Physics 80 Lab Manual

January 22, 2019

Chapter 1

Introduction to Plotting

1.1 Introduction

This lab will introduce calculations and plotting techniques using numpy arrays within Scientific Python.

1.2 Plotting discrete data and continuous functions

Consider the Jupyter notebook example in Fig. 1.1 which plots a sine function sampled at discrete values. Note the following key features, which you will use repeatedly today and in future labs:

- Use of global variables `UPPER` and `STEP` at the top of the snippet, allowing for easy adjustment of parameters that affect the plot.
- Use of `np.arange` to define an array of x values.
- Creation of the array y, defined by $y = \sin(2\pi x/5)$ for each value of x. One of the great joys of using numpy is the ability to avoid getting bogged down with explicit for loops.
- Use of slicing techniques `x[:5]` to show only the first five entries for debugging.
- Plotting the arrays of x and y values with `plt.plot` using the `"bo"` option for blue circles.
- Defining appropriate axis labels with `plt.xlabel` and `plt.xlabel`.
- Creation of a legend using the `label` option to `plt.plot` and the plot.legend() command.

Notice that even in this simple example, I've added some intermediate feedback from my code in the form of the screen dumps of the first few values of x and y. It's a common pitfall to try and rush ahead to the final product when programming. But it is much faster and reliable to break your task into small steps, and establish feedback at each small step. To plot a continuous function with Scientific Python, you will still use discrete data but:

- Use much finer binning of the x-axis variable to draw a smooth curve.
- Use the line option `"-"` or dashed line `"--"` instead of points with `"o"`.

Plot 1: Plot the quadratic function $y = x^2$ with the following requirements:

```python
# plot a sin function
UPPER = 10
STEP  = 0.25
x = np.arange(0,UPPER,STEP)
y = sin(2*pi*x/ 5.0)
print("dumping first five entries:")
print("x[:5]:", x[:5], "...")
print("y[:5]:", np.around(y[:5],2), "...")
plt.plot(x,y,"bo",label="sin")
plt.xlabel("x")
plt.ylabel("y")
plt.legend()
```

```
dumping first five entries:
x[:5]: [0.   0.25 0.5  0.75 1.  ] ...
y[:5]: [0.   0.31 0.59 0.81 0.95] ...

<matplotlib.legend.Legend at 0x11781ef98>
```

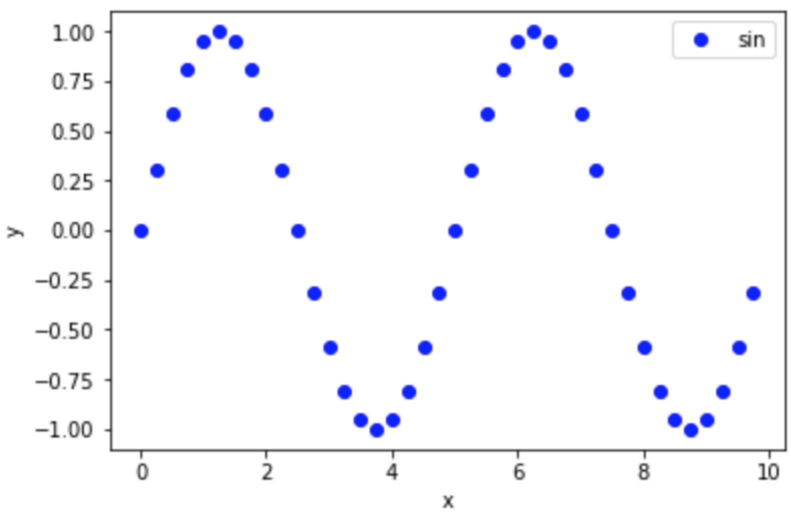


Figure 1.1: Circuit for verifying Ohm's law as a (a) circuit diagram, and (b) implemented using your lab equipment.

- Plot in the range $x = [0, 20)$.
- Plot discrete samples with a step size of 2 using blue circles.
- On the same axis, plot the corresponding smooth function using a red solid line.
- Add appropriate axis labels.
- Add a legend for the "discrete" and the "smooth" function.

1.3 Multivariate analysis using boolean masks

```
x = np.array([1,2,3,4,5,6])
y = np.array([1,2,1,2,1,2])
cut = (y > 1)
print("cut:  ", cut)
print("y subject to cut:  ", y[cut])
print("x subject to cut:  ", x[cut])
```

```
cut:   [False  True False  True False  True]
y subject to cut:   [2 2 2]
x subject to cut:   [2 4 6]
```

Figure 1.2: Using boolean masks to cut on variable y.

A powerful technique in Scientific Python for performing analysis involving multiple variables uses boolean masks as shown in Fig. 1.2. In the example:

- Two numpy arrays x and y `of the same length` are defined to contain the collected data.
- The cut defined by $y > 1$ is a boolean array of the same length as x and y which is true at indices where the condition is met and false where it is not.
- The subset of the entire y array defined by `y[cut]` consists only of those entries of y for which the condition $y > 1$ is met.
- The subset of the entire x array defined by `x[cut]` consists only of those entries of x for which the condition $y > 1$ is met for the corresponding y value.

The last item shows the real power of this technique, one can look at one variable subject to constraints on another variable.

Next consider the sample data in Table 1.1 which comes from experimental measurements of a voltage level v at discrete times t. The measurement is subject to a high-frequency noise monitoring by the variable n. The noise is only present for $n > 6.0$. A straightforward way to load this data into scientific python is by defining numpy arrays for each variable as follows:

Table 1.1: Sample data for a voltage measurement subject to high frequency noise.

t (s)	v (V)	n
0.4	0.25	2.8
1.1	2.37	7.3
1.4	1.69	9.7
1.9	0.93	1.3
2.5	-1.0	6.2
3.0	0.95	4.8
3.4	1.22	6.9
4.1	0.54	4.0
4.4	0.37	1.9
4.8	0.13	4.0
5.5	-2.04	9.5
6.2	-2.06	8.7
6.5	-0.81	2.3
7.0	-0.95	5.3
7.5	0.98	9.7
7.9	0.27	8.3
8.5	-0.81	0.1
9.0	-0.59	5.1
9.4	-0.37	4.4
9.9	0.56	9.9

```
t = np.array([0.4, 1.1, 1.4, 1.9, 2.5, 3.0, 3.4, 4.1, 4.4, 4.8,
                     5.5, 6.2, 6.5, 7.0, 7.5, 7.9, 8.5, 9.0, 9.4, 9.9])
v = np.array([ 0.25, 2.37, 1.69, 0.93, -1.0, 0.95, 1.22,
                      0.54, 0.37, 0.13, -2.04, -2.06, -0.81, -0.95,
                      0.98, 0.27, -0.81, -0.59, -0.37, 0.56])
n = np.array([2.8, 7.3, 9.7, 1.3, 6.2, 4.8, 6.9, 4.0, 1.9, 4.0,
                      9.5, 8.7, 2.3, 5.3, 9.7, 8.3, 0.1, 5.1, 4.4, 9.9])
```

Plot 2 Prepare a plot of the sample data subject to the following:

- Plot the voltage as a function of time as discrete data using red points.
- Define the boolean array `keep` based on the noise reducing condition $n <= 6.0$.
- Plot the voltage as a function of time, subject to the noise reducing condition using blue points.
- Plot the function $\sin(2\pi x/10)$ as a smooth function.
- Add appropriate axis labels.
- Add a legend for "raw" data with no cut, "clean" data with noise removed, and your continuous "sin" function.

Your plot will reveal a clear sine function in the discrete data (after noise removal) consistent with the continuous function. **This is a sign-off point for the lab.** (Each student should be prepared to explain any line in your code.)

