

Ph 3 - INTRODUCTORY PHYSICS LABORATORY

– California Institute of Technology –

Electronics Test Equipment and Digital Data Acquisition

1 Introduction

The 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. In addition, you will examine some of the principles of digital data acquisition as applied to electronic signals. A principal 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 *waveform generator* (also called a *function generator* or *signal generator*) does what the name suggests – it generates electronic waveforms 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, 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 $V(t)$. There are many useful ways to plot and analyze electronic signals, so the oscilloscope has become a fairly complex piece of equipment, with quite a few knobs, buttons, menus, and sub-menus that provide a great deal of functionality for observing and measuring a wide variety of electronic signals. 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).

Our oscilloscopes are also networked, so you can send your recorded data to a computer for additional analysis. Figure 2 shows the oscilloscope you will be using in Ph3. At first the oscilloscope may seem complicated and cumbersome, with all its menus and sub-menus. But once you gain some experience using an oscilloscope, it becomes an indispensable tool for quickly and easily examining just about anything electronic. This first lab is a basic oscilloscope tutorial. In subsequent labs you will gain more experience using it in a laboratory setting.

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 the Ph3 website), 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 also the Ph3 website 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 screen shot (by pressing the print button on the oscilloscope) and tape it into your lab notebook.
- If the handout instructs you to view some signal on the oscilloscope, include a screen shot in your notebook, along with a brief description.
- 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. There are several computers in the Ph3 lab you can use to print 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 the Ph3 website.

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 while, you should ask for help before it chews up too much time.

- To get started, turn on the power to your waveform generator and oscilloscope. While they 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 *Autoset* button on the oscilloscope (all the way on the right, near the center). You should see a yellow sine wave plotted on the ‘scope. If you also see a blue line, press the blue Ch 2 button on the ‘scope, then the nearby *OFF* button to remove it, and then press the *Autoset* 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; we’ll get to that below.



Figure 1. The Rigol DG1022 waveform generator.



Figure 2. The Tektronix TDS3012B oscilloscope.

- Next turn the *Vertical Position* knob and see what that does. You should see a yellow “flag” on the left side of the screen that moves as you turn the 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 blue flag for Ch2.
- Now adjust the *Vertical Scale* knob, which changes the gain setting inside the oscilloscope. You will see something like “Ch1 1.00V” on the lower left corner of the display. This tells you the “volts per division”, where one division is about one centimeter on the oscilloscope screen, delineated by dotted lines. Next press the *Ampl* button on the waveform generator, which probably says “5.000 V_{pp}”, meaning 5 volts peak-to-peak. Change the signal amplitude on the waveform generator and adjust the oscilloscope’s *Vertical Scale* and *Vertical Position* knobs to verify that indeed you can use the screen divisions to measure what the waveform generator is supplying. Thus an oscilloscope can be used as a voltmeter.
- Now turn the *Horizontal Scale* knob and see what it does. At the bottom of the ‘scope you will see something like “M 400 μ s”, telling you that the oscilloscope is plotting the waveform at 400 microseconds per division. Press the *Freq* button on the waveform generator and you should see 1.000 kHz (i.e., the frequency in kilohertz) on the display. Press the *Freq* button a few more times and you see the display toggle between frequency and period. Change the waveform period, and verify that you can measure the correct period on the oscilloscope. Thus an oscilloscope can be used to measure the signal period (or equivalently the signal frequency).
- Just a few 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 3. 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

- Another way to measure signal amplitudes and frequencies is to use the oscilloscope’s built-in *Measure* feature. To access this, press the *Measure* button (top row on the ‘scope). Press the *Remove Measrmnt* menu button, and then *All Measurements*, to clear any existing measurements. Next press *Select Measrmnt* and you will see many possibilities to select from. Select *Frequency* and *CycleRMS* measurements, and you should see these

