Lab 6: AC Analysis

Reference Resource (*optional*):
Practical Electronics for Inventors, Paul Scherz and Simon Monk, 2016, ISBN 978-1-25-958754-2

Lab Overview:

In order to build electrical circuits such as the solar power meter in the final project, an understanding of how to analyze, simulate, and experimentally measure circuit networks in the AC domain is necessary. Let's begin with a simple RC circuit. Be sure to not just follow directions in this lab, as you will be expected to design your own simulations and take the measurements you need when designing the Final Design Project at the end of the semester.

Inventory List:

The following is a list of available inventories for this Lab

Part	Details
Resistors	100Ω
Capacitors	0.1μF
Inductors	1mH
Breadboard	830 Tie Points
Alligator clips	Connectors
Assorted jumper wire	Red, black, white, green, blue, orange
Wavefunction Generator	AD2
Oscilloscope	AD2
Digital Multimeter	MS8217, DT-830B, DT838, or similar
Software	LTspice

Lab Instructions:

A Sparky Solar business executive knows you are a circuits expert and asks you a seemingly simple question: "Why in your engineering field is there such an emphasis on sinusoidal signals and waves?" What would you say? Here are couple possible replies:

- *"Any time varying signal can be written as a sum of various sinusoidal waves of different amplitudes and frequencies. So, if we can solve a problem for a sinusoid, we can determine the answer for any time varying signal." You may study this in EEE 203.*

- *"Furthermore, when we want to transmit signal from cell phones, or TV/radio stations, sinusoidal electromagnetic waves decay as inverse distance whereas DC signals decay as inverse distance squared. At 10km distance, a DC signal (electric field) is 10,000x weaker than a radio frequency or sinusoidal signal". You may study this in EEE 241 and EEE 341.*

The Sparky Solar executive is impressed with your knowledge and asks, "Okay, why is the power in my house a 60Hz AC (sinusoidal) signal when it would be much more convenient to have DC voltages throughout my house?" You might reply:

- *"Until very recently there weren't efficient methods to convert from one DC voltage to another, however AC or sinusoidal voltages can be converted easily with a transformer. When the power company moves power (P=VI) around the nation on large transmission lines, they must use high voltage to minimize the current, otherwise, there would be large losses in the power lines (I^2R). To minimize the losses, they would have to reduce R by using large diameter copper cables which are very expensive when extending over many miles. So, the power company uses high voltage (i.e 500kV sinusoids or AC) distribution networks and transform down to 120V sinusoids in stages for your house". You may study this in EEE360.*

The business executive is really impressed and finally asks, "Is there a technical way to determine the next hit song for the Sparky Music Company?" You reply that, "*hit music is an art, not a science*". You think to yourself that you might take a few liberal art classes for a balanced education… and you might investigate artificial intelligence generation of music (Generative Deep Learning, Foster, O'Reilly).

So, you dive into this lab on time varying and sinusoidal signals with enthusiasm! Sparky Solar has provided your team with a Lab equipped with LTSPICE and the inventory listed above. Using the detailed instructions provided, simulate, build, and measure first-order circuits which are circuits containing a single capacitor or inductor as well as voltage and/or current sources, resistors, and switches. A circuit containing multiple capacitors or inductors can be treated as a first-order circuit if a single equivalent capacitance or inductance can be found.

In this lab exercise, you will be measuring first – order circuits using your EIC-106 Breadboard, the wavefunction generator, the oscilloscope, and the multimeter.

Part 1 – Prelab Calculations and LTSpice Simulation Work:

Transient Analysis

Transient analysis is used when the signal in the circuit (current or voltage) changes with time.

Question 1: Resistive Circuits - Sinusoids

Using LTspice, build the circuit below. Use SINE as the source function and run a transient analysis for 10ms.

