Ph 3 - INTRODUCTORY PHYSICS LABORATORY
– California Institute of Technology –

Electronics Test Equipment and Digital Data Acquisition

1 Introduction

The primary purpose of this first lab is for you to learn about the oscilloscope and waveform generator, which are widely used tools in the category of electronics “test equipment.” Test equipment is mainly used for debugging electronic devices during development, or figuring out how to fix electronic devices when they break. An overarching goal of this course (along with the other laboratory courses in physics) is that you should be able to walk into any modern physics research lab and have a reasonable level of familiarity and proficiency with the electronic equipment and measurement techniques being used.

The *signal generator* (also called a *function generator* or *waveform generator*) does what the name suggests – it generates electronic signals that can be written as $V(t)$, the voltage as a function of time. Examples include sinusoidal signals $V(t) = A\cos(\omega t)$, square-wave signals, pulse signals, triangle waves, and a host of others, including “noise” waveforms (where $V(t)$ looks like electronic noise with tunable properties). Figure 1 shows the waveform generator you will be using in this lab.

The *oscilloscope* is used to examine electronic signals by plotting $V(t)$.The oscilloscope provides many useful ways to plot and analyze electronic signals, with quite a few knobs, buttons, menus, and sub-menus included in its user interface. Oscilloscopes range in price from a few hundred dollars (for basic low-frequency models) to several hundred thousand dollars (for fancy models capable of viewing and analyzing very small, very fast signals). They all operate on similar principles, so once you learn one you will have little trouble using others (although high-end oscilloscopes typically have many additional complex features to learn).

Figure 2 shows the oscilloscope you will be using in Ph3. It takes some time to become familiar with the oscilloscope and signal generator in the lab, but they soon become indispensable tools for quickly and easily examining just about anything electronic. This first lab is a basic test-equipment tutorial, and in subsequent labs you will get more practice using these devices incorporated into several physics experiments.

Figure 1. The Rigol DG1022Z waveform generator.

Figure 2. The Keysight EDUX1052A oscilloscope.

1.1 What to write in your lab notebook

Reproducibility is essential in science, and reproducibility requires good record keeping. In this course, you need to keep a lab notebook that describes what you did in each lab, and what results you obtained. You will turn in your notebook about every two weeks (the schedule is on Canvas), and its contents will be used to determine your grade for each lab.

In modern science and industry, hundreds or even thousands of people might be working on different aspects of a project simultaneously, so it becomes extremely important to record exactly who did what when. The same goes for writing large computer codes, as you can easily imagine. In these situations, personal lab notebooks are being superseded by elaborate software tools, and large scientific or industrial groups often have strict rules and protocols. We are not so rigid in Ph3, but you should get into the habit of keeping good records in the lab.

Here are some things you should write in your Ph3 lab notebook (see Canvas for more on this subject):

- Include apparatus diagrams as appropriate, including electrical connections, optical layouts, etc. These can be drawings, photographs, or schematics copied from the handouts. Include enough information that another person could reconstruct the apparatus from your notes.
- Record all the settings from the oscilloscope, function generator, etc., even if you do not change them. Note that a convenient way to include many important oscilloscope settings is to print a screenshot and tape it into your lab notebook.
- If the handout instructs you to view some signal on the oscilloscope, include a screenshot in your notebook, along with a suitable description. Please make sure that your handwriting is *easily* legible, and that each image/graph/result includes at least a brief explanation or caption.
- Include photographs as appropriate. Your camera phone is a terrific laboratory tool, so use it to record your overall experimental set-up or other details. Computers are available in the Ph3 lab for printing your photos, so you can tape them into your notebook.
- Include all data taking and analysis steps, especially graphs taped into your notebook.

Beyond these guidelines, use your best judgement to decide what to put in your notebook. If the handout asks you to perform a specific task, record it (via a screenshot, data plot, etc.) in your notebook for grading. If you tape a figure or photo into your notebook, include a caption alongside that clearly describes what the image depicts. You can find an example of a well-kept Ph3 lab notebook on Canvas.

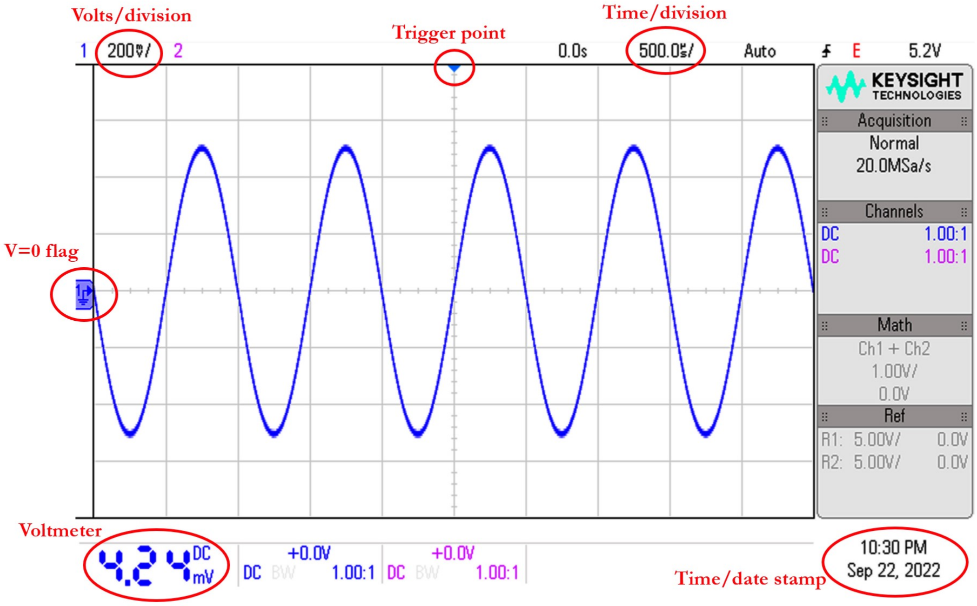


Figure 3. A typical screenshot of the Keysight EDUX1052A oscilloscope. Several important settings are circled in red, as they appear on the live 'scope screen.

