ECE 212 Spring 2017 M / Th / F Names: ______________________________
Circuit Analysis II Group A / B ______________________________

Lab 2: Gain and Phase Measurements of RC, RL, and RLC Networks

Objectives

In this lab exercise you will analyze and measure the phase and amplitude relationships of input voltages and responses in circuits comprised of resistors, capacitors, and inductors. An op-amp with a capacitor in the feedback loop will also be analyzed and measured. Prior to performing the lab, you will analyze each circuit by hand and check your results using PSpice.

The purpose of the pre-lab work is to make sure you understand how each circuit works and what results to expect when you make measurements in the lab. This should allow you to spend nearly all of your time in the lab building circuits and making measurements. If you know what results to expect, you will know when you have done each experiment correctly. If your hand calculations and PSpice results agree, you will know that your pre-lab work is correct.

Pre-Lab Instructions (10pts)

Calculations and comparisons

1. Using circuit analysis techniques such as nodal analysis and/or voltage dividers with phasors, calculate the theoretical gains and phase relationships asked for in each part of this lab (see the Appendix A for an example). **All calculations should be shown on a separate sheet to be handed in**; that is, your results should **not** be recorded in the spaces designated throughout this assignment until you are in lab (or just before lab) and comparing your prelab results with those of your lab partner. You must use Excel to speed up these hand calculations. In other words, program each formula once, then copy the formula in Excel and change R or the frequency to compute the other required answers. Also, note that you must hand in the sheet of results and the sheet of formulas using the <ctrl-backquote/tilde> key (above the <tab> key).

2. Verify every hand calculation by simulating each circuit in PSpice using the given parameters. (See Appendix B for some tips on how to use PSpice efficiently.) Record the simulated results in the appropriate spaces. Specifically, you should add a column in your spreadsheet to record the simulated results. The hand calculations and simulated results should be the same (typically within 1%) and will be used when looking at the actual circuit measurements made in the lab. The schematic for every circuit needs to be printed out. If a parameter of a circuit is varied (for example, the value of a resistor changes), only print the circuit once for its initial parameters. In addition, print out the input/output waveforms for part 4 and part 5.2, as well as the circuit schematics for part 5.2. Attach these printouts to the lab. In the interest of saving paper, you may print more than one circuit schematic per page. (**NOTE: Your name must appear in the filename of all circuit and waveform printouts!**)

3. After both you and your lab partner have completed parts 1 and 2 above, compare answers, determine what the best answers are, and fill in the appropriate parts of the table for one copy of the In-Lab portion. (Only one copy needs to be handed in.) NOTE: All questions asked in this lab should be answered while actually doing the lab, after you have built the corresponding circuits and made measurements. In other words, you should not answer them before going to lab!

In-Lab Instructions (15pts)

Part One: RC Networks

1. Assume $v_i(t) = \cos(\omega t)$ in the circuit in Figure 1, such that $f = \omega/(2\pi) = 10kHz$. Measure the phase shift in degrees between v_i and v_o for the following values of R in the RC network shown in Figure 1.

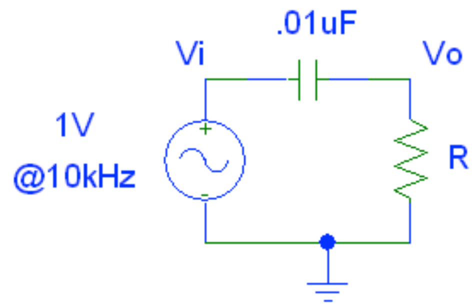


Figure 1: First RC Network

Value of R (Ω)	Hand Calculation	PSpice Simulation	Circuit Measurement
R = 1k	φ =	φ =	φ =
R = 2k	φ =	φ =	φ =
R = 5.6k	φ =	φ =	φ =
R = 10k	φ =	φ =	φ =

To calculate phase shift on the oscilloscope, display both waveforms on the screen. Then press the "Cursor" button, set the "Type" to "Time," and adjust the two position knobs until one vertical cursor intersects the peak of the waveform of CH1 and the other cursor intersects the peak of the waveform of CH2. If you can't see the vertical cursor, adjust the knob until the cursor appears. The oscilloscope should show the time difference between these two peaks. Multiplying this time by the frequency (f, in Hz) of the signal and then again by 360 will give the phase shift in degrees.