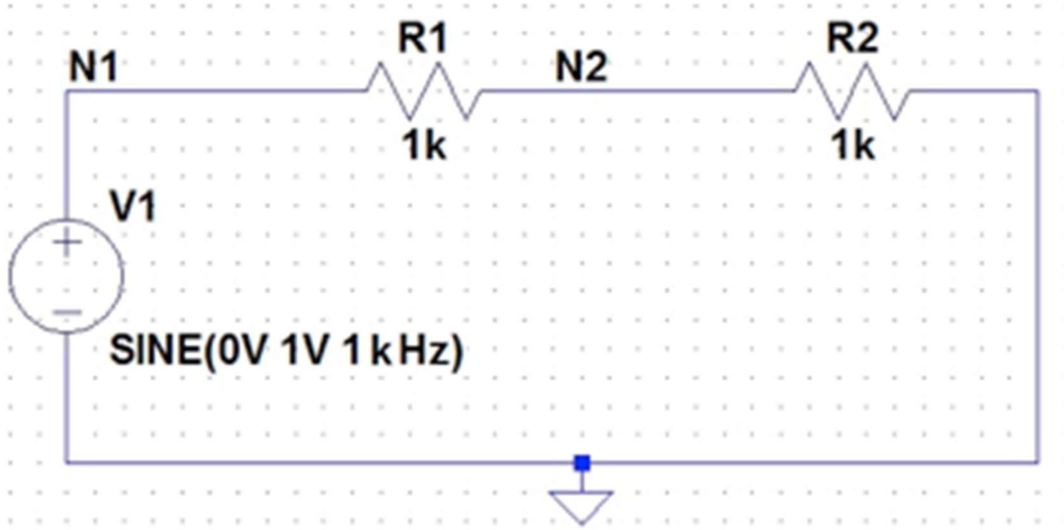

Figure 1: Sine Wave Transient Analysis

Set the SINE source to have a frequency of 1kHz, with a peak-to-peak voltage of 2V (magnitude of 1V) and 0V offset.