2 Lab Procedures – Week One

What follows are step-by-step instructions for this lab. Each paragraph has a small task, and in series they will guide you through the lab. As with most of the Ph3 labs, there are a lot of steps, so you will need to work efficiently. If you find that one step is taking quite a bit of time, ask for help. Often many experimental parameters must be set correctly to obtain a desired result, and it can be easy to overlook something.

- To get started, turn on the power to your signal generator and oscilloscope. Note that the signal generator has two primary output channels, and the oscilloscope has two primary input channels. While the instruments boot up, connect the Ch1 output of the waveform generator to the Ch1 input of the oscilloscope using a BNC cable. (If you are not familiar with BNC cables, they have "bayonet" connectors on each end that you push on and then give a quarter turn to lock. Ask if you need help.)

- Press the Ch1 *Output* button on the waveform generator, which will light up, indicating that a signal is being applied to the output. Then press the white *Auto Scale* button on the oscilloscope. You should see a yellow sine wave plotted on the 'scope. If you also see a green line, press the green illuminated "2" button on the 'scope to turn this channel off, and then press the *Auto Scale* button again. As always, just ask if this does not seem to be working, or if you have any questions. Do not record any of this in your notebook just yet; for now you just need to become familiar the various buttons and features of the instruments.

- Turn the large *Horizontal scale* knob and see what it does. At the top of the 'scope screen you will see something like "100 μs/" (see Figure 3), telling you that the oscilloscope is plotting the waveform at 100 microseconds per division. One division is about one centimeter on the oscilloscope screen, delineated by vertical lines, with a total of 10 divisions being displayed. As you turn the Horizontal knob, note that the 'scope uses a 1-2-5-10 sequence for the time/division. This is fairly standard for a logarithmic sequence, giving three knob clicks per decade.

- Press the Horizontal scale knob and you will toggle the Coarse/Fine modes. The Coarse mode gives three clicks per decade, while Fine mode gives 80 clicks per decade. If you ever find yourself cranking away on this knob, check the time/division setting; you probably want to toggle to Coarse mode. Go back to Coarse mode now, but remember that the Fine mode exists. Also, many of the knobs on this 'scope can be pressed for additional functionality.

- On the signal generator, note that the Ch1 frequency is set to 1 kHz (i.e., the frequency in kilohertz) on the display.

This number has a lot of digits (1.000,000,000 kHz in this case) because the frequency can be set to a resolution of 1 μHz with an absolute accuracy of about 100 parts-per-million (ppm) on this instrument. If you set the 'scope to 1 msec/div, you should see that a single cycle fills one horizontal division. Makes sense.

- There are several ways to set the frequency on the signal generator: 1) Enter a number on the keypad and choose the units; 2) Enter the period of the waveform (by toggling the *Freq/Period* button to select which you want); and 3) Turn the large knob on the instrument and use the arrow keys below it to select which digit you want to change. Try these and see that they all work as advertised.
- Now adjust the large *Vertical scale* knob (also called the *Vertical gain* knob), which changes an amplifier gain setting inside the oscilloscope. You will see something like "1.00V/" on the upper left corner of the display. This tells you the "volts per division", where eight divisions are displayed on the oscilloscope screen. On the waveform generator, you probably see a signal amplitude of "5.000 Vpp", meaning "5 volts peak-to-peak." Change the signal amplitude to 6 Vpp and confirm that the signal fills six vertical divisions when the gain is set to 1 V/division. (If this does not the case, press the "1" button to bring up the *Channel 1* menu, then press *Probe* and set the probe to 1:1 using the *Entry* button. If something is still off, please ask someone.) Note that the Vertical scale knob uses the same 1-2-5-10 logarithmic sequence.
- Next try the *Vertical position* knob below the Vertical scale knob (labeled with "up/down" symbols). You should see a yellow "flag" on the left side of the screen that moves as you turn the Vertical position knob. This is a most useful flag, as it shows you the position of zero volts on the oscilloscope screen. The yellow flag is for Ch1, and later you will see there is also a green flag for Ch2. When you turn the Vertical position knob, the numerical voltage offset is displayed on the screen. Press the knob and this value resets to 0V.
- Try the *Horizontal position* knob and you will see the same general behavior. People have been building digital oscilloscopes for many years, so the user interface is pretty intuitive (at least for the basic features).
- When working with electronics test equipment, you will often encounter the acronyms AC and DC. These originally stood for "Alternating Current" and "Direct Current," but the acronyms have evolved to broader meanings. In the electronics world, DC typically means "the very-low-frequency part of the signal" And AC typically means "the signal that remains after subtracting the DC part." Now "low" is a subjective term, so to quantify it one talks about a "crossover" frequency $f_{crossover}$. The signal components below $f_{crossover}$ make up the DC part, while signal components above $f_{crossover}$ make up the AC part. More about all that later.
- Now that you have a signal on the 'scope, try changing from a sine wave to a square wave, ramp wave, pulse wave on the signal generator. For the ramp signal, try changing the symmetry to see what happens (no need to record any of this in your lab notebook). For the pulse signal, try changing the duty cycle. After a quick explore of these signal features, go back to a sine-wave signal for the next steps.
- Also, a few quick words about cables. The BNC cables we use are "coaxial" cables, with one inner conductor surrounded by a cylindrical outer conductor, as shown in Figure 4. The outer conductor typically has a braided wire construction, and it is held at zero volts, also called "ground". (Ground is a voltage reference point, often connected to the third prong of the electrical plug, which is connected to pipes in the building that include water pipes that are literally connected to the ground.) The outer conductor acts as an electrical shield to reduce noise on the central "signal" line. The coaxial cable also acts as a "transmission line" for high-frequency signals, but we will not get into that in Ph3. BNC is an acronym that once stood for something, but the internet tells us that no one is quite sure anymore what it stood for; so it has become just BNC.

2.1 Measurements

Once you have a signal displayed on the oscilloscope, there are many ways you can measure the signal amplitudes, frequencies, and other properties.