Phase shift (degrees): $\varphi = t_{p\text{-}p} * f * 360$

Does v_o lead or lag v_i? (Circle one.) lead / lag

2. Using a 1kΩ resistor for R, record the amplitude (1/2 the peak to peak) of the output voltage for the following three input voltage frequencies. (The amplitude of the input voltage should remain at 1V peak, i.e. 2V peak to peak)

Frequency	Hand Calculation	PSpice Simulation	Circuit Measurement
1kHz	v_o =	v_o =	v_o =
10kHz	v_o =	v_o =	v_o =
100kHz	v_o =	v_o =	v_o =

Based on the relationship between the frequency of the input voltage and the amplitude of the output voltage, why is this RC circuit considered a "high-pass" filter?

3. In the circuit in Figure 2, again assume that $v_i(t) = \cos(\omega t)$, such that $f = \omega/(2\pi) =$ 10kHz. Measure the phase shift between v_i and v_o for the following values of R in the RC network shown in Figure 2.

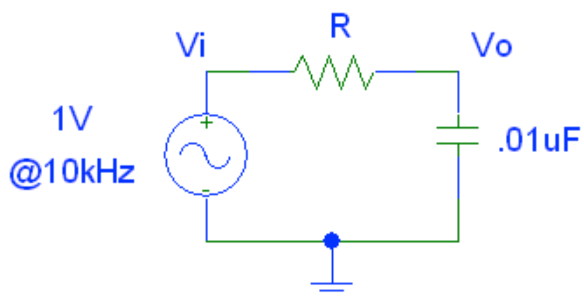


Figure 2: Second RC Network

Value of R (Ω)	Hand Calculation	PSpice Simulation	Circuit Measurement
R = 1k	φ =	φ =	φ =
R = 2k	φ =	φ =	φ =
R = 5.6k	φ =	φ =	φ =
R = 10k	φ =	φ =	φ =

Does v_o lead or lag v_i? (Circle one.) lead / lag

4. Using a 1kΩ resistor for R, record the output voltage for the following three input voltage frequencies. (The amplitude of the input voltage should remain at 1V peak.)

Frequency	**Hand Calculation**	**PSpice Simulation**	**Circuit Measurement**
1kHz	v_o =	v_o =	v_o =
10kHz	v_o =	v_o =	v_o =
100kHz	v_o =	v_o =	v_o =

Based on the relationship between the frequency of the input voltage and the amplitude of the output voltage, why is this RC circuit considered a "low-pass" filter?

Part Two: RL Networks

1. Record the amplitude of the output voltage for the circuit shown in Figure 3, where $v_i(t) = \cos(\omega t)$, and f is the following three input voltage frequencies.

Frequency	**Hand Calculation**	**PSpice Simulation**	**Circuit Measurement**
1kHz	v_o =	v_o =	v_o =
10kHz	v_o =	v_o =	v_o =
100kHz	v_o =	v_o =	v_o =

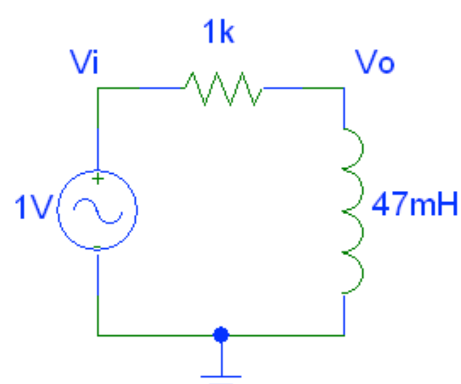


Figure 3: First RL Network

Based on the relationship between the frequency of the input voltage and the amplitude of the output voltage, would this circuit be considered a "low-pass" or a "high-pass" filter?

high pass / low pass

Does v_o lead or lag v_i? (Circle one.) lead / lag

2. Record the amplitude of the output voltage for the circuit shown in Figure 4, where $v_i(t) = \cos(\omega t)$, and f is the following three input voltage frequencies.

