Physics 80 Lab Manual

December 23, 2018

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 the sin function sampled a 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 must 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:

```
# 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>
```

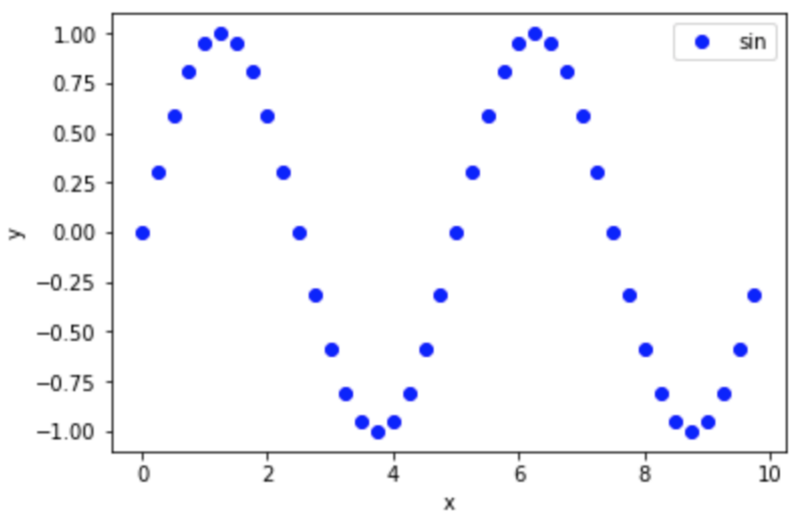


Figure 1.1: Circuit for verifying Ohm's law as a (a) circuit diagram, and (b) implemented using your lab equipment.

- 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 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.

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 $[0, 4]$.

This logistic 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.

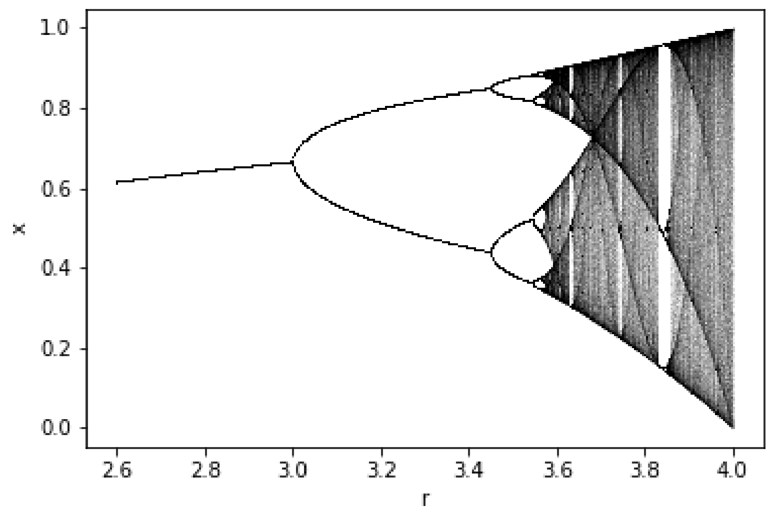


Figure 1.3: Long term behavior of the logistics map.