- To begin, press the *Meas* button to bring up the *Measurement menu*, so indicated on the right side of the screen. To get a fresh start, press the *Clear Meas* button and then *Clear All*. Note that when a menu button takes you to a sub-menu, the Back button will take you back one menu level.
- In the Measurement menu, the *Source* refers to the signal being measured, and this should be Ch1 at this point. Press the *Type* button and you will see many possible measurement options. Use the *Entry* knob to select *Peak-*

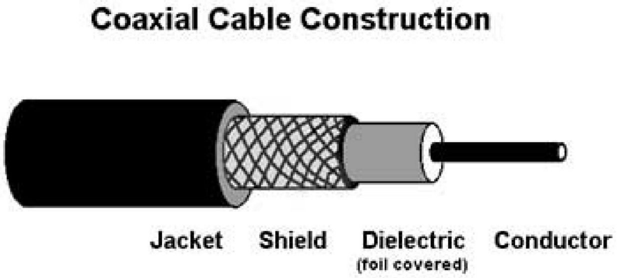


Figure 4. Typical construction of a coaxial cable.

Peak, press *Add Measurement*, and you should see the measurement appear on the screen. This measurement corresponds to Vpp on the signal generator, so these numbers should be about the same.

- You probably see some orange horizontal lines on the screen that indicate the measurement being made. The lines usually bounce around a bit because of voltage noise on the signal. The Peak-Peak measurement includes this noise, so the measurement is probably a few percent higher than what you set on the signal generator.
- Next add a frequency measurement and you will see both measurements on the screen (there is room for up to four). Note that the 'scope only measures what it sees on the screen, so you will need at least a few wave cycles to make a frequency measurement. Try displaying less than one full wavelength on the screen (using the Horizontal knob) and you will see that the frequency measurement does not work. Best to keep about ten cycles on the screen for measuring frequencies. Although the frequency measurement fluctuates, it should be accurate to about 0.1%. Turn up the vertical gain and you will see that the Pk-Pk measurement also does not work if the signal is too large to fit on the 'scope screen.
- With measurements, always remember that the 'scope only sees what you see. If you could not make a measurement using only the information you see on the screen, than neither can the 'scope.
- Next add an *AC RMS - N Cycles* measurement so you see all three measurements being displayed. RMS stands for the *root-mean-squared* average of the signal amplitude, averaged over one cycle:

  $$V_{RMS} = \left\langle V^2 \right\rangle^{1/2} = \left[\frac{1}{T_{cycle}} \int_{cycle} V(t)^2 dt \right]^{1/2}$$

  For a perfect sine wave, the peak-to-peak voltage is $2\sqrt{2}$ times the RMS average.
- Set the signal generator to 10mVrms at 1kHz, set the oscilloscope's Vertical Scale to 5mV/division so the signal fills the screen, and set the Horizontal Scale to 1msec/division so you have a good number of sine-wave periods on the screen. Record the 'scope measurements in your notebook. How far off (in percent) are both your amplitude measurements compared to what you expect from the Signal Generator? In general, an *RMS* measurement is more accurate than a *Pk-Pk* measurement, because the former involves an average that reduces the noise contribution. When you record your numbers in your notebook, be sure to include a brief description of what you are doing. A sentence or two should be sufficient.
- Before changing any settings, press the *Quick Action* button on the ‘scope and a screenshot should appear on one of the Ph3 printers. (If *Quick Action* does not print, try pressing *Save/Recall* and then *Print*. If that also does not work, ask for help.) Print a copy for each lab partner, so each of you can tape it into their own lab notebook (there is a paper cutter near the printer, and tape all around the Ph3 lab). Again, be sure to add some kind of caption and/or explanation with your screenshot. Explain (briefly) that this exercise is about measuring the signal frequency and amplitude using the *Measure* feature. Note that the measurements appear in the screenshot, so this is a convenient way to record both the 'scope settings (See Figure 3) and a set of signal measurements.
- Just for fun, add an *Average/Cycle* measurement to the mix, and then add a DC offset of 2 mV on the signal generator. From the *Measurement* menu, select *Statistics* to view a table of measurement data. Press *Reset Statistics* and you will see the *Count* (on the right) count up from zero. For each of your measurements, you can

now see the Present-value (Current), Mean, Min, Max, and Standard Deviation. Observe that a single frequency measurement is accurate to about ± 1 Hz (the measured standard deviation), but the average is spot on. Both the 'scope and the signal generator have quartz crystal oscillators built in, and these are accurate to about 100ppm or better. Voltage RMS is not too bad, better than 1% in the mean. Print another screenshot for your notebook, now showing the measurement statistics.

- Note how the Statistics feature turns your oscilloscope into an accurate digital multimeter, giving you a time-averaged measurements and standard deviations. In particular, the *Average/Cycle* measurement can be used as a DC voltmeter even when there is no AC signal present. (Your signal generator, oddly, cannot produce a pure DC signal with zero AC component. Try it.)
- In general, the *Measure* features work well on high-amplitude, low-noise signals. However, the built-in 'scope measurements can sometimes yield erroneous results when the signal is weak or noisy.

2.2 Cursors

If the *Measurement* menu is not giving you good results (typically because the signal is too weak or noisy), all is not lost. Another option is to essentially "hold a ruler" up to the screen (the digital equivalent of that) using the *Cursors* menu.

- To see how this works, first remove all the measurements (just press the *Meas* button once or twice) to get them out of the way, set the signal generator to 1Vpp at 1kHz, and press the *Auto Scale* button to get you back to a clean display. From there, press the *Cursors* button to bring up the Cursors menu. Set the *Mode* to *Manual* to start. The small knob to the right of the *Cursors* button will move the vertical and horizontal cursor lines on the screen. There are four such lines, labeled X1, X2, Y1, and Y2. Press the move-the-cursors knob to select which line you want, and adjust things so all four lines (two vertical and two horizontal) are visible on the screen.
- Note that several numbers are now displayed at the bottom of the 'scope screen:

1) X1 and X2 – these are the time coordinates of the X1 and X2 cursor lines, relative to the trigger time (marked by the small triangle at the top of the screen; we'll look at what that triangle means below).

2) ΔX – the time difference between the two cursor lines.

3) $1/\Delta X$ – the frequency when the period equals ΔX.

