Chapter 2

Characterizing Components Using Lab Tools and LTSpice

2.1 Pre-Lab

The answers to the following questions are due at the beginning of the lab. If they are not done by the beginning of the lab, no points will be awarded.

1. In an inductive circuit, which leads, voltage or current?

2. Explain and fill in the table below with how V_{OUT} differs between the two circuits in figures 2.1 and 2.2 as frequency approaches DC and ∞.

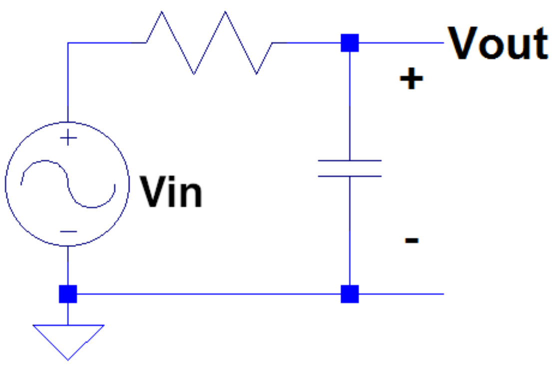


Figure 2.1: Pre-lab Circuit A

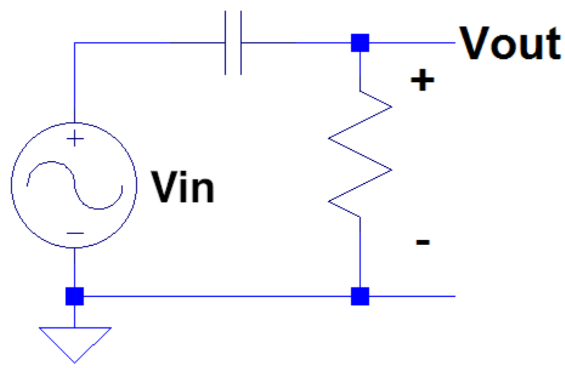


Figure 2.2: Pre-lab Circuit B

	DC	∞
Circuit A	Vin?	
Circuit B		

2.2 Section Overview

In this lab, differences between simulated values and real values from circuits will be explored in addition to practice using the o-scope to measure current and phase angle.

2.3 Determining Current through a Simple Circuit

Begin by setting the function generator to generate a 5V$_{PP}$ sine wave oscillating at 5kHz. Record what this source voltage is in phasor notation. Since this is the source for the circuit, a 0° phase angle can be assumed.

Source Voltage Amplitude:______________________________**V** (1)

Recall from lecture the behavior of current and voltage in capacitive and inductive circuits. In an inductive circuit current lags voltage, as in Figure 2.3. In a capacitive circuit, voltage lags current, Figure 2.4.

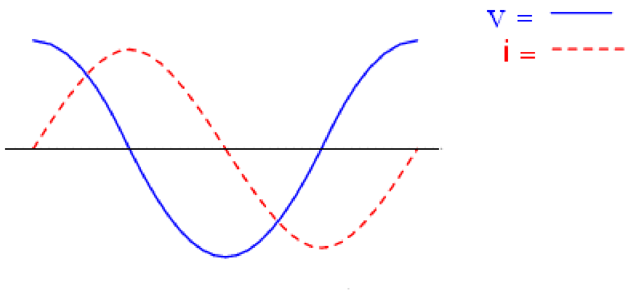


Figure 2.3: Inductive Circuit

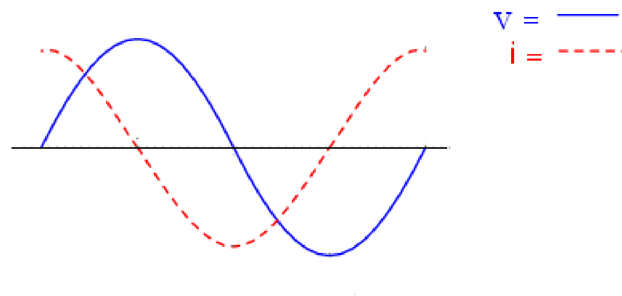


Figure 2.4: Capacitive Circuit

To demonstrate this, construct the circuit shown in Figure 2.5 using a 5V$_{PP}$ sine wave from the function generator as V$_S$.

