

EEE 117L Laboratory – Network Analysis

Lab #5: PSpice

Lab Day and Time: Wednesday 1:30-4:10 PM

Group Number: # 03

Group Members: (Last Name, First Name)

Member #1: Algador, Vigomar Kim

Member #2: Chan, Casey

Member #3: Trinh, Bon

Total Score: /100

General Instructions:

- 1) You will be graded on the neatness and clarity of your schematics, so make sure you can see the details of your circuits when you include them in your report. Similarly, when you include the pictures of the results of your simulations, make sure to provide a brief, accurate, and informative description of the graphs you created. For each output graph, make sure to include the titles (“y” versus “x”), the labeled axes with appropriate units, the domain and ranges shown in the graph, and any relevant information that will help the reader understand the output of the simulations.
- 2) You will need to modify this document in order to complete the exercises.

Work Breakdown Structure: It is important that every group member do their share of the work in these labs. Remember that you will receive no credit for the lab worksheet if you did not contribute. Write in the Table provided below, which group member(s) contributed to the solution of each problem in the lab worksheet. Also remember that only on lab worksheet per group will be turned in to Canvas. If there was any group member that did not contribute, then write their name in the space provided below.

Problem Number	Group member(s) that worked on the problem.
Part I.1	Algador, Vigomar Kim Chan, Casey Trinh, Bon
Part I.2	Algador, Vigomar Kim Chan, Casey Trinh, Bon
Part II.1	Algador, Vigomar Kim Chan, Casey Trinh, Bon
Part II.2	Algador, Vigomar Kim Chan, Casey Trinh, Bon
Part III	Algador, Vigomar Kim Chan, Casey Trinh, Bon

Absent member(s): _____

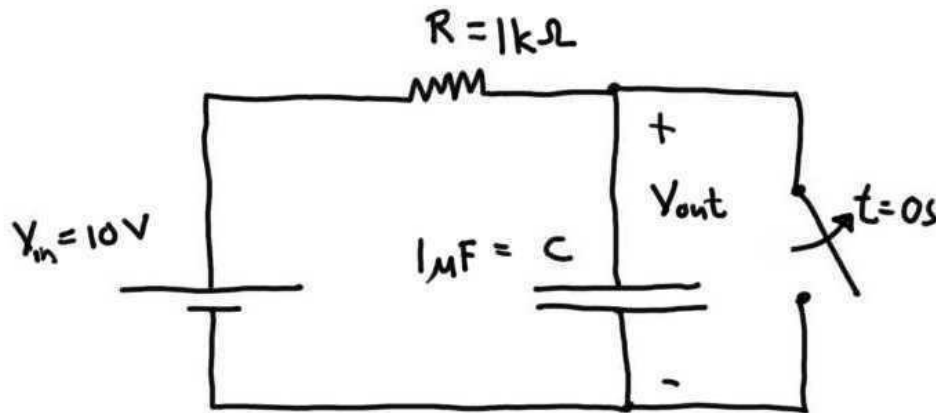


Figure 1. Transient Analysis for an RC Circuit

1) Transient Analysis of an RC circuit

Score: /20

Construct the circuit shown in Figure 1 in PSpice, without the switch. Since PSpice does not have a switch component, simulate the switch by setting the “Initial Condition” or “IC” of $V_{out} = 0\text{ V}$. In the simulation select the run time to be

$0.0 \leq t \leq 5\tau$, where τ is the time constant for the RC circuit that you calculated in the prelab. Calculate the “maximum step size” such that the output graph is composed of 500 straight lines by using the formula: $\text{maximum step size} = \frac{\text{total run time}}{\text{number of steps}} = \frac{5\tau}{500}$.