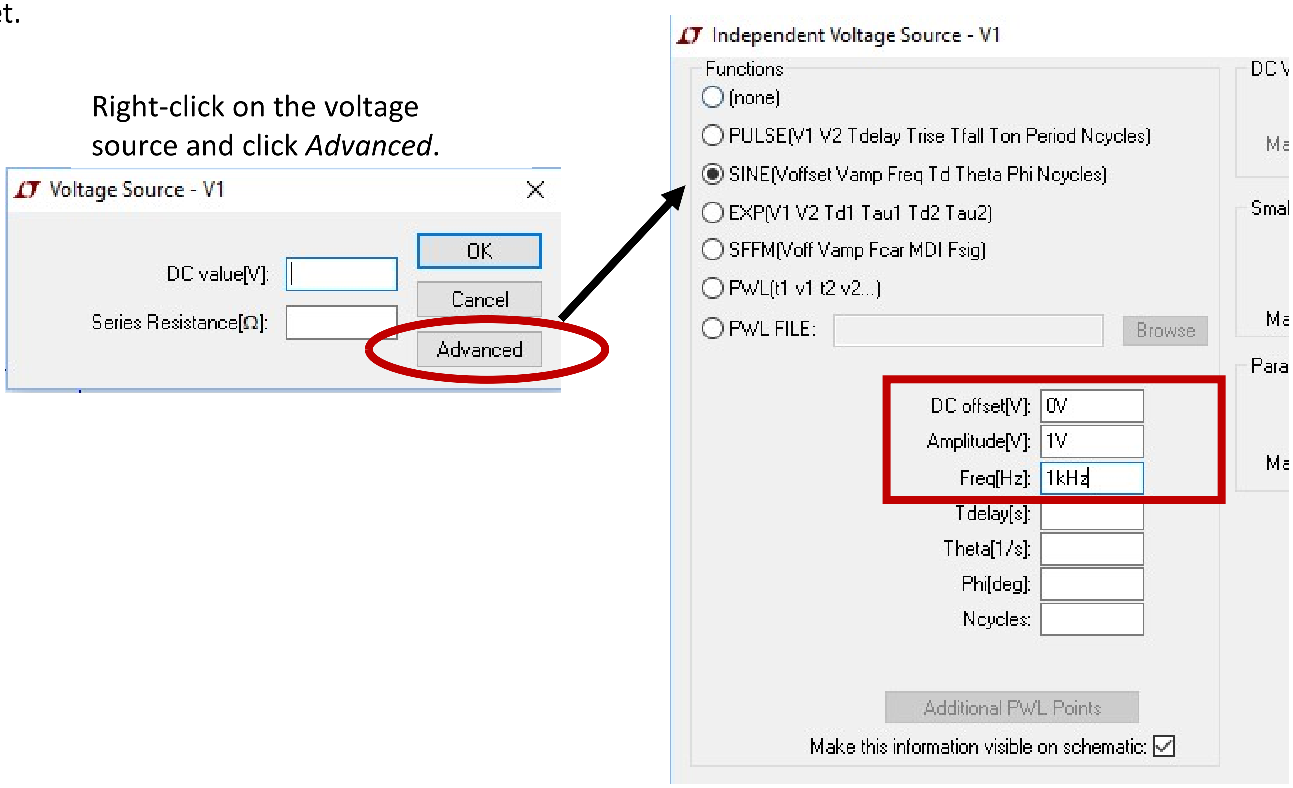

Figure 2: Setting up the AC Supply

Under the *Simulate* menu, click *Edit Simulation Command*. Click on the Transient tab. Enter 10ms for Final time. Click OK. *[SPICE directive = .tran <stop time>]*