Frequency	Hand Calculation	PSpice Simulation	Circuit Measurement
1kHz	v_o =	v_o =	v_o =
10kHz	v_o =	v_o =	v_o =
100kHz	v_o =	v_o =	v_o =

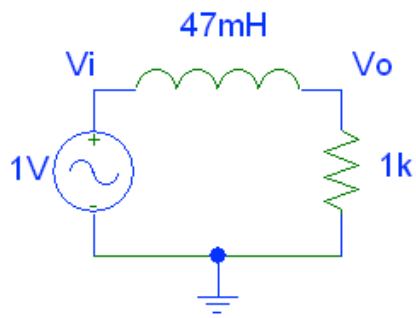


Figure 4: Second RL Network

Based on the relationship between the frequency of the input voltage and the amplitude of the output voltage, would this RL circuit be considered a "low-pass" or a "high-pass" filter?

high pass / low pass

Does v_o lead or lag v_i? (Circle one.) lead / lag

Part Three: Analysis of an RLC Circuit

1. Construct the RLC circuit shown in Figure 5.

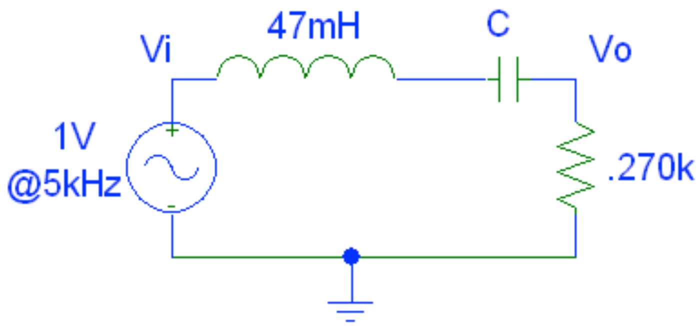


Figure 5: RLC Circuit

2. Assuming that $v_i(t) = \cos(\omega t)$, such that $f = \omega/(2\pi) = 5\text{kHz}$, measure the phase shift between v_i and v_o for $C = 0.01\mu F$.

Hand Calculation	PSpice Simulation	Circuit Measurement
φ =	φ =	φ =

Does v_o lead or lag v_i? (Circle one.) lead / lag

3. Measure the phase shift between v_i and v_o for $C = 0.047\mu F$.

Hand Calculation	PSpice Simulation	Circuit Measurement
φ =	φ =	φ =

Does v_o lead or lag v_i? (Circle one.) lead / lag

Part Four: Design Problem

1. Suppose the output voltage of Figure 6 is desired to lag the input voltage by 30°. Calculate the value of R that would yield such a phase shift of the output voltage. Looking at the table of "Standard 5 Percent Tolerance Resistor Values" on the inside cover of the textbook; choose a value of R that is closest to the calculated value.

R (calculated): _______

R (chosen): _______

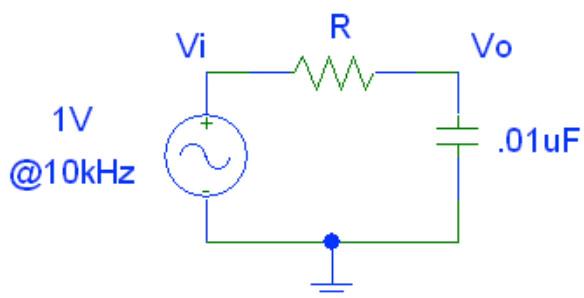


Figure 6: RC Lag Network

2. With the chosen value of R in place, measure the phase shift between v_i and v_o.

Hand Calculation	PSpice Simulation	Circuit Measurement
φ =	φ =	φ =

3. Compute and record the percent error between the measured phase shift and the expected value of 30°.

% Error: _______

What could be the sources of this error?

How much can the phase vary if R and C each have +/– 5% tolerances?

Part Five: Analysis of an Op-Amp with a Capacitor in the Feedback Loop

1. Recall that the gain of the op-amp circuit in Figure 8 is:

$$v_o/v_i = 1 + R_2/R_1 \qquad \text{(where } R_2 \text{ is the } 2k\Omega \text{ resistor, and } R_1 \text{ is the } 1k\Omega\text{)}$$

Measure the gain and the phase shift between the output voltage and input voltage of the op-amp circuit shown in Figure 8, assuming that $v_i(t) = \cos(\omega t)$, such that $f = \omega/(2\pi) = 10kHz$. If there is any clipping on the output waveform, the DC offset on the function generator may need to be adjusted.

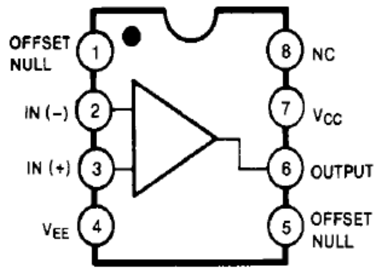


Figure 7: Pin Layout of the LM741 Op-Amp

NOTE: Set V_{EE} to –15V and V_{CC} to 15V.