- 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')
```

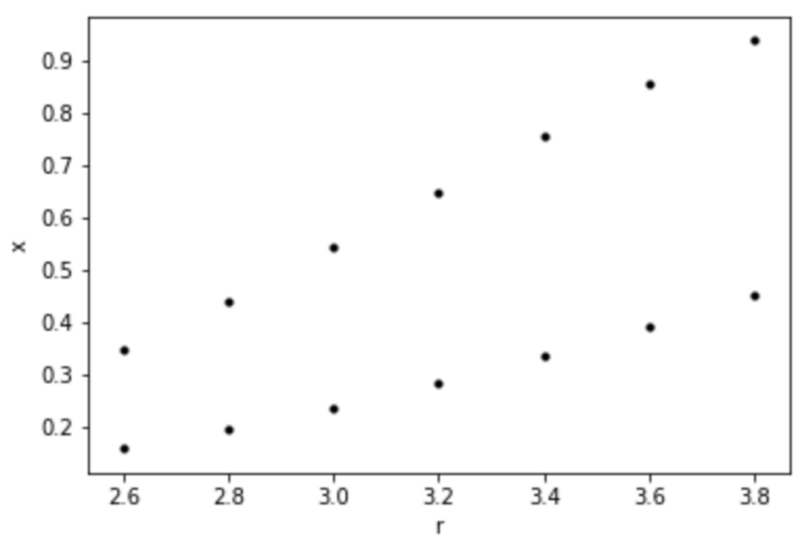


Figure 1.4: Modeling the logistics map.

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. You will solder two resistor circuits to explore the Δ-Y transformation for three terminal networks.

2.2 Benchtop Power Supply

2.3 Digital Multimeter

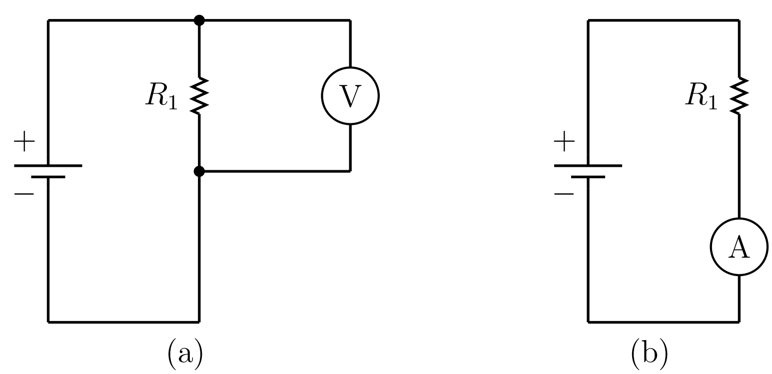


Figure 2.1: Circuits for verifying Ohm's law.

In this lab, you will use two new pieces of lab equipment: your bench-top power supply and your digital multimeter (DMM).

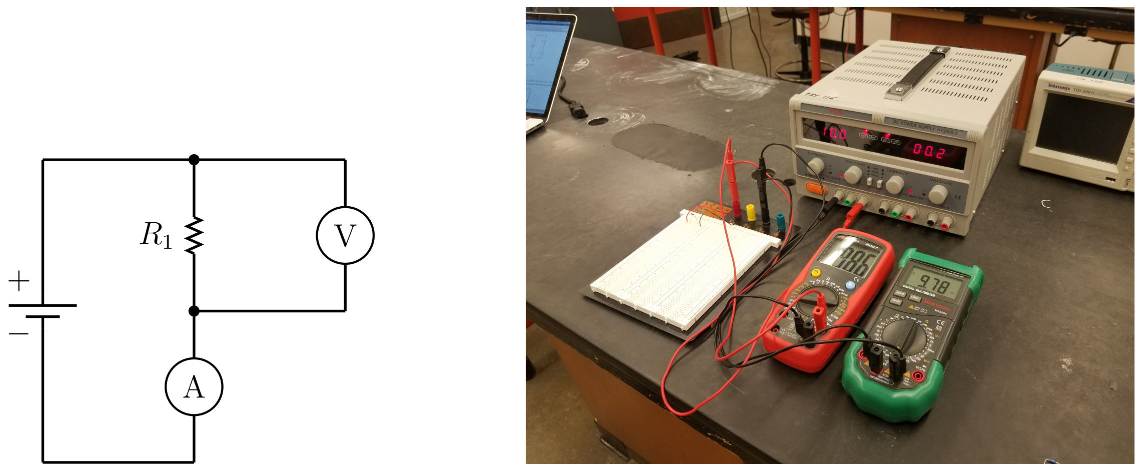
Your bench-top supply provides up to XX volts of DC power on two independent channels. For this lab, we'll only be using a single channel. Each supply has two associated knobs, one controlling the maximum allowed current, and one controlling the maximum voltage. The supply will provide the maximum voltage subject to those constraints. Usually your are primarily concerned with setting the voltage by the voltage knob, but it is a good habit, which will save you from losing components, to set the current as low as possible. An LED indicates if your circuit is current limited or voltage limited. You can adjust the current limit until just beyond the point where your circuit become voltage limited.

The supply is floating, it provides the specified voltage between the black and red outputs without referencing either to ground. If you want to provide a ground referenced voltage you explicitly connect the green output to the red (positive) terminal or the black (negative) terminal.

(Discuss limits)

Your DMM can measure voltage, resistance, and current.

2.4 Verification of Ohm's Law



(a) (b)

Figure 2.2: Circuit for verifying Ohm's law as a (a) circuit diagram, and (b) implemented using your lab equipment.

Build the circuit in Fig. 2.2. 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 voltage.

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.

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?

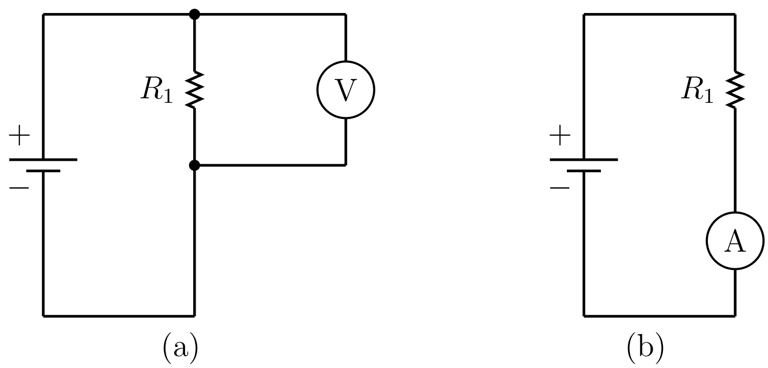


Figure 2.3: Circuits for verifying Ohm's law.

2.5 Voltage Divider

One circuit you will encounter again and again is the humble voltage divider circuit of Fig. 2.4a. 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 total current through both resistors. Compare the measured current to your expectation.

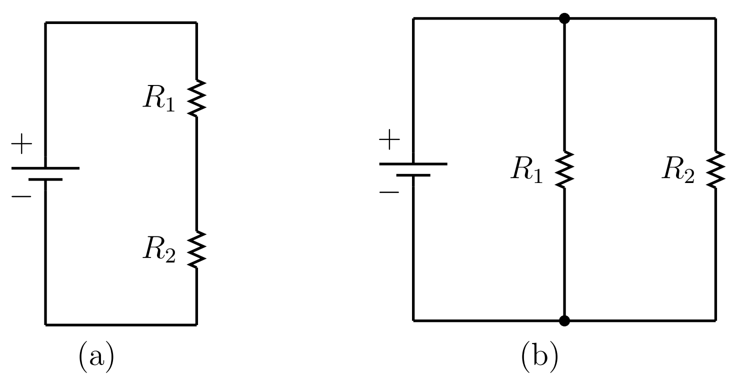


Figure 2.4: Circuits for driving an LED (a) directly from the signal voltage and (b) using a diode switch.

2.6 Δ-Y transformation

Consider the two different circuits shown in Fig. 2.5. If we are willing to neglect the central vertex in the left hand circuit, the two circuits are equivalent in the case that $R_1 = R_2/3$. Using your soldering iron,

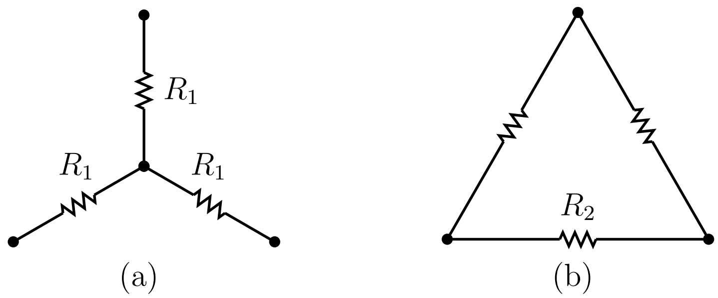


Figure 2.5: Equivalent three-node circuits.

2.7 Additions

Effect of resistance measurement with current in resistor?

Loading of circuit by DMM.

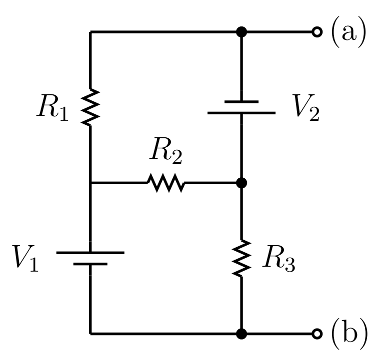
Chapter 3

Thevenin Equivalent Circuits

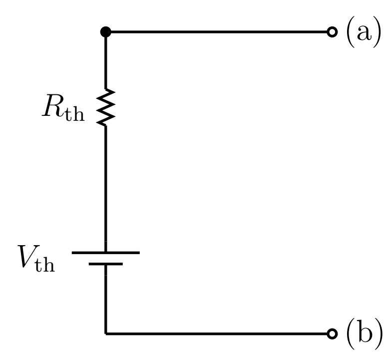
3.1 Pre-lab Calculation

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.



(a)



(b)

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, $R_2 = 4.7$ kΩ, and $R_3 = 3.9$ 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, $1.2\ \mathrm{k}\Omega, 4.7\ \mathrm{k}\Omega$.

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.

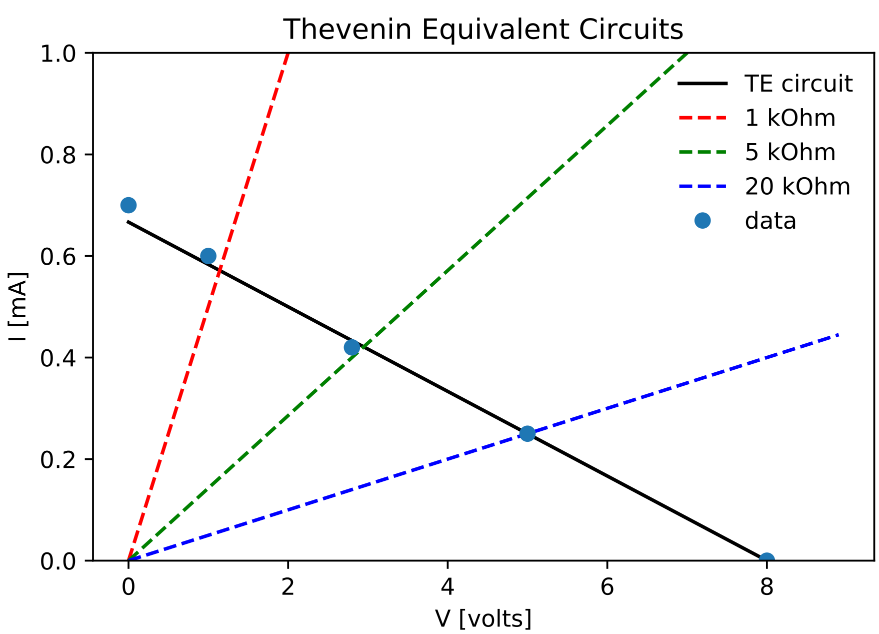


Figure 3.2: Using boolean masks to cut on variable y.

Chapter 4

Alternating Current and Time Varying Signals

4.1 Introduction

In this lab you will use two essential new pieces of lab equipment (scope and function generator) to study alternating current and time varying signals.

4.2 Function Generator

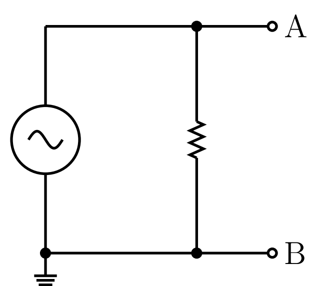


Figure 4.1: A function generator driving a resistor.

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. **??**. Set your DMM to the 20 V AC scale (the V with a squiggly line) .

Now 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.

The factory default settings place the function generator in Sine function mode, so leave that set as is.

Adjust the frequency to 10 kHz by turning the large knob. Press the Freq/Period choice button twice to see how you can switch between specifying frequency and period. Leave it as Frequency for now.

You can likewise switch between Ampl and Offset to specify an amplitude and an offset, or set the high and low values. Adjust the amplitude to 1 V RMS by:

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

Your function generator should now be set to provide a 10 kHz Sine function with an RMS amplitude of 1 V. Confirm this from the status screen.

Note that your DMM reads 0 V. Because the output is not yet enabled! Press the "On/Off" button above Channel 1 to enable the output.

Turn the output frequency of your function generator down to 2 kHz. Now your DMM should read an RMS voltage near 2 V, about twice the value that we expect. (EXPLAIN)

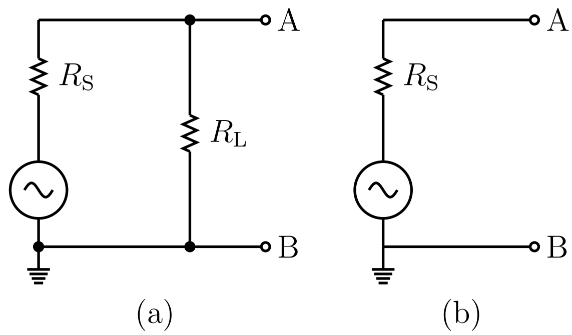


Figure 4.2: A function generator with source impedance explicitly shown.

(Adjust DC offset and measure DC voltage with DMM)

4.3 Oscilloscope

Trigger...

AC coupled versus DC bias...

Grounding???

To observe the time dependence of signals, the tool of choice is the oscilloscope.

Press the default setup function of yours scope...

Adjust the trigger level and observe that the phase of the channel 1...

4.4 Lissajous Figures

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. 4.3.

To begin, adjust channel 2 of your scope as you did channel 1: probe to 1x, with 500 mV range. Connect the output of Channel 2 on your function generator to the channel 2 input on your ... Set the amplitude of both Channel 1 and Channel 2 to 3 V peak-to-peak.

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:

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

You should now have reproduced the "start" pattern from Fig. 4.3a.

Adjust the phase of Channel 2 until the pattern collapses into a Fish pattern (or greek letter α). 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 the table, 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. ??b. For the right proportions, you'll need to 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. 4.4.

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~\rm kHz
```