1.4 The Logistics Map

The logistics map is the recurrence relation

$$x_{n+1} = r\, x_n\, (1 - x_n)$$

with the variable x between 0 and 1. The variable x can be thought to represent the ratio of a population to its maximum possible value. The population increases due to birth and decreases due to starvation as the population approaches it's maximum value (x near 1). This leads to the non-linear relationship that defines the logistic map. The mapping keeps the variable x between 0 and 1 as long as the parameter r is in the range $[0, 4]$.

The logistics map is frequently encountered as a simple example of a chaotic system emerging from a simple non-linear system. If we consider the long term behavior of the population x as a function of the parameter r, as shown in Fig. 1.3, we see that for values of r less than 3 the population approaches a single fixed value. At the value $r = 3$ the non-linear system exhibits bifurcation with the population oscillating between two values. As r increase, further bifurcations occur at an ever increasing rate until the systems exhibits chaotic behavior alternating with occasional returns to stable oscillations.

The long term behavior of the logistics map can be easily modeled in Scientific Python. A start is shown in Fig. 1.4 where you should understand:

- An array of r values is defined.

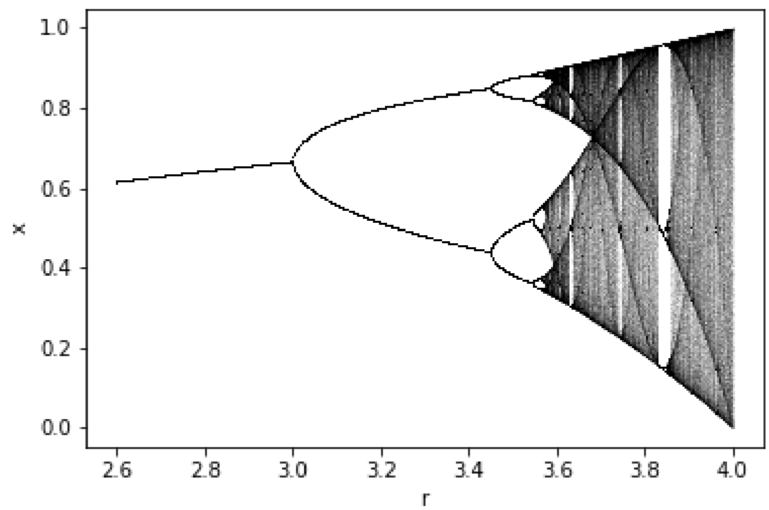


Figure 1.3: Long term behavior of the logistics map.

- An array of x values of the same size as r is defined and initialized to an arbitrary non-zero value (0.01).
- Two example iterations of the logistic map are applied.
- The next two iterations of the values of x are plotted as function of r on the same plot.

Plot 3: Reproduce the figure in Fig. 1.3 by doing the following:

- Define two global variables `ITER = 10` and `PLOT = 5`.
- Apply the logistics map `ITER` times by using a for loop.
- Apply the logistics map an additional `PLOT` times, plotting the values of x as a function of r, as in the example, each time.

You'll observe the long term behavior by increasing the value of `ITER` to a large value, such as 10,000. You'll see the full dependence on r by decreasing the step size in the initialization of the numpy array r to something like 0.001. You'll observe the chaotic behavior by increasing the value of `PLOT` to 100 or even 1000 iterations. To make a prettier plot using finer points (once you have a large number of points) you can reduce the size by adjusting the `s=10` parameter in the call to `plt.scatter` to something like `s=0.0001`.

```
r = np.arange(2.6,4.0,0.2)
print("r:  ", r)
R_SIZE = r.size
x = np.full(R_SIZE, 0.01)
print("initial x:  ", x)
x = r * x*(1.0 - x)
print("one iteration x:  ", np.around(x,2))
x = r * x*(1.0 - x)
print("two iterations x:  ", np.around(x,2))
# plot the next two iterations:
x = r * x*(1.0 - x)
plt.scatter(r,x,s=10,color="black")
x = r * x*(1.0 - x)
plt.scatter(r,x,s=10,color="black")
plt.xlabel("r")
plt.ylabel("x")
```

```
r:   [2.6 2.8 3.  3.2 3.4 3.6 3.8]
initial x:   [0.01 0.01 0.01 0.01 0.01 0.01 0.01]
one iteration x:   [0.03 0.03 0.03 0.03 0.03 0.04 0.04]
two iterations x:   [0.07 0.08 0.09 0.1  0.11 0.12 0.14]

Text(0,0.5,'x')
```

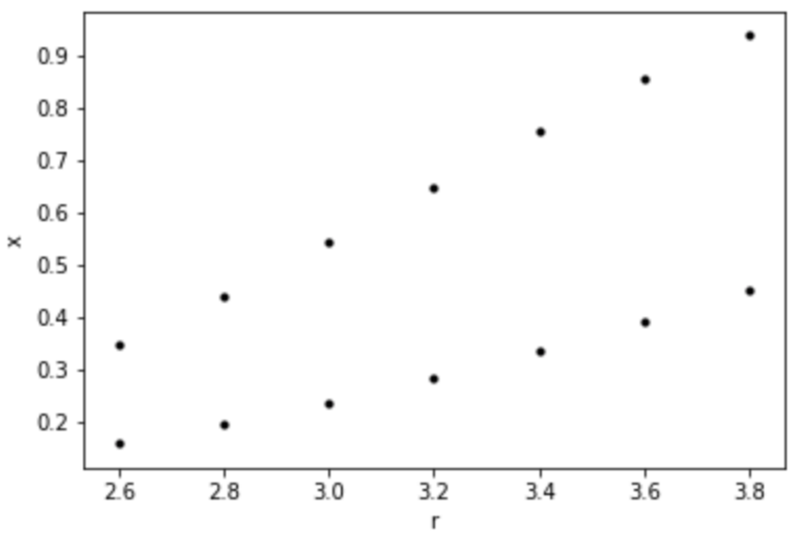


Figure 1.4: Modeling the logistics map.

Chapter 2

DC Circuits

2.1 Introduction

In this lab, you will learn how to use your digital multimeter (DMM) and bench-top DC power supply to explore DC circuits involving resistors. You will experimentally verify Ohm's law and the equivalent resistance for resistors in series and parallel.

If time permits, you will also solder two resistor circuits to explore the Δ-Y transformation for three terminal networks.

2.2 Benchtop Power Supply

Figure 2.1: Your bench-top DC power supply, the MASTECH 3005F-3.

In this lab you will use one channel of your MASTECH 3005F-3 DC power-supply as the voltage source in your experimental circuits. See Fig. 2.1 for the location of the control features. Since you won't know initially what settings your power-supply was left at, leave the supply disconnected while you configure the supply to provide 10 V.

First, check that the supply is set to provide two independent outputs (you will only use one for this lab) by making sure both bush buttons are out. Next, power the device by pressing the power button. Your power supply has both a current limit knob and a voltage limit knob. Usually, we specify the voltage you want in your circuit, but even in this case, the current limit is useful for protecting your circuit and measurement equipment. When the LED labeled "CV" is lit, the voltage limit is controlling the output. When the LED labeled "CC" is lit, the current limit is controlling the output.

Turn the voltage limit knob clockwise and watch the voltage increase on the display. If the CC LED lights, it indicates that the current limit is active, and you will not be able to raise the voltage. Turn the current limit knob until just a bit after the CV LED lights, indicating the voltage limit is active. Continue raising the voltage until you reach the 10 V. Once you reach your target voltage, turn the current limit counter-clockwise, to lower the current limit, until the LED labeled "CC" lights, indicated you have reached the current limit. Turn the current limit clockwise just past the point that the device goes back to being voltage limited. If you get in the habit of doing this every time you use your bench-top supply, you will safe many components for accidental damage!

