highly sophisticated and technologically mature devices. The oldest forms of them date back to the beginnings of electronic engineering, and their modern descendants are often digitally based, multifunction devices costing thousands of dollars. This collection of exercises is intended to get you started on some of the basics of operating 'scopes and generators, but it takes a good deal of experience to learn how to operate them well and take full advantage of their capabilities.

Function generator basics

Function generators, whether the old analog type or the newer digital type, have a few common features:

- A way to select a waveform type: sine, square, and triangle are most common, but some will give ramps, pulses, “noise”, or allow you to program a particular arbitrary shape.
- A way to select the waveform frequency. Typical frequency ranges are from 0.01 Hz to 10 MHz.
- A way to select the waveform amplitude.
- At least two outputs. The “main” output, which is where you find the desired waveform, typically has a maximum voltage of 20 volts peak-to-peak, or ± 10 volts range. The most common output impedance of the main output is 50 ohms, although lower output impedances can sometimes be found. A second output, sometimes called “sync”, “aux” or “TTL” produces a square wave with standard 0 and 5 volt digital signal levels. It is used for synchronizing another device (such as an oscilloscope) to the possibly variable main output signal.

A wide variety of other features are available on most modern function generators, such as “frequency sweep”—the ability to automatically vary the frequency between a minimum and maximum value, “DC offset”—a knob that adds a specified amount of DC voltage to the time-varying

waveform, and extra inputs or outputs that can be used to control these extra features by other instruments.

Oscilloscope basics

All oscilloscopes share certain basic features. Refer to Fig. 1 to see where typical controls may be found.

- The most recognizable feature: a screen. On older analog scopes this is a cathode-ray tube or CRT; the signal creates a moving dot or “trace” across the screen. On newer digital scopes the screen is a CRT or flat-panel display that operates like a computer monitor. The basic use of the screen is to display the signals in a voltage versus time graph. The screen usually has a graticule on it of about 1 cm squares.
- At least two (maybe more) signal inputs, or “channels”, typically called “CH1”, “CH2”, etc., and one external “trigger” input, typically called “EXT TRIG”.
- A collection of controls related to vertical part of the display associated with the input signals. These control the kind of coupling to the input: direct—“DC”, through a capacitor—“AC”, or disconnected—“GND”. The amount of amplification applied to the signal is controlled by a knob, and is specified in terms of screen units: a “10mV/div” setting means that a 10 millivolt change in the input signal will move the trace vertically by one major division.
- A collection of controls related to the horizontal part of the display. These controls set the time axis and are calibrated in seconds per division, e.g., $1\mu\text{s}/\text{div}$ means that one major division corresponds to 1 microsecond. The horizontal controls are sometimes called the “timebase” and the setting is called the “sweep rate”.
- A collection of controls called the “trigger” that are used to synchronize the input signal to the horizontal display. Because there is no fixed relationship between an external signal and the internal timebase, the trigger makes the scope wait until some prescribed level in an input is reached before beginning its display. Triggering controls are discussed in more detail on page 6.

In addition to the above features which are common to both analog and digital varieties, digital oscilloscopes typically come with the ability to save data and control settings to memory, perform mathematical operations on data traces, average many cycles together to reduce the effect of random fluctuations, and carry out automatic measurements of frequency and amplitude of the input signals. One can also make printouts of the display screen to keep a paper record of the measurement. The basic and advanced features of oscilloscopes will be explored in the following exercises.

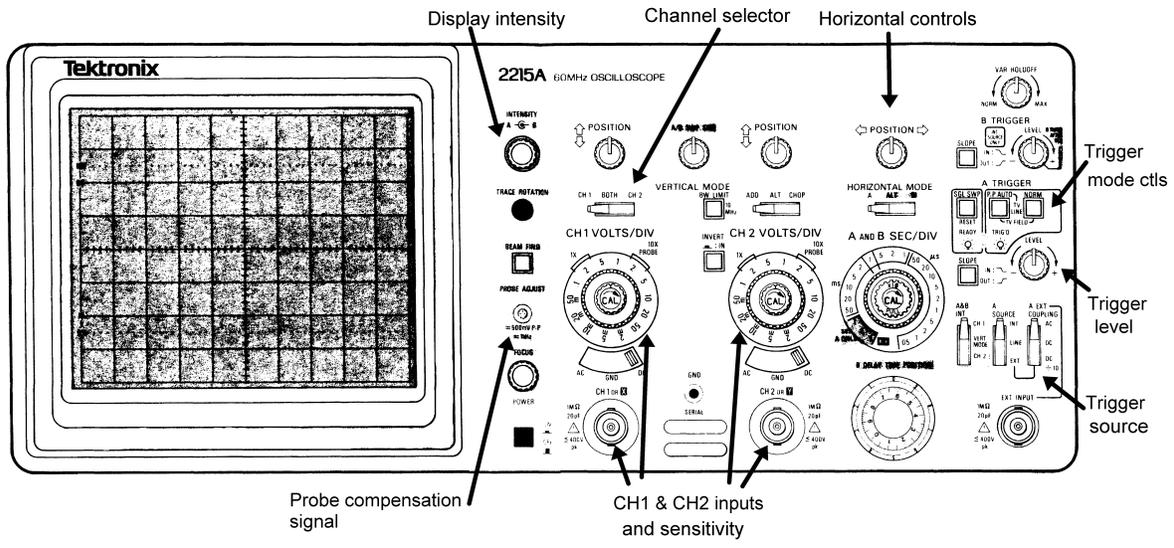
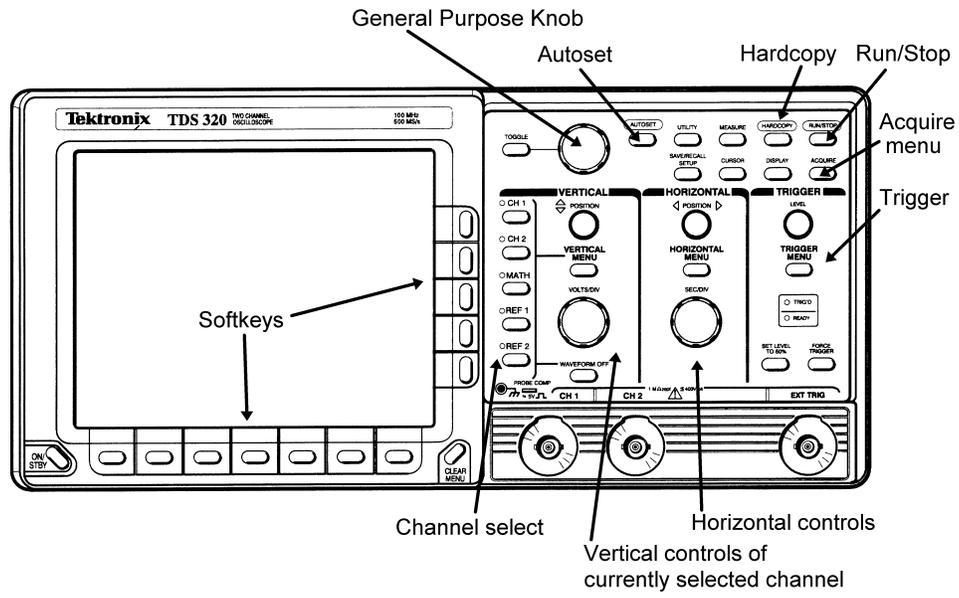


Figure 1: Front panel layouts of the digital scope (top) and the analog scope (bottom).