With these definitions, we would define:

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

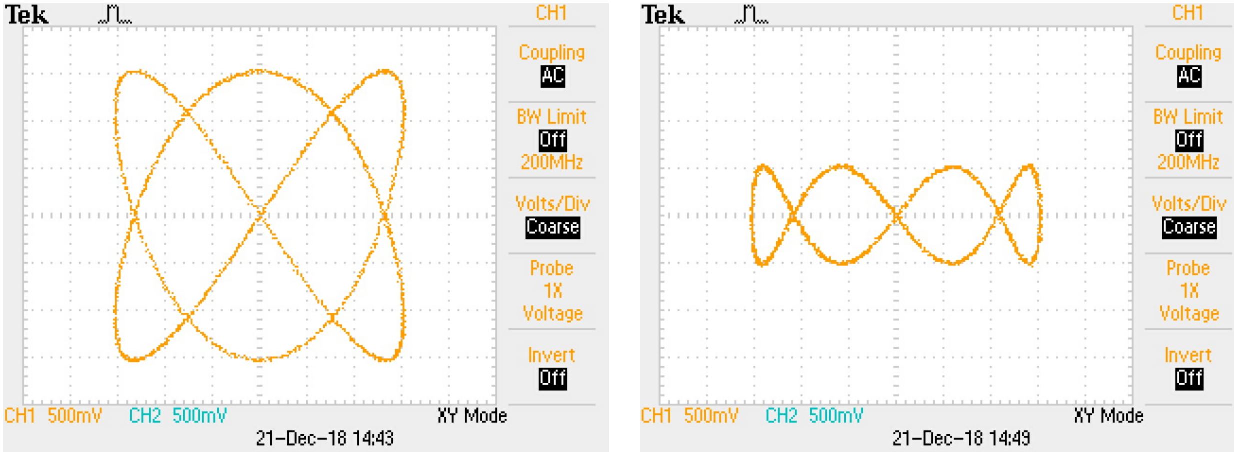
to obtain x points corresponding to $f = 4$ kHz sine function.