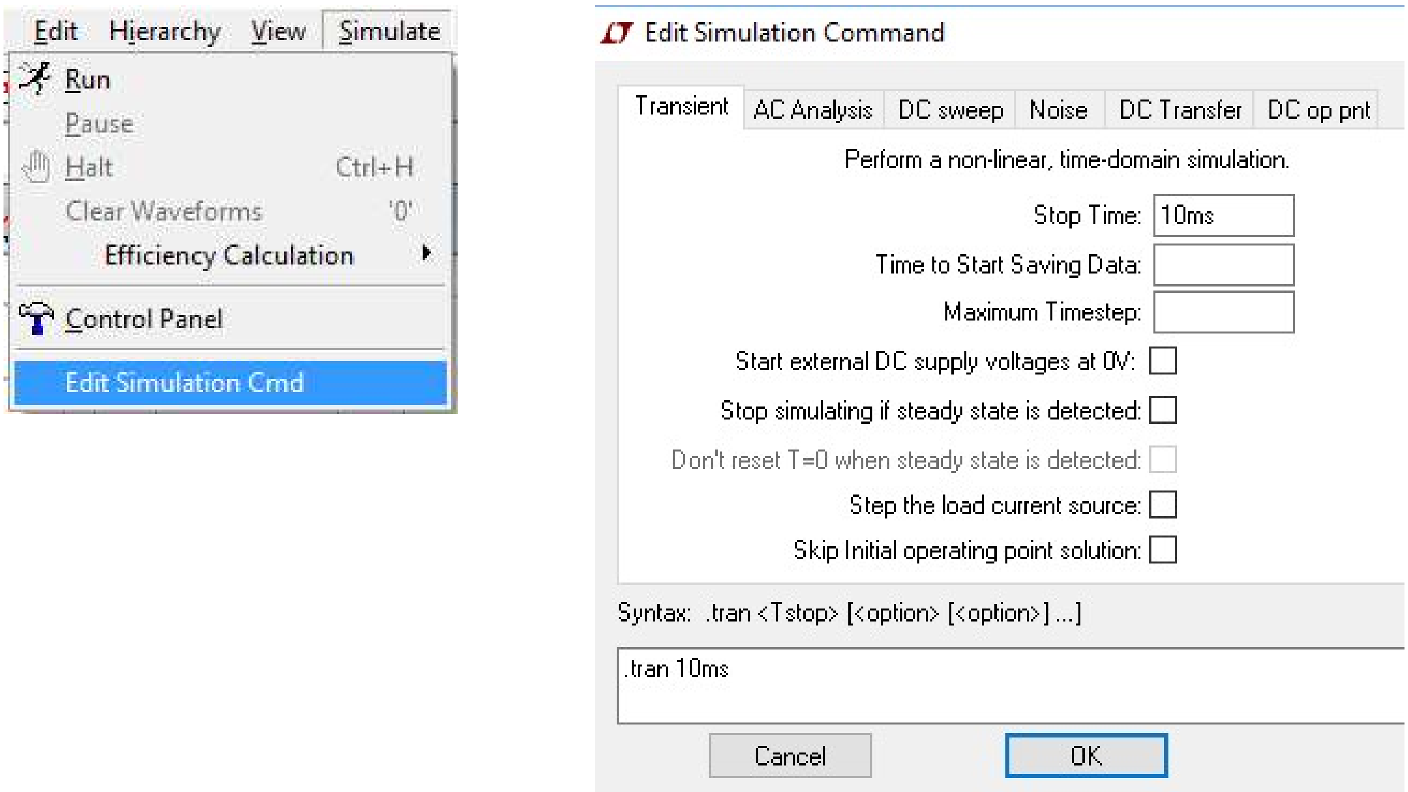

Figure 3: Running a Transient Analysis

Simulate the circuit and plot the voltage dropped across the R1 resistor, and the current through the R1 resistor. *Be sure to check that the resistor current is set in the correct direction in the schematic (current left to right).* Include your schematic and plot with your datasheet.

On your Data sheet, sketch the plot of the **current** through resistor R1 and the **voltage** across R1 and explain why the current is in-phase or out-of-phase with the voltage.

Question 2: RC Circuits – Pulse Source

Using the circuit shown below:

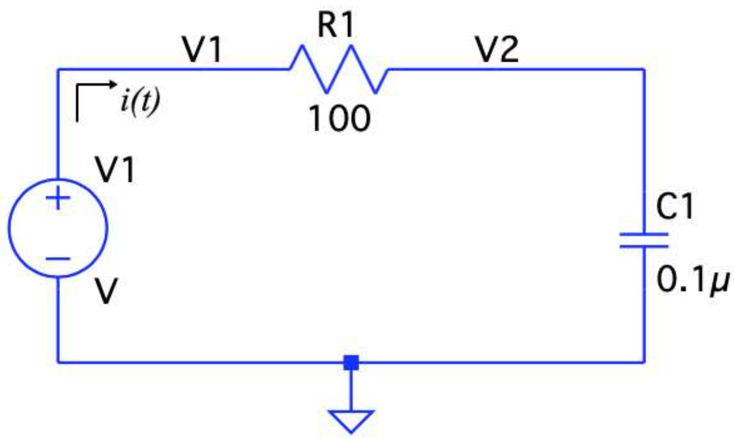

Figure 1: RC Circuit

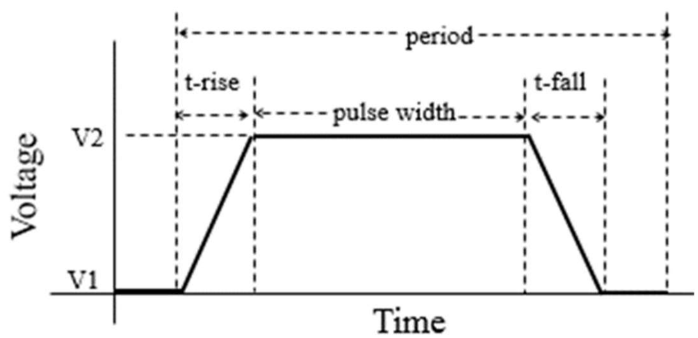

Figure 2: Pulse Source

(V1=0V, V2=1V, t-rise=1µs, t-fall=1µs, pulse width (t-on)=100µs, period=200µs)

As a reminder, duty cycle is defined as a percentage of the period in which the signal is active.

$$Duty\ Cycle = \frac{pulse\ width}{period} \times 100\%.$$

In this lab, the pulse waveform has a duty cycle of 50% (**Figure *2***).

