Name: _____

Basic AC Lab - ELEC 105L

LAB #11: RC Lag-Lead Band-pass Filter Design and Testing

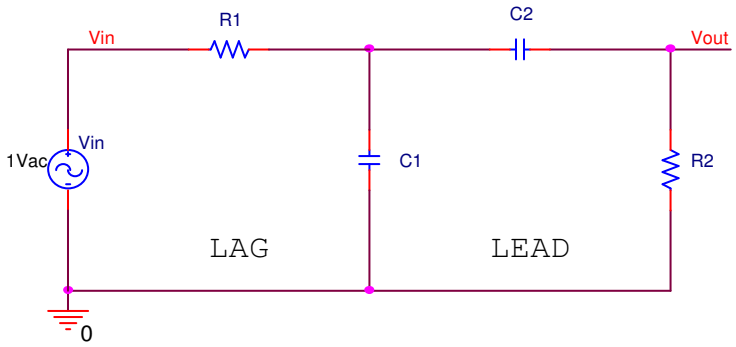
INTRODUCTION: The objective of this lab is to design an RC lag-lead band-pass filter, simulate it with PSpice and verify its operation by testing it in the lab.

MATERIALS:

- PSpice Circuit Simulator
- 2 – capacitors of same value
- 2 – resistors of different values
- HP33401A Digital Multi-meter
- HP54603B Oscilloscope
- HP33120A Wave form Generator

Design Specifications:

1. Pass band low frequency cutoff of 500 Hz (Lead filter cutoff frequency)
2. Pass band high frequency cutoff of 25 kHz (Lag filter cutoff frequency)
3. Capacitors are the same value and in the range of 100 pF – 0.1 uF
4. Resistors are in the range of 100Ω – 1MΩ
5. The circuit should have the form shown below



PART I: Component Selection

The low and high pass frequencies are determined from the formula

$$f_L = \frac{1}{2\pi R_2 C} \qquad f_H = \frac{1}{2\pi R_1 C}$$

The first step is to select an initial capacitor value in the range specified (HINT: if there is no other constraint for the capacitor value pick a value around the middle of the range).

$C =$ _____

Once this is done the resistor values for the low and high pass frequencies can be calculated.

$R_2 =$ _____ $R_1 =$ _____

An alternate method for determining the components would be to choose one resistor value, calculate the capacitor value, then calculate the second resistor value.

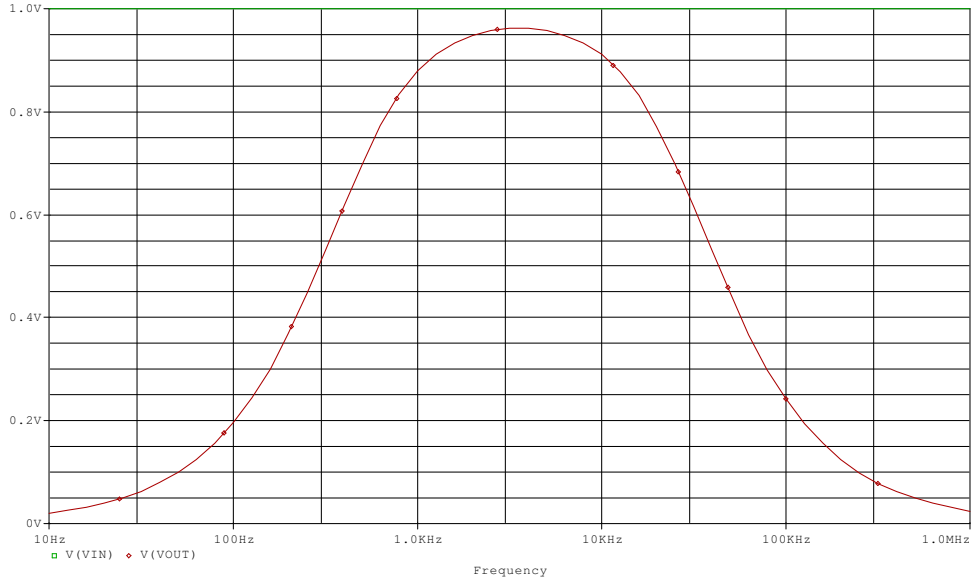
PART II: Circuit Simulation

If the resistor values fall within the range specified the circuit can be simulated in PSpice. If the resistors are above or below the range allowed then the capacitor value will need to be changed and the resistors recalculated so they are within the range specified.

Build the lag lead RC circuit in PSpice. Next select the Voltage/Level Marker and place voltage probes on the input and output of the circuit. Before simulating, an AC Sweep will need to be Setup in the simulation settings.

In ***PSpice Schematics***, click on the Setup Analysis button or use the **Analysis>Setup…** menu. Next enable only the AC Sweep checkbox. Click the **AC Sweep…** button. Set the **AC Sweep Type** to *Decade*, the **Pts/Decade** to *100*, the **Start Freq.** to *10*, and the **End Freq.** to *1000k*. Click OK and close the Analysis Setup box.

Run the simulation and switch to the ***PSpice A/D*** window to observe the results. The graph should look similar to the one below. One change may be needed. On the toolbar above the graph find the Log X Axis button and make sure it is depressed. This will change the scale of the frequency to logarithmic so the pass band may be observed better. If **Log Y Axis** is selected also the graph will more closely resemble the straight line approximation discussed in class.



Check the accuracy of the design using the Toggle cursor button. To select the V_{out} graph click on the ◊ V(VOUT) legend below the graph. Move the cursor to the 500Hz and 25kHz locations, and measure the voltage. Record the cursor voltages below.

V_{500Hz} = _____ V_{25kHz} = _____

As a final step copy the schematic and graph and paste them into your lab report. To copy the schematic select the **Edit>Copy to Clipboard…** menu in the ***PSpice Schematics*** window. To copy the graph select the **Window>Copy to Clipboard…** menu in the ***Pspice A/D*** window. Creating reports with integrated data is very valuable. If there is ever any need to refer to the results in the future, they can be found in one location.

PART III: Circuit Measurements

The next step in this lab is to build the circuit that's been designed and measure the output voltage at several different frequencies.

Once the circuit is built, set the input voltage to 1Vrms, and the frequency to 3kHz. Using the DMM, V_{out} should measure close to 1V (~960mV). If so, change the frequency to 100kHz and measure V_{out} again. If its close to 250 mV the circuit has passed the operation test. Complete the table below by setting the function generator to each frequency and recording the DMM's AC voltage reading.

Freq. (Hz)	Vout (V)
10	
30	
100	
300	
1000	
3000	
10000	
30000	
100000	

Using the oscilloscope capture a screen image to the computer. Set the scope up with the V_{in} on Ch.1 and V_{out} on Ch.2. Set the waveform to f=30 kHz. Set the time base so there are 1-4 cycles on the screen.

Plot a graph of the data here, and include a similar one in the lab report (using the LeadLag_Graph.xls spreadsheet on the server, enter your data and copy that graph to your report).