4) Similar numbers for the Y cursors.

- Display a 1 kHz sine wave, 2 volts peak-to-peak, and position the four cursors to measure Vpp and frequency. Verify that the measurements agree reasonably well with expectations, then print a screenshot for your notebook. Once again you see that all the cursor numbers are included in the screenshot. Again, add some kind of caption and/or explanation with your screenshot, explaining that this exercise shows how to measure the signal frequency and amplitude using Cursors.
- Next set the Cursor Mode to *Track* and see what happens when you move the Cursors around. When you move X1, the Y1 cursor follows the waveform at the X1 point. This allows you to make the same measurement you just did, but you only have to position X1 and X2, while Y1 and Y2 take care of themselves. (No need to add another screenshot to your notebook.)
- In summary, you see that there are three ways to measure signal properties: 1) read directly from the grid divisions on the screen, 2) use the *Measure* feature, and 3) use the *Cursors*. The grid is good for quick estimates, the *Measure* feature is good for strong signals, and the *Cursors* are useful when *Measure* doesn't work well. To remove the cursor/measurement lines from the screen (just to clean up the display), press the *Meas* button once or twice.

2.3 Triggering

With a sine-wave signal on the 'scope, press *Trigger/Trigger Type/Source/External* to see what this signal really looks like. The signal generator puts out a simple sinusoidal signal, $V(t) = A\cos(\omega t)$, and the 'scope now shows this signal as it just whizzes by at high speed. The magic of an oscilloscope is its ability to display time-varying period signals as stationary functions, thus making such signals easy to view and analyze. This trick is accomplished using a mechanism called "triggering", and this one of the subtler aspects of using an oscilloscope. Go back to *Source/Ch1* to produce a properly triggered signal.

- To see how triggering works, press the *Trigger* button and turn the nearby *Trigger level* knob until the value displayed on the 'scope screen (upper right) equals zero volts. With this setting, the 'scope waits until it sees the signal go through zero volts, with a positive slope. This occurrence is called a "trigger" event. When the 'scope detects a trigger event, it immediately plots the signal forward and backward in time until it fills the screen. Then it stops plotting and waits for the next trigger event. The triangle flag at the top of the screen shows you when the trigger event happened. You can move this flag using the *Horizontal Position* knob (to the right of the main *Horizontal* knob). Plotting forward and backward in time allows you to see the signal immediately before and after a trigger event, which can be useful. Of course, at 1 kHz the 'scope has to wait at most 1 msec to see the next trigger, so what you see is a continuously updated, stationary, sinusoidal signal. Indeed, you should see that the signal has zero volts and a positive slope at the trigger point.
- Next change the *Trigger Level* knob (above the Trigger button) and see what happens. Now you see another flag on the left side of the screen that shows you the trigger level, which is the voltage the 'scope is waiting for. The triangle flag still shows the position of the trigger event; it just happens at a different voltage now, not zero volts as it was previously. And you see the sine-wave signal move as the trigger level changes. You can also change the slope of the trigger event by pressing the *Trigger Type* button (in the *Trigger* menu) and selecting the *Slope*. Note that the slope changes at the trigger position.
- Is this is not making sense, talk it over with your lab partner, or ask someone. Triggering can be remarkably subtle at time, and you need to understand how it works if you want to use any oscilloscope effectively.
- Whenever you use an oscilloscope, remember this important operating point: if you are ever examining a nice, clean periodic signal (like a sinusoidal waveform), and you see it drifting across the oscilloscope screen (i.e., the waveform is not stationary), then you are doing it wrong! You need to adjust the trigger so the signal looks stationary on the screen. The *Auto Scale* button can be your friend here, and it usually works well if you have a strong signal plugged into Ch1. But *Auto Scale* is not a cure-all, so you also need to know your way around the Trigger menu.
- For another quick triggering example, adjust the *Trigger Level* knob until the trigger level is higher than any point on the sine wave. Now the 'scope never sees a trigger event (because it is waiting to see a high voltage that never happens), so it gives up and just plots something. In this case the signal is "untriggered" (or free-running). The 'scope is trying to "trigger off of Ch1" (i.e., use the Ch1 signal to find trigger events), but it is not working. Lower the *Trigger Level* again, and proper triggering is restored.
- You will also see a *Run/Stop* button on the oscilloscope, which simply toggles between *Run* (plotting the data every time a trigger event occurs) and *Stop* (leave the last plotted signal on the display and stop updating). Right below *Run/Stop* is *Single*, which tells the 'scope to wait for the next trigger event, plot a single trace, and then stop.
- Next use a second BNC cable to connect Ch2 on the waveform generator to Ch2 on the oscilloscope, Make sure the *Output* light is on for both channels. Press the "2" button on the oscilloscope to display both channels. Adjust the *Vertical* and *Vertical position* knobs so both the Ch1 (yellow) and Ch2 (green) traces are visible on the screen, although both might not be stationary at this point. Note that the light below the *Vertical* button shows which channel the scale and position buttons are addressing.
- Next set Ch1 to 1000 Hz and Ch2 to 1001 Hz, using the Ch1/Ch2 button to toggle between the two channels. Now you should see that Ch1 is stationary on the 'scope while Ch2 is slowly drifting across the display. That is because the 'scope is using trigger events found on the Ch1 signal to display both channels simultaneously. With a single trigger, however, both signals cannot appear stationary at once, because they have different frequencies.
- Now pull up the *Trigger* menu, and change the *Source* from Ch1 to Ch2. If what you see makes sense, then you probably understand triggering. If not, talk about it with your lab partner, or ask someone for help. Note that the Ch1 and Ch2 signals are roughly in phase every Δt seconds, where $\Delta t = 1/\Delta f$ and Δf is the frequency difference between Ch1 and Ch2. Try changing Δf to see this in action.