Getting started

In this exercise you will look at the types of signals available from a generator and explore the basic controls of the generator and digital oscilloscope. First check to see that you have the following equipment at your station:

- One digital oscilloscope. You may have one of the following, all made by Tektronix: TDS320, TDS340, TDS360, TDS620B or TDS3012.
- One analog oscilloscope. These are Tektronix models 2213A, 2215A, or 2235.
- Two function generators. Analog types are made by Krohn-Hite, Wavetek, or Exact and are identifiable by their prominent frequency-adjust knob and fairly simple panel controls. Digital types are made by Wavetek and Stanford Research Systems (SRS) and are identifiable by the digital display and large number of panel buttons.
- An oscilloscope probe. A scope probe is *not just a wire with a clip*, but it has other circuitry in it to modify the signal before it passes to the scope input. In most of the advanced lab experiments, the scope is wired into the setup using BNC cables, so you won't use a probe in these cases, but it is standard equipment when using a scope to study the internals of a circuit.
- A box of small parts, which should have a small screwdriver (possibly made of plastic), a few resistors, a capacitor, some wires and some BNC adaptors.
- A plug-in circuit board ("breadboard") that can be used to wire up test circuits.
- BNC cables. You will need a few of these. There are a variety of lengths available on racks attached to the back side of the mobile whiteboard.

Let's get started by looking at some function generator signals.

1. Turn on a function generator. The digital ones will take a few seconds to complete internal tests, but the analog ones will be ready right away.
2. Turn on the **digital** oscilloscope. After the scope completes its internal tests, press the "Clear menu" button at the lower right hand corner of the display screen.
3. Get a couple of BNC cables from the cable rack; six-foot lengths should be adequate. Use the BNC cables to connect the "main" output of the generator to CH1 of the oscilloscope and the "aux" (or "sync" or "TTL") output to CH2.
4. Set the controls of the function generator to produce a sine wave of about 1000 Hz frequency and a few volts amplitude. Depending on your generator, here's how:

ANALOG (Krohn-Hite, Wavetek, Exact) Use the "waveform" or "function" switch to select the sine (curvy-line) waveform type. Use the frequency adjust knob and multiplier button/switch to select the frequency, and then set the amplitude knob ("Ampl" or "Attenuator") at about 12 o'clock, with any attenuator switches set at 0dB.

DIGITAL (SRS DS345) Use the Δ , ∇ buttons under the FUNCTION menu to highlight the sine function symbol. Press the **FREQ** button, and then key in "1" followed by the

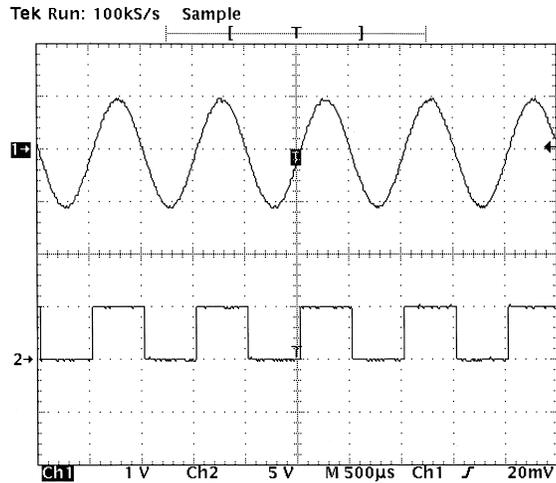


Figure 2: Digital scope display of a 1 kHz sine wave from a function generator and its associated “sync” signal. Note these important features: the channel 0-volt markers (“1→” and “2→”); the trigger indicators: the “T” in the middle of the screen and the small arrow on the right; the channel sensitivities (volts/div), sweep time (sec/div) and trigger settings are shown along the bottom of screen: channel 1 (“Ch1 1V”) is showing a 2 volt peak-to-peak sine wave, and channel 2 (“Ch2 5V”) is showing a 0 to 5 volt square wave. The sweep time is 500 $\mu\text{s}/\text{div}$ (“M 500 μs ”), and the trigger is set for a 20 mV positive-going level in channel 1.

“kHz/Vrms” units button. To set the amplitude, press the AMPL button followed by “2” and the “kHz/Vrms” units button.

DIGITAL (Wavetek 29) Select the “Sine” option under the FUNCTION menu. Press FREQ/PER under the SET menu, followed by “1” and the “kHz us mV” units button. Set the amplitude by pressing AMPL followed by “2” and “MHz ns V”. **IMPORTANT:** the Wavetek 29 will not activate its main output until you press the OUTPUT button at the lower right, above the MAIN OUT connector.

5. In order to start from a uniform setting, reset the oscilloscope to its factory default values by pressing the SAVE/RECALL button, and then using the softkeys to choose the “factory default” setup. Then press the CH1 and CH2 buttons to turn on both of these channels. Finally, press AUTOSET on the oscilloscope panel. Autoset tell the scope to examine the inputs and choose a best guess at reasonable input gain and timebase settings. You should see a display similar to that shown in Fig. 2. If you don’t get this, *ask for help now*.

Oscilloscope vertical, horizontal and trigger controls

Now that you have a signal, explore the effects of the various scope controls. Refer to Fig. 1 to help you locate the controls on your scope.

Vertical

1. Press the CH1 button and see what happens when you turn the POSITION and VOLTS/DIV knobs. Pay attention to both the changes in the displayed waveform and the text indicators

at the bottom of the screen. You should be able to deduce the meaning of these indicators.

2. Press the CH2 button and repeat the above step.
3. You can turn a waveform off by first pressing the channel selector button (turning the adjacent green LED on), followed by the WAVEFORM OFF button. Try this, and turn off and then on the waveforms from channel 1 and channel 2.
4. Press the VERTICAL MENU button and study the various options that are controllable by the softkeys.