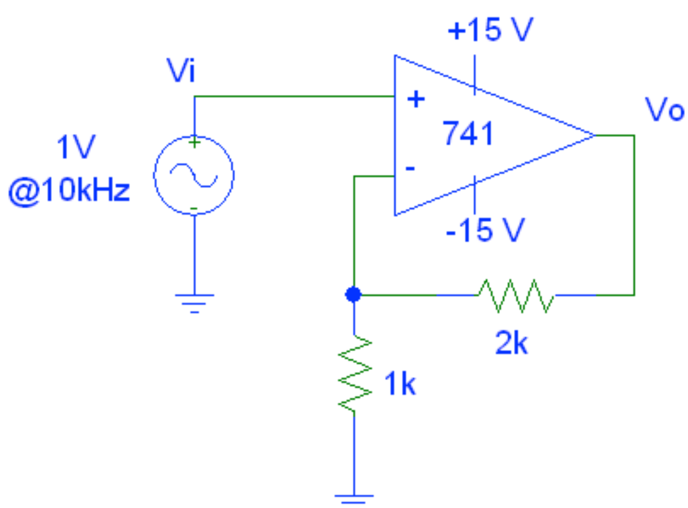


Figure 8: Non-Inverting Op-Amp Circuit

Hand Calculation	PSpice Simulation	Circuit Measurement
$v_o/v_i =$	$v_o/v_i =$	$v_o/v_i =$
$\varphi =$	$\varphi =$	$\varphi =$

2. Replace the 2kΩ resistor with a 0.01μF capacitor in parallel with a 100kΩ resistor. Measure the phase shift between the output voltage and input voltage, as well as the gain. By writing everything in terms of phasors, the same gain equation given in part 5.1 can be used to find both the gain and the phase. **Make sure your simulation time is long enough for the circuit to reach steady state.**

Hand Calculation	PSpice Simulation	Circuit Measurement
v_o/v_i =	v_o/v_i =	v_o/v_i =
φ =	φ =	φ =

3. Replace the 1kΩ resistor with each of the following resistors and measure the gain and phase shift.

Value of R (Ω)	Hand Calculation	PSpice Simulation	Circuit Measurement
R = 2k	v_o/v_i =	v_o/v_i =	v_o/v_i =
	φ =	φ =	φ =
R = 5.6k	v_o/v_i =	v_o/v_i =	v_o/v_i =
	φ =	φ =	φ =
R = 10k	v_o/v_i =	v_o/v_i =	v_o/v_i =
	φ =	φ =	φ =

What trends do you notice in phase and gain as the resistor value increases?

Does v_o lead or lag v_i? (Circle one.) lead / lag

What would you expect to happen if a 10MΩ resistor were put in place of the 1kΩ resistor?

When you have completed the lab, sign and print your names below and have the TA initial next to each name.

TA

____ ________

____ ________

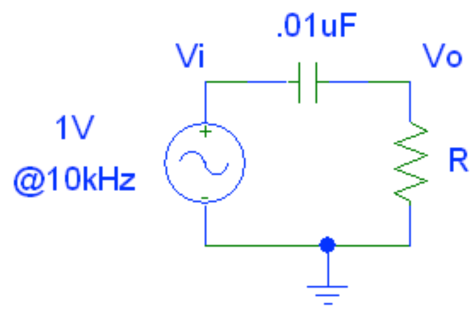
Skills Assessment (5pts)

Everybody must demonstrate that he/she can perform certain lab skills, as described below.

Fill in your names in the spaces provided. A TA will initial the appropriate space when a skill has been successfully demonstrated.

	Name #1	Name #2
	________________________	________________________
1. Demonstrate how to determine the time difference between two traces on the oscilloscope.	_____	_____
2. Explain the difference between the peak voltage (amplitude) and the peak-to-peak voltage.	_____	_____
3. Given a drawing of two sinusoidal signals, identify which is leading and which is lagging.	_____	_____
4. Given a sinusoidal voltage signal on the oscilloscope, predict what the amplitude will be when measured using the multimeter, and show that you can measure it with the multimeter.	_____	_____
5. Demonstrate how to read the amplitude of an AC signal using the MAX / MIN / RMS on the oscilloscope.	_____	_____