The voltage on the display is maintained between the black (-) terminal and the red (+) terminal. It is floating, meaning that only the difference is set by the device, just like a battery, and you may connect ground wherever you choose (within the limits of the supply). If you wish to provide a ground referenced DC voltage, you connect the green terminal to the appropriate terminal. For example, connecting green to red would provide -10 V referenced to ground at the negative terminal. You will leave the supply floating for this experiment.

2.3 Voltage Measurement

Your primary digital multimeter (DMM), the Triplett 9007 shown in Fig 2.2, can be used to make a number of measurements including resistance, DC current, and DC voltage. We'll measure a DC voltage to start. This lab will be most convenient if you use alligator clip probes in your DMM. Install a black alligator clip probe in the "Common" terminal, and a red alligator clip probe in the "Voltage" terminal directly above the Common terminal. Clip the alligator clips together so that you expect to measure 0 V. Now power the DMM by pressing the power button, and turn the voltage dial to the DC voltage measurement, the V with one straight and one dashed line next to it. Most measurements on your DMM have a number of different full range settings. For the highest precision, you should use the smallest full-range setting larger than your measurement. We'll be measuring 10 V, so use the 20 V (DC) setting.

With the alligator clip probes connected together, your DMM should read 0.00 V. Now touch the probes to the terminals on your bench-top DC voltage supply: red probe to red terminal, and black probe to black terminal. You should read 10 V. Next touch red probe to black terminal and black probe to red terminal. You should read -10 V. This is because the "Common" (black) probe is the reference point for the measurement, and the red probe is currently connected to a point 10 V lower than this reference point.

(a) (b)

Figure 2.2: Your (a) digital multimeter (DMM), the Triplett 9007, and (b) alligator clip probes.

2.4 Resistance Measurement

Pick two different resistors from the random collection of resistors at the front of the lab. Connect the alligator clip across one of the resistors, and turn the dial to the resistance measurement, marked Ω. When the measurement reads "1" the resistance is larger than the full range. Find the smallest range larger than your resistor and record the measured value. Using the resistor color coding guide in Fig. 2.3, determine the nominal resistance of your selected resistor. In your logbook, record your resistor color pattern, the nominal value you determined, and the measured value. Repeat this for a second resistor.

2.5 Current Measurement

Your DMM is designed to have minimal impact on the circuit your are measuring. When used to measure the voltage across a component, such as a resistor, it is connected in parallel, as in Fig. 2.4a. The ideal voltmeter therefore has infinite resistance, so that no current from the circuit is diverted through the voltmeter. Your DMM has a 10 MΩ resistance when used as a voltmeter.

When used to measure the current through a component, your DMM is connected in series, allowing the current to pass through the device. The ideal ammeter therefore has zero resistance, so that there is no voltage drop across the voltmeter. For this reason, most DMMs have separate terminals for measuring currents.

You make a current measurement with a DMM by installing the red probe in a current terminal (instead of the voltage terminal) the black probe in the common terminal as before, and installing the probe in series with the component you wish to measure the current through.

However, the current measurement in your Triplett 9007 is fused, and you can fairly safely assume that the fuse is blown. Because the current measurement is nearly a short circuit, it is very

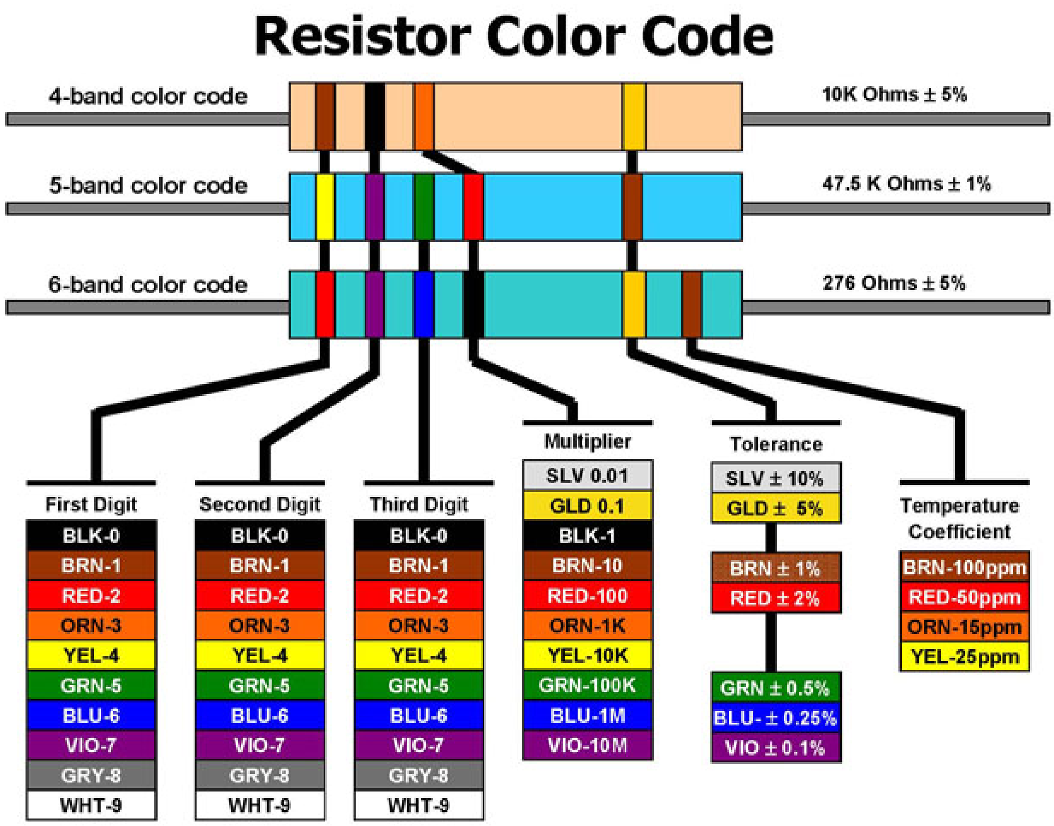


Figure 2.3: The resistor color code.

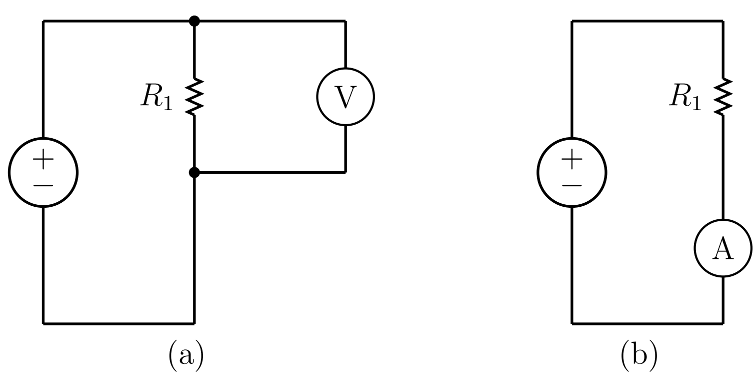


Figure 2.4: Appropriate connections for measuring (a) the voltage across resistor R_1 and (b) the current through the resistor R_1.

easy to introduce an unintentionally large current by connecting a voltage across the terminals, as if to measure a voltage. Even experience is no sure remedy for this common lab mishap.

Instead, we will use the Mastech MS8264 which features a resettable fuse for making current measurements.