Coaxial Cable Construction

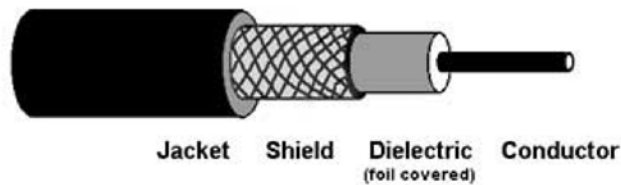


Figure 3. Typical construction of a coaxial cable.

measurements displayed on the screen. Set the signal amplitude to 1 kHz, 1 V_{RMS}, on the waveform generator, and verify that you see these values on the oscilloscope. (With the amplitude tab selected on the waveform generator, press 1 and you will see a range of units to choose from, including V_{RMS}.) RMS stands for the root-mean-squared average of the signal amplitude, averaged over one cycle:

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

For a sine wave, the peak-to-peak voltage is $2\sqrt{2}$ times higher than the RMS average.

- Move the yellow signal trace down, and reduce its amplitude (using the *Scale* and *Position* knobs) so that the trace is displayed on the screen but is not overlapping the measurement numbers. Press the *Print* button on the 'scope (far left button) and a screen shot should appear in one of the Ph3 printers. 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).
- Now add the *Pk-Pk* measurement on the screen, which gives the difference between the highest and lowest voltages in the trace. Because there is some noise on the signal, you will find that the *Pk-Pk* measurement is a bit higher than the expected $2\sqrt{2}V_{RMS} = 2.8$ volts. The *CycleRMS* measurement is less affected by noise.
- If you turn up the *Scale* on the oscilloscope, you will see that the measurements tell you when the signal no longer fits on the screen. This is a feature of this oscilloscope – the entire signal has to be visible on the display for the measurements to work correctly. In general, the *Measure* feature works best on high-amplitude, low-noise signals. Frequency measurements are especially problematic when the signal is weak or noisy.

2.2 Cursors

- Next remove the measurements (*Measure* menu, *Remove Measrmnt*, *All Measurements*) to get them out of the way, and press the *Autoset* button to get you back to a clean display. From there, press the *Cursor* button (right below *Measure*) and you will find another measurement tool. Select *V bars* and you will see two vertical lines on the screen. Press the *Select* button (upper left of the main panel of buttons) to select which cursor to move, then move it using the large knob nearby.
- Note that four numbers are displayed:
 - 1) @ – the time coordinate of the cursor line, relative to the trigger time (marked by the red flag on top of the screen; we'll look at what that flag means below)
 - 2) Δ – the time difference between the two cursor lines (this Δ can also show frequency = (time difference)⁻¹, and you can select one or the other in the *Cursor* menu)
 - 3) @ – the signal voltage at the chosen time (note the short horizontal line on the signal)
 - 4) Δ – the voltage difference between the two cursor points
- Exercise: Display a 1 kHz sine wave, 2 volts peak-to-peak, and position one cursor at a signal minimum, and the

other cursor at an adjacent signal maximum. Verify that the cursor measurements agree with expectations, then print a screen shot for 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.
- For completeness, select *H Bars* from the *Cursor* menu and you can see that the functionality is similar to the *V Bars*.

2.3 Triggering

Note that the waveform generator puts out a simple sinusoidal signal, $V(t) = A \cos(\omega t)$, which clearly varies with time. A most useful feature of all oscilloscopes is that they display time-varying signals as stationary functions. That is, the signal on the display is stationary, making it easy to view and analyze. This trick is accomplished using a mechanism called “triggering”, which one of the subtler aspects of using an oscilloscope.