Answer the following questions:

a) Consider the RC Circuit shown in **Figure *1*: RC Circuit** Assume a pulse source as shown in **Figure *2*: Pulse Source**. Derive an expression for the current, ***i*(*t*)** (for t values during the on time) flowing clockwise in the single loop circuit. Neglect the fall and rise times in your derivation. **Include all your hand calculations on the data sheet.**

b) Obtain expressions for the voltages across the <u>resistor</u> and the <u>capacitor</u> (in the direction of assumed current flow). Neglect the fall and rise times in your derivation. **Include all your hand calculations on the data sheet.**

c) Create a simulation of the RC circuit in LTSPICE and include a screenshot with the data sheet. Assume that the pulse source defined in question 1 has rise and fall times of 1 µS and a period of 200µs. (Refer to **Figure *2*: Pulse Source** for definitions of these parameters.)

d) Capture the plots of the current, i(t), and the voltages across the resistor, $v_R(t)$, and the capacitor, $v_C(t)$, all 3 plotted on the same plot, and include it in the data sheet.

e) Repeat the LTSpice simulations you carried out in question 1 but with t_{ON}, rise time, fall time, and the period all reduced by a factor of 10.
   a) Include a picture of your schematic showing the new parameters.
   b) Include the plot of i(t), $v_R(t)$, and $v_C(t)$.

Question 3: RC Circuits – Sinusoids

Simulate in LTSPICE the RC Circuit shown in **Figure *1*: RC Circuit** again but with V1 as a sine wave with period of 2ms (as shown in **Figure *3*: Two sinusoids**). Calculate the corresponding frequency.

a) Plot the voltages across the source($v_s(t)$), the resistor ($v_R(t)$), and the capacitor ($v_C(t)$), all 3 plotted on the same plot. Capture the schematic and plot and include it with your data sheet.

b) From the plot, find the capacitor voltage magnitude and the phase difference, in degrees, between the source and capacitor voltages, where:

The Magnitude: is the value measured at the peak of the sine wave.

The Phase difference: is measured by measuring the time difference between two "similar points" on the two curves and converting to degrees through:

$$\Delta\phi = \frac{\Delta t}{T} \times 360^o$$

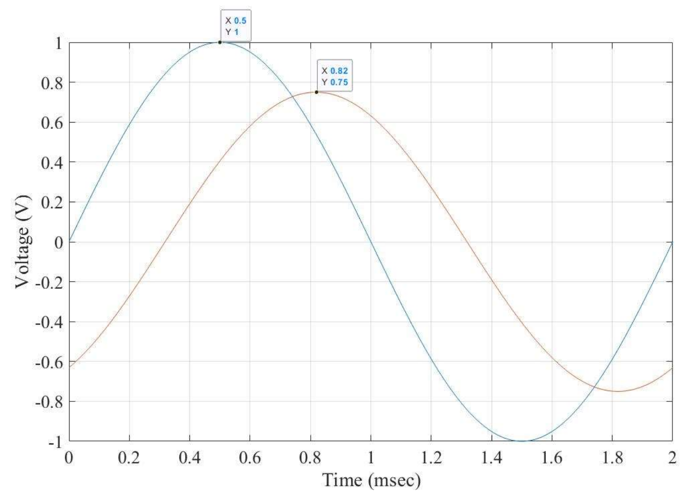

Figure 3: Two sinusoids with equal frequencies but have different magnitudes and a phase difference

where T is the period of the sine wave while Δt is the time difference you measured between two peaks or two zero-crossings of the two signals. As an example, **Figure *3*: Two sinusoids** has a phase difference:

$\Delta\phi = \frac{0.82- \ .5}{2} \times 360^o = 57.6^o$.