2.6 Breadboard

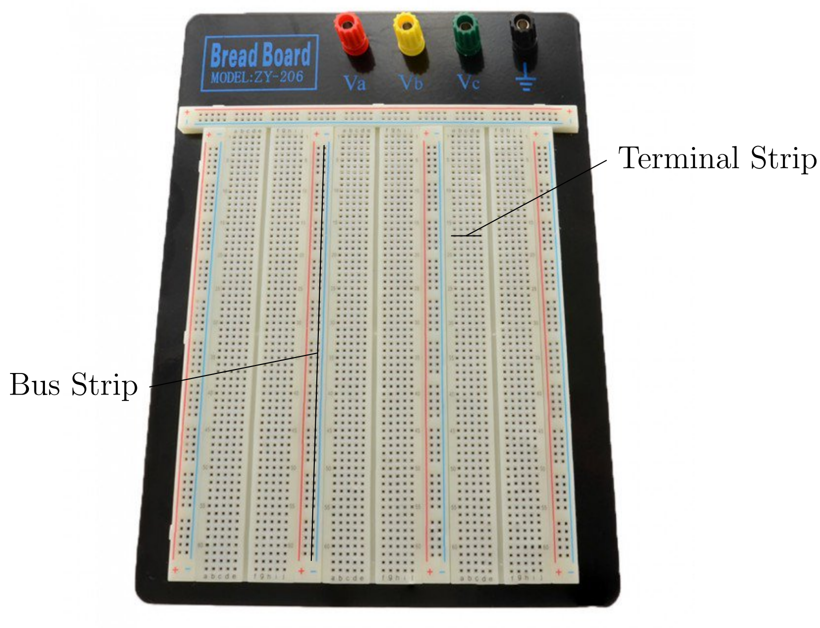


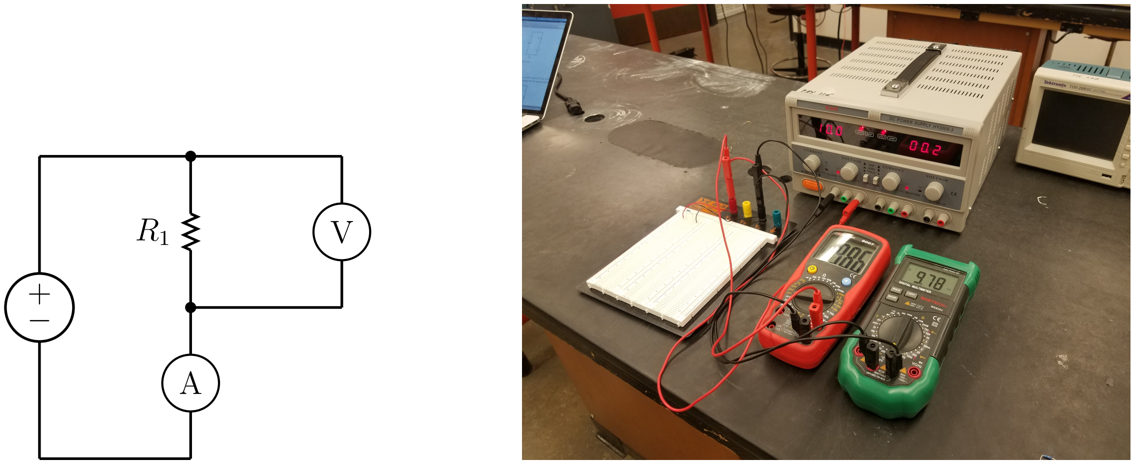
Figure 2.5: A typical breadboard.

Breadboards are a convenient way to prototype circuits without having to solder. When the leads of discrete components are inserted in the holes in the breadboard, they make electrical contact with metal strips inside the breadboard that connect with additional holes. The short terminal strips are used to make electrical connections between components. The longer bus strips are a convenient way to bring in ground or voltage supplies that need to connect to many places in circuit.

2.7 Verification of Ohm's Law

Build the circuit in Fig. 2.6. Use a resistor $R_1 = 1.0$ kΩ with a 1% tolerance. Use your Triplett 9007 as the voltmeter and the Mastech MS8624 as the current meter. Use your benchtop power supply to provide the DC voltage. Use banana plug connectors for the current measurement, installing the Mastech MS8624 between the supply and the breadboard.

By adjusting the voltage setting of the power supply, take a series of voltage and current measurements with voltage across the resistor at target voltages from 1 to 10 V in steps of 1 V. Generally, you can measure more precisely than you can control, so never fuss about trying to measure the voltage at exactly the target value, instead, simply record e.g. $V = 1.04$ V along with your current measurement and move on to the next target value.



(a) (b)

Figure 2.6: Circuit for verifying Ohm's law as a (a) circuit diagram, and (b) implemented using your lab equipment.

While recording data, check that the current values you measure are consistent with what you expect given the voltage across the resistor and resistance. You should *always* make quick sanity calculations when collecting data, otherwise you risk wasting time collecting useless data!

Plot 1: Plot the current versus voltage of your ten data points (using option `"o"`). Draw a line (using option `"-"`) for the current versus voltage curve of a 1.0 kΩ resistor. Make certain your plot has appropriate axis labels, including appropriate units in parenthesis, and a legend distinguishing data from your expectation ("expected").

Measurement 1: After taking your last measurement, leave all the connections in place and the power-supply at 10 V. Record in your log book the resistance of the resistor R_1 reported by your DMM. Is this a reasonable measurement? **Measurement 2:** Turn off the DC supply and record the resistance reported by the DMM. Is this accurate? **Measurement 3:** Remove the resistor from your circuit and measure the resistance with your DMM. Is this accurate?

2.8 Voltage Divider

One circuit you will encounter again and again is the humble voltage divider circuit of Fig. 2.7 a. Modify your setup to include an additional resistor $R_2 = 4.7$ kΩ in series with your resistor $R_1 =$ 1 kΩ. Before installing it in your circuit, record the actual value of your resistor R_2 in your log book.

Measurement 4: adjust the supply voltage to 10 V and record the voltage across resistor R_1, the voltage across resistor R_2, and the current through the divider. Compare these measured values to your expectation.

Now adjust your circuit so that R_1 and R_2 are in parallel and set the supply to 10 V **Measurement 5:** Record the voltage across the resistors R_1 and R_2 and the current through each resistor. Compare the measured currents to your expectation.

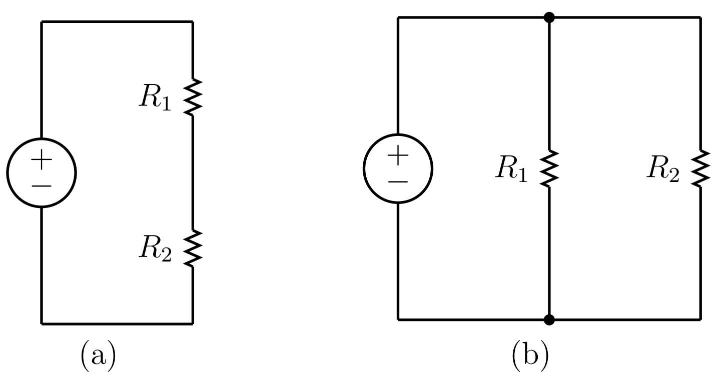


Figure 2.7: Circuits for studying resistors (a) in series, and (b) in parallel.