- To see an untriggered signal, press the *Trigger Menu* button, and then press the *Ext* button near the display. Now the signal is free-running, and clearly much harder to examine. In this mode, the oscilloscope plots the signal over and over again at essentially random times. Press the *Ch1* menu button to go back into triggered mode.
- Next adjust the *Trigger Level* knob until the *Level* value displayed on the ‘scope screen 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. You will see a red “T” flag at the top of the screen, which shows you when the trigger event happened. You can move this flag using the *Horizontal Position* 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, yet 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 and see what happens. Now you see another flag on the right side of the screen that shows you the trigger level, which is the voltage the ‘scope is waiting for. The red 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 *Slope* button on the bottom of the screen (in the *Trigger* menu). Note that the slope changes at the trigger position (indicated by the red trigger flag).
- Now remember this: if you are ever examining a nice, clean periodic signal (like a sinusoidal waveform), and you see it whizzing across the oscilloscope screen (i.e., it is not stationary), then you are doing it wrong. You need to adjust the trigger so the signal looks stationary on the screen. The *Autoset* button is your friend here, and it usually works well, provided you have a strong signal plugged into Ch1. But *Autoset* is not a cure-all, so you also need to know how triggering works.
- 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 occurs) and *Stop* (leave the last plotted signal on the display and stop updating). Right below *Run/Stop* is *Single Seq*. This 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 blue button on the oscilloscope to display both channels. Adjust the *Vertical Scale* and *Vertical Position* knobs so both the Ch1 (yellow) and Ch2 (blue) traces are visible on the screen, although both might not be stationary at this point. (Pro tip: if both Ch1 and Ch2 are displayed, then

pressing *Autoset* will try to arrange them both nicely on the screen.)

- Next press the Ch1/Ch2 button on the waveform generator and set Ch1 to 1000 Hz and Ch2 to 1001 Hz. 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. Change the Ch2 frequency from 1001 Hz to 999 and 1000 Hz (easiest if you use the large knob on the upper right of the waveform generator) and again what you see should make sense. 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.
- 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 *Sync Out* connector on the back of the waveform generator. Press *Utility*, then *Syncoff* to turn the *Sync* signal on. This might be a good time to hit the *Autoset* 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 5V.
- Trigger using Ch1 (from the *Trigger* menu), and then set the Ch1 signal amplitude to 2mV RMS. This is the lowest amplitude the waveform generator can produce. Turn the Ch1 *Scale* 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, and thus 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.
- 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 like a third channel, but it does less and is thus less expensive to manufacture than the full-function signal channels. Some newer and/or more expensive oscilloscopes just have additional signal channels, and any can be used for triggering, dispensing with the *Ext Trig* input.)
- There are some other triggering tricks if you do not want to use a *Sync* signal with external triggering. Remove the blue trace (hit the blue button, then off) and use the waveform generator to send a 10mV RMS sine-wave signal to Ch1. Adjust the scale so the signal is nicely displayed on the screen, and trigger from Ch1 (in the *Trigger* menu). You should see once again that the weak signal is difficult to trigger from.
- While an external trigger is the best fix for this, you can also try some smoothing to reduce the noise in the trigger signal. Select the *Trigger* menu, then change the *Coupling* from *DC* to *HF Reject*. This sends the signal through a low-pass filter that smooths the signal, making it easier to trigger from. (The displayed signal is unchanged; it is copied, and the copied signal is smoothed and used for triggering.) Sometimes this trick works, and sometimes it does not. The same goes for the other *Coupling* selections. Using an oscilloscope becomes a bit of a black art when the signals are small and noisy, and sometimes you have to try a variety of tricks to obtain stable triggering. As with everything, it takes some experience using the oscilloscope to know what tricks to try when.

2.4 Normal and Auto Triggering modes

- Next send a 50 Hz sine wave signal to the oscilloscope and trigger off Ch1 (*Trigger* menu, *Source*, Ch1) to display the signal. Easy, but now change the *Horizontal Scale* to 40 milliseconds per division, and you will see that the

triggering no longer works – the signal is not stationary. You can 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 300 msec for the next triggering event. If no triggering event is seen in that time, the 'scope times out and plots the signal anyway, at some random time.
- One typically stays in *Auto* mode (the default when you press *Autoset*), because then there is always something being displayed on the screen in real time. If you turn the signal off, the display shows it going off. Now turn off the signal (press the *Output* button on the waveform generator) with the 'scope in *Normal* mode. The screen shows what was last plotted, while it sits and waits for the next triggering event, which never arrives. This can cause confusion, because you see a signal on the 'scope when it is not actually receiving that signal.
- As a rule of thumb, you want to stay with *Auto* mode except at low frequencies (when the *Horizontal Scale* is slower than 20 msec/division), or in other circumstances when there may be long times between triggering events. You will encounter such circumstances when working with electronic signals, so you should know that the oscilloscope can handle them.