Question 4: RL Circuits – Pulse Source

Simulate in LTSpice the RL Circuit shown in **Figure 4: RL Circuit**. Similar to question 3, assume that the source is a sine wave with a period of 20 ms.

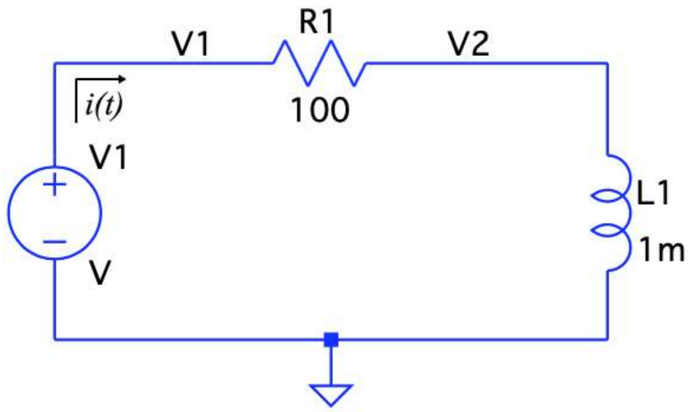

Figure 4: RL Circuit

a) Plot the voltages across the source($v_s(t)$), the resistor ($v_R(t)$), and the inductor ($v_L(t)$), all 3 plotted on the same plot. Capture the schematic and plot and include it with your data sheet.

b) From the plot, find the inductor voltage magnitude (in Volts) and the phase difference (in degrees), between the source and inductor voltages.

Question 5: RMS Calculations

Derive the RMS value of a $2V_{pp}$ a) sine wave, b) square wave, and c) sawtooth wave. Note that an RMS value can be defined for a current signal as well.

$$V_{RMS} \text{ or } I_{RMS} = \sqrt{\frac{1}{T}\int_0^T x^2(t)dt}$$

Question 6: AC Power Calculations

If the following signals are applied directly to a 1kOhm resistor, what is the power consumption in the resistors? *<u>Hint</u>: What is the average I^2R or V^2/R over time? Do you see the benefit of RMS values*?

a) $2V_{pp}$ sine wave
b) $2V_{pp}$ square wave
c) $2V_{pp}$ sawtooth wave

Part 2 – Hardware Lab Work:

1. Generate the following functions on the wavefunction generator. Verify their peak-to-peak voltages on an oscilloscope (refer to Lab 5 for how to use a Waveform Generator and Oscilloscope). Measure their voltages on the multimeter.
    a. Sine wave $2V_{pp}$, 1kHz
    b. Square wave $2V_{pp}$, 3kHz
    c. Sawtooth wave $2V_{pp}$, 5kHz
    d. Does the multimeter provide peak-to-peak voltages or RMS voltages?

2. Build the RC Circuit in **Figure 5** on your breadboard. Use a "Wavefunction Generator" as V1 and use the same R and C values you used in the prelab exercises (**Figure *1*: RC Circuit**). Assume the pulse source parameters defined above in **Figure *2*: Pulse Source**. Neglect the rise and fall times since they will be non-zeros anyways since the Analog Discovery 2 is non-ideal. Connect the oscilloscope channels as shown in the following figure (some hints are provided on the next page). After connecting the circuit and powering on the source, the oscilloscope should display two waves such as those you got on the LTSPICE simulations (1 cycle displayed is sufficient for this hardware part). Take a screenshot of the plot from your oscilloscope waveforms and include it with the data sheet. Make sure the numbers in your photo are readable.