Chapter 3

Equivalent Circuits

3.1 Pre-lab Calculation

1) Determine an equation for the Thevenin equivalent voltage V_{th} and resistance R_{th} from the values V_1, V_2, R_1, R_2, R_3 for the circuit shown in Fig. 3.1. Hint: Use the superposition principle. Find the equivalent resistance by setting the voltage V_1 and V_2 to zero, i.e. shorting them in the circuit. Then calculate two contributions to the Thevenin voltage, one with V_1 set to zero and one with V_2 set to zero. The actual Thevenin voltage is the sum of these two contributions. Play close attention to the polarity of V_2 as drawn, i.e. that a positive value of V_2 tends to make the voltage V_{ab} negative.

2) Compute V_{th}, R_{th}, and the short-circuit current I_{sc} for the particular values of R_1,R_2,R_3,V_1, and V_2 you will be using in the lab.

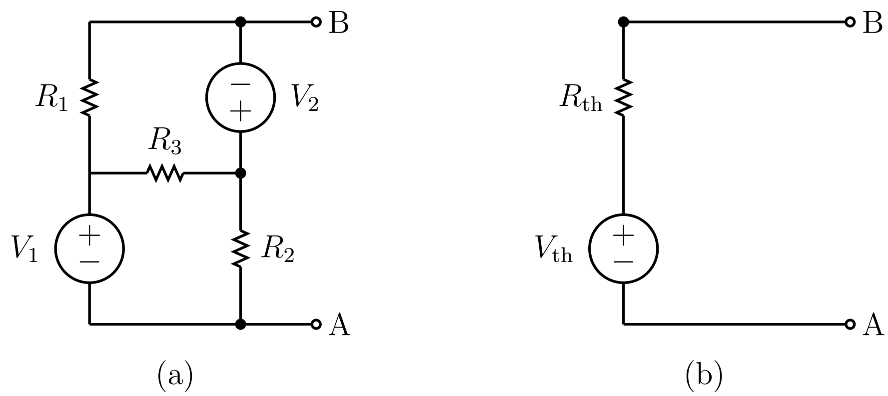


Figure 3.1: The circuit (a) you will be building in lab and it's (b) Thevenin Equilvalent.

3.2 Thevenin Equivalent Circuit

Build the circuit in Fig. 3.1 using $R_1 = 3.3$ kΩ, $R_2 = 3.9$ kΩ, and $R_3 = 4.7$ kΩ. Supply $V_1 = 10$ V and $V_2 = 5$ V using your two channel bench-top power supply. In the diagram, the supplies are not referenced to ground or each other, so make certain that your supply is set to provide independent outputs and do not add any jumpers to ground. Take careful note of the polarity of the supplies, so e.g. the negative (black) output of V_1 is connected to point (b) whereas the negative (black) output of V_2 is connected to point (a).

Use your Triplett 9007 as a voltmeter and the Mastech MS8624 as a current meter. First measure the open circuit voltage $V_{\rm ab}$. Next short the points (a) and (b) through your current meter. These values should closely match the Thevenin voltage and short-circuit current which you have already calculated. If not, you should check your work and find the discrepancy before proceeding.

Next you will measure the voltage across and current through a load resistor connected between the terminals at (a) and (b) to experimentally determine the IV curve for your circuit. Recall from the previous lab that you measure the current by connecting your meter in series and the voltage by connecting your meter in parallel. As before, use your Triplett 9007 as a voltmeter and the Mastech MS8624 as a current meter.

Make simultaneous current and voltage measurements for three different values of the load resistance $R = 470\ \Omega, 1.2\ \text{k}\Omega, 4.7\ \text{k}\Omega$.

3.3 Analysis

Plot 1: To present your analysis you should produce a part like that of Fig. 3.2. Your plot should show the Thevenin equivalent source IV curve for the circuit you built in lab. You should also draw theoretical load IV curves for the three resistor values you used to make current and voltage. Finally, you include data points for the five measurements you made.

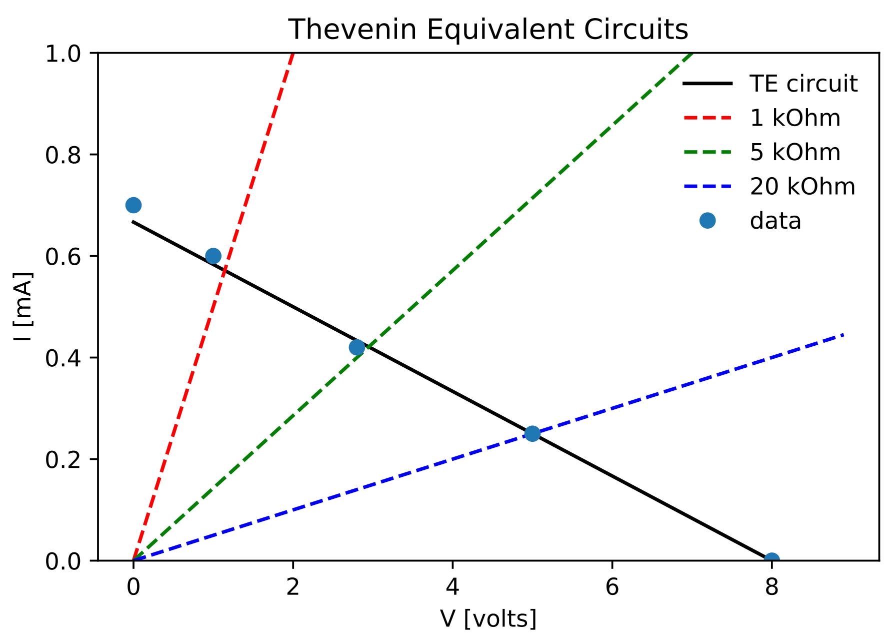


Figure 3.2: Using boolean masks to cut on variable y.

3.4 $\Delta - Y$ Transformation

Consider the two different networks shown in Fig. 3.3. If there are no external connections to the central node in the left-hand circuit, the two networks are equivalent if:

$$R_A = \frac{R_{AC} R_{AB}}{R_{AB} + R_{AC} + R_{BC}}$$

as well as two similar equations for R_B and R_C. Going in the other direction we have:

$$R_{AB} = \frac{R_A R_B + R_A R_C + R_B R_C}{R_C}.$$

These transformations are more general than the series and parallel laws, which you can derive by considering the case that $R_{BC} = 0$ for parallel resistors, and $R_C \to \infty$ for series resistors. They allow one to simplify more complicated networks for which the series and parallel equivalence relations are insufficient.

In the special case that $R_A = R_B = R_C = R$ it follows that

$$R_{AB} = R_{AC} = R_{BC} = 3R.$$

If time permits, use your soldering iron to construct the left-hand network using $R_A = R_B = R_C = 1$ kΩ. Then construct the equivalent right-hand network using $R_{AB} = R_{AC} = R_{BC} = 3.0$ kΩ. Since 3 kΩ is not a standard sized 10% resistor, you can construct one by using a 33 kΩ in parallel with a 3.3 kΩ resistor.

Make sure the soldering iron is on, and the sponge is moist. Twist the leads of the resistor together to make initial connections, then hold the arrangement securely in the clamp. Wipe the tip of the hot iron on the sponge to clean it, then apply a small amount of solder to the tip by touching the hot iron to the solder wire.

Heat the connection by holding the soldering iron against it, then bring the solder wire in contact with the heated connection (not the soldering iron) You want the iron to heat the connection, and then the connection to melt and draw in the solder. The little bit of solder on the tip is only there to ensure good thermal conduct between the tip and the connection: don't "paint" the solder onto the connection.

Measurement 1: Check the resistance between pairs of terminals on your creations, and compare with your expectation. You can bring your creations home if you like.