When plotting your curves, use:

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

to keep the unit aspect ratio used by your scope.

figs/labs/lissajous/pythonlissajous.pdf



(a) (b)

Figure 4.3: Scope traces from Lissajous figures from settings for (a) start, and (b) crown.

Table 4.1: Settings for various Lissajous figures.

pattern	f_1 (kHz)	f_2 (kHz)	δ_1
start	2	3	0
fish	2	3	30°
parabola	1	2	45°
lace	13	12	0
crown	1 kHz	4 kHz	0

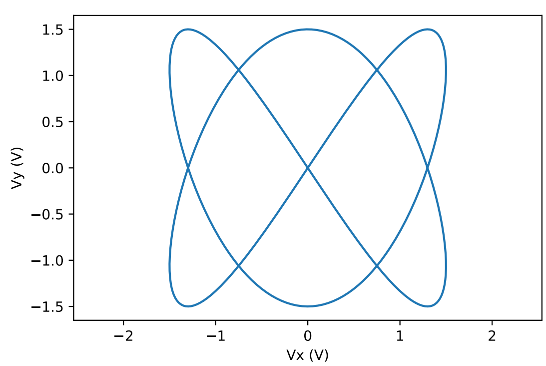


Figure 4.4: Lissajous curve constructed using Scientific Python corresponding to the scope trace in Fig. 4.3a.

Chapter 5

RC and RL Transient Signals

5.1 Pre-lab Calculation

1) Show that for an exponential decay with time constant τ, the rise-time, when defined as the time interval between 10% and 90% values, is given by:

$$t_{90} = \ln(9)\,\tau \sim 2.2\tau$$

2) Calculate the inductance of a solenoid with N=20 turns, length $\ell = 4$ cm, a radius of 1 cm^2 using the formula:

$$L = \frac{\mu N^2 A}{\ell}$$

where A is the cross-sectional area and $\mu = 1.257 \times 10^{-6}$.

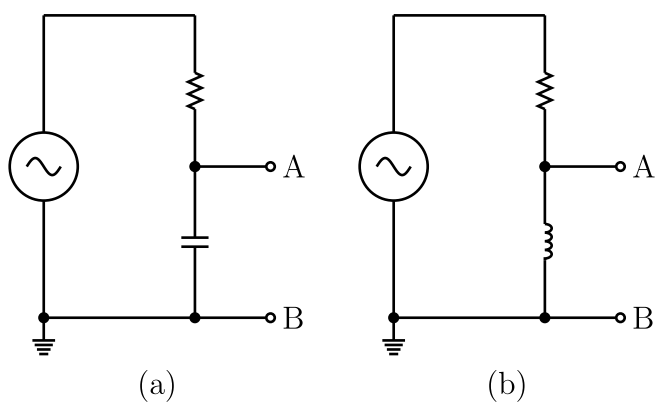


Figure 5.1: A function generator driving an RC circuit.

5.2 Transient response of an RC circuit

Build the circuit in Fig. 5.1a using a precision $R = 10$ kΩ resistor and an 1 nF capacitor. Window appropriately. Measure time dependence, rise time according to procedure, and using automated measure function. Determine time constant. (Plot exponential.)

5.3 Transient response of an RL circuit

Wrap an inductor around the provided dowel, and estimate it's inductance by modifying your pre-lab calculation accordingly.

Turn down the supply to 2.5 V peak-to-peak. Build the circuit in Fig. 5.1b using your homemade inductor and a resistor of $R = 47$ Ohms.

Determine the inductance of your coil and compare to your theoretical estimate.

5.4 Analysis

Plot your collected data and compare with an exponential function using the measured life-time.

Chapter 6

Passive Filters and Resonance

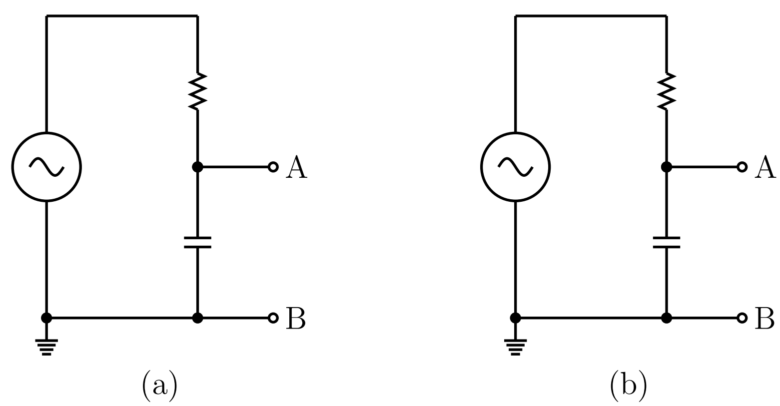


Figure 6.1: A function generator driving a resistor.

6.1 Resonant Band-pass Filter

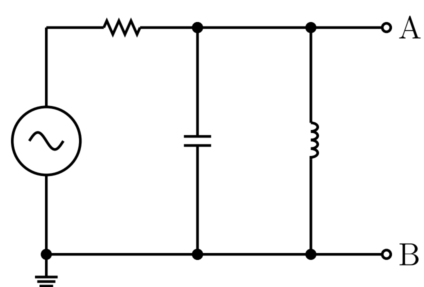


Figure 6.2: A function generator driving a resistor.