- But wait, we are not done with triggering yet. You can also use what is called an “external” trigger when you are looking at small signals. To see what this is, connect Ch2 on the oscilloscope to the *Ch1 Sync* connector on the back of the waveform generator. This might be a good time to hit the *Auto Scale* button to display both channels on the oscilloscope. You should see a 1 kHz sine wave on Ch1, plus a 1 kHz square wave on Ch2. The two signals are “synchronous”, so have the same frequency and are locked in phase.
- If you change the Ch1 signal amplitude on the waveform generator, then the Ch1 signal changes as you would expect, but the *Sync* signal amplitude is unchanged. Moreover it is a whopping big signal, varying from 0V to Vmax. Use the cursors to measure Vmax, and add this to your notebook
- Trigger using Ch1 (from the *Trigger* menu), and then set the Ch1 signal amplitude to 2mVpp. This is the lowest amplitude the waveform generator can produce. Turn the Ch1 *Vertical* way up on the ‘scope to view the signal. You will probably see that the ‘scope has trouble locking onto the waveform. That is because the amplitude is so low that the signal is noisy, so triggering events are hard to recognize. To fix this problem, switch to triggering using Ch2. Because the *Sync* signal is huge, triggering is easy, and both signals are now stationary.
- In a typical physics experiment, you might use the *Sync* signal to turn a digital switch on and off repeatedly, which in turn turns some signal on and off in the experiment. If this is a cutting-edge experiment, then the output signal is probably very weak. But you know that the signal should be synchronous with the *Sync* signal that drives it. By using the *Sync* to trigger the ‘scope along with the experiment, you can display a weak Ch1 signal easily, and you can average this signal to increase the signal-to-noise ratio.
- Now move the *Sync* signal from Ch2 on the ‘scope to the *Ext Trig* input, and change the trigger source to *Ext*. You may have to change the trigger level to a couple hundred mV to get the ‘scope to trigger properly, but again you should see a stationary Ch1 signal. This allows you to display weak signals on both Ch1 and Ch2, while using the third *Ext Trig* channel to trigger. (The *Ext Trig* input is essentially like a third channel, but it does less and is thus less expensive to manufacture compared to the full-function signal channels. Some newer and/or more expensive oscilloscopes just have additional signal channels, so any can be used for triggering, dispensing with the separate *Ext Trig* input.)

2.4 Normal and Auto triggering modes

- Next send a 10 Hz sine-wave signal to the oscilloscope and hit *Auto Scale* to display the signal and get a clean start. Easy, but now change to a Pulse signal with a duty cycle of 1% and set the *Horizontal Scale* to 1 milliseconds per division. You will see that the triggering no longer works, so the signal is not stationary. Fix this by selecting *Mode* in the *Trigger* menu, and then changing from *Auto* to *Normal*. Change the wave amplitude on the waveform generator and you can verify that the signal is being displayed in real time, as it should be.
- The difference between *Auto* and *Normal* modes involves the triggering algorithm in the oscilloscope. In *Normal* mode, the ‘scope waits until it sees a triggering event to plot the signal, and it will wait forever if necessary. And it will not update the plot if it does not see a triggering event. In *Auto* mode, the ‘scope will wait only about 70 msec for the next triggering event. If no triggering event is seen in that time, the ‘scope times out and plots the signal anyway, usually at some random time.
- With the signal triggering in Normal mode, try turning the signal off (press the *Output* button on the waveform generator). Now the screen shows what was last plotted, while it sits and waits for the next triggering event... which never arrives. You can see how this might cause confusion, because you see a signal on the ‘scope even when there is no input signal whatsoever. This is the biggest occupational hazard when working in *Normal* mode.
- So, as a rule of thumb, you want to stay with *Auto* triggering mode except at low frequencies or in other circumstances when there might be long times between triggering events. When you do switch over to *Normal* mode, you just have to be wary of possible hazards.

This is the mid-point in this two-week lab. You are welcome to stop here and leave if you wish, or you may keep going and get a head start on next week (but only until 4:00, when we close for the day).

3 Lab Procedures – Week Two

3.1 DC and AC coupling

Another common circumstance that arises when using oscilloscopes is wanting to look at a small AC signal sitting on top of a large DC offset. So let us examine this next.

- To begin, set up a sine-wave signal with a frequency 1 kHz, amplitude 50 mVpp, and offset 5V using the waveform generator, and view this on Ch1 of the oscilloscope. Open the *Ch1* menu (press the "1" button) and make sure the *Coupling* is set to DC. Open the Trigger menu and make sure the *Mode* is set to Auto. Press the *Vertical position* knob (to set the Vertical offset to 0V) and you should see the *Ground* flag centered vertically on the screen. You will see that the large offset makes it difficult to see the sine-wave signal. If you turn the *Vertical scale* knob up, the signal just disappears off the top of the screen. You can bring the trace down some using the *Vertical position* knob, but this adjustment does not have a large range.

- The remedy for this common problem is to select *AC* input coupling in the Ch1 menu. With AC coupling, the large offset is removed, so now you can turn up the *Vertical scale* to see the small sine-wave signal. Triggering may not be working at this point, however, and pressing *Auto Scale* will just put you back into DC input coupling. To regain triggering, press the Trigger knob (which sets the trigger level to the midpoint of the input signal) and then set the *Horizontal Scale* to 1mV/div. With the *Vertical scale* set to 10mV/div, now you can clearly see the 50 mVpp signal. When this looks good, print a screenshot and add it you notebook. Note that the *Ground* flag now shows zero volts after subtracting the signal average, so the signal is always centered on the *Ground* flag when using AC coupling.

- To understand AC coupling better, you can measure what happens to a signal when you select AC coupling on the oscilloscope. Send a 1kHz sine-wave signal with 1Vrms amplitude to Ch1 on the 'scope (and zero offset), and select AC coupling. Using the oscilloscope's *Measure* feature, select *CycleRMS* to measure the RMS signal amplitude. At a frequency of 1 kHz, the measurement should agree with the signal-generator setting.

- With Ch1 set to AC input coupling, record the *CycleRMS* measurements when you set the signal frequency to 128, 64, 32, 16, 8, 4, 2, 1, 0.5, and 0.25 Hz. Note that you will have to change the *Horizontal scale* setting to keep several periods visible on the oscilloscope screen. Write all the numbers down in your notebook.