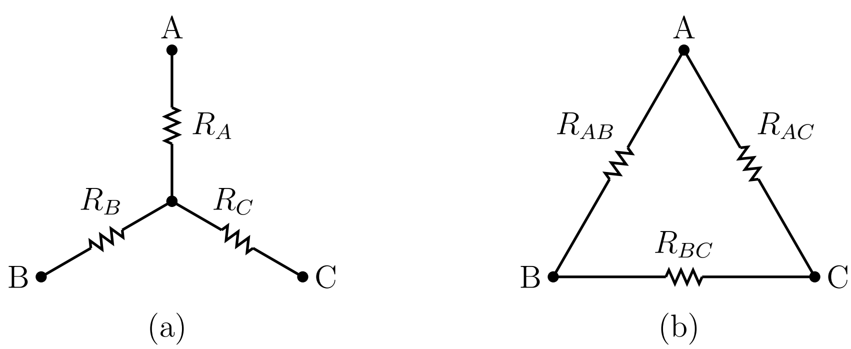


Figure 3.3: Equivalent three-node circuits.

Chapter 4

Alternating Current and Time Varying Signals

4.1 Introduction

In this lab you will use two essential new pieces of lab equipment: the digital oscilloscope and the function generator. You will learn how to measure AC voltages with your DMM, as well as how to view time-dependent wave forms on your digital oscilloscope. You'll view Lissajous curves in 2-D using the X-Y mode of your scope and reproduce the same curves using parameterized equations in Scientific Python.

4.2 Function Generator

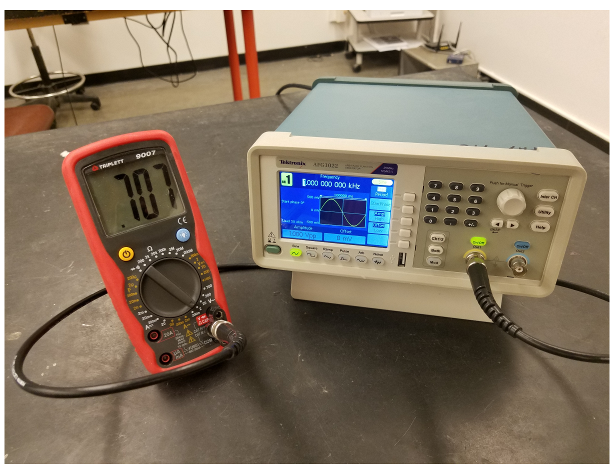


Figure 4.1: Connect the Channel 1 Output of your function generator directly to your DMM.

Connect the output of Channel 1 directly to the Voltage measurement input of your Triplett 9007 DMM, using a BNC to banana plug adapter as shown in Fig. 4.1. Turn on power to the function generator. Then set your function generator to the factory default:

Utility Button → System → Set to Default → Select.

You must perform this step today for the instructions that follow to make sense. With shared equipment, it is essential to know how to restore the factory default, in case another user has left the device with strange settings. You don't need to start with this step every lab, but it is a fast way to recover when you encounter strange behavior in your equipment.

The factory default settings are set to produce a Sine function with a peak-to-peak voltage $v_{\rm pp} = 1.0$ V and a frequency $f = 1$ kHz. We'll leave that as is for now. To turn on the output, push the "On/Off" directly above the coaxial output for Channel 1, and then ensure that the button is lit.

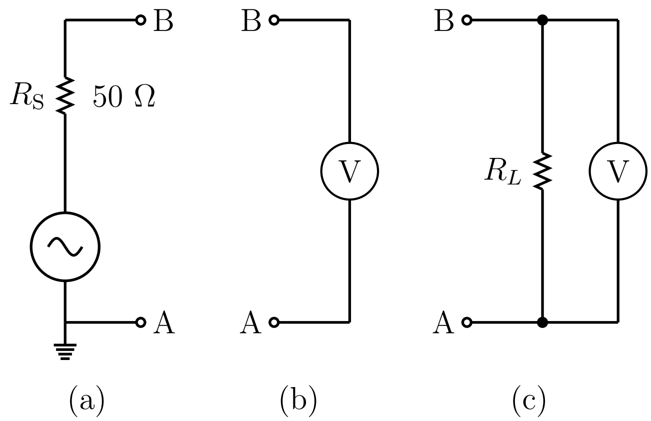


Figure 4.2: Equivalent circuit for (a) your function generator, which includes a 50 Ω source resistance, and two typical terminations for a coaxial signal: (b) infinite resistance voltmeter or scope, or (c) a terminating resistorin parallel.

Set your DMM to the 2 V (AC) scale (the V with a squiggly line). And you should now measure the AC voltage

$$v_{\rm rms} = 0.707 \text{ V} \sim \frac{1}{\sqrt{2}} \text{ V}$$

However, recalling the relationships between the peak-to-peak voltage, the peak-voltage, and the RMS voltage of an AC sine wave:

$$v_{\rm pp} = 2\, v_{\rm p} = 2\sqrt{2}\, v_{\rm rms}$$

we expect our function generator, set to $v_{\rm pp} = 1.0$, to produce output with:

$$v_{\rm rms} = \frac{v_{\rm pp}}{2\sqrt{2}} \sim 0.353 \text{ V}$$

Clearly someone is lying to us! In fact, we've encountered a very common source of factor of two mistakes. The equivalent circuit for your function generator is shown in Fig. 4.2a. Notice that it includes a 50 Ω source resistance in series with the AC voltage produced by the function generator. This internal resistance is important for a number of reasons, most notably making it impossible to short-circuit the output and destroy the equipment!

In our setup, we've connected the function generator output directly to your DMM, which has a very high input resistance, effectively infinite, as shown in Fig. 4.2b. However, the standard termination for coaxial cables is 50 Ω, and the default setting for your function generator expects the load shown in Fig. 4.2c with $R_{\rm L} = 50$ Ω. In this case, the internal resistance and load resistance form a voltage divider, so that the output voltage $V_{\rm AB}$ seen by the user is 1/2 the internal AC voltage. The function generator is designed to produce an internal AC voltage which is twice the value selected by the user, so that the output voltage is precisely the value specified by the user. We are seeing twice our requested value, because we have no load resistor, and so no voltage divider, and instead see the full value of internal AC voltage. To fix this discrepancy, we simply have to configure our generator to expect a high load resistance at both outputs:

$$\begin{aligned} \text{Utility Button} \rightarrow \text{CH1Load} &\rightarrow \text{HighZ} \\ \text{CH2Load} &\rightarrow \text{HighZ} \end{aligned}$$

Press the "Ch1/2" button until you return to the Channel 1 menu. Adjust the amplitude to 1 V peak-to-peak by:

$$\text{Ampl} \rightarrow 1 \rightarrow \text{Vpp}$$

Your DMM should now read the expected value:

$$v_{\text{rms}} \sim 0.353 \text{ V}$$

Press the button next to Ampl a couple of times. There is a slightly annoying feature of your function generator which allows you to specify either the Amplitude and Offset or the High and Low voltage values. So if you want to adjust the Amplitude, you have to press the button next to Ampl until the Ampl label is highlighted. Often you'll end up setting the wrong value by mistake. But in general, whatever parameter is highlighted along the side of the screen is the parameter which you can specify by either the knob or the key pad. Keeping this is mind, set your function generator to produce 1 V RMS output:

$$\text{Ampl} \rightarrow 1 \rightarrow \text{Vrms}.$$

Now your DMM should also read a value quite close to one.

Now let's adjust the frequency. Highlight the frequency parameter by pressing the button next to the "Freq" option until it is highlighted:

$$\text{Freq} \rightarrow 10 \rightarrow \text{kHz}.$$

You can also adjust the selected parameter with the multipurpose knob. Turn the multipurpose knob until the frequency is around 100 kHz and observe what happens to your DMM measurement. The reason your measurement is now inconsistent with the setting in the function generator, is that your DMM is only rated to 2 kHz. It isn't intended for measuring high-frequency AC signals. Turn the frequency back down to 1 kHz.