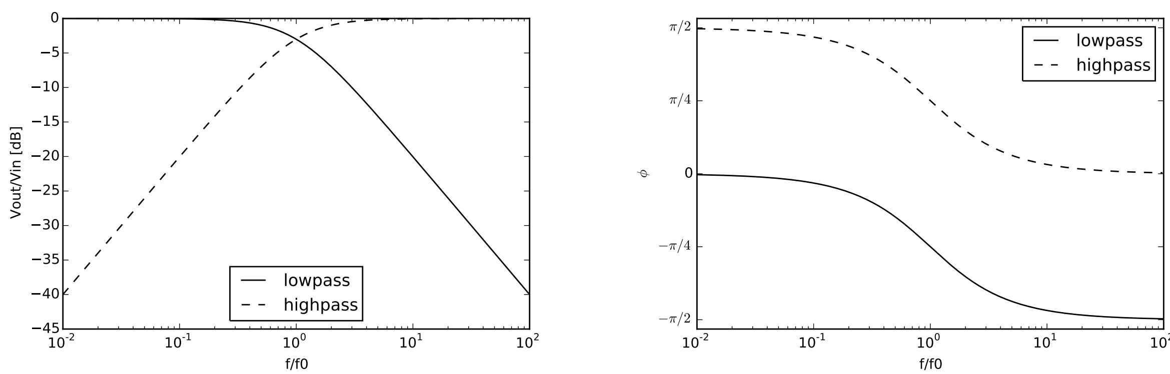
6.2 Pre-lab Calculations

1) Calculate the crossover frequency (aka the -3 dB point) for the RC filter shown in Fig. **??**a.

2) Using the formula derived in lecture, calculate the resonant angular frequency ω_0 and resonant frequency f_0 of the resonant circuit in Fig. **??**. *A very common mistake is to mixup frequency and angular frequency in the lab.*

3) Using the formula derived in lecture, calculate the Q-factor for the resonant circuit in Fig. **??**.

6.3 Introduction



(a) (b)

Figure 6.3: Bode plots for highpass and lowpass filters showing the (a) gain on a dB scale, and (b) phase, both as a function of the ratio of frequency f to the crossover frequency f_0 on a log scale.

In this lab, you will build and measure the performance of a lowpass and a highpass RC filter, and produce Bode plots to compare your circuit with the impedance model derived in class and shown in Fig. 6.3. You will build an RLC bandpass filter, determine its resonant frequency and quality factor, and see how limitations from non-ideal components dramatically affect the performance of real circuits.

6.4 Lowpass Filters

You'll be using two scope probes in this lab, and some care must be taken when using their grounding clips. The only place you may connect the grounding clip from a scope probe is to the ground in your circuit. For a battery powered device not connected to earth ground, you could choose this ground point to be anywhere, as long as it was consistent for both scope channels. But in our circuit, the function generator is already referenced to earth ground, and therefore the only possible choice for the ground location is at the negative terminal of the function generator, as already shown in the circuit diagrams. Always remember that when you connect a scope probe grounding clip, you short that point to earth ground through the scope! **In these circuits, the only place you may attach a scope probe grounding clip is at the point P_1, i.e. at the negative terminal of the function generator.**

We'll be measuring two potentials simultaneously for all of the filters in this lab. The input voltage V_{in} is measured between ground at P_1 and the point V_{in} in the circuit. This can be measured by using a "BNC T" to connect both your scope and your circuit to the function generator output.

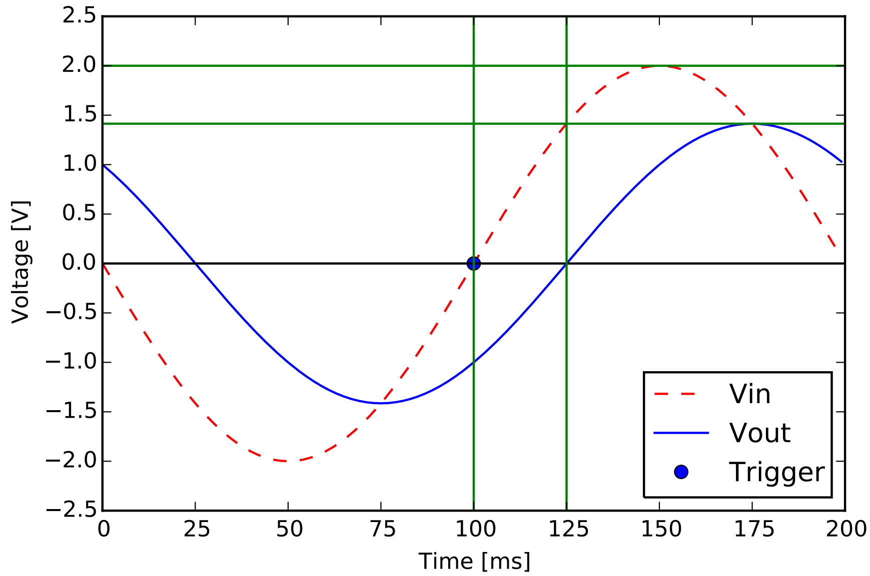


Figure 6.4: The gain and phase change measurement using an oscilloscope. Here the gain is $1.4/2.0 = 0.7 \sim 1/\sqrt{2}$. Later times appear toward the right, so the output is lagging the input, and therefore the phase shift is negative. The size of the time offset is 25 ms in a period of 200 ms so the phase shift is $\phi = -\pi/4$. This is what engineers like to call the -3 dB or even just the 3 dB point, because $20 \log_{10} \sqrt{2} = 3.01$. Just to confuse physicists!

Alternatively, use a scope probe with the grounding clip at P_1 and the the probe tip at $V_{\rm in}$. The output voltage $V_{\rm out}$ is measured using a scope probe with the grounding clip at P_1 and the probe tip at $V_{\rm out}$. We will be measuring the voltage gain $G = V_{\rm out}/V_{\rm in}$ and the phase shift ϕ of $V_{\rm out}(t)$ relative to $V_{\rm in}(t)$.

Build the circuit shown in Fig. **??**a using $R = 1.5$ kΩ and $C = 0.01$ μF. Use your function generator in AC mode with a peak-to-peak voltage of 4 V. Either put the function generator in high impedance output mode, or select the 50 Ω output impedance and use a 50 Ω terminator.

Vary the frequency above and below the cross-over frequency which you calculated, and verify the behavior qualitatively before proceeding with the data taking. As this is a low-pass filter, you should expect to see the gain $V_{\rm out}/V_{\rm in} = 1/\sqrt{2}$ with a phase shift $\phi = -\pi/4$ at the crossover frequency. An example measurement at the crossover frequency is shown in Fig. 6.4. Below this frequency you should approach unit gain, and above this frequency the gain should fall rapidly.

To make these measurements accurately, you should first confirm that both channel 1 and channel 2 on your scope are vertically aligned with the x-axis (i.e. the thick central horizontal line on the display), which you do by temporarily setting the coupling of each channel to ground (i.e. V=0) and adjusting the horizontal offset as needed. Next, adjust the trigger setting to trigger on the rising edge of $V_{\rm in}$ and set the trigger threshold at zero. Learn to use your scope's cursor function. For instance, when making the phase shift measurement, you would place the reference cursor at the point where $V_{\rm in}$ crosses zero with positive slope (the trigger point) and measure the point where $V_{\rm out}$ crosses zero with positive slope using the cursor.