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

- Set up your next task by creating a sine-wave signal with a frequency 1 kHz, amplitude 50 mV_{PP}, and offset 1V using the waveform generator, and view this on Ch1 of the oscilloscope. Open the *Vertical* menu and select *DC* input coupling. You will see that the large offset makes it difficult to see the sine-wave signal, because the *Vertical Position* adjustment does not have a large range. This is a common problem, easily remedied by selecting *AC* input coupling. With *AC* coupling, the large offset is removed, so now you can turn up the *Scale* to see the sine-wave signal easily. Do this, print out a screen shot, and put it in your notebook. Note that the *Position* flag now shows zero volts after subtracting the signal average, so the signal is always centered on the *Position* flag when using *AC* coupling.
- The nomenclature has evolved somewhat from its historical roots. *DC* traditionally stood for “direct current,” but in the electronics world *DC* has evolved to mean “the low-frequency part of the signal.” And *AC* means “the signal 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.
- To understand this better, you can measure what happens to a signal when you select *AC* coupling on the oscilloscope. Send a sine-wave signal with 5V_{RMS} amplitude to Ch1 on the 'scope, and select *AC* coupling. Using the oscilloscope's *Measure* feature, select *CycleRMS* to measure the RMS voltage. At a frequency of 1 kHz, the measurement should agree with the waveform generator setting.
- Now 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.

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

$$V(f) = \frac{fV_0}{\sqrt{f^2 + f_{crossover}^2}}$$

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 before doing a proper numerical fit, especially a non-linear fit like this one, because non-linear fitting algorithms can be tricky to use.)

- 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 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 $V(f) \sim f^{-1}$, 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 good-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 $f_{crossover}$.

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. To see this, 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 (remember you must use the *Utility* menu on the waveform generator to turn on the *Sync* signal). 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. Display both channels at 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 (press the blue button, then off). Once again, display both traces at 50mV per division. Print a screen shot and tape it into 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 $2\text{mV}/20\text{mV} = 0.1$.
- Now re-scale so both traces are displayed at 5mV per division. You can think of Ch1 as being the “theory” curve, while the red trace is the noisy experimental data. To start averaging data, press the *Acquire* menu (all the way on the lower right), and select *Average*. Turn the large knob (top row on the ‘scope) to select 512 traces to average. That is a lot of traces, so it takes about 10 seconds to do the average. (To start the process over, select *Sample* for no averaging, then select *Average* again.) Averaging 512 traces reduces the noise by about a factor of $\sqrt{512} \approx 23$, so the signal-to-noise ratio goes from 0.1 to 2.3, and the signal is now clearly visible in the data. Print out another screen shot 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 always somewhat limited (in this case you can average at most 512 traces), but it allows you to quickly observe and analyze just about anything that produces an electronic signal.