Next highlight the Offset parameter on your function generator and adjust it to 2 V. This will add a DC offset to your function generator output. After settling down, the measured value of the AC voltage on your DMM should be unchanged at 1 V. Switch your DMM to measure the DC voltage and you should now measure the 2 V DC offset. Pay attention to the sign. If you see a negative value, it is because you installed your BNC-to-banana adapter incorrectly. Notice that one side of the adapter has a small raised tab, indicating which side connects to the coaxial cable shield. The side with the raised tab should be plugged into the Common socket. Whether you got lucky this time or not, change the orientation of the adapter a few times and observe how the sign of the voltage changes, finally plugging it back in with the correct orientation. Now adjust the DC level with the multipurpose knob and observe the change on your DMM. When satisfied, set the offset back to zero.

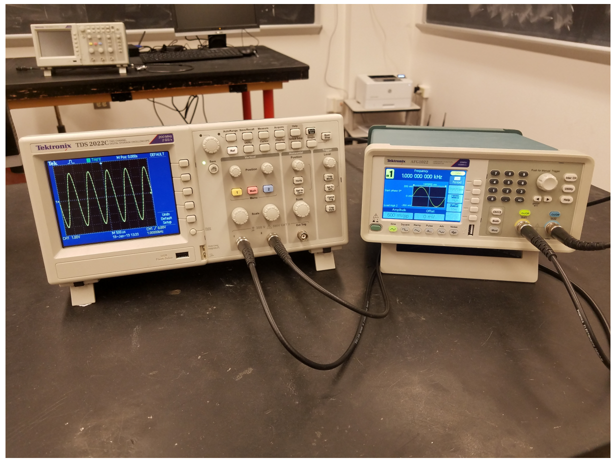


Figure 4.3: Connect the Channel 1 Output of your function generator directly to your DMM.

4.3 Oscilloscope

Put your DMM aside. Connect the Channel 1 output of your function generator to the Channel 1 input of your digital oscilloscope. Do the same for Channel 2. The setup is shown in Fig. 4.3. Set your function generator to provide a 1 kHz sine wave with peak-to-peak voltage of 600 mV. (A DC offset of zero is implied unless otherwise stated) Press the "Default Setup" button on your digital scope. You should immediately observe a sine wave on your Digital scope just as in Fig. 4.3.

Press the button labeled "Square" on your function generator to change the output from a Sine wave to a square wave and observe the waveform on your scope. Do the same for the Ramp and Noise functions. Then return to a Sine wave.

Press the yellow button labeled "1" several times. This button turns on and off the display of channel 1, and brings up the Channel 1 parameter menu. Notice that the voltage scale for Channel 1 is indicated as 1.0 V. This means that the difference between each pair of consecutive horizontal lines corresponds to 1 V. We say "One volt per division". By counting divisions, you should be able to see that your waveform has a peak-to-peak voltage of 6 V. Yet your function generator is set to produce 600 mV = 0.6 V. Clearly someone is lying to us!

Although we won't be using them in this lab, most sensitive measurements with an oscilloscope are made using a scope probe, as shown in Fig. 4.4. To protect the circuit being measured from being effected by the insertion of the probe, there is usually a large resistance in the probe. This means that the oscilloscope itself measures the output of a voltage divider, and the signal is attenuated, most often by a factor of 10. The oscilloscope simply adjusts the voltage scale so that values you read are not attenuated. To make consistent measurements, you simply have to make sure that the oscilloscope is configured for the attenuation factor we are using.

In our case, we are connecting coaxial cables directly between the oscilloscope and the function generator, and so their is no attenuation. But the default setup for the scope assumes that you are using a probe with a 1/10 attenuation, called a 10X probe. Look at the options next to the menu buttons and find the one that says "Probe 10X Voltage". Press this menu button, and then press the Attenuation button until it reads 1X, appropriate for a coaxial cable with no attenuation factor. The waveform is unchanged, but now the voltage scale is correctly set to 100 mV. And your signal now appears to be 600 mV, consistent with the setting from your function generator, as shown in Fig. 4.5. Next turn the knob labeled "scale" located under the yellow channel "1" button.

Figure 4.4: An example scope probe.

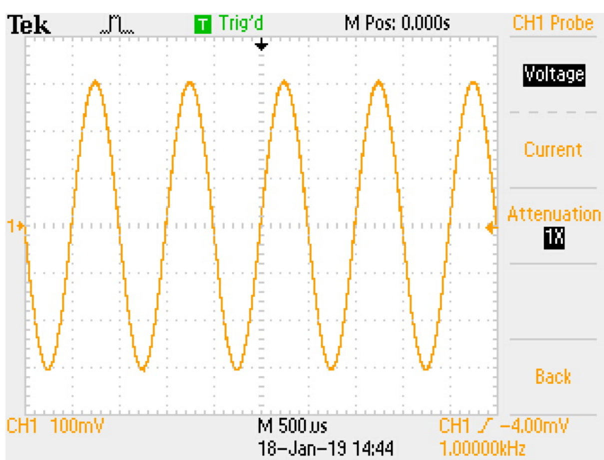


Figure 4.5: Correctly scaled scope output.

Adjust this knob until the scale for CH1 is listed as 200 mV per division. The apparent size of the waveform will be reduced by a factor of two, because each division is now 200 mV and so your 600 mV signal appears three divisions high.

Next note that the function repeats every two divisions. Since the time scale is listed as 500 μs, the period is therefore 1 ms, corresponding to a frequency of 1 kHz. Adjust the time scale, using the large knob in the Horizontal column, until the time scale is 100 μs per division. This is still a 1 kHz signal, but one period now takes up the entire display.

Using the multipurpose knob on your function generator, adjust the frequency up to 10 kHz, and observe how the waveform changes. Then adjust the voltage between about 100 mV and 2 V peak-to-peak. When finished, leave the function generator producing a 5 kHz sine wave with 600 mV peak-to-peak voltage. Your scope should remain at a voltage scale of 200 mV and time scale of 100 μs. Next, set the DC offset of the signal on the function generator by pressing:

$$\text{Offset} \rightarrow 10 \rightarrow \text{mV}$$

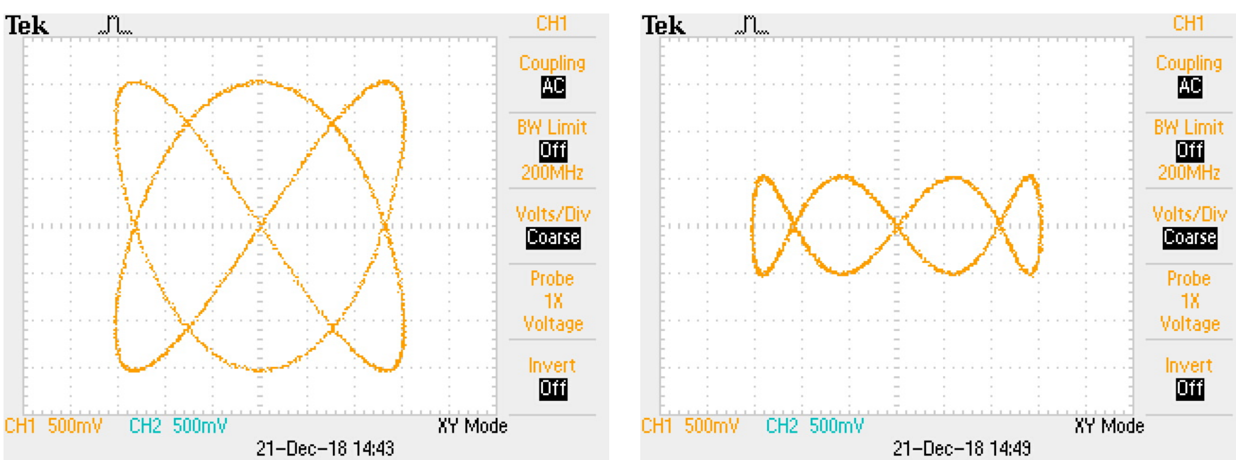
Turn the multipurpose knob to adjust the DC offset between -100 mV and 100 mV. Your waveform will rise and fall on your display. By default, your scope includes the DC offset, but often this is not what you want. On your scope, press the button labeled "Coupling DC", until the DC becomes AC. When AC coupled, the DC component of your waveform is ignored. When AC coupled, observe that changing the DC offset on the function generator does not change the position of the waveform.