Once the simulation is run, then graph the capacitor voltage, V_{out} , as a function of time for $0.0 \leq t \leq 5\tau$. Include pictures of a) the circuit schematic and b) the output graph of the simulation below.

a) Circuit Schematic:

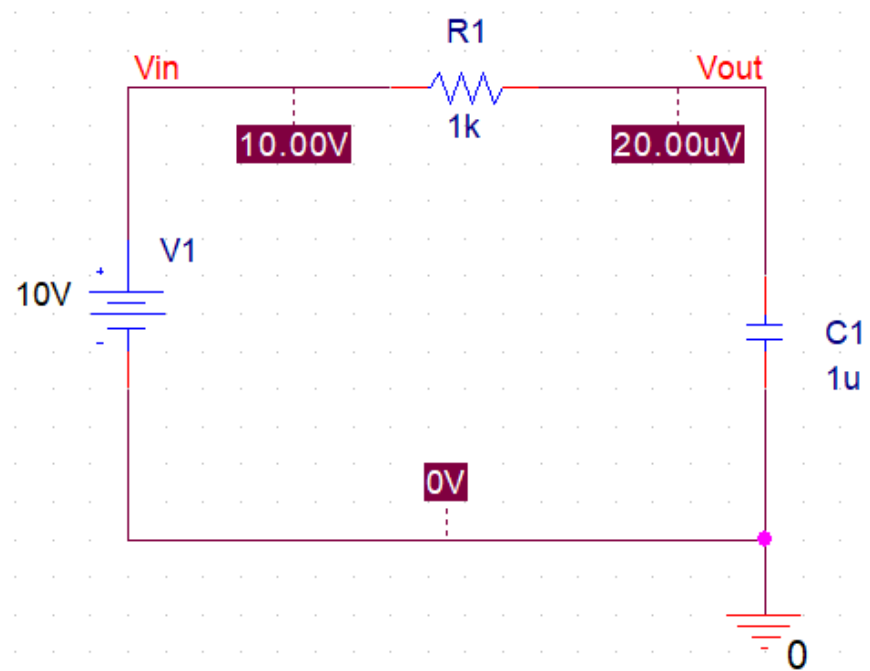


Figure 2. RC Circuit for the Transient Simulation

b) Output:

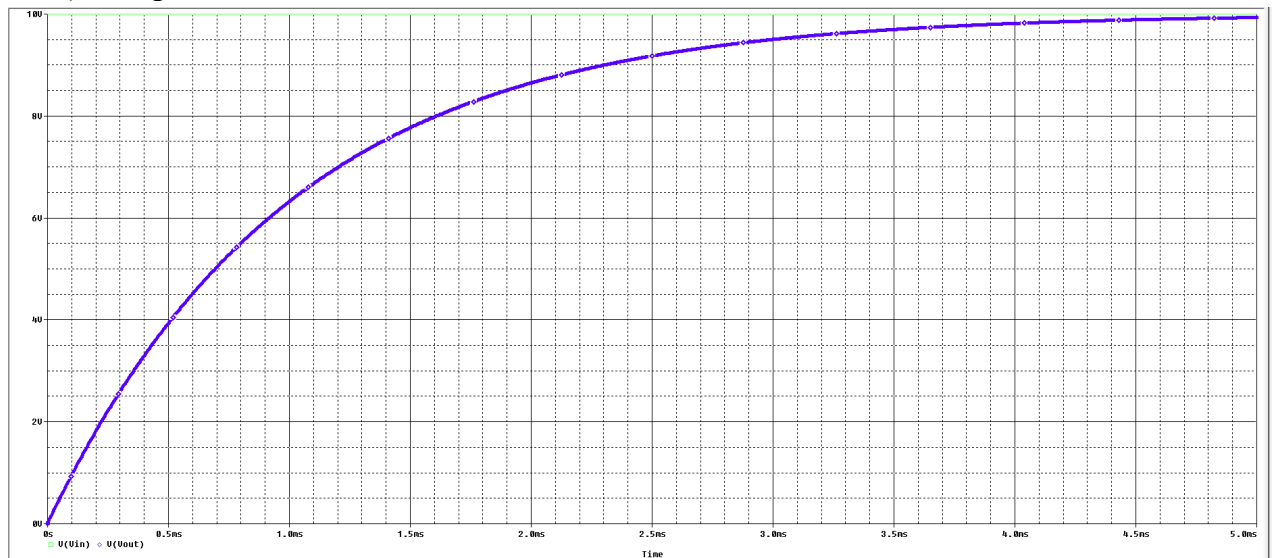


Figure 3. V_{out} vs. Time

2) Transient Analysis with R and C Switched

Score: /20

Construct a new circuit by interchanging the resistor and the capacitor in the circuit shown in Figure 1. Repeat the simulation this time with V_{out} being the voltage across the resistor. Also make sure to change the IC for the capacitor voltage to 0 V. Include pictures of a) the circuit schematic and b) the output graph of the simulation below.