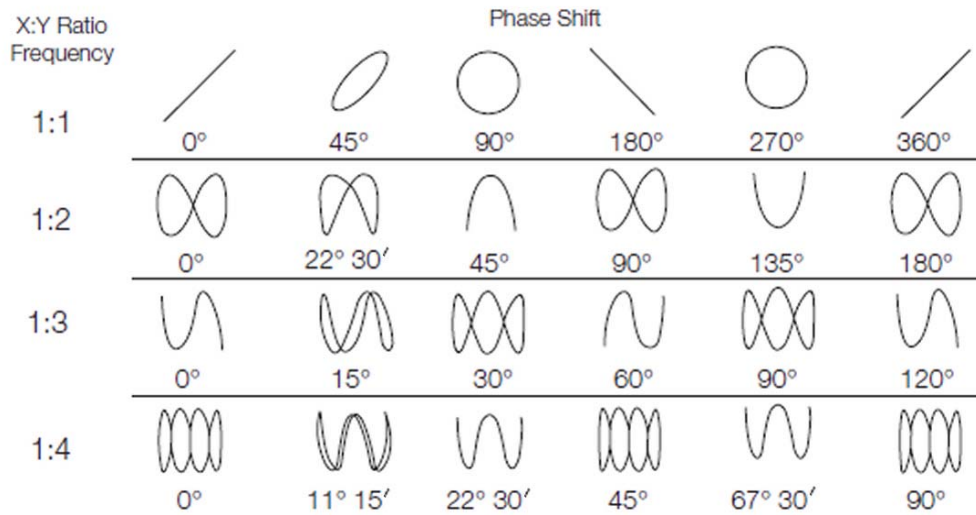


Figure 70. Lissajous patterns.

Figure 4. 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>).

3.3 YT and XY plotting

- 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 V_{pp} and zero offset. Display them both as usual (again, *Autoset* may be helpful here). Press the *AlignPha* 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.)
- Then press the *Display* button (near the top right on the 'scope), select *XY Display*, and you will see two choices: YT mode and XY mode. YT is the default, which plots both traces as a function of time, and this mode is selected when you press *Autoset*. Select XY mode, and the oscilloscope plots Ch1 along the X axis and Ch2 along the Y axis of the display. Turn the *Vertical Position* knob, and you will see one of the zero-voltage flags move, and the display will change accordingly. You can move the other flag by first selecting that channel using the yellow or blue buttons. Center the signal on the oscilloscope screen, and adjust the *Scale* for both axes also. 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 4. (If not, check that both channels are set to zero phase.) Print a screen shot 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 4 is not perfectly accurate, although it gets the basic point across.)

3.4 Digital Sampling

We begin this section with the Nyquist-Shannon sampling theorem, which states that: *Any continuous function $g(t)$ is only uniquely determined by a set of discretely sampled points if the sampling rate R is greater than twice the highest-frequency component of $g(t)$.*

Put another way, a data set sampled at a rate of R samples per second cannot uniquely identify signal components having frequencies above $f_{Nyquist} = R/2 = 1/2\Delta t$, where $f_{Nyquist}$ is called the *Nyquist frequency*, and $\Delta t = R^{-1}$ is the spacing between sampled points. We now look at what all this means using a digital oscilloscope.

- First set the oscilloscope time base (*Horizontal Scale*) to 1 msec per division. The TDS3012B samples at a rate of 1000 points per division, regardless of the *Horizontal Scale* setting, so in this case sampling is 10^6 points per second, or 1 μ sec per point. If you send a 1 kHz signal to Ch1, you will see that this looks good on the display,

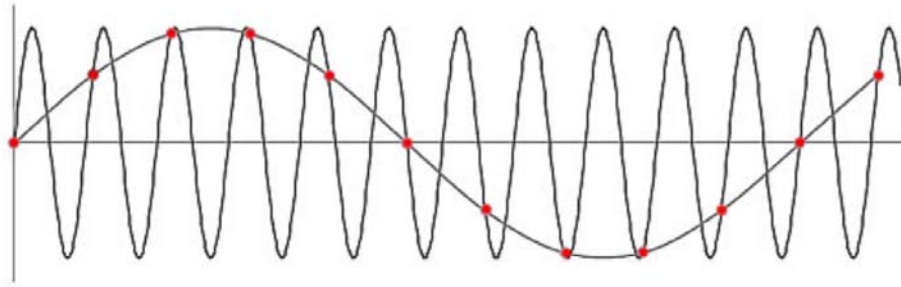


Figure 5. *Aliasing*, also known as *undersampling*, can cause a very high-frequency signal to be misinterpreted as a lower-frequency one, as shown by the red dots in this figure. This problem is avoided by insuring at least two samples per period. (Image from industrial-needs.com).

because there are lots of sample points per period.