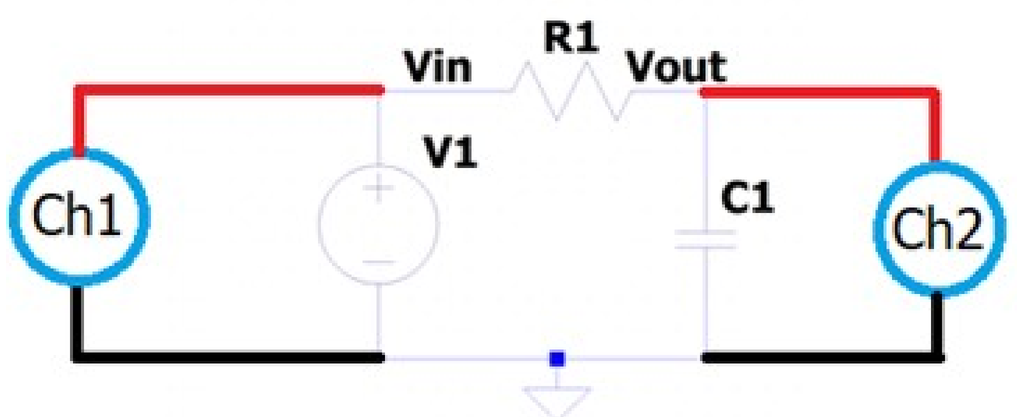

Figure 5: RC Circuit with oscilloscope connections

3. Now set the voltage source to a sine wave with the same frequency. Take a screenshot of the plot from the oscilloscope and include it with your data sheet. From the plot, measure the following:
    a. The magnitude of the voltage of the capacitor
    b. The RMS of the voltage of the source
    c. The phase difference, in degrees, between the source and the capacitor voltages.
    d. Is the capacitor voltage leading or lagging the source?

4. Replace the capacitor with an inductor as shown in **Figure 4: RL Circuit** with the same parameters as in Part 1 Question 4. Make sure you first **turn your source and oscilloscope off** (*this is a mandatory step before replacing any components in any electrical circuit*). Set the voltage source to a sine wave with

the same frequency. Take a screenshot of the plot from the oscilloscope and include it with your data sheet. From the plot, measure the following:

a. The magnitude of the voltage of the inductor
b. The RMS of the voltage of the source
c. The phase difference, in degrees, between the source and the inductor voltages.
d. Is the inductor voltage leading or lagging the source?

Hints on how to build a circuit on hardware and take measurements from oscilloscopes (see Lab 5):

1. **Building the circuit**: When building your circuit, make sure to connect the two channels of the oscilloscope as shown in **Figure 5**. Make sure that the two black wires of both channels are connected to the ground. Note that if you connected only one of these two black wires, the oscilloscope would still work on both channels since the black wires are already connected to each other internally inside the oscilloscope.

2. **Adjusting the oscilloscope scale**:
   a. X-axis (time scale): set the "time/division" to a value that lets you see 1 full sine-wave cycle on the oscilloscope screen. Hint: The screen has 10 divisions which should correspond to 1 time-period.
   b. Y-axis (voltage scale): set the "voltage/division" to a value that makes the peak-to-peak of the signal **of the voltage V_1** span 4-6 divisions (or any suitable value you deem fit).

You should eventually see two curves as in **Figure 3** (your numbers might be different).

Part 3 - Extra credit (Bode Plots):

In the previous tasks, you were plotting the voltages versus time. Instead, we would like to see how the amplitude and phase change as a function of frequency. On LTSPICE, build the circuit in

Figure *6* and do an AC simulation defining an AC amplitude of 1V with phase of 0˚. For the AC analysis, do 20 points per decade starting at 1Hz and ending at 100kHz (as seen in the figure below). **Plot V_{out} vs. frequency and capture your plot in a photo and include it with your data sheet.**