a) Circuit Schematic:

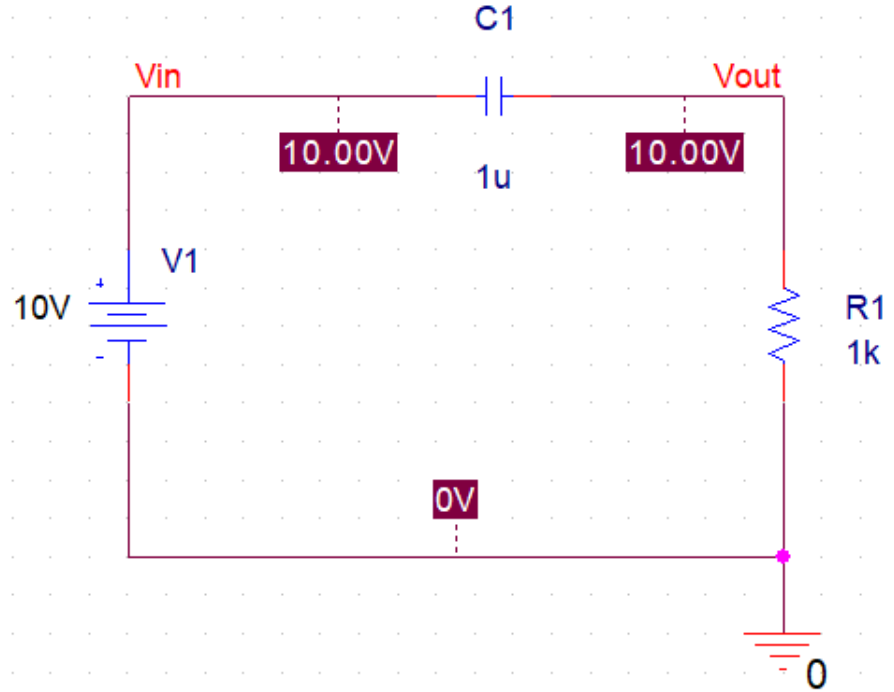


Figure 4. RC switch circuit for Transient Simulation

b) Output:

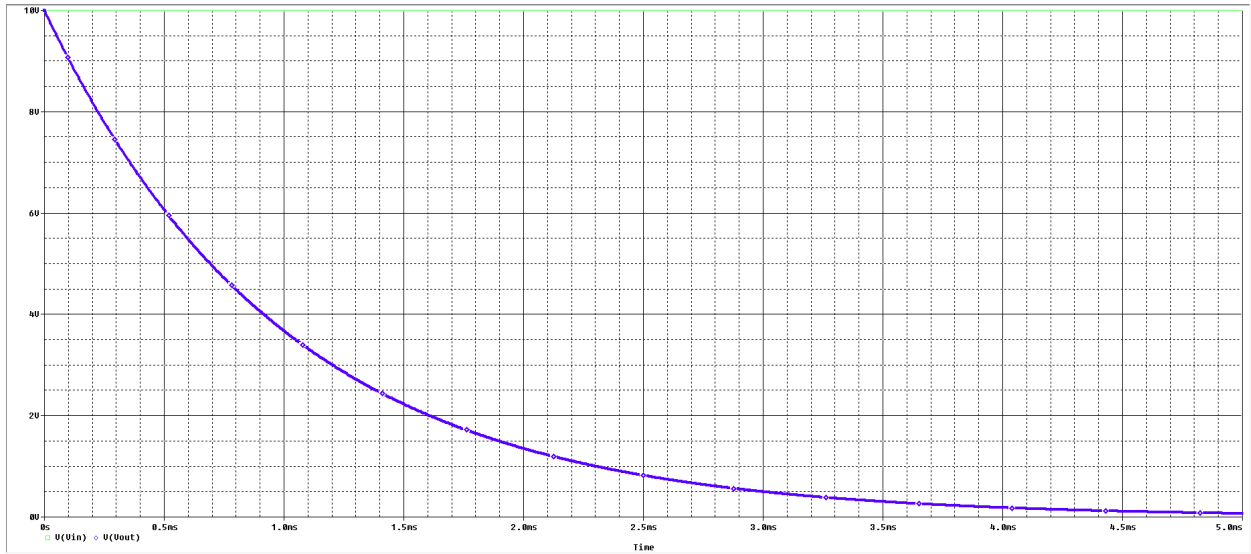


Figure 5. V_{out} Vs. Time Graph for RC Switched Circuit

Part II: AC Sweep Simulations

Total Score: /50

1) AC Sweep/Analysis of an RC Circuit

Score: /25

This analysis uses the two circuits simulated in the transient analysis section. However, the DC source is replaced with an AC (VAC) source. Adjust the source for 1 Volt with 0° phase shift. Do not set the IC for the capacitor voltage in these simulations. In the Simulation Profile use the AC Sweep. For sweep type, select decades. The sweep parameters should be 50-points/decade, with a start frequency of 1Hz and an end frequency of 100 kHz. PSPICE uses the phasor method to calculate the amplitude and phase of the circuit responses at each of the 50 points in the frequency range specified.

The first circuit (Figure 1 without the switch) is now ready for simulation. For the first trace, use the magnitude of the output voltage. Select M (for magnitude) from the functions and V(Vout) from the variables. The next trace will be the phase. Since the values of the magnitude and phase are so different, a plot should be used for the phase. Use “Add Plot to Window” (under “Plot” on the tool bar.) Select the new plot and add the trace “P[V(Vout)].” If the simulation is correct this RC circuit should have a magnitude of 0.707 and a phase shift of -45° at $\omega = 1 \text{ krad/s}$ ($f = 1000/2\pi \text{ Hz}$.) You can use the “Toggle Cursor” button in order to confirm this. Note that PSPICE plots the graphs in Hertz and not radians per second.

Include pictures of the circuit schematic and two output graphs below.

a) Circuit Schematic:

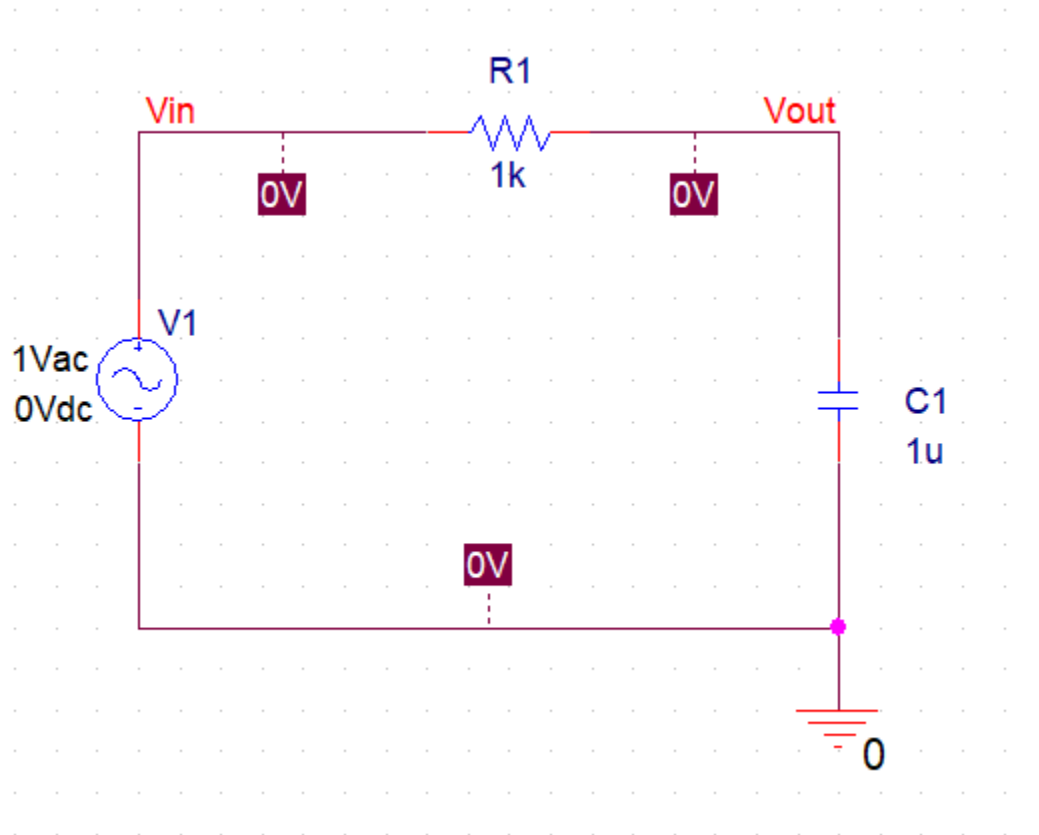
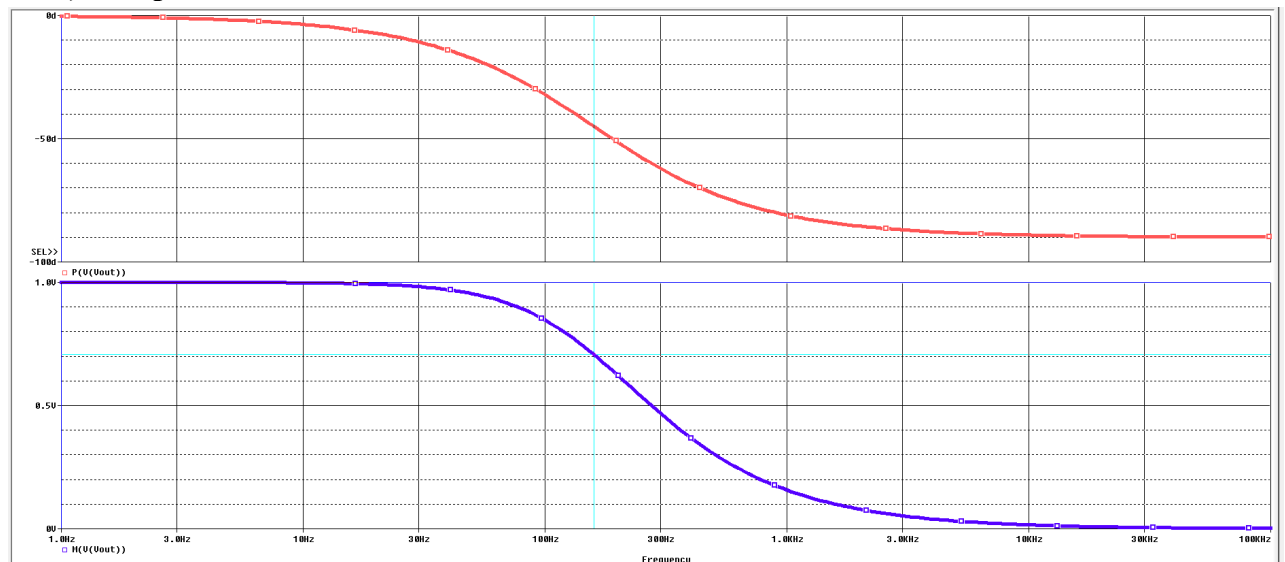


Figure 6. AC Sweep of RC Circuit

b) Output #1 and #2:



Red Line: Phase of V_{out} Vs. Frequency (Hz)

Purple Line: Magnitude of V_{out} Vs. Frequency(Hz)

c) Output Table:

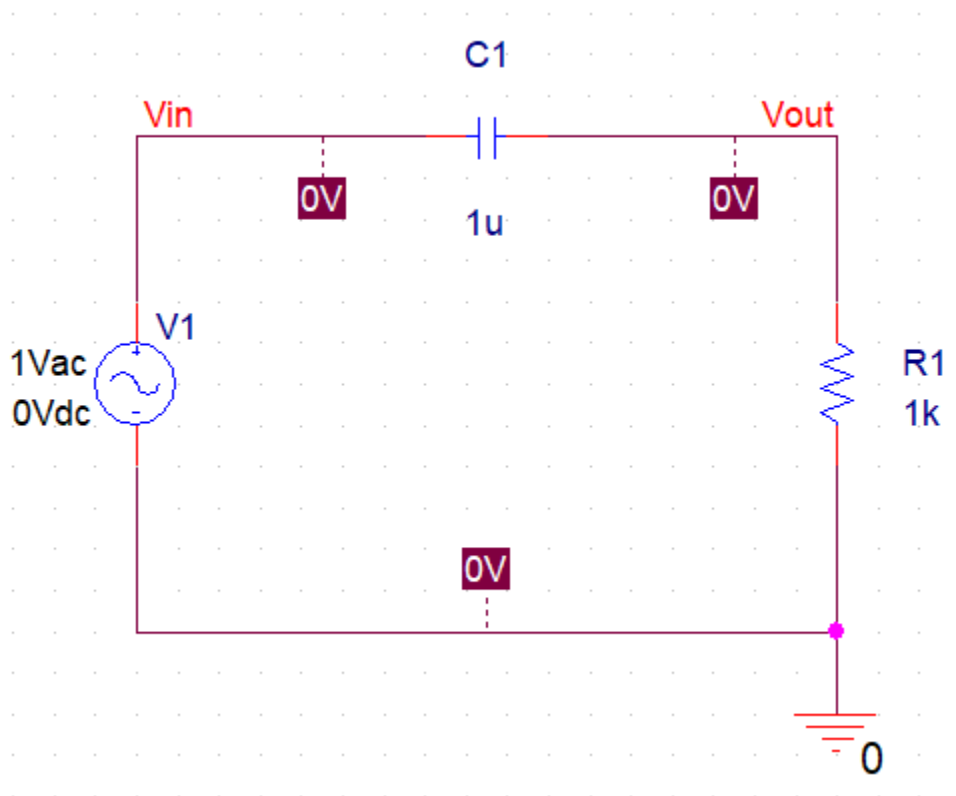
	Trace Color	Trace Name	Y1	Y2	Y1 - Y2		Y1(Cursor1) - Y2(Cursor2)	
		X Values	159.312	1.0000	158.312		Y1 - Y1(Cursor1)	Y2 - Y2(Cursor2)
	CURSOR 1,2	M(V(Vout))	706.741m	1.0000	-293.239m		0.000	0.000
		P(V(Vout))	-45.028	-359.995m	-44.668		-45.735	-1.3600

2) AC Sweep/Analysis with R and C Swapped

Score: /25

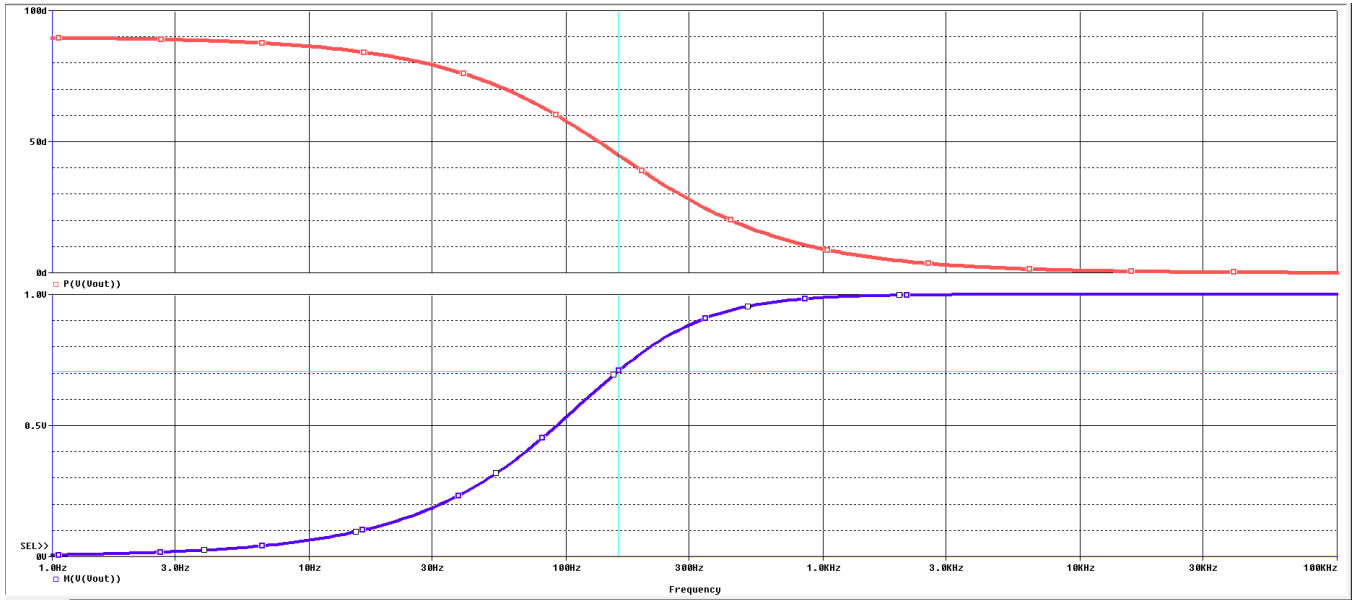
Repeat the simulation and plot of magnitude and phase for the modified circuit by simulating the resistor voltage, after having swapped the capacitor and the resistor. Include pictures of the circuit schematic and two output graphs below.