Once you are certain your circuit is working properly and you scope is configured to measure the magnitude and gain accurately, take measurements at 9 different frequencies, chosen to cover four decades of frequency range and uniform in the log of the frequency:

$$f = \{f_0/100, f_0/30, f_0/10, f_0/3, f_0, 3f_0, 10f_0, 30f_0, 100f_0\}$$

where f_0 is the cross-over frequency. At each frequency, you will measure the gain: $G = V_{\text{out}}/V_{\text{in}}$ and the phase shift ϕ of V_{out} relative to V_{in}.

If you prefer, you can use your scope's measurement features (which include a phase measurement) after making a few measurements from the wave form as described above. When using automated features, it is usually a good idea to make a few manual measurements first to make sure you are measuring what you think you are.

6.5 Highpass Filter

Using the same components as in the previous section, build the high-pass filter of Fig. **??**a. Verify the operation of the circuit at the crossover frequency (aka 3 db point) where we expect the gain to be $1/\sqrt{2}$ as before but the phase shift to be *positive* $\phi = \pi/4$.

As in the preceding section, measure the magnitude and phase of the gain as a function of frequency, but to save time, you may omit some measurements:

$$f = \{f_0/10, f_0/3, f_0, 3f_0, 10f_0\}$$

where f_0 is the crossover frequency.

6.6 Bandpass Filter

In lecture, we showed that the resonant angular frequency of an RLC bandpass filter is given by

$$\omega_0 = \frac{1}{\sqrt{LC}}. \tag{6.1}$$

At this frequency, the RLC resonant circuit has unit gain and no phase shift. As the frequency moves away from the resonant frequency, the gain drops. We define two points ω_+ and ω_- as the two frequencies, one above and one below ω_0, at which the gain drops below $1/\sqrt{2}$. These are the two -3 dB points which define the bandwidth of the system. We define the quality of the resonance by the ratio of resonant frequency to this bandwidth:

$$Q = \frac{\omega_0}{\omega_+ - \omega_-} = \frac{f_0}{f_+ - f_-}$$

For the RLC resonant circuit, we showed in lecture that:

$$Q = \omega_0 RC \tag{6.2}$$

Build the circuit in Fig. **??** using $R = 1$ kΩ , $C = 4.7$ μF, and $L = 150$ μH. Set the frequency of the function generator to $f_0 = \omega_0/2\pi$ and the peak-to-peak voltage 10 V.

We'll determine the resonant frequency using a trick. During normal use, your oscilloscope displays the voltage of each channel versus time. But there is an "XY" mode available under the display options. In this mode, the scope displays the voltage of channel one versus the voltage of channel two. When in XY mode, two out-of-phase signals yield an ellipse. But, as shown in Fig. 6.5

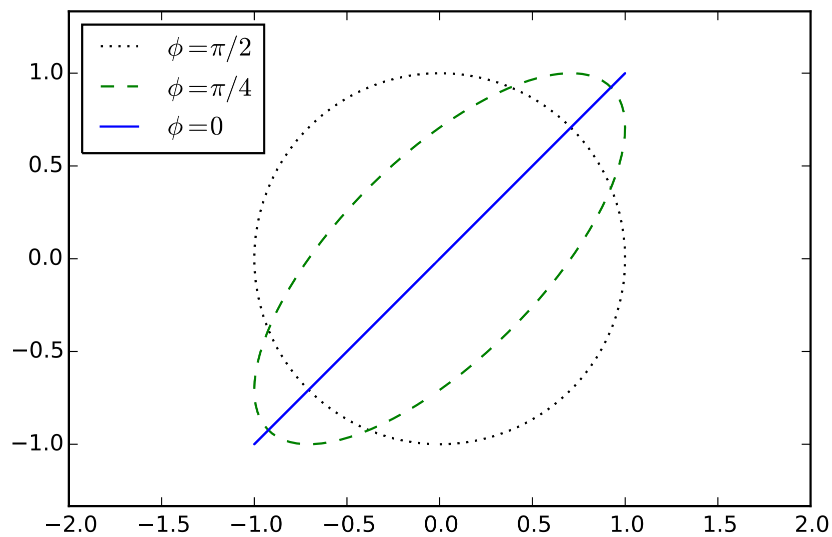


Figure 6.5: Example scope traces in XY display mode for a relative phase $\phi = \pi/2$, $\phi = \pi/4$, and $\phi = 0$. It is easy, accurate, and somehow deeply satisfying to tune the frequency until the ellipse collapses into itself, forming a line.

when both channels are perfectly in phase, the ellipse collapses to make a diagonal line. You can set you scope in XY mode and adjust the frequency until the ellipse collapses to quickly and accurately find the resonance frequency.

One you determine the resonant frequency, switch back to the normal display mode and determine the bandwidth. The easiest way to obtain this is to use the curser to measure the peak voltage of your output signal. Multiply this by a factor of $1/\sqrt{2} = 0.7$ to determine the -3 dB amplitude and set the second curser at this value. Now adjust the frequency above and below the resonant frequency and note at which frequencies the amplitude drops below this line. The difference between these two points is your f_+ and f_-.

From your measurement of f_0, f_+, and f_- you can calculate the measured Q-factor of your circuit. It will be quite different than what you calculated in pre-lab!

6.7 Degradation of the Q-factor

The Q-factor that you measured in the previous section is significantly lower than the theoretical Q-factor for the circuit in Fig. **??**. In fact, the inductor is non-ideal in a number of ways. In this case, there is a parasitic parallel resistance that makes the circuit effectively that of Fig. **??**. The resistance R_{P} is already present (unfortunately!) in your circuit.

It turns out that the Q factor is degraded in this case to the value:

$$Q = \omega_0 C \frac{R R_{\mathrm{P}}}{R + R_{\mathrm{P}}} \tag{6.3}$$

To determine R_P, note that at the resonant frequency, the parallel impedance of L and C are infinite, and so can be treated as open circuits. The circuit becomes a voltage divider with two resistors R and R_P. By measuring the gain $V_{\text{out}}/V_{\text{in}}$ at the resonant frequency, you should be able to determine R_P, and correct your gain calculation accordingly. Now how does your calculated Q-factor compare with what you measured?

6.8 Lab Report

Your report should include all of your calculations, and the plotted response of your lowpass and highpass circuits as in Fig. 6.3. In addition to the gain in dB, also produce a plot with $V_{\text{out}}/V_{\text{in}}$. Answer all questions in the text.