- Next use *Mathematica* to plot your measured *CycleRMS* values in volts as a function of the frequency f in Hz. If you are not yet proficient using *Mathematica*, follow the example shown in the Appendix below. Plot the data on a log-log scale, as shown in the Appendix. [Pro tip: ChatGPT and other AI tools can write code for you in *Mathematica*, *Python*, or whatever other language you want. Or, if you know how to ask, they might even do the whole analysis for you. AI code writing can be a time-saver, but only if you know what you are doing. AI tools can easily generate gibberish if the problem is even slightly subtle or unusual.]

- Add a line that goes through the data using the functional form

  $$V(f) = \frac{f V_0}{\sqrt{f^2 + f_{crossover}^2}}$$

  and adjust the two parameters V_0 and $f_{crossover}$ to get a reasonable fit. (We call this "chi-by-eye" – putting a line through the data without doing a full-blown numerical fit. This is quite handy for getting a feel for a data set without getting too deep into the analysis, especially in cases like this one when a non-linear fit is required, because non-linear fitting algorithms can be tricky to use.) Tape a copy of your final plot into your notebook.

- These data show what happens when the AC-coupled signal in Ch1 is sent through a *high-pass filter* inside the oscilloscope. Far above $f_{crossover}$, the signal goes through without significant change, while the signal is reduced at lower frequencies. When you look at the mathematics of frequency-dependent filters (we will not be doing that in Ph3), you find that you cannot have a filter that cuts off abruptly at $f_{crossover}$. There is always some gradual transition around $f_{crossover}$, as you see here.

- At low frequencies, you see that $V(f) \sim f$, which in log-log space looks like a straight line. That is why it is best to plot the data with log-log axes. And this is why you were asked to take data using frequencies in a geometric progression, each frequency point 2x smaller than the previous one. Doing so gives a nice-looking log-log plot with a fairly small number of data points, thus minimizing effort... just some tricks of the experimental-physics trade.

- For one last example of AC/DC coupling, send a 10 Hz square wave into Ch1 and compare DC and AC coupling. The waveform is distorted with AC coupling because its low-frequency components are reduced in amplitude. At 1 kHz (try it), the distortion is negligible, because the frequency is so much higher than $f_{crossover}$. Add a screenshot or two describing this exercise in your notebook.

3.2 Signal averaging

If you have a stationary signal combined with random noise, then averaging is often a powerful technique for reducing the noise and thus enhancing the "Signal-to-Noise Ratio" (SNR). Your oscilloscope can accomplish this by averaging many traces, provided the signal is stationary on the 'screen while the noise is random.

- To see this in action, start by sending a *Ramp* (triangle-wave) signal to the oscilloscope, with a frequency of 1 kHz and an amplitude of 2 mV RMS. Use external triggering to display the signal on the oscilloscope. If you have trouble getting a stationary trace on the 'scope, consult your notes from last week or ask for help.
- Next generate a Noise waveform on Ch2 of the waveform generator with an amplitude of 20mV RMS and zero offset. Send this signal to Ch2 on the oscilloscope and display both on the screen. Set both channels to 50mV per division, and you will see that the noise is much greater than the triangle-wave signal.
- Now you can use the oscilloscope to add these two signals together in real time. Press the *Math* button, set the 1st source to Ch1 and the 2nd source to Ch2, and set the operator to *Add*. To clean up the screen a bit, remove Ch2 (by pressing the 2 button). Using the separate Math scale and offset knobs, display both traces at 20mV per division. Arrange the two traces so both fit nicely on the screen, then print a screenshot and add it to your notebook. The combined (Math) trace does contain some signal, but it is clearly buried in the noise. The signal-to-noise ratio in this case is about 2mV/20mV = 0.1.
- You can think of Ch1 as being the "theory" curve, while the Math trace is the noisy experimental data. To start averaging data, press the *Acquire* button and change the *Acq Mode* to *High Res*. This does some temporal averaging between adjacent time points, which noticeably reduces the noise.
- Next change the *Acq Mode* to *Averaging* and select 512 traces to average. That is a lot of traces, so it takes about 10 seconds to do the average (because each trace takes about 10 times the time/div). If you want to start the process over for any reason, select *Normal* for no averaging, then select *Averaging* again. Averaging 512 traces reduces the noise by about a factor of $\sqrt{N}$, in this case $\sqrt{512} \approx 23$, so the signal-to-noise ratio goes from 0.1 to 2.3, and the signal is now clearly visible in the Math trace. Print out another screenshot for your notebook.
- This exercise demonstrates the power of signal averaging, and it shows you how this can be done quickly and easily using the oscilloscope. Electronics test equipment is popular in the lab because it allows you to quickly observe and analyze just about anything that produces an electronic signal, without having to create your own data-acquisition system.

3.3 Math

As long as we have the Math menu open, let us explore this feature a bit more.

- With Averaging turned off, set up both Ch1 and Ch2 using a 100 kHz sine-wave signal to Ch1 and a 1 kHz sine-wave signal to Ch2, both with 1 Vpp amplitudes. And, of course, display them both on the oscilloscope to make sure the waveforms are as expected. Trigger from Ch2, and you can see some phase jitter in the Ch1 signal. This arises because noise on the Ch2 signal affects the position of the trigger events.
- Now press the *Math* button, set the first source to Ch1, and set the second source to Ch2. Select each of the four math operators and you should be able to understand qualitatively what you see. Discuss with your lab partner, and change the horizontal and vertical scales to look at the signals more closely.
- Finally, select the multiplication operator, then adjust the *Horizontal Scale* to 100 μsec per division. Make sure all three traces are nicely displayed, then print a screenshot for your notebook.

3.4 Reference traces

It is often useful to display saved traces on an oscilloscope so you can compare signals taken under different conditions. With Math turned off, start with a 1 kHz sine-wave signal displayed on the oscilloscope as usual. Then press *Analyze/Features* and save a trace to R1. (The user interface is a bit cumbersome here, but manageable. Oddly, while two reference signals can be saved, only one can be displayed at a time). Next change the signal frequency to 2kHz and display both ch1 and R1 on the screen. Separate the two waveforms so ch1 is on the top half of the screen and R1 is on the lower half. (Use Analyze/Offset to move the reference trace.) When things look good, print a screenshot for your notebook. As you can imagine, the utility of this feature becomes apparent when you want to compare "before" and "after" waveforms on the same screen. Most oscilloscopes have this features somewhere within all the various menus and submenus. To remove the Reference traces, uncheck the appropriate boxes in the *Analyze/Features* menu.