a) Circuit Schematic:



RC Circuit with Resistor and Capacitor Swapped from Figure 6.

b) Output #1 and #2:



Output Table

	Trace Color	Trace Name	Y1	Y2	Y1 - Y2		Y1(Cursor1) - Y2(Cursor2)	701.152m		
		X Values	159.312	1.0000	158.312		Y1 - Y1(Cursor1)	Y2 - Y2(Cursor2)	Max Y	M
	CURSOR 1,2	M(V(Vout))	707.435m	6.2831m	701.152m		0.000	0.000	707.435m	6.2
		P(V(Vout))	44.972	89.640	-44.668		44.264	89.634	89.640	44.

Part III: Conclusions

Total Score: /10

Explain in a few paragraphs the purpose of the lab, the experimental set up and methodology, and central results of the lab and these experiments. **You should be quantitative** in this summary. Include any important equations used and explain their significance. Write the conclusion as if you were writing an English essay. This is an important portion of the lab, so make sure to do a good and thorough job.

In this lab, we used PSpice to create RC circuits. We then ran simulations to see the output of the circuit. The first part of the lab we did was transient analysis. The first circuit was constructed normally. It starts at 0 V, then increases to 10 V. The second circuit was with the resistor and the capacitor switched. It starts at 10V, then decreases to 0 V. The output shown is a logarithmic line. The difference is that the line is flipped.

For the second part of the lab, we used the same circuit, except we changed the voltage source to VAC source. The simulation we did was an AC Sweep. In the first simulation, the magnitude and phase were parallel. When the resistor and the capacitor were switched, the magnitude and phasor were inverses of each other. When the magnitude increased, the frequency would decrease.

Appendix to PSpice - Getting Started in Cadence PSpice

This is a very brief description of the steps necessary to get started in Cadence PSpice. For more detail you are referred to the PSpice tutorial from Purdue University referenced in the syllabus.

Drawing a Schematic

1. Open the “Cadence Release 17.2-2016” folder followed by the “Capture CIS” program.
2. Choose the “Allegro PCB Design CIS L” option.
3. Open “New Project”.
4. Select “Analog or Mixed A/D”.
5. Select an appropriate name (like RC-Circuit) and select a save location of your choice.
6. Select “Create a Blank Project”.
7. Once the workspace is open, press “P”, which is the shortcut key for “Place part”. In the portion of the user interface opens (on the right hand side of the screen) find the button “Add Library” under the “Libraries” menu. The shortcut key for “Add Library” is “Alt+A”. Highlight all of the files with the “.olb” extension then click the “Open” button.
8. To begin placing parts use “Part” under the “Place” menu –left click to place the part and use ESC to change to a new part.
9. For ground use “0/CAPSYM”. Every PSPICE program must have a ground.
10. To change a part name or value click on the desired name or value.
11. To add or change the IC on a capacitor click on the circuit symbol.
12. Use “NET ALIAS” to add names to frequently used nodes (such as Vout).
13. After you have the schematic in the desired form do a “SAVE”.

Doing a Circuit Simulation

1. Under PSPICE on the menu bar choose “New Simulation Profile”.
2. Choose a name for the simulation such as RC_Transient or RC_AC.
3. In the (hidden) menu that just popped up behind your workspace, choose “Allegro PSpice Simulator”. You can access this menu by minimizing PSpice.
4. For the transient response choose “RUN TO” 5ms with a maximum step size of 5us. For the AC sweep use a logarithmic sweep with a starting frequency of 1Hz and a final frequency of 100kHz. Use 50 steps per decade.
5. Choose “RUN” under PSPICE on the menu bar.
6. If you have a successful simulation PROBE should open. Use ADD under TRACE on the menu bar to add the desired traces.

Use the “Snipping Tool” in order to create pictures of both the schematics and output results so that you can include them in the worksheet.