- Next change the signal frequency to 250 kHz, while keeping the 'scope at 1 msec per division. Press the *Single Seq* button to record a single trace. With a single trace displayed (and not being updated), you can now change the *Horizontal Scale* to zoom in, showing you the individual sampled points that are saved in memory. (Note that the time required to read the input voltage is of order 1 nsec, which is essentially instantaneous for a 1 MHz signal frequency.) With a sampling of 1 μ sec per point and a frequency of 250 kHz, you should see that there are four samples per cycle.
- Now set the 'scope back to 1 msec per division and change the signal frequency to 1 MHz. Again press the *Single Seq* button to record a single trace, and zoom to see the individual sampled points. Because you are now sampling at a rate of one point per cycle, the signal appears essentially flat. Figure 5 shows what is happening.
- Go back to a *Horizontal Scale* of 1 msec per division, and change the frequency to 1.001 MHz. Now acquire a single trace and use the *Measure* feature to measure the signal frequency. As you can infer from Figure 5, the measured frequency is equal to the signal frequency minus 1 MHz. Add a screen shot of this measurement to your notebook.
- This exercise shows you the potential for sampling ambiguities when you acquire too few samples per period of a signal. The take-home message is that you should always, if possible, sample fast enough that all significant signal frequencies lie well below the Nyquist frequency.

3.5 Reference Traces

- It is often useful to display saved traces on an oscilloscope so you can compare signals taken under different conditions. To see how this works, start with a 1 kHz sine-wave signal displayed on the oscilloscope as usual. Then press the white *Ref* button, select Ref1, and then select Save Ch1 to Ref1. Now change the signal frequency to 2 kHz and save a trace to Ref2, and similarly put a 4 kHz signal in Ref3.
- To straighten out the mess on the screen, start by removing Ch1 (press the Ch1 button, then the nearby off button). Then press the Ref button, select Ref1, and use the *Vertical Position* and *Scale* adjustments to put it in the top third of the display. Similarly, put Ref2 in the center of the screen, and Ref3 in the bottom third. When all three traces are nicely separated on the display, print a screen shot for your notebook. As you can imagine, the power of this feature becomes apparent when you want to compare “before” and “after” waveforms on the same screen.

3.6 Math

- Remove all the *Ref* waveforms from the screen by pressing the *Ref* button followed by the *Off* button, as you do to remove signal traces. Turn on both Ch1 and Ch2, then send a 100 kHz sine-wave signal to Ch1, and send a 1 kHz sine-wave signal to Ch2, both with 1 V_{pp} 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. Select the multiplication operator, then adjust the *Horizontal Scale* to 1 msec per division. Make sure all three traces are nicely displayed, then print a screen shot for your notebook.
- Save the *Math* trace to Ref1, then remove the Ch1 and Ch2 traces from the screen, leaving the *Math* and Ref1 traces. In the *Math* menu, select the FFT (Fast Fourier Transform) option, and select *Set FFT Source* to Ref1. Now you see the multiplied signal (Ref1) and a frequency decomposition (FFT) of that signal. Center the largest FFT peak in the screen. Use a vertical cursor to show that this largest peak corresponds to 100 kHz. Thus, as you might expect, the largest frequency component in the signal comes from Ch1 at 100 kHz. If you zoom in on this peak (using the *Horizontal Scale* and *Position* knobs), you can see that this peak is actually two peaks. Again use the cursors to show that one peak is at 99 kHz and the other is at 101 kHz. Again, print out a screenshot for you notebook.
- To explain this, note that multiplying two sinusoidal signals gives

$$\cos(\omega_1 t) \cos(\omega_2 t) = \frac{1}{2} \{ \cos [(\omega_1 + \omega_2) t] + \cos [(\omega_1 - \omega_2) t] \}$$

Thus what you are seeing is the sum and difference of the Ch1 and Ch2 frequencies. This is the small tip of a very large iceberg – it is common to modulate high-frequency signals in amplitude or frequency (called AM and FM), and going down this path will lead you to the entire telecom industry. (And along the way you would become very familiar with using oscilloscopes!)