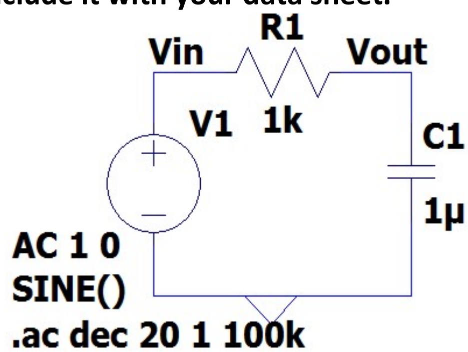

Figure 6: RC circuit for Bode Plots

Note that you get both the amplitude in dB (*dB = 20log(Vout/Vin)*) and the phase in degrees (phase = phase shift of Vout relative to Vin). Does the behavior of amplitude and phase vs. frequency agree with the trends you observed above?

Some helpful explanations:

Let's do the math and see why we get these results. Derive V_{out}/V_{in} treating the capacitance as a complex impedance and using voltage division. With some algebra, you will get the following expression, but make sure you can derive it from scratch.

$$\frac{Vout}{Vin} = \frac{1}{1 + j\omega RC}$$

Derive the magnitude and phase of this function:

$$\left|\frac{Vout}{Vin}\right| = \frac{1}{(1 + (\omega RC)^2)^{1/2}}$$

$$Phase\ of\ \frac{Vout}{Vin} = -\tan^{-1}(\omega RC)$$

The shape of your plots should look something like seen below. Your actual numbers will be different.

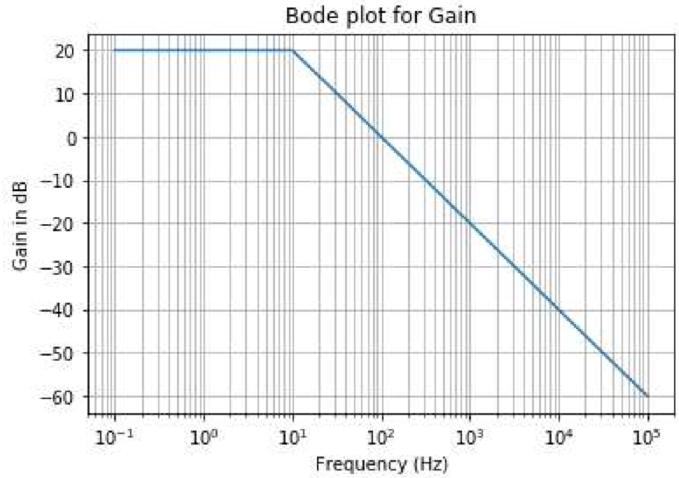

Figure 7: Bode Plot - Magnitude

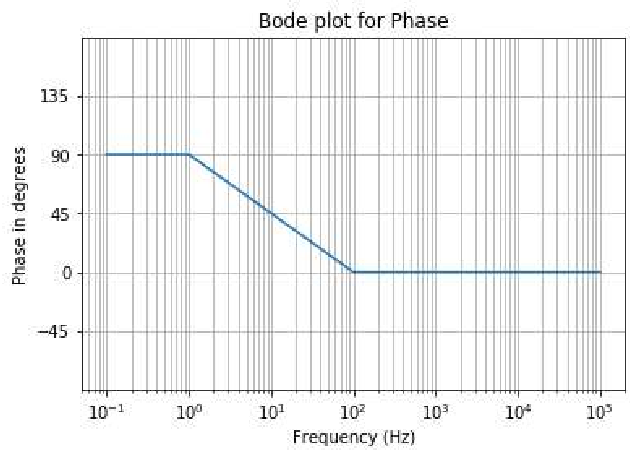

Figure 8: Bode Plot - Phase

Lab 6 CLOS:

- Understand the concepts of passive RC and RL circuits
- Using SPICE to analyze a circuit and calculate expected values
- Proficient in SPICE simulation
- Proficient in measurement of electrical systems using an oscilloscope
- Understands the difference between calculated and measured results
- Use AC steady state analysis to find currents & voltage within circuits driven by sinusoidal sources
- Understands how elements of an ecosystem are connected