Horizontal

1. Adjust the POSITION and SEC/DIV knobs and see what happens. Notice that both the sample rate and the sweep rate depend on the setting of the SEC/DIV knob. (The sample rate is the rate at which the waveform is digitized, and it must be notably higher than the sweep rate for the trace to be accurately drawn.) Also notice how the bar indicator and trigger-position indicator at the top of the screen changes as you move the POSITION knob: the bar represents the scope's memory, and the region between the square brackets [] represents the amount of the memory shown on the screen. In the TDS300 series, the screen always looks at a portion of the scope's total memory; in the TDS600 series, the amount of memory visible on the screen is adjustable (as an option under the HORIZONTAL MENU); in the TDS3000 series, the screen always looks at all of the scope's memory, but you can zoom in on a small part of it with the "magnify" option (under the button with the magnifying glass symbol).
2. (TDS300 and TDS600 series only) Press the HORIZONTAL MENU button and study the various options. Note in particular the presets for the trigger location on the screen. Other options are used less frequently, so you can ignore them for now.

Trigger

The purpose of a trigger is to synchronize the scope's measurement cycle with an input signal. You can think of triggering as analogous to the following example. Imagine that you want to measure the time duration that a runner takes to run the first 100 meters on an 800 meter track while she runs laps. You would first position yourself so that you could see the runner approach a specific marker and then start your timer as she passed it. After timing the 100-meter stretch, you would wait until the runner made a full lap and returned to the marker again before timing it again. Because the events you are measuring (100-meter time duration over successive laps) take a different amount of time than the duration of a full lap, and may start at different time points from lap to lap, you need to wait for a part of each lap for the runner to reach the marker before starting your timer. The oscilloscope trigger works in a similar way: the scope waits until the input waveform reaches a particular level or produces a particular pattern before starting its "trace" or measurement cycle. Subsequent traces occur after waiting until the level/pattern repeats.

The operation of the oscilloscope trigger is the most difficult skill for the novice to learn. Before we look at the trigger controls, let's review some terms associated with triggering.

Trace Jargon term for the visible part of a scope's measurement cycle. The term originates from the older analog scopes in which a moving bright dot is created on a CRT screen by an electron

beam deflected under the influence of the vertical and horizontal amplifiers. In digital scopes the trace is the visual display of the scope's data memory. In general, a proper trigger must occur before a trace is created or updated.

Trigger Level The voltage that a trigger signal must cross before the trace will start.

Slope The trigger level coupled with the “slope” defines the pattern that must be sensed by the trigger circuit before it will start the trace. A “positive slope” means that the trigger level must be crossed going from a lower to a higher voltage, i.e., “going up”; likewise a “negative slope” means that the trigger level must be crossed going from a higher to a lower voltage: “going down”.

Source The signal from which the trigger circuit takes its input. If the trigger comes from one of the channel inputs (CH1 or CH2), the source is “internal” or INT. This is the most common choice. The second most common choice is from a separate signal connected to a special trigger input, called EXT TRIG (or just EXT). The other choices are from the 60Hz power-line signal (LINE) or from an oscillator internal to the scope (AUTO).

Mode Refers to additional conditions under which a trace is triggered. The two basic modes are “normal” (NORM) and “automatic” (AUTO). Under the normal triggering mode, no trace will be made until the triggering condition is met. Under automatic triggering, the scope looks for a valid triggering condition within a certain time, but if none is found, it will trigger anyway. Automatic triggering is useful when you are looking for a signal that you do not know much about—you may see some movement in the trace but it won't be stable. Normal triggering gives you more control over when a trace happens, and is the best mode to use when you know that a usable trigger signal exists. With analog scopes the screen will be *blank* under normal triggering unless a valid trigger occurs; with digital scopes the screen will freeze the last trace recorded.

Coupling DC coupling means that the trigger signal is passed to the trigger circuit unchanged. AC coupling means that any constant or very slowly varying voltage is blocked from the trigger circuit. Some scopes also have coupling options for low-frequency or high-frequency rejection in order to reduce the effects of noise that may be present in those frequency ranges.

Holdoff A time period following a trigger event where subsequent triggers are blocked. The holdoff control can help stabilize traces when looking at more complex waveforms that have many possible trigger points.

Delay A waiting time between trigger event and the start of a trace. Trigger delay is used when you want to look at a part of a signal that happens well after the desired trigger point.

Pretrigger The time before the trigger event. Digital scopes can show a fairly long pretrigger period. You can use this period to see the signal leading up to the trigger condition. Analog scopes have a very short pretrigger period.

Most of the time you need only worry about four settings: the trigger *level*, the *slope*, the *mode* and the *source*. Let's see how these work.

1. Press AUTOSSET again to reestablish the settings so that the traces are similar to Fig. 2.
2. Press the TRIGGER MENU button.

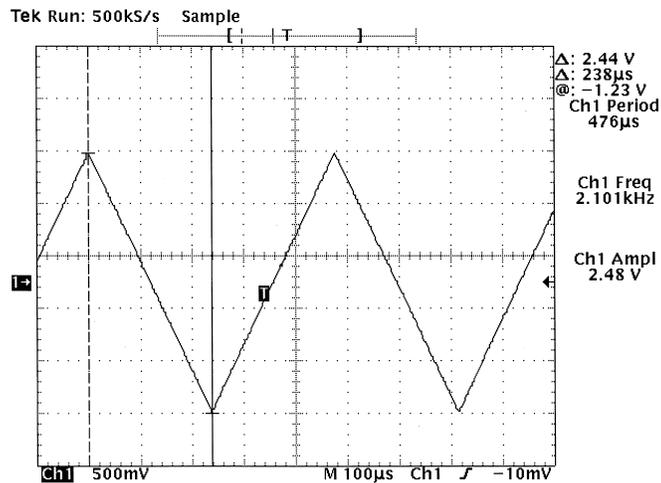


Figure 3: Digital scope measurements of a triangle wave.

3. Look at the softkey selections along the bottom of the display, select the “Level” menu, and turn the LEVEL knob. The source for the trigger should be CH1. Note what happens on the screen. In particular, note how the sine wave shifts horizontally as you change the level.
4. Select the “Slope” menu, and note how the waveform changes when you switch between the positive and negative slope settings (from the softkeys on the side of the display). Pay attention to the shape of the waveform right at the “T” marker.
5. Select the “Mode” menu, and make sure the mode is set to “Auto (Untriggered Roll)”. Turn the LEVEL knob clockwise until the arrow marker denoting the trigger level is well above the sine wave. You should see the trace lose stability and drift horizontally. Now switch the mode to “Normal”. The trace should freeze. When you adjust the level back down, the trace will become active again once the scope senses a valid trigger.
6. Select the “Source” menu, and select “Ch 2”. Notice how the trigger markers change to the second trace. Now disconnect the sync signal going into CH2 and plug it into the EXT TRIG input. You should lose triggering first, but recover it when you select “Ext” from the source menu.

Exercise 1: Making measurements with a scope

Now that we have reviewed the basics, you should be able to make the following setup: Use the generator to create a 2100 Hz triangle wave of approximately 2.4 volts peak-to-peak amplitude. Feed this signal into CH1 of the digital scope and get a stable trace showing this signal. You should see something similar to Fig. 3. When you make this setup, think about what kind of trigger settings you need to make a stable trace.