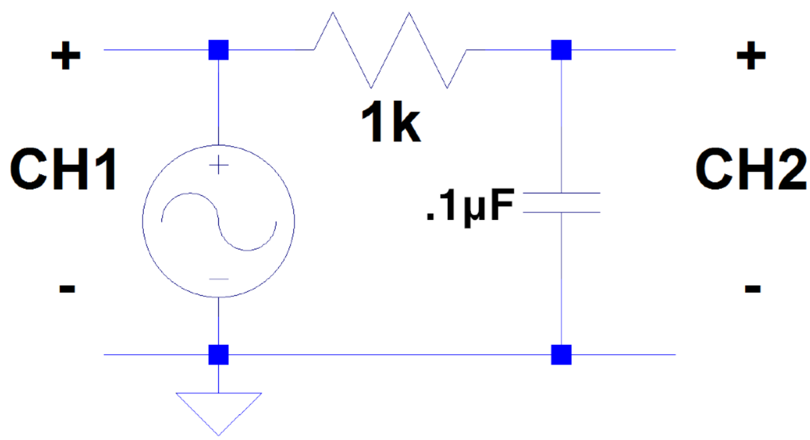


Figure 2.5: Schematic of Capacitive Circuit

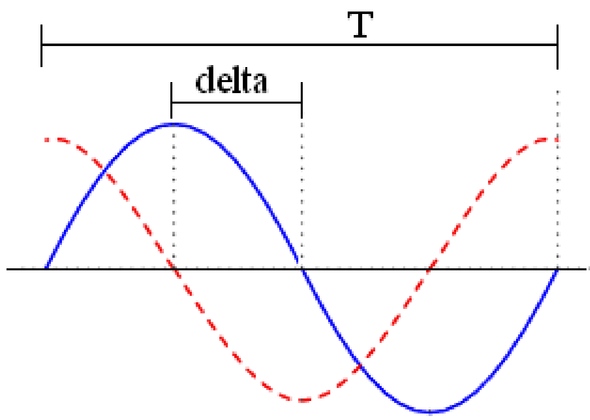
To measure the voltage and current in the circuit using the o-scope both channel 1 and 2 need to be connected to the circuit. See figure 2.5 for how to connect the different channels.

1. CH1 of the oscilloscope will measure the voltage across the source (V$_S$).
2. CH2 of the oscilloscope to measure the voltage across the capacitor.
3. Connect the ground of the probe for CH2 to the side of the capacitor not connected to the resistor. This will be considered the ground reference from now on.
4. Ground should always be the same, so connect all the ground leads to the same node.

In order to solve for the phase angle, the two signals on the o-scope must be investigated. Use the oscilloscope to solve for the phase shift between the voltage and current and use this to determine the phase shift. To do this, follow these steps:

1. Zoom in until only one period is visible. This allows for the cursors to be accurately adjusted giving the best measurement.

2. Press the cursor button to make two cursors appear on the screen (one for each waveform).
3. Press the type button so that the cursors can measure small differences in time.
4. Move each cursor to adjacent 'zero crossings' of the two corresponding waves. This step is very important because you are trying to measure the difference in time between the two waves. In order to do this, the cursors must be on the same parts of their own waves.
5. Record the time difference between the two cursor locations from the o-scope(delta).__________________(2)
6. Find the period of the waveforms (T = 1/f)__________________(3)
7. Find the fraction of a period that the wave forms are separated by (delta/T)__________________(4)
8. Multiply this value by 360 to get the number of degrees that the two waveforms are different__________________(5)



$$\frac{\text{delta}}{\text{T}} * 360 = \text{Phase Shift}$$

Figure 2.6: Calculating Phase Shift

9. Since the resistor is 1kΩ, according to Ohm's Law (V = IR), the current can be easily calculated.
   NOTE: The voltage across the resistor needs to be calculated from V_S and V_{CAP}
10. View both of these waves on the oscope and answer the following questions.