3.5 Spectral analysis

Oscilloscopes are mainly set up to display and analyze signal voltages as a function of time, written $V(t)$. However, it is also useful to analyze signals in the frequency domain, displaying the power in a signal as a function of frequency, $V(f)$. Transforming between $V(t)$ and $V(f)$ is accomplished using a *Fourier transform*, and the math behind all that is beyond the scope of this Intro lab. However, one possibly familiar example is the audio frequency spectrum, where an acoustic signal as a function of time is converted to an acoustic frequency spectrum. When analyzing music, it is often useful to look at the frequency spectrum in detail. And so it for analyzing many voltage signals.

- To demonstrate this feature briefly on your oscilloscope, set things up with 100kHz square wave at 1Vpp and press the FFT (Fast Fourier Transform) button. Adjust the setting so Source = 1, Settings/Vertical = Vrms, Span = 1MHz, and Center = 500kHz, and then adjust the main *Horizontal scale* knob so *FFT Resolution* = 3.81kHz. (If this knob is in Fine mode, press the knob to go back to Coarse mode.) You should see a large peak at 100 kHz (the spectrum goes from 0 to 1 MHz) and additional peaks at 300, 500, 700, and 900 kHz.

- What the FFT graph shows is the spectral content of the input signal. Switch to a sine-wave signal and you will see that there is now just a single peak in the FFT at 100 kHz, indicating that the entire signal is concentrated as this single frequency. A square wave signal at frequency $\nu = \omega/2\pi$ can be written as a *Fourier Series*

  $$f(t) = \frac{4}{\pi} \sum_{n=1,3,5...}^{\infty} \frac{1}{n} \sin(n\omega t)$$

  which shows that a square-wave signal consists of a sum of all the odd harmonics of the primary frequency.

- With Ch1 and the FFT on the 'scope, print a screenshot for your notebook. To analyze this further, turn off Ch1 (press the "1" button) and go to the Cursors menu. Set the Mode to Track and measure the heights of the different peaks. With the signal set to 1 Vpp, the peaks heights (according to theory) should be

  $$a_n = \frac{1}{n}\left(\frac{2}{\pi}\right)^2$$

  Make a table that compares theory with experiment for the five visible peaks. Agreement at the 5-10 mV level is typical.

- Next remove the Cursors, turn Ch1 back on, and go back to the FFT menu. Change the FFT Resolution and observe how the peak widths change. This happens because it takes time to measure a signal frequency, and the best you can do is a resolution of roughly $\Delta f \simeq 1/\Delta T$, where T is the total observation time. And, once again, the 'scope can only measure what is on the screen. Therefore, the more cycles you put on the screen, the sharper the FFT peaks become.

- Looking at signals as a function of frequency is called *spectroscopy*, and spectral techniques are widely used across all areas of science and technology. Most oscilloscopes have some capacity for viewing spectral information, usually limited by the computational power built into the 'scope. High-end oscilloscopes include many features for doing complex spectral analysis. These techniques are mostly beyond the scope of Ph3, but this exercise gives you a brief sample.

3.6 Acquire modes

This oscilloscope includes some other handy feature that are useful from time to time, so let us explore these briefly.

- **Zoom**. First set up a Pulse signal with: 1kHz, 1Vrms, and 1% duty cycle. Display this signal at 5msec/div and you see a nice train of pulses on the screen. Press *Acquire/Zoom* to get a split screen, where the lower half shows a zoomed-in portion of the upper half. Use the *Horizontal scale* knob (in Zoom mode) to vary the width of the zoom portion. Set this to 5μsec/div (see the top of screen) to zoom in on a single pulse. Press the large *Run/Stop* or *Single* buttons to see a single sweep of this signal. Save a screenshot to notebook as usual. Zoom is something of a “bonus” feature on the ’scope ... not super useful, but sometimes nice to have.
- **Roll mode**. Moving on, get out of *Zoom* mode and set the system up to view a 1 kHz sine wave at 500msec/div. Just to have something more interesting to look at, press the *Mod* button on the signal generator to set up amplitude modulation (AM) of the signal. Set *MFreq* to 10 Hz. As you can see, the ’scope gives you a stationary view of the signal with these settings, but there is a lot of waiting. At 500msec/div, it just takes several seconds to refresh the screen. In the Acquire menu, select *Time Mode = Role* for another view. Now you see the signal running slowly across the screen in real time. Press *Run/Stop* to reset the clock. Role mode is useful if you happen to be looking at signals that only varying with time. In role mode you can view a signal moving across the screen at very low speeds, down to 50sec/div. Again, save a screenshot for your notebook.

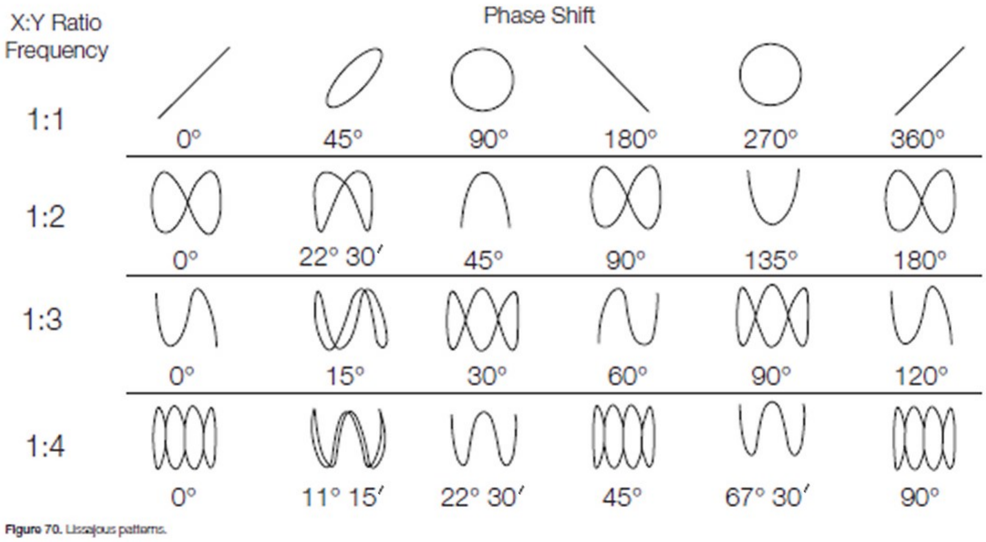


Figure 5. Plots of various Lissajous figures for different X:Y frequency ratios and phase relationships (from https://www.tek.com/support/faqs/how-do-i-utilize-xy-display-feature-dpo-mso-mdo4000-series-oscilloscope).