Now, measure the amplitude and frequency of this signal by three different methods:

- **Use the graticule.** This method works with any oscilloscope. Compare your waveform to the markings on the screen and use them along with the horizontal and vertical settings

to calculate the amplitude and frequency. For example, in Fig. 3 the vertical peak-to-peak extent of the waveform is a little less than 5 large divisions (more accurately, about 4.9 large divisions), and the vertical sensitivity is 500mV/div, so the peak-to-peak amplitude is $4.9 \times 500\text{mV} = 2.45\text{V}$. The horizontal extent corresponding to one full period is a bit less than 4.8 large divisions, and the sweep rate is $100\mu\text{s}/\text{div}$, so the period is $4.8 \times 100\mu\text{s} = 480\mu\text{s}$, which gives a frequency of $1/(480 \times 10^{-6}\text{s}) = 2080\text{ Hz}$.

- **Use the cursors.** The digital scopes (and some advanced analog scopes) have cursor markers that can be used to ease the challenge of converting graticule distances to time and voltage units. To access the cursors on our digital scopes, press the CURSOR button near the upper right of the front panel. Notice that you can select either “H bars” (horizontal bars: voltage only), “V bars” (vertical bars: time only) or “Paired” (both time and voltage). The cursor positions are controlled by the unmarked “General Purpose Knob” that is linked to the TOGGLE button; see Fig. 1 to help you locate it. Choose one of the cursor options, and see what happens when you turn the General Purpose Knob and press the TOGGLE button. Notice also the appearance of the Δ and @ symbols at the right side of the screen. The Δ symbol denotes the difference between the cursors, while the @ symbol denotes the position of the active cursor (denoted by the solid line) relative to the 0 volts indicator for voltage or the trigger point for time.

In the example in Fig. 3, the cursors shown are the “paired” type that are positioned to measure the peak-to-peak voltage and one-half of the period. The Δ indicators (at the top right) show a voltage difference of 2.44 V and a time difference of $238\mu\text{s}$ (which would imply a period of $476\mu\text{s}$ or frequency of 2100 Hz).

Use the cursors to measure the period and peak-to-peak amplitude of your waveform, and compare these numbers with what you got from the graticule examination method.

- **Use the automated measurement function.** If you are using a digital scope *and* you have a very uniform stable waveform, like a sine, triangle or square wave, you can use the easiest method of all: automated measurements. Press the MEASURE button in the upper part of the scope’s panel (near the CURSOR button). Then press (if not already highlighted) the “Select” softkey. You should see a wide selection of measurement possibilities from the menu at the right side of the screen. By scrolling through this menu, you should be able to select a frequency measurement and an amplitude measurement that will be applied to the active channel. Try it, see what you get, and compare the results with those from the previous two methods.

In the example of Fig. 3, automated measurements of “Ch1 Period”, “Ch1 Freq” and “Ch1 Ampl” have been applied. You can see that the automated measurements give a peak-to-peak amplitude of 2.48 V and frequency of 2.101kHz. These compare well with the cursor results.

Exercise 2: Measurement of an RC time constant

In this exercise you will use some of the measurement methods you learned above to find the time constant of a simple resistor-capacitor, or *RC*, circuit. You will also learn a few more tricks you can do with the digital scope.

An *RC* “low-pass” filter circuit is shown in Fig. 4. Circuit theory shows that if the circuit is fed a step input (or low frequency square wave) that the output will show an *RC* charging curve that

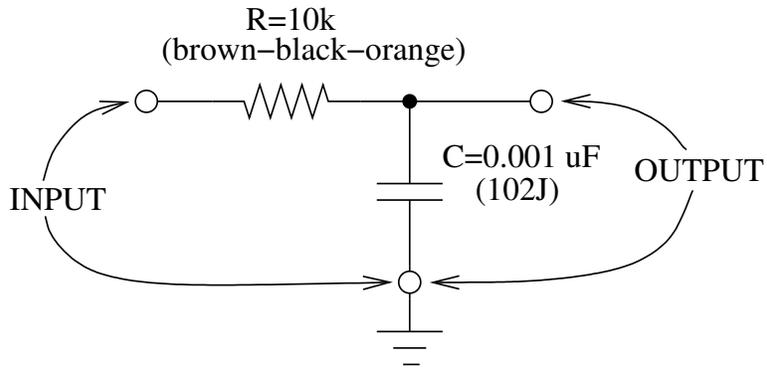


Figure 4: Schematic of the RC circuit used in Exercise 2.

approaches the limiting DC voltage exponentially with a time constant of RC . In other words, if the initial voltage across the capacitor is some positive voltage V_0 and then the input drops to zero volts, the capacitor voltage V_C will decay according to

$$V_C = V_0 e^{-t/RC} . \quad (1)$$

If the input were not a step or low-frequency square wave but a sine wave, the ratio of the output amplitude V_{out} to the input amplitude V_{in} would depend on the frequency f according to

$$\frac{V_{\text{out}}}{V_{\text{in}}} = \frac{1}{\sqrt{1 + (2\pi fRC)^2}} . \quad (2)$$

We can obtain the RC constant in two ways: either by measuring the “half-life” of the decay, or by measuring the frequency at which the output amplitude drops by a specific amount. In the following, we’ll use the digital scope to try both methods.

1. Build the RC circuit shown in Fig. 4 on the breadboard using the parts in the plastic box. *If you have never used a breadboard, or are otherwise confused about how to do this, please ask for help.*
2. Attach a BNC tee to the main output of the function generator. Then connect a BNC cable between the one end of tee and CH1 of the scope. Connect another BNC cable to the other end of the tee, and attach a BNC to banana-plug adapter to it, and plug it into a pair of banana terminals on the breadboard.
3. Use some short jumper wires to connect the banana terminals to the input of the RC circuit. The “input” is shown in Fig. 4.
4. With a second BNC to banana-plug adaptor and jumper-wire arrangement, connect the output of the RC circuit to CH2 of the scope (see Fig. 4).
5. Set up the function generator to produce a 5000 Hz square wave of 8 volts peak-to-peak amplitude, and look at both the input and output signals of the RC circuit on the digital scope. You should see something like Fig. 5a. Notice that triggering is on CH1, with a negative slope.