- What is V$_{CAP}$ in phasor notation __________________(6)
- Calculate V$_R$ in phasor notation __________________(7)
- Using the **MATH** function on the oscope measure and record V$_R$.__________________(8)
- What is the difference between calculated and measured? __________________(9)
- Calculate the current phasor. __________________(10)
  NOTE: Use Ohms Law.
- Is the voltage lagging the current, or is the current lagging the voltage? __________________(11)
- Is this consistent with what you learned in lecture? __________________(12)

2.4 Determining Impedance

Recall that the impedance **Z** of a circuit is a measure of the resistance and reactance for that particular circuit, measured in ohms. You can use impedance along with phasor voltages and currents to come up with an updated version of Ohm's Law. It is interesting to note that ohms law always applies to every circuit. If the circuit is being investigated strictly as a DC circuit, R is used. If the circuit is looked at in the AC domain, Z is used. The same rules even hold when analysis in the transient (S) domain occurs. The same equations work, just with a few different terms.

$$Z = \frac{V}{I}$$

Recall that impedance for capacitors and inductors is dependent on frequency according to the table below (where ω = 2πf).

Element	Impedance(Ω)
R	Z = R
L	Z = jωL
C	Z = 1/jωC

Use the same process for the inductive circuit in figure 2.7. Choose a frequency that allows for a phase shift of at least 10 degrees, though about 45 degrees will provide the most accurate results.

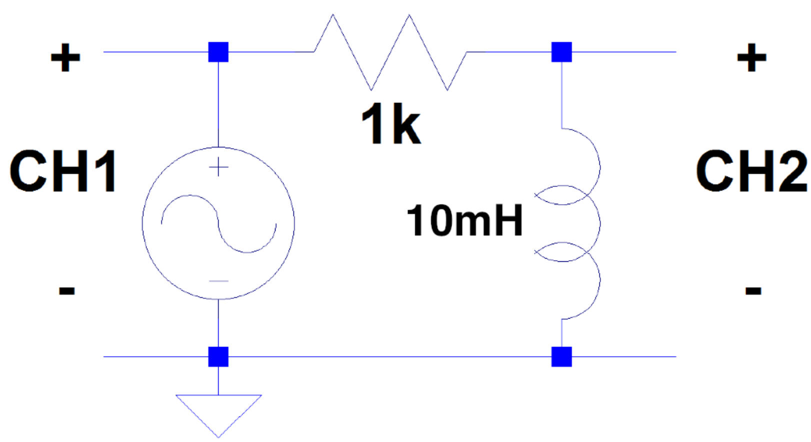


Figure 2.7: Schematic of Inductive Circuit

Connect the oscope probes as explained in the last circuit. Take the needed measurements and calculate the requested values.

Frequency:____________________(13)

Voltage Across Circuit:____________________(14)

Voltage Across Resistor:____________________(15)

Now use Ohm's Law and the voltage across the resistor to calculate the current through the entire circuit. With both the voltage and current, you can now calculate the impedance using the modified version of Ohm's Law.

Current Through Circuit:________________________(16)

Impedance Of Inductor:________________________(17) Hint: $Z = \frac{V}{I}$ *Do not use given value of inductor to calculate impedance. Instead, use the voltage across and current through the inductor.*

Now that you have both the magnitude and phase angle of the total impedance, you can break it up into the resistance (impedance due entirely to the resistor) and the reactance (impedance due entirely to the inductor). Draw the impedance of the circuit (Imaginary and Real components) in figure 2.8. Be sure to include labels on the axis and actual values.

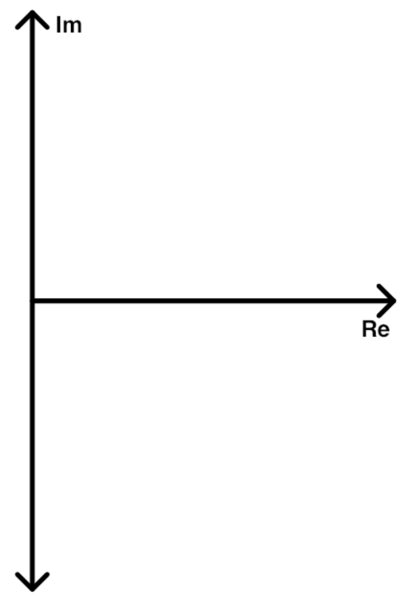


Figure 2.8: Magnitude of Impedance

Reactance of Inductor:________________________(18)
Finally, you can calculate the actual inductance of the inductor, not just the labeled value, using the formula that relates impedance to inductance: $Z = j * \omega * L$ where $\omega = 2 * \pi * f$.