Appendix A: Sample hand calculation



Assume R = 1kΩ. Replace the resistance and capacitance with their equivalent impedance (R for the resistor, 1/jωC for the capacitor). To find V_o, use the voltage division equation:

$$V_o = \left(\frac{R}{R + 1/j\omega C} \right) V_i$$

so:

$$\frac{V_o}{V_i} = \frac{R}{R + 1/j\omega C}$$

$$= \frac{(j\omega C)R}{(j\omega C)R + (j\omega C)1/j\omega C}$$

$$= \frac{j\omega CR}{j\omega CR + 1}$$

To find the gain, find |**N**| / |**D**|, where |**N**| is the amplitude of the numerator and |**D**| is the amplitude of the denominator

|**N**| = $\text{sqrt}[\text{Re}^2 + \text{Im}^2]$ = $\text{sqrt}[0^2 + (\omega CR)^2]$
= ωCR
= $2\pi * 10\text{kHz} * .01\mu\text{F} * 1\text{k}\Omega$
= 0.628

|**D**| = $\text{sqrt}[\text{Re}^2 + \text{Im}^2]$ = $\text{sqrt}[1^2 + (\omega CR)^2]$
= $\text{sqrt}[1 + (2\pi * 10\text{kHz} * .01\mu\text{F} * 1\text{k}\Omega)^2]$
= 1.18

Vo/Vi = 0.628/1.18 = 0.532

Vo = .532V = 532mV (since Vi = 1V)

To find the phase of **N/D**, find ∠**N** – ∠**D**, where ∠**N** is the phase of the numerator and ∠**D** is the phase of the denominator:

∠**N** $= \tan^{-1}[\text{Im/Re}]$
$= \tan^{-1}[\omega CR/0]$
$= 90°$
∠**D** $= \tan^{-1}[\text{Im/Re}]$
$= \tan^{-1}[\omega CR/1]$
$= \tan^{-1}[2\pi * 10\text{kHz} * .01\mu\text{F} * 1\text{k}\Omega]$
$= 32.14°$

So, ∠**N** – ∠**D** = 90° – 32.14° = 57.86°

To calculate gain in PSpice, simply divide the output voltage amplitude by the input voltage amplitude. To calculate phase shift, measure the time difference between a peak on the input waveform and the nearest peak on the output waveform. Then use the equation in Part One ($\varphi = t_{p\text{-}p} * f * 360$) to find the phase shift in degrees.

Appendix B: Tips for using PSpice to measure the time difference between two peaks

Changing step size

If the trace doesn't look smooth when graphed, it will be very hard to get an accurate measurement of the time difference between two sinusoids. Changing the step size should help:

1. After creating a new simulation profile, open the simulation settings by selecting the "Edit Simulation Profile" under the "PSpice" menu on the schematic page.
2. Click the "Analysis" tab in the "Simulation Settings" window and adjust the "Maximum step size" value by entering in a numerical time increment.
3. Play around with this setting before each simulation until a reasonable smoothness is achieved. If you start with a maximum step size value that is 3 orders of magnitude smaller than the signal period it should get you close.

Increasing the duration of the simulation

Another factor in getting an accurate measurement of the time difference is making sure that the transient responses have decayed away. In other words, PSpice assumes you have connected the circuit at $t = 0$, which means it can take many cycles for the responses to reach steady state. If your traces have not reached steady state when graphed, try the following:

1. After creating a new simulation profile, open the simulation settings by selecting the "Edit Simulation Profile" under the "PSpice" menu on the schematic page.
2. Click the "Analysis" tab in the "Simulation Settings" window and adjust the "Start saving data after" value by entering in a numerical amount.
3. Try increasing this setting before each simulation until the transient response is no longer present in the graphed waveform.

Measuring the time difference directly

Although it is relatively easy to get PSpice to display the time corresponding to a particular instant on your trace, it is even more convenient to get PSpice to show you the time <u>difference</u>:

1. After running the simulation, click "Trace" from the main menu bar and select "Display" from the "Cursor" submenu. A small window will appear labeled "Probe Cursor" having three rows: "A1", "A2", and "dif". A1 will be controlled by left clicks of the mouse while A2 will be controlled by right clicks of the mouse. Row "dif" is the difference between A1 and A2. The first column is time and the second column is voltage.
2. On the graph, left click the output signal from the legend in the bottom left and then select "Peak" from the "Trace/Cursor" menu used above. If you have a hard time seeing where the cursor is you can turn the X grid off by selecting the "Axis Settings" under the "Plot" menu and clicking "None" for both the major and minor grids in the "X Grid" tab.
3. Now right click the input signal from the legend in the bottom left. Using the same process as step 2, get the peak closest to the peak chosen for the output signal.
4. Read off the first column of the "dif" row to determine the time difference.