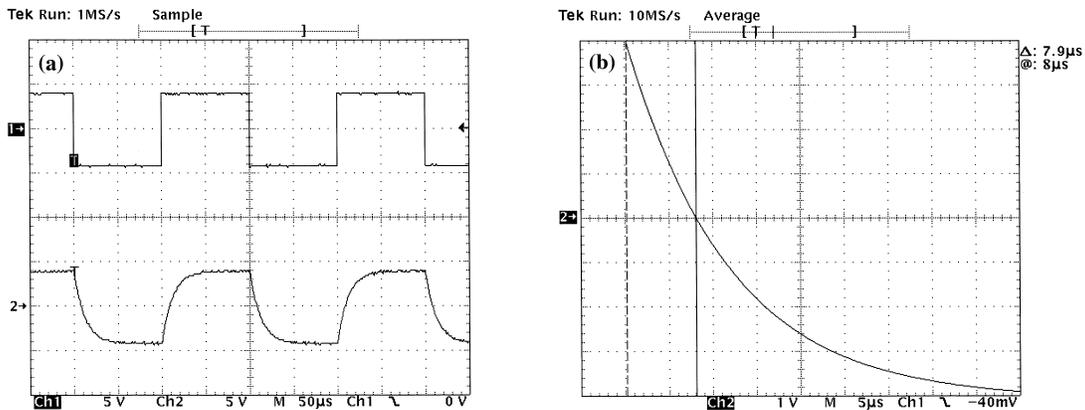


Figure 5: Digital scope measurements of RC decay. (a) Display showing both the input square wave (CH1) and the output waveform that is rounded due to RC charging (CH2). (b) Expanded view of the signal in CH2 showing the cursors placed to measure one half-life time of the decay ($7.9\mu\text{s}$).

- Next, turn CH1 off, and use the SEC/DIV knob, the VOLTS/DIV knob and the horizontal and vertical POSITION knobs to expand the amount of screen space taken up by a downward-going part of the RC -circuit output waveform, as is shown in Fig. 5b. Note that the vertical sensitivity on CH2 is 1 V/div, so that an 8 volt peak-to-peak signal takes up the full vertical range of the screen.
- Now you should be ready to measure the decay “half-life”. The half-life time $T_{1/2}$ is defined as the amount of time it takes for the signal to drop by one-half of wherever it was when you started the measurement. So if the full peak-to-peak value is 8 volts, and we start $t = 0$ at the beginning of its drop from +4 volts towards -4 volts it will drop by 4 volts (to 0 volts) at $t = T_{1/2}$, and then it will drop by another 2 volts (to -2 volts) at $t = 2T_{1/2}$, and then by another 1 volt (to -3 volts) at $t = 3T_{1/2}$. These points correspond well with the grid markings on the scope. As shown in Fig. 5b, the cursors are positioned to measure the first $T_{1/2}$ interval of $7.9\mu\text{s}$. Use the cursors to measure $T_{1/2}$ for your circuit.
- From Eq. (1), it is easy to show that $RC = T_{1/2}/\ln 2$. Calculate RC for your circuit from your measurement, and compare it to the value you expect from the part values. In the example shown in Fig. 5b, we would find that $RC = 7.9/0.693 = 11.4\mu\text{s}$. This is close to the expected value of $10\mu\text{s}$ ($10\text{k}\Omega \times 0.001\mu\text{F}$), if we remember that capacitor tolerances are typically 10%–20% and resistor tolerances are 5%.

To try the second method of comparing the input to the output using sine waves and Eq. (2), note that when $2\pi fRC = 1$, $V_{\text{out}}/V_{\text{in}} = 1/\sqrt{2} = 0.707$.

- Set the function generator to produce a sine wave of 10 volts peak-to-peak amplitude.
- Turn on both CH1 and CH2 so that you can see the input and the output.
- Set up automated measurements to make the following: CH1 peak-to-peak amplitude, CH2 peak-to-peak amplitude, CH1 frequency and/or period. You should see that the CH1 amplitude measurement gives close to 10 volts.

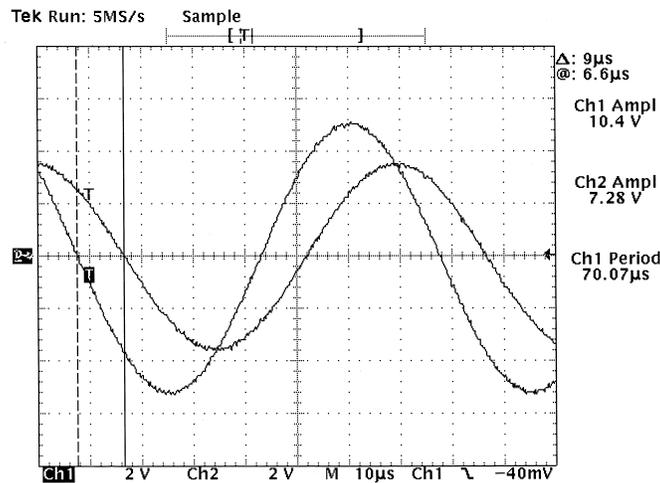


Figure 6: Digital scope measurements of sine wave response of an RC circuit. The larger waveform is the signal at the input of the RC circuit, and the smaller waveform is the signal at the output. The frequency of the sine wave is set so that the ratio of the output to input amplitudes is about 0.7.

4. Position the 0 volt markers of both channels at the horizontal centerline of the display, so that each trace is symmetric about the center.
5. Now turn the frequency knob up until you see the CH2 amplitude drop to (about) 7.07 volts. This is the frequency at which $2\pi fRC \approx 1$. This is the condition that is shown in Fig. 6.
6. From this frequency, calculate the value of RC . In the example of Fig. 6, the period of $70.07\mu\text{s}$ corresponds to a frequency of $14,270\text{ Hz}$, which gives $RC = 1/(2\pi \times 14,270)\text{s}$ or $11.2\mu\text{s}$, which is close to the value obtained through the half-life measurement.
7. AC circuit theory also predicts that at this frequency there should be a phase shift between the input and output of 45° . You can measure this with the cursors. First use the cursors to measure the points at which the two waveforms cross the 0 volt line. Then use the cursors (or automated measurement) to find the full period of the sine wave. The ratio of these two values times 360 gives the phase shift in degrees. From Fig. 6, one obtains a phase shift of $9\mu\text{s}/70\mu\text{s} \times 360 = 46^\circ$. What do you get for your circuit?

Exercise 3: Signal averaging

You have seen that the cursor and automated measurement functions of the digital scopes can be very handy. But by far one of the most used features of these scopes in the advanced labs is their capability to perform *signal averaging*. If the signal that you want to measure is periodic but accompanied by a large amount of random noise (or noise that is periodic with a different frequency), you can reduce the noise by averaging together many sweep records; the repeated part of the signal will increase relative to the non-periodic part of the signal. The next exercise shows how this works.

