Name:	Lab section:	Total score for lab: ______ /10

Lab 3: AC Phasors

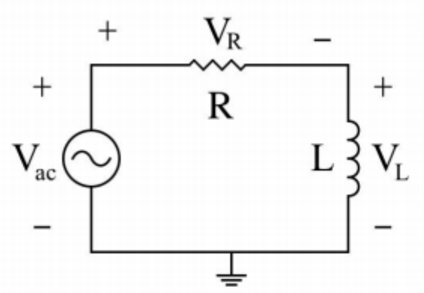
INTRODUCTION:

In this lab, you will apply your knowledge on phasor techniques to analyze some basic circuits comprising resistors and reactive components (i.e. **inductors**). The objective here is to relate a theoretical understanding of impedance and phase to what can be observed using an oscilloscope. In the process, you will learn how to simulate a sinusoid output of an RLC circuit with LTSpice.

Learning outcomes

- ☐ Simulate output AC signals of an RLC circuit
- ☐ Explain from observation the dependence of impedance on the AC signal frequency
- ☐ Observe the superposition of multiple frequency sources in a basic circuit with reactive components

Lab Task 1: Applying phasor techniques using single source



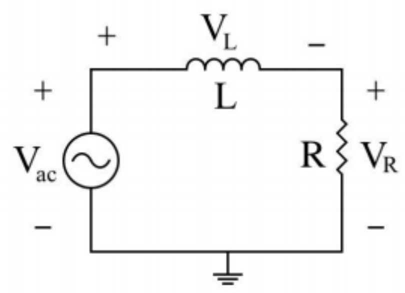


Fig 1a (left) & Fig 1b (right): Schematic of an RL circuit

1.1 Assemble the circuit of Fig. 1 in LTSpice, given $V_S = 5 \sin(2\pi f t)$ V.

1.2 TA will announce in class on the resistance R = __________ kΩ and inductance = __________ mH to be used.

1.3 To simulate AC signal (V_{ac}), right click on the voltage source and select "SINE". Set the DC offset, amplitude and frequency to 0V, 5V, and 10kHz respectively. Use shortcut "s" to add a SPICE directive then right click in text box and choose "Help me edit". You just need to select the tab "Transient" and set the stop time to 5e-3 (s). Then press OK to put the text command anywhere on the schematic. Press the Run button ( ) to initiate the simulation.

1.4 Right click anywhere on the schematic, select "View", and then select "Waveform Data". Record the maximum amplitude V_L (using Fig 1a) and V_R (using Fig 1b) of the last cycle on at 10 kHz. To measure phase, the wave form of V_{ac} should also be compared when you measure V_L and V_R.

1.5 Then set the frequency of the signal generator to 20 kHz and study the waveforms of V_L and V_R again.

Table 2: Simulated values for Task 1

	Simulated V_L (Fig 1a)		Simulated V_R (Fig 1b)		I (Calculate using V_R)	
	Amplitude	**Phase**	**Amplitude**	**Phase**	**Amplitude**	**Phase**
@ 10 kHz						
@ 20 kHz						

What is the closest phase of the capacitor voltage relative to the current used in your circuit at 2.5kHz and 5kHz?

For 10 kHz: ☐0° ☐-45° ☐45° ☐-90° ☐90°

For 20 kHz: ☐0° ☐-45° ☐45° ☐-90° ☐90° ______ / 5

Lab Task 2: Dual source

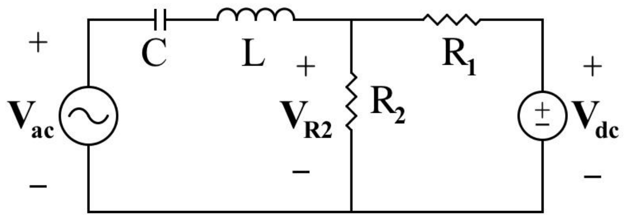


Fig 2: Schematic of circuit with two input sources with different frequencies

2.1 Assemble the circuit you have analyzed in Fig. 2 in LTSpice, giving R_1 = 2 kΩ, R_2 = 1 kΩ, L = 10 mH, C = 33 nF, $V_{ac} = 5 \sin(2\pi ft)$ V, and V_{dc} = 9V. Use an additional voltage source to apply a DC voltage of V_{dc} = 9V.

2.2 Keep the amplitude of the AC signal at 5V and set the frequency to 5 kHz.

2.3 Measure V_{R2}. Observe the waveform of V_{R2} in both AC mode and DC mode. You should notice two frequency components on the waveform display in LTSpice.

2.4 Capture the screenshots two waveforms of V_{ac} and V_{R2} in the space provided below. Identify the various frequency components on your graph. Note down the amplitude of the AC wave from V_{R2} and its phase relative to V_{ac}.

2.5 Then adjust the AC signal frequency to 20 kHz and simulate V_{R2} again.

Capture a screenshot of waveforms of V_{ac} and V_{R2} at 5 kHz and insert below:

At 5 kHz, V_{R2} = __________ (DC component) + _______________ (AC component)

Capture a screenshot of waveforms of V_{ac} and V_{R2} at 20 kHz and insert below:

At 20 kHz, V_{R2} = __________ (DC component) + _______________ (AC component)

2.6 What is the difference in DC offset between your results from Lab Task 2 and Lab Task 1?

☐ No difference ☐ positive DC offset ☐ negative DC offset _______/ **5**