Notice that the difference between the RC highpass and lowpass filter just amounts to which component we measure the voltage V_{out} across. Therefore, one might reasonably expect the crossing point of the Bode plots to occur where $V_{\text{out}}/V_{\text{in}} = 0.5$, but, in fact, it occurs at the 3 dB point where $V_{\text{out}}/V_{\text{in}} = 0.7$. How is this be possible? Does the voltage across the resistor plus the voltage across the capacitor add to 1.4 times the input voltage?!

Chapter 7

The Diode

7.1 Pre-lab Calculations

1) Suppose a diode is in forward bias with a resistor $R = 10\ k\Omega$ in series while connected to a 10 V DC source. Estimate the effective resistance of the diode. Hint: assume a typical diode drop of 0.6 V and consider an equivalent voltage divider consisting entirely of resistors.

2) Consider the circuit in Fig. **??**a and assume $R_1 = 1.8$ kΩ and the peak-to-peak voltage is $V_{\rm pp} = 5$ V. What is the peak current through the diode? The math is easier if you assume a diode drop of 0.7 V, so go ahead and do so!

3) What is the ripple current for an AC source with amplitude 10 V and frequency 100 Hz driving a load of $R_L = 18$ kΩ in the circuit in Fig. **??** for (A) $C = 1\ \mu F$ and (B) $C = 100\ \mu F$?

7.2 Introduction

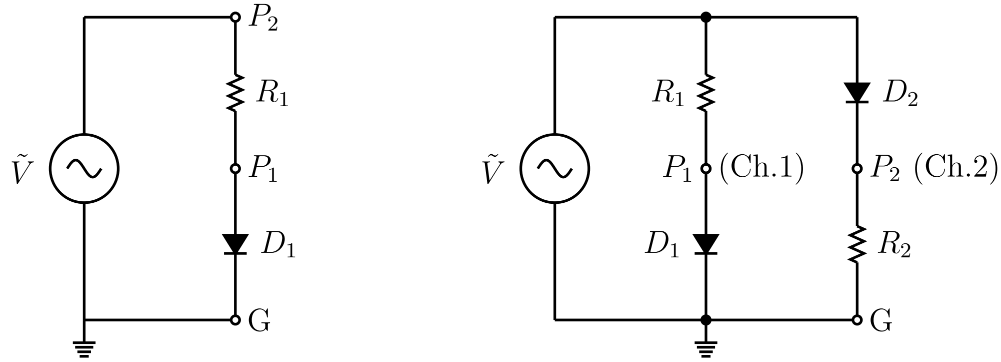
In this lab, you will measure the IV curve of a diode, use it to predict the operating point of a circuit, and use rectification to provide a DC current source with low ripple voltage. In the process, you will learn how to use the Math mode of your scope to make a differential voltage measurement.

7.3 Measuring the I-V Curve of a Diode

In this section you will measure the I-V curve of a 1N914 diode, and compare your results to the curves available from the device data sheet. To avoid taking a bunch of measurements by hand, we will use a trick to plot the curve directly on your oscilloscope using the XY mode.

Consider (but don't build!) the circuit in Fig. **??**a. The voltage between points P_2 and P_1 is proportional to the current passing through the diode, and the voltage between points P_1 and G is the voltage across the diode. So if we could display $P_2 - P_1$ versus $P_1 - G$ on your scope we could use this circuit. Unfortunately, this is not possible on your scope, because (1) the only valid place to put the scope probe ground shield clips is at the point G (Why?) and (2) you can only display Channel 1 versus Channel 2 in XY mode.

The solution is to drive two copies of the diode in series resistor, with the component order reversed, as in Fig. **??**b. This way, we can connect the probe ground shields as required at point G, put the voltage across the diode on scope Channel 1 by connecting the probe tip at P_1, and put

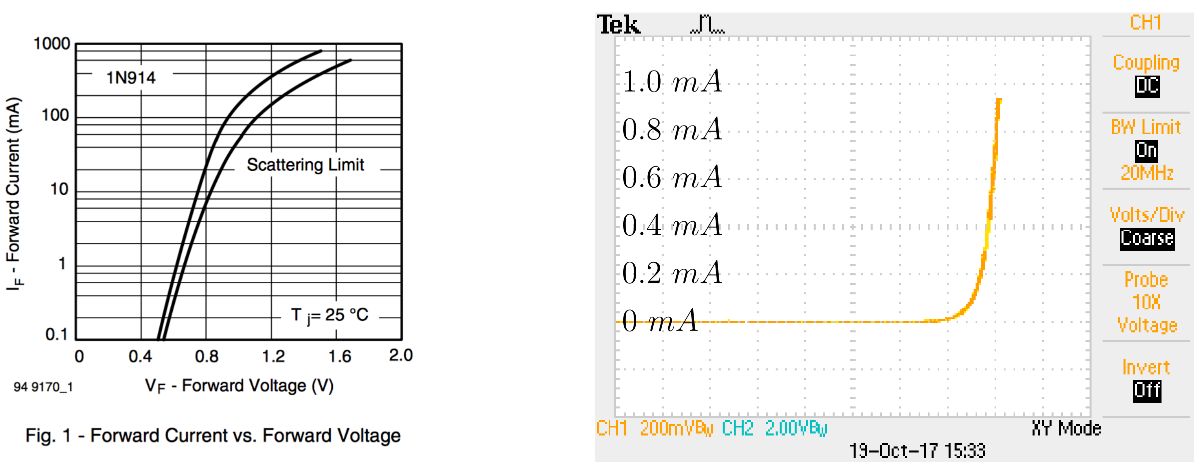


(a) (b)

Figure 7.1: Diode circuits for (a) demonstrating rectification and (b) plotting the diode IV curve on your oscilloscope.

the voltage across the resistor (proportional to current through the diode) on scope Channel 2 by connecting the probe tip at P_2.

Build the circuit in Fig. ??b using a 1N914 fast switching diode for D_1 and D_2 and $R_1 = R_2 = 10$ kΩ. Set your function generator for high-impedance output, providing AC with peak to peak voltage of 20 V at a frequency of 100 Hz. Before switching to XY mode, make certain that your Channel 1 has no voltage offset (that is, zero voltage is located at the origin) or else your diode output voltage won't be calibrated properly in your output plot. Once you set this, try not to adjust the offset of Channel 1 or you'll have to redo it! To minimize noise, set the bandwidth limit "On" for both channels (this is available in the menu for each input channel as "BW Limit").