Inductance of Inductor:________________________(19)

Is this consistent with the known value of the inductor? If not, why are they different?
________________________(20)

2.5 LTSpice Analysis

Now that you have measured some basic inductive and capacitive circuits for yourself, it's time to simulate one of them using LTSpice.

1. Begin by opening LTSpice and starting a new schematic.

2. Begin by adding an AC voltage source. To do this, press F2 on the keyboard and select "voltage" from the list of options. Click OK to place the source on the new schematic. Once the voltage source has been placed on the schematic, right click on it and then click "advanced" to bring up the options shown in Figure 2.9.

3. On this screen, choose the "SINE" option with an amplitude of 5V_{PP} and a frequency of 5000Hz. Finally click OK to finish.

4. Next, press the "R" key to bring up a resistor and place it on the diagram in the correct location. Again, right click on the resistor to set the resistance to 1K ohm.

5. Finally, add an inductor to complete the circuit. To add an inductor press "L". Place it and set the value of 10mH.

6. Once all of the components are placed, it's time to add the wires that connect them. To add wires press F3 and place them with the mouse. Right click when finished.

7. Before running the simulation designate a ground node, by pressing the "G" key and placing the symbol.

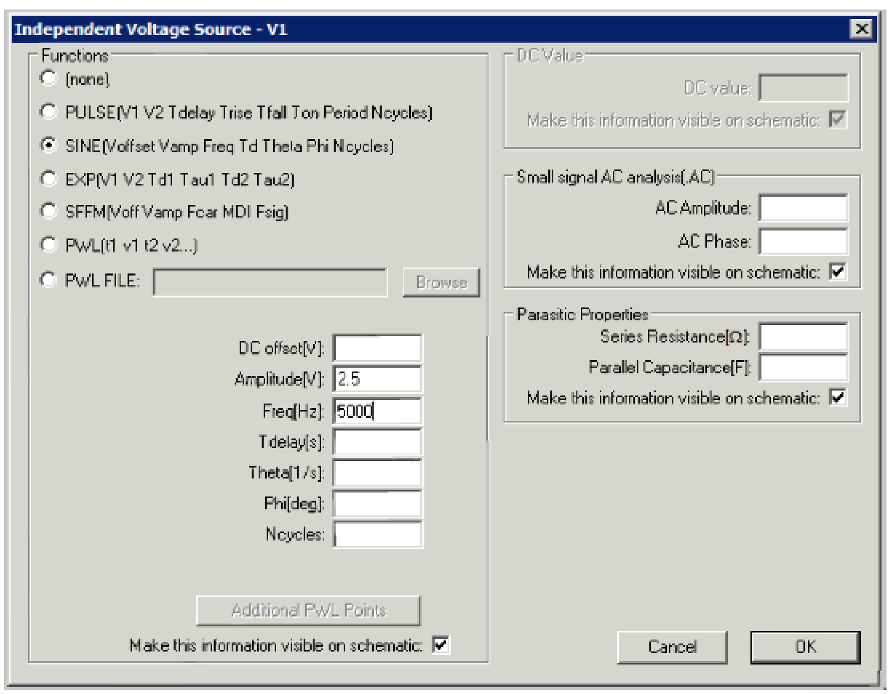


Figure 2.9: LTSpice Voltage Source Options

8. To run the simulation, go to Simulate and then Run to bring up the run settings window. Since the voltage source is alternating at such a high frequency only a very small window of time is needed to see the waveform. To begin, set the Start time at 1 and the Stop time at 1.01 and leave the Max Timestep blank. Select OK when ready.