1. Power up two generators, and connect the main outputs of each one to CH1 and CH2 of the

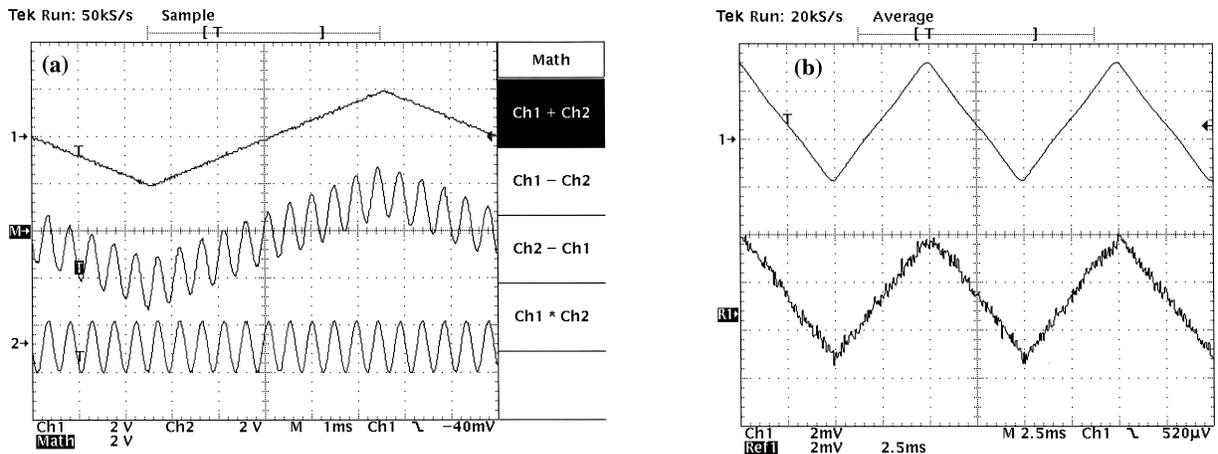


Figure 7: Signal averaging with the digital scope. (a) CH1 and CH2 show signals from two different signal generators. The center trace (M) is the sum of the two signals as indicated by the Math menu. (b) The lower trace shows a weak signal (note the 2mV/div sensitivity) with lots of noise on it. The upper trace shows the same signal averaged over many scans.

digital scope.

2. For CH1, set up a triangle wave of about 100 Hz and 4 volts peak-to-peak amplitude.
3. For CH2, set up a sine wave of a little more than 2100 Hz and about 2 volts peak-to-peak amplitude.
4. Set the digital scope so that you can see both signals easily. Then set up triggering so that its source is CH1. You should see that the signal in CH1 is stable, but that CH2 is not: there is no fixed relationship between CH1 and CH2.
5. Switch the triggering source to CH2, and notice the difference. Does it make sense?
6. Press the MATH button near the vertical channel controls, and select the “Ch1+Ch2” option. You should see something like Fig. 7a, where the center waveform shows the sum of CH1 and CH2.
7. Now press the ACQUIRE button, and select the “Average” option. The number of waveform records to average is controlled by the General Purpose Knob. Turn it and see what happens.
8. You should notice that if you trigger on CH1, the signal from CH2 is averaged out, leaving mostly CH1 in the Math waveform, but that if you trigger on CH2, you will average out the signal from CH1.
9. The signal averaging is most useful when it works on real noise. You can get a taste of this: Disconnect one of the generators and turn the amplitude of the other way down. Turn the sensitivity of the live channel on the scope up. (You may need to use the AUX output on the generator and the EXT input on the scope, with external triggering set up in order to get a stable trace). This is shown in Fig. 7b, where the lower trace shows a record saved from one sweep of CH1 into the memory location Ref1, and the upper trace is the same signal averaged over 256 sweep records.

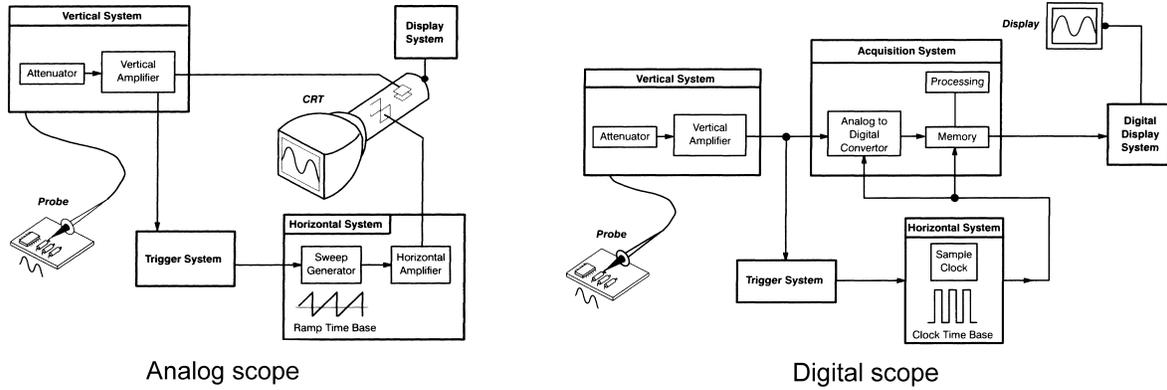


Figure 8: Block diagrams of analog and digital scopes. Taken from reference [1].

The analog oscilloscope

The original oscilloscope was the analog type, and although digital technology has largely replaced the older designs, analog scopes are still widely used. Since you will encounter both types in many physics labs, you should know something about how they work. Figure 8, which is taken from “The XYZ’s of Oscilloscopes” [1], shows functional diagrams of the two types of oscilloscopes for comparison.

Both analog and digital scopes have certain circuit blocks in common: the vertical system, the trigger system, and the horizontal system. In the digital scope there is another component, the acquisition system, that converts the signal to digital form and processes it before passing the result to the display system. The acquisition system is where signal averaging and automated measurements are performed.

Analog scopes, which do not have a digital acquisition circuit block (although some do), send the horizontal and vertical signals directly to the CRT display. The vertical signal is applied to metal plates within the CRT that displace the electron beam vertically. The horizontal signal, which is configured to have a repeating “ramp” waveform, causes the electron beam to scan horizontally across the screen at a rate equal to the ramp frequency.

It should be evident that the electronics of analog oscilloscopes are simpler than those of digital scopes; there is considerably less hardware between the input signal and the displayed picture. In an analog scope, the image you see is the accumulated effect of many repeated measurements, each displayed once; whereas the image you see from a digital scope is from more infrequent measurements that are captured and stored by the acquisition system.

Thus, an analog scope may be a better choice when you want to observe a rapid signal that may change over time or is itself aperiodic. An important example of such a signal is the output of a particle detector, such as the proportional tube in the Mössbauer spectroscopy experiment. The detector signal may be described as a sequence of randomly occurring but well-shaped pulses whose heights are proportional to the energy of the detected particle. When this signal is fed to an analog scope, the pulses become superimposed on each other, with the more frequent particle energies appearing brighter than the less frequent energies; the overall picture is a collection of overlapping peak traces seen as “bands”, where important particle energies can be easily distinguished by their brightness. The same signal fed into an a digital scope produces only a single trace that appears

to fluctuate randomly; since the time taken to process a pulse signal and display it is much longer than the time between successive pulses, it is very difficult for the user to spot which pulse heights are most frequent.