(a) (b)

Figure 7.2: IV curves for the1N914 from (a) data sheet, and (b) as you will measure in this lab. In the scope trace, the Channel 2 (Y) with scale set to 2 V measures the voltage across a 10 kΩ resistor, so each division corresponds to 200 μA as indicated.

Set the scope into XY mode, and see if you can reproduce the diode IV curve in Fig ??b. Beats jotting down voltages in your logbook doesn't it? Now jot down the voltage you expect across the diode for a current of 1 mA in your logbook. Where they overlap, does your measured IV curve agree with the curve from the component data sheet in Fig. ??a?

7.4 Rectifying an AC Signal

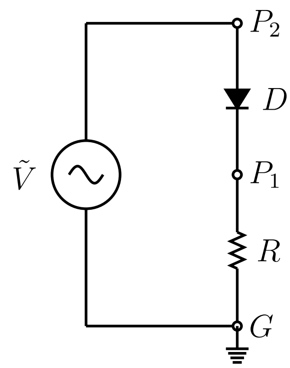


Figure 7.3: A diode rectification circuit.

Set your function generator to provide an AC source with frequency 100 Hz and peak-to-peak voltage $V_{\mathrm{pp}} = 5$ V. Build the circuit in Fig. **??** using a 1N914 diode for D and $R = 1.8$ kΩ.

With your scope probe ground shield clips both properly connected to the ground at G, monitor the voltage at points P_1 and P_2. Sketch the voltage across the resistor R and the voltage supplied by the function generator versus time on the same plot in your lab book.

Using your scopes amplitude measurement feature, measure precisely (i.e. to within 50 mV precision) the voltage drop across the diode at the peak current value, by measuring the difference between Channel 1 and Channel 2 of your scope at the peak. Is this operating point consistent with your results from the previous section and the pre-lab calculations?

7.5 Building a DC voltage source

Now build the DC source circuit in Fig. **??** using a 1N914 diode for D and $R_{\mathrm{L}} = 18$ kΩ. Adjust your function generator to provide a peak-to-peak voltage $V_{\mathrm{pp}} = 20$ V.

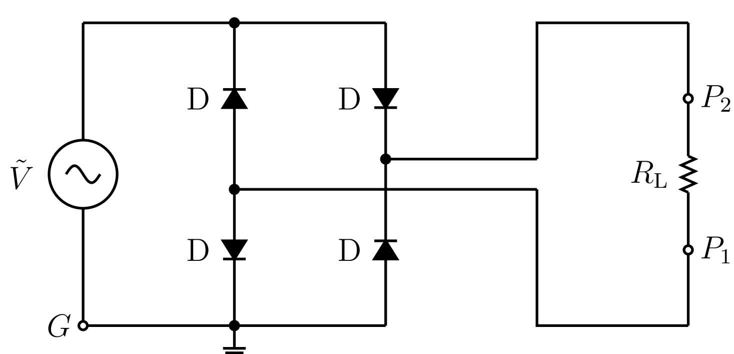


Figure 7.4: A full-wave rectifier. Note that crossed lines without a dot are *not connected.*

To measure the performance of our DC source, we would like to measure the voltage across the resistor R_L on the scope. However, notice that the ground for the circuit is located at point G, so you cannot measure the voltage between P_1 and P_2 using a single probe. To make the measurement, connect both probe ground shield clips to the point G as required, and connect the probe tips to points P_1 and P_2. Next, use your scope's Math mode to subtract Channel 1 to from Channel 2. The result of this operation is the voltage across the resistor R_L.

Sketch the current as a function of time for a few cycles, and measure the amplitude. In your lab report, explain the shape and the amplitude.

7.6 Controlling the Ripple

In class, we derived the following formula for the ripple voltage (the residual AC voltage after rectification) for a full-wave rectifier with a capacitance C:

$$\Delta V = \frac{I_\mathrm{max}}{2fC}$$

Add a capacitor with $C = 1\ \mu$F to your circuit, as in Fig. **??** and sketch the resulting waveform for the voltage across the load resistor as measured with your scope. Estimate the ripple voltage. As your DMM is a handheld device that is not DC coupled, you may use it to measure the voltage across R_L directly. Using your DMM, measure the voltage across R_L in both AC and DC mode. How does the AC measurement relate to the ripple voltage ΔV? How do you various measurements compare to the voltages you calculated in pre-lab calculations?

For the last tweak, you are going to use a large electrolytic capacitor. **These capacitors are polarized, and will likely "let the smoke out" if you install them the wrong way.** Making sure the negative terminal is connected as indicated in Fig. **??**, install a $C = 100\ \mu$F electrolytic capacitor in your circuit and measure the ripple voltage.

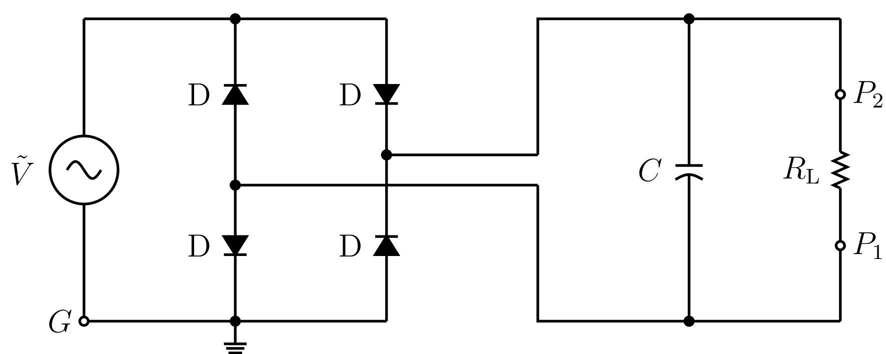


Figure 7.5: A full-wave rectifier with ripple voltage limiting capacitor. When using a polarized electrolytic capacitor, make certain that the negative terminal is connected to the lower half of the figure, as indicated.

7.7 Lab Report

Your report should include all of your measurements and a comparison with your calculation.

Chapter 8

Complete List of Labs

Electron:

1. DC Circuits
2. Thevenin Equivalent Circuits
3. Alternating Current and Time Varying Signals
4. RC and RL Transient Signals
5. Passive Filters and Resonance
6. The Diode

Scientific Python Analysis Labs:

7. Introduction to Plotting
8. Histograms and Distributions
9. The Central Limit Theorem
10. Error Propogation
11. Curve Fitting
12. Fourier Transforms
13. Monte Carlo Techniques

Advanced Labs:

14. Geiger Counter
15. Planck's Constant
16. Speed of Light in Cable
17. Speed of Light in Vacuum
18. Speed of Light Analysis
19. Muon Lifetime Analysis