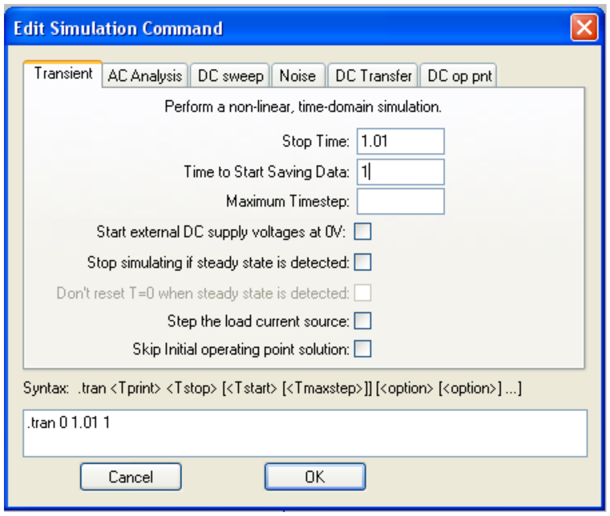


Figure 2.10: Edit Simulation Command Window

9. Now the simulation should have run, and a blank chart will be displayed. To see the voltage and current graphs use the mouse as a probe on the circuit. By moving the mouse over any component an ammeter probe will appear to measure current. Place the mouse over the resistor and select to view the waveform of the current going through it.

10. To view voltage at any node, move the mouse over the node and select when the voltmeter probe appears. Place the mouse over the node connected to the positive terminal of the voltage source and select.

11. There should be a couple of waveforms, one for voltage and one for current displayed. Are the waveforms you see consistent with what you saw on the oscilloscope earlier?

12. To measure phase difference use the same techniques used with the oscilloscope. Right select on the trace labels at the top of the graph and assign a cursor to each waveform.

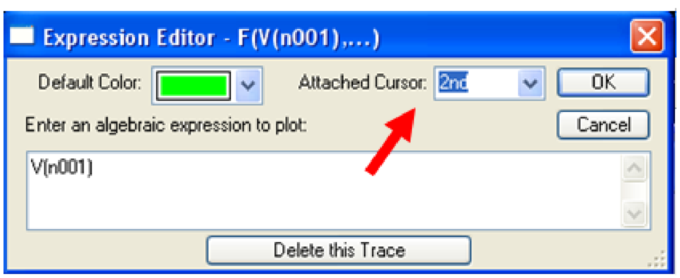


Figure 2.11: Assigning Cursors

13. Drag each cursor to adjacent peaks of the waveforms. To get the amplitude and time difference between the two waves, look at the window with the cursor information (pictured in Figure 2.12).

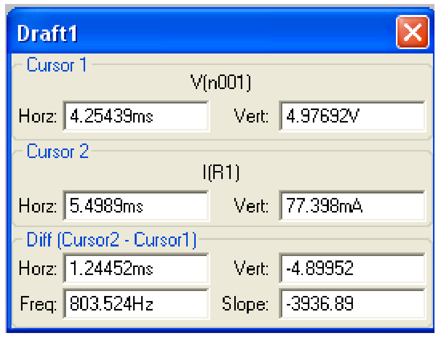


Figure 2.12: Reading Amplitude and Time Differences

14. The ”Vert” sections show the value of the waveform at that point in time. With the cursors at the peaks of the waves, they will be showing the full amplitude of each wave.

    Amplitude of Voltage Waveform:____________________________(21)

    Amplitude of Current Waveform:____________________________(22)

15. Under the Diff(Cursor2 & Cursor1) section you can read the time difference between the two waves under the ”Horz” section.

    Time Difference Between the Waves:____________________________(23)

16. Is this the same value that you got for your delta value from the oscilloscope? If not, why not? (24)

17. Print out simulation results and schematic and turn in with your lab report. However, make sure that everyone in your group is comfortable working with LTSpice. *Make sure schematic and simulation results are printed with a white background and the different signals can be differentiated*

2.6 Study Questions

1. If you had used a voltage source with 100kHz frequency instead of 1kHz, would your current waveform have been shifted more or less in the inductor circuit, figure 2.7? How about the capacitor circuit, figure 2.5?

2. What value capacitor would you have to add in series with the inductor in figure 2.7 to shift the current waveform until it is back in phase with the voltage and a frequency of 5 Khz?

3. Explain the process for solving for the inductor value in Section 2.4 in your own words.