3.7 Saving data

Digital oscilloscopes can also be used to save signals for analysis elsewhere. Often this is done using a built-in USB port, but our 'scopes allow saving data via the network, just as you have been sending screenshots to our networked printers. (We will not be using the TDS3012B's built-in floppy-disk drive!)

- To save a trace, first send a 1 kHz sine-wave signal with 1V_{PP} amplitude to your oscilloscope, and display it so at least ten periods are visible.
- Then locate your oscilloscope's IP address, which should be on a label just above the screen. If not, it may be printed on your previous screenshots, or you can find it by pressing the *Utility* button, then *System I/O*, then *I/O*, then *Ethernet Network Settings*, and finally *Change Instrument Settings*.
- Enter this IP address as the URL in a web browser, either on your computer or one of the computers in the Ph3 lab. This should pull up a page that shows the current oscilloscope screen. From this you can download the data from any channel (including the Ref and Math channels), control the 'scope remotely, or even transfer data to be displayed on the 'scope. We will focus here on downloading data.
- At the top of the screen you should see a number of tabs. Select the *Data* tab, select Ch1 to download, select *Spreadsheet Format*, and click on the *Get* button. We have found that this doesn't always work on every computer in every browser. Firefox seems to work best, especially on one of the iMac computers in the Ph3 lab. Rebooting the oscilloscope sometimes helps. Send the data file to your computer, as you will need it for next week's homework.

Ph3 Oscilloscope Lab

Appendix

Plotting High-Pass Filter Data, with Theory Line

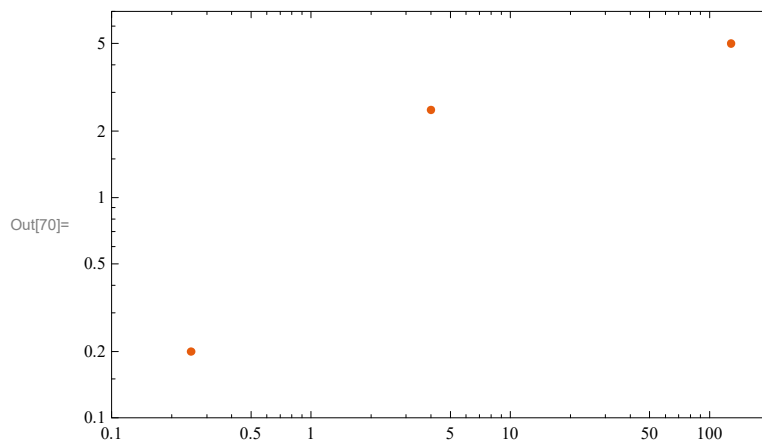
First define the data table, format {frequency_1, voltage_1}

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

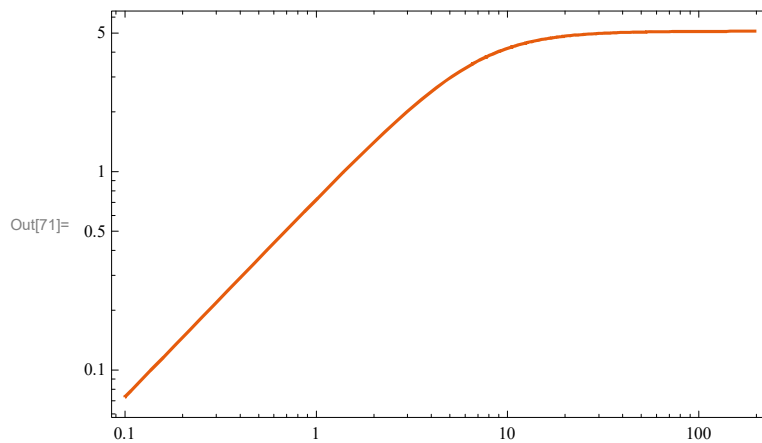
```
In[76]:= data1 = {{128, 5}, {4, 2.5}, {0.25, 0.2}}
```

```
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 *)
```



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



```

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 *)

```