- **XY mode**. Next send a 1 kHz sine-wave signal to Ch1 on the oscilloscope, and send a 2 kHz sine-wave signal to Ch2, both at 1 Vrms and zero offset. Display both signals on the ’scope as usual. Press the *Align Phase* button on the waveform generator, and you will see that the two signals are in phase at the trigger point. (If not, check that both channels are set to zero phase, and set the trigger level to zero volts.)
- On the signal generator, go to *Start Phase* (either channel) and observe that you can change the relative phase between the two signals. Set both phases back to zero and press *Align Phase* to continue.
- Next press the *Acquire* button, select *Time Mode/ XY*, and the oscilloscope plots Ch1 along the X axis and Ch2 along the Y axis of the display. For best results, make sure nothing is connected to the EXT trigger input (this seems to confuse the ’scope). Turn the *Vertical* and *Vertical position* knobs for both Ch1 and Ch2 to adjust the scales and offsets. Adjust all four settings to center the signal on the oscilloscope screen, with it nearly filling the screen on both axes. Plotting two sine-wave signals against one another in this way generates what is called a *Lissajous* pattern, and your ‘scope trace should look like sketch (1:2)(0 deg) in Figure 5. (If not, check that both

channels are set to zero phase.) Print a screenshot for your notebook.

- Select different waveform parameters to also generate traces that look like sketches (1:1)(90 deg) and (1:3)(15 deg), and add these to your notebook. (You will find that the (1:3)(15 deg) sketch in Figure 5 is not perfectly accurate, although it gets the basic point across.) Lissajous patterns can be useful for observing phase relationships between two signals.
- Just for fun, Set Ch1 to 1000.3 Hz and observe what happens in both XY and Normal modes. Now the relative phase shift between Ch1 and Ch2 changes slowly with time, The Lissajous pattern moves horizontally in Figure 5 with time.

Ph3 Oscilloscope Lab

Appendix

Plotting High-Pass Filter Data, with Theory Line

First define the data table, format {frequency_1, voltage_1}

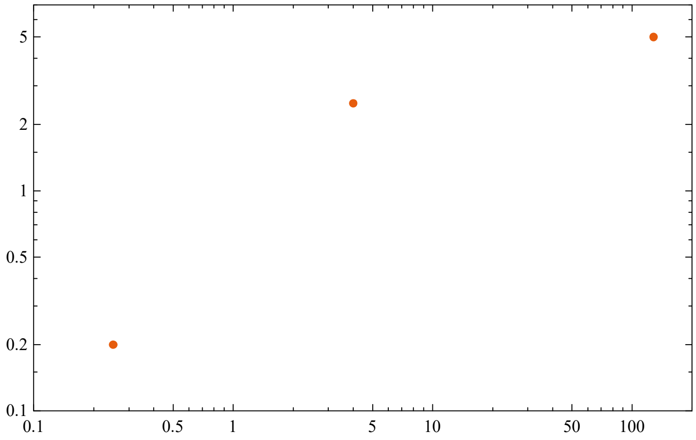
This is an example only; you will have more than 3 data points to plot!

```
In[76]:= data1 = {{128, 5}, {4, 2.5}, {0.25, 0.2}}

Out[76]= {{128, 5}, {4, 2.5}, {0.25, 0.2}}
```

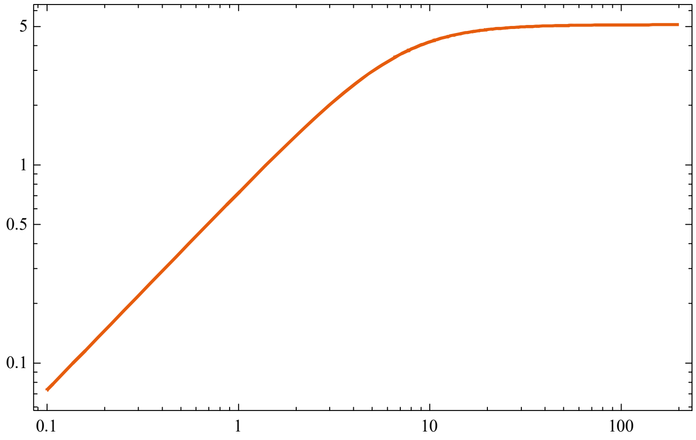
```
In[70]:= ListLogLogPlot[data1, PlotRange → {{0.1, 200}, {0.1, 7}}, PlotTheme → "Scientific"]
         (* Plots points only *)
```

Out[70]=



```
In[71]:= LogLogPlot[5.1 * x / Sqrt[7^2 + x^2], {x, 0.1, 200}, PlotTheme → "Scientific"]
         (* Plots line only *)
```

Out[71]=



In[90]:=
```
Show[LogLogPlot[5.1 * x / Sqrt[7^2 + x^2], {x, 0.1, 200},
   PlotTheme → "Scientific", BaseStyle → {FontSize → 14}],
  ListLogLogPlot[data1, PlotRange → {{0.1, 200}, {0.1, 10}}],
  FrameLabel → {Style["Frequency  (Hz)", 16], Style["CycleRMS  (volts)", 16]}]
 (* Plots both line and points, with axis lables and larger fonts *)
```

Out[90]=