Exercise 4: Observe pulse signals on an analog scope

There should be one or more experiments set up that use a particle detector in the apparatus. Examples are Mössbauer spectroscopy, X-ray fluorescence, and the Hanle effect. Your instructor will show you which one is to be used for this exercise.

Note: since there is only one setup for this part, if you find that it is being used, move on to the next exercise and return to this one later when the apparatus is free. This exercise is short.

1. Locate the apparatus and observe the display showing the pulse signal. In particular, you should be able to see that some pulse heights are much more common than others. If the pulses are very rapid, there should be one or more notable bright bands on the screen.
2. Turn the SEC/DIV knob down to around 0.1 ms/div, and notice that the signal is made up of randomly occurring pulses of varying height. You cannot see any periodicity in the pulse signal, which is due to the detection of randomly occurring radioactive decays.
3. Turn the SEC/DIV knob back down to around 2 μ s/div. Make sure that the trigger mode is set to NORM. Now vary the trigger level knob, and note the effect. You should see that when the knob is set high, only the larger peaks are displayed; as the knob is turned down, the smaller peaks appear. If the knob is turned too low, the trace will disappear completely.

Exercise 5: Oscilloscope probes

When using an oscilloscope to study or troubleshoot an electronic circuit, one normally attaches an oscilloscope probe to the input channel. *A scope probe is not just a piece of wire attached to a special clip!* It has additional circuitry inside it to increase the input impedance as seen by the circuit under study in order to minimize the effect of the scope on that circuit. In most cases, one would use a “times-ten” (10X) probe. This probe increases the impedance by a factor of ten, but it also reduces the signal measured by the scope by the same amount. However, except when the signal is very small, this reduction can be overcome by the channel amplifier.

This exercise will introduce you to a scope probe and the important operation of “probe compensation”.

Basic operation

1. Turn on the analog scope at your station. Set the vertical and horizontal position knobs to 12 O'clock, the CH1 and CH2 input coupling switches to DC, and the VOLTS/DIV knobs to “1” relative to the “1X” marker. Set the SEC/DIV knob to 2 ms.
2. Attach a BNC cable between a function generator output and the input of CH1 on the analog oscilloscope. Set up a sine wave signal of 2 volts peak-to-peak at a frequency of 100 Hz. Verify the signal by using the display graticule on the scope. Set the triggering to CH1 with the

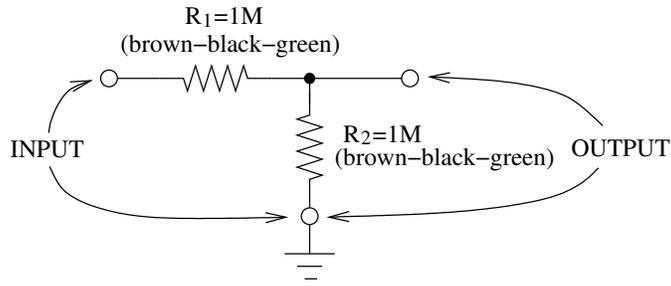


Figure 9: Circuit diagram of a high-impedance divider.

source switch to INT. Make sure that the Horizontal Mode switch is set to A (called NO DLY on the 2213A). Adjust the trigger controls (A TRIGGER) until you see a stable trace. If you have trouble triggering the analog scope, please ask for help.

3. Attach a 10X probe to CH2 of the analog scope and turn the vertical mode to BOTH and CHOP. You should see two traces. If you only see one, try adjusting the vertical position knobs; one of the traces may be off screen.
4. Set both channel sensitivities to be the same, for example, 1 volt/div, as measured from the “1X” marker on the dial.
5. Now disconnect the BNC cable from the function generator output, and connect a BNC tee between the cable and the function generator. After doing this, you should still see the signal on CH1.
6. The open end of the tee will give you access to the signal that you can measure with the probe. Touch the end of the probe clip to the center pin of the BNC tee. (You may prefer to pull the clip end off of the probe and insert the exposed pin into the BNC center pin.)
7. Compare the signal measured by CH2 (the probe channel) to that in CH1. Now set the knob for CH2 so that it shows the same amplitude waveform on the screen as CH1. Note the setting of the CH2 knob compared to CH1; you should see that they *are* set the same if you use the reference marker for 10X on CH2, but 1X on CH1.

Oscilloscope “loading”

To get a sense of why one would use a 10X oscilloscope probe, we will turn now to the circuit in Fig. 9. This is a simple voltage divider made with high-resistance resistors. If the input voltage to the circuit is V_{in} , the output voltage is given by

$$V_{out} = V_{in} \frac{R_2}{R_1 + R_2}, \quad (3)$$

where R_1 and R_2 are as shown in Fig. 9. (Can you prove this equation? Try it!) In the following steps, you will make measurements to verify this equation.

1. Construct the divider circuit on the breadboard. (It is very similar to the circuit in Fig. 4.)

2. Set up a 1000 Hz sine wave of 2 volts peak-to-peak amplitude on the function generator and feed this signal into CH1 of the scope and into the input of the divider circuit using a BNC cable, BNC-to-banana adaptor and jumper wires, as you did in the RC circuit exercise.
3. Set up triggering on CH1 and observe the signal. This is V_{in} .
4. Connect CH2 of the analog scope to the output of the divider circuit with another BNC cable, adaptor and jumper wires. This gives V_{out} .
5. Turn both channels on (DC coupled), and use the vertical controls along with the graticule to measure V_{in} and V_{out} .
6. From the measurements, calculate the ratio V_{out}/V_{in} and compare this with what you expect from the formula, Eq. (3).
7. The agreement between measurement and theory should have been poor. To figure out why, look very closely at the small print next to the CH1 and CH2 input connectors on the oscilloscope. What do you see? Does it make sense? Ask for help if you do not understand.
8. With the BNC cable to CH2 still attached, turn the *frequency* of the generator up to around 15 kHz, but leave the amplitude the same. What happens?
9. Now repeat the above measurements, but replace the BNC cable connected to CH2 with a 10X probe. You will need to reset the VOLTS/DIV knob on CH2 according to the 10X probe. Explain to yourself and your partners why the result is different, and indeed, better.

What you have just seen are the effects of oscilloscope “loading”, where the input impedance of the scope becomes significant in affecting the measurement. One should always be aware of the impedance of the circuit under study, or at least remember that there may be times when one cannot ignore the the oscilloscope or other measuring instruments. Question: why did the input impedance of the oscilloscope not affect the signal seen on CH1?

Probe compensation

You may have noticed that the text near the input of the oscilloscope states both a resistance and a capacitance: on the 2215A model this is $1M\Omega$ and $20pF$. These are the effective resistance and capacitance to ground seen by a circuit attached to the input. Because of the capacitance, the impedance of the scope input is a function of the input signal’s frequency. It drops from its DC value of $1M\Omega$ as the frequency increases, and by the time the signal reaches the nominal maximum of 60 MHz, the input impedance is only 133 ohms! That is why you saw such a large drop in the amplitude in the above exercise as you turned the frequency up.