You can adjust the position of the waveform using the small knobs labeled "Position" to adjust the offset in vertical and horizontal. Try this out. To return a waveform to (0,0), notice that the offset is displayed while you are turning the knob.

On your function generator, set the output to a 5 V peak-to-peak sine wave with frequency of 100 kHz. Adjust the voltage scale and time scale until you can clearly see the sine wave.

You now know most of the key features of your scope apart from the trigger, which we'll leave for another day!

4.4 Lissajous Figures



(a) (b)

Figure 4.6: Scope traces from Lissajous figures from settings for (a) start, and (b) crown.

Lissajous figures are the graph of system of two parameterized functions:

$$
\begin{aligned}
x &= A_1 \sin(2\pi f_1 t + \delta) \\
y &= A_2 \sin(2\pi f_2 t)
\end{aligned}
$$

which produces a closed loop if the ratio A_1/A_2 is rational. The appearance of the figure is of a 3 dimensional knot with the viewing angle determined by the parameter δ. Two examples are shown in Fig. 4.6.

To produce these figures on your scope, we'll need to use two channels. To begin, enable the output of both Channel 1 and Channel 2 on your function generator, and set them both to produce sine waves with amplitude 3 V peak-to-peak. Adjust the frequency of channel 1 to 2 kHz and the channel 2 to 3 kHz. Note that you can switch between the Channel 1 and Channel 2 parameter menus with the button labeled "Ch1/2".

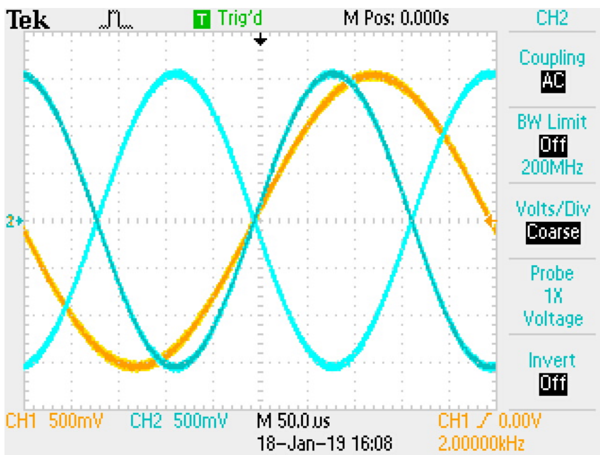


Figure 4.7: Correctly scaled scope output.

On your scope, switch to the Channel 2 parameter menu by pressing the blue button labeled "2". Set the coupling of Channel 2 to AC, and probe attenuation to 1x, just as you did previously for Channel 1. Next adjust the voltage scales of each channel to 500 mV and set the common time scale to something appropriate, so that you can view both Sine waves on the display. As shown in Fig. 4.7, you will see two versions of the Channel 2 output, inverted with respect to each other, because the frequency of Channel 2 is 1.5 times the frequency of Channel 1.

The relative phase between the two output channels of your function generator shifts whenever you adjust the frequency of one of the signals. For consistent results with offline plots and the scope traces shown here, you'll need to align the phase of the two channels every time you adjust the frequency on the function generator:

$$\text{InterChbutton} \rightarrow \text{AlignPhase}.$$

Usually, scopes are used to display the inputs as a function of time. In this case, the voltage level is along the y-axis, and time is the x-axis. This mode is called YT mode. Occasionally, however, it is useful to display things in XY mode. In this mode, the x-axis is used for the voltage of Channel 1 and the y-axis is used for the voltage of Channel 2. Each point on the curve represents a particular point in time. Switch to XY mode by pressing the Display button and then pressing the button next to the Format menu item until the mode is XY. You should reproduce Fig. 4.6a exactly. If

Table 4.1: Settings for various Lissajous figures.

pattern	f_1 (kHz)	f_2 (kHz)	δ_1
start	2	3	0
fish	2	3	135°
parabola	1	2	45°
lace	13	12	0
crown	1 kHz	4 kHz	0

not, check that you have aligned the phase as described above and that frequencies are set correctly as in Table.

Adjust the phase of Channel 2, under menu item StartPhase, until the pattern collapses into a Fish pattern (or greek letter α) at 135 degrees. Save a scope trace by inserting your USB drive into the scope and pressing the Save button. Then produce the parabola and lace patterns, according to the settings in Table 4.1, saving a scope trace each time. Remember to align the phase each time you change the frequency.

Next, produce the crown pattern, shown in Fig. 4.6b. For the right proportions, adjust the amplitude of Channel 2 to 1 V peak-to-peak, leaving Channel 1 at 3 V peak-to-peak. Notice that as you adjust the phase of Channel 1, the crown appears to rotate. Adjust the frequency of Channel 2 to 4.0002 kHz. The crown should now appear to rotate constantly at low speed. This is a **sign off** point in the lab.

4.5 Analysis

From the previous section, you should have scope traces for the fish, parabola, and crown. Reproduce each of these figures using scientific python to draw the parameterized shape. For example. Fig. 4.8.

One way to approach this problem is to set the period to 1 μs, with fundamental angular frequency $\omega = 2\pi$ kHz.

One way to approach this problem is to set the period to 1 μs. The functions should be evaluated at 1000 discrete times within the interval from 0 to 1 μs.

```
t = np.linspace(0,1,num=1000)
```

Define a fundamental angular frequency $\omega_0 = 2\pi$ kHz:

```
w = 2*np.pi
```

With these definitions, we would define:

```
x = np.sin(4*w*t)
```

to obtain x points corresponding to $f = 4$ kHz sine function.

When plotting your curves, use:

```
plt.axis('equal')
```

to keep the unit aspect ratio used by your scope. You can display your scope traces in python using the Image library:

```
import Image

image = Image.open('myscope.jpg')
image.show()
```

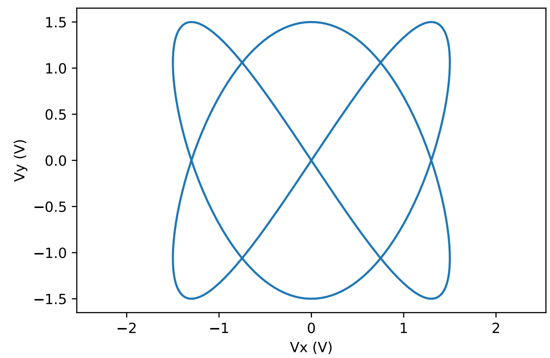


Figure 4.8: Lissajous curve constructed using Scientific Python corresponding to the scope trace in Fig. 4.6a.