The capacitance of the scope input means that a 10X probe cannot be made by a resistor alone, there must also be some carefully chosen probe capacitance, so that the fraction of the signal that the scope measures (1/10) is constant over the frequency span of the scope. Specifically, the capacitance of the probe must be set so that the capacitive reactance $X_C = 1/(2\pi fC)$ of the probe is in the same proportion to the capacitive reactance of the scope as the resistance of the probe is to the resistance of the scope. The resistance of the probe is $9M\Omega$, which, when combined with the resistance of the scope makes a 1/10 voltage divider with an input resistance of $10M\Omega$. Thus the capacitance of the probe must be set to 1/9th the capacitance of the scope input. However, it

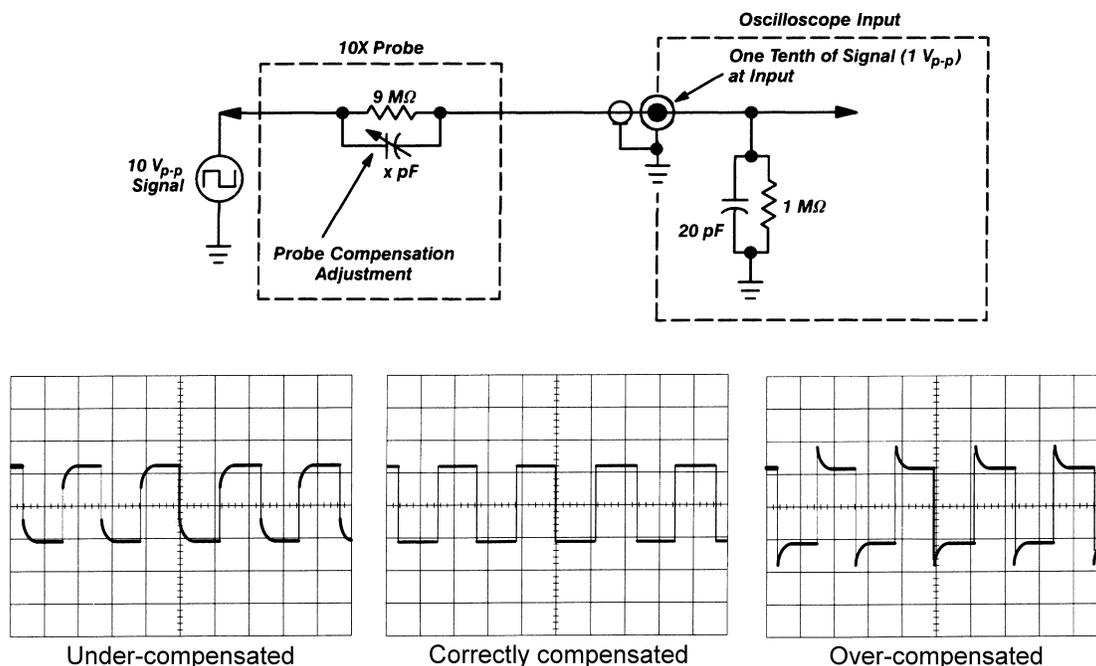


Figure 10: Top: Effective circuit of an oscilloscope input and associated 10X probe. Bottom: Typical waveforms seen during the adjustment of probe compensation. The probe is correctly compensated when the waveform is as square-shaped as possible. Figures taken from reference [1].

is hard to make accurate capacitors of very small values, and the extra capacitance of the probe clip and cable may vary according to manufacture and use, so the probe manufacturer makes this capacitance adjustable.

Figure 10 shows the equivalent circuit of a scope probe and scope input. The action of adjusting the variable capacitance is called setting the “probe compensation”. You should *always* check the probe compensation before making any measurements of signals above a few kilohertz frequency, and especially so if you want those measurements to be as accurate as possible for any frequency signal. Since probe compensation is so important, the scope manufacturers make it easy by providing a test signal we can use to check and set the probe capacitor. Let’s try it.

1. Locate the small screwdriver or tiny plastic adjusting tool in your parts box. Also locate a small gold pin.
2. Connect the 10X probe to one of the channels of the analog scope, and set the trigger controls to AUTO and INT, with the corresponding channel selected as the trigger source.
3. Insert the gold pin into the small PROBE ADJUST test point near the scope display. Attach the probe to the pin.
4. You should now see a square wave signal displayed on the screen. If you don’t, double check the settings of the controls to make sure that they are consistent with the stated test signal of 500 mV peak-to-peak at 1 kHz. As usual, ask for help if you have trouble.

5. Now, find the small hole with a screw adjustment somewhere on the probe. Some probes have the adjustment in the body of the clip, and some have the adjustment in a small box near the BNC connector.
6. *Carefully* insert the adjusting tool into the hole and turn the screw head slowly while watching the waveform shape on the display. Compare what you see with the pictures in Fig. 10.
7. After exploring the range of adjustment available and its effects, carefully set the screw so that waveform shape is as square as possible. The probe is now correctly compensated.

Finishing up

When you are done with your explorations, please put all parts back where you found them:

- Resistors, capacitors, connecting (jumper) wires, and BNC-to-banana adaptors go back in the plastic box.
- BNC cables should be put back on the rack *in the part of the rack that has similar lengths*. Please remove any adaptors that remain on the plugs and put them away.
- Turn off all electronics.

References

- [1] *The XYZ's of Oscilloscopes*, Tektronix, Inc., Beaverton, OR (1992). This book gives a very thorough introduction to oscilloscopes and their use. A copy is available online, and paper copies can be inspected the lab.
- [2] *Tektronix 2215A Oscilloscope Operators Instruction Manual*, Tektronix, Inc., Beaverton, OR (1984). Operation and specifications of one of the most common analog scopes in our lab. Available for inspection by request.
- [3] *Tektronix TDS310, TDS320 & TDS350 Two Channel Oscilloscopes Instruction Manual*, Tektronix, Inc., Beaverton, OR (1994). Operation and specifications of our most common digital oscilloscope. Available online (large PDF) or by request.
- [4] *Wavetek Model 29 Operator's Manual*, Wavetek, Ltd., Norwich, UK (1997). A portion of this manual giving the basic operation is available online.
- [5] *Model DS345 Synthesized Function Generator*, Stanford Research Systems, Sunnyvale, CA (1995). Operations and programming manual for the DS345. Available online.
- [6] *The Art of Electronics, 2nd edition*, by Paul Horowitz and Winfield Hill, Cambridge University Press, New York (1989). Authoritative and accessible electronics text. Appendix A gives an overview of analog oscilloscopes.

Prepared by D. Pengra
scope_ex.tex -- Updated 26 September 2007