

# **EECE 2105**

## **Electric Circuits I Laboratory**

### **Lab 2: pSPice Simulation & MATLAB**

Fall 2025

Student(s) Names/ID: Gavino Martinez / 20678524

# Lab 2: pSPice Simulation & MATLAB

## I. Introduction

PSPICE is a simulation software for electric circuits based on SPICE that is distributed under license of OrCAD Systems Corporation. It is a tool used by engineers, students and other professionals related to electrical and electronic circuits, that allow them to analyze, design and create a circuit that is planned to be assembled, with the objective of having a better understanding of the operation of the specified circuit, and find faults in it. The way that this type of software operates is using mathematical models to replicate the behavior of an electric circuit. PSPICE is a software based on SPICE (Simulation Program with Integrated Circuit Emphasis), which is an Open-Source simulation software that perform simulation of circuits in time domain, while some other software simulators work on the frequency domain.

## II. Objectives

At the end of this lab assignment student will be able to use pSpice for the analysis of resistive networks on direct current and alternating current.

## III. Equipment Required

- Computer/Workstation
- pSpice Simulation Software
- MATLAB Software

## **IV. Prior To the Lab**

### **1) PSpice and MATLAB**

This laboratory session involves the use of a desktop computer and is oriented toward developing innovative methods of analysis using PSpice and MATLAB software, and providing an increased challenge to those students using a different language to develop an analysis routine. The general procedure for the analysis to be performed is outlined in the Tutorial: Introduction to PSpice handout (uploaded to Blackboard) and should be read carefully while completing the necessary steps included in this experiment. The first few sections, up to Performing DC Circuit Simulations, should be read before coming to the laboratory session.

PSpice will be used to simulate a given electrical circuit. The output results after a simulation include nodal voltages, branch currents, graphical plots, etc. These results can be used to compare with hand-made calculations and measurements, as a means of verification.

MATLAB will be used to solve for a system of linear equations. This allows the student to avoid solving for systems of equations by hand, which can be very time consuming if there are a large number of unknown variables.

## 2) Resistor Combinations

There are two general rules for simplifying (reducing) a resistive circuit:

**Series Combination:** Two resistors  $R_1$  and  $R_2$  connected together so that one terminal of one of the resistors is connected to one terminal of the other resistor, forming a node. This node should not connect to any other path for the current to flow (Figure 1). This setup can be reduced to a single resistor  $R_{eq}$  by using the series resistance equation:

$$R_{eq} = R_1 + R_2$$

For  $N$  resistors placed in series, then:

$$R_{eq} = R_1 + R_2 + \dots + R_N$$

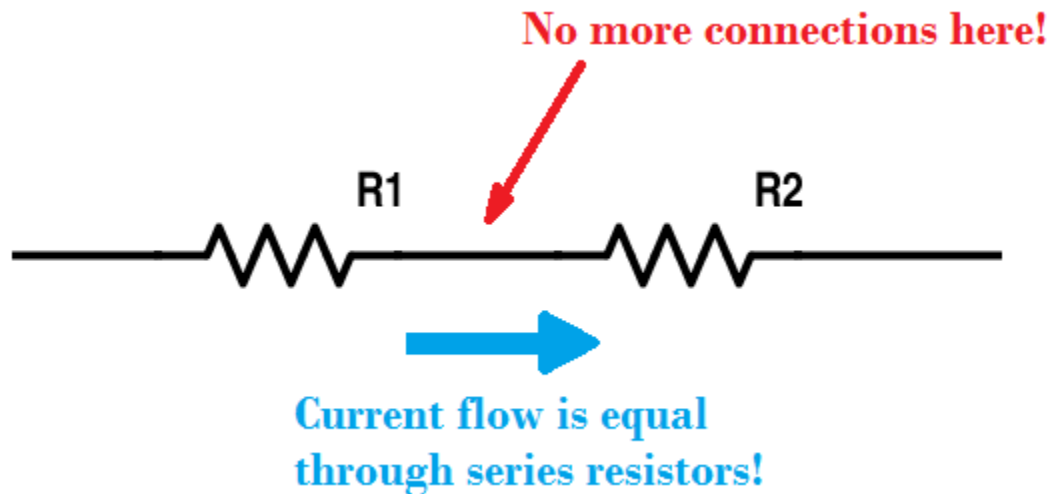


Figure 1: Series Combination

Figure 2 shows an example circuit with two resistors in series. To determine the voltage across each resistor in this circuit, we utilize the voltage division equation:

$$v_1 = \frac{R_1}{R_1 + R_2} v, \quad v_2 = \frac{R_2}{R_1 + R_2} v$$

Notice that the source voltage  $v$  is divided among the resistors in direct proportion to their resistances; the larger the resistance, the larger the voltage drop. This is called the principle of voltage division, and the circuit in Figure 2 is called a voltage divider.

In general, if a voltage divider has  $N$  resistors ( $R_1, R_2, \dots, R_N$ ) in series with the source voltage  $v$ , the  $n$ th resistor ( $R_n$ ) will have a voltage drop of:

$$v_n = \frac{R_n}{R_1 + R_2 + \dots + R_N} v$$

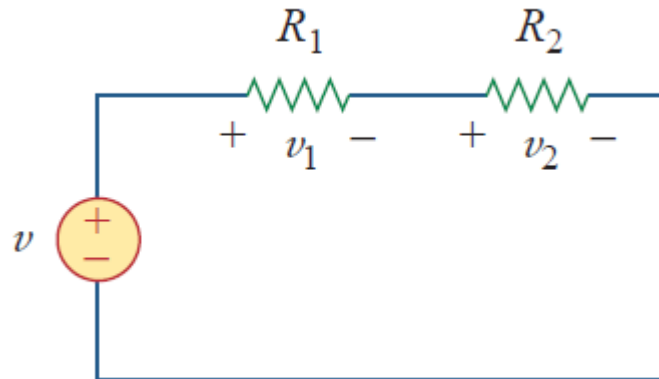


Figure 2: A single-loop circuit with two resistors in series

**Parallel Combinations:** Two resistors  $R_1$  and  $R_2$  connected together using both of their terminals, forming two nodes. These nodes could have other connections/paths, and the resistors would still be in parallel with each other (Figure 3). This setup can be reduced to a single resistor  $R_{eq}$  by using the parallel resistance equation:

$$R_{eq} = \left( \frac{1}{R_1} + \frac{1}{R_2} \right)^{-1}$$

For  $N$  resistors placed in parallel, then:

$$R_{eq} = \left( \frac{1}{R_1} + \frac{1}{R_2} + \dots + \frac{1}{R_N} \right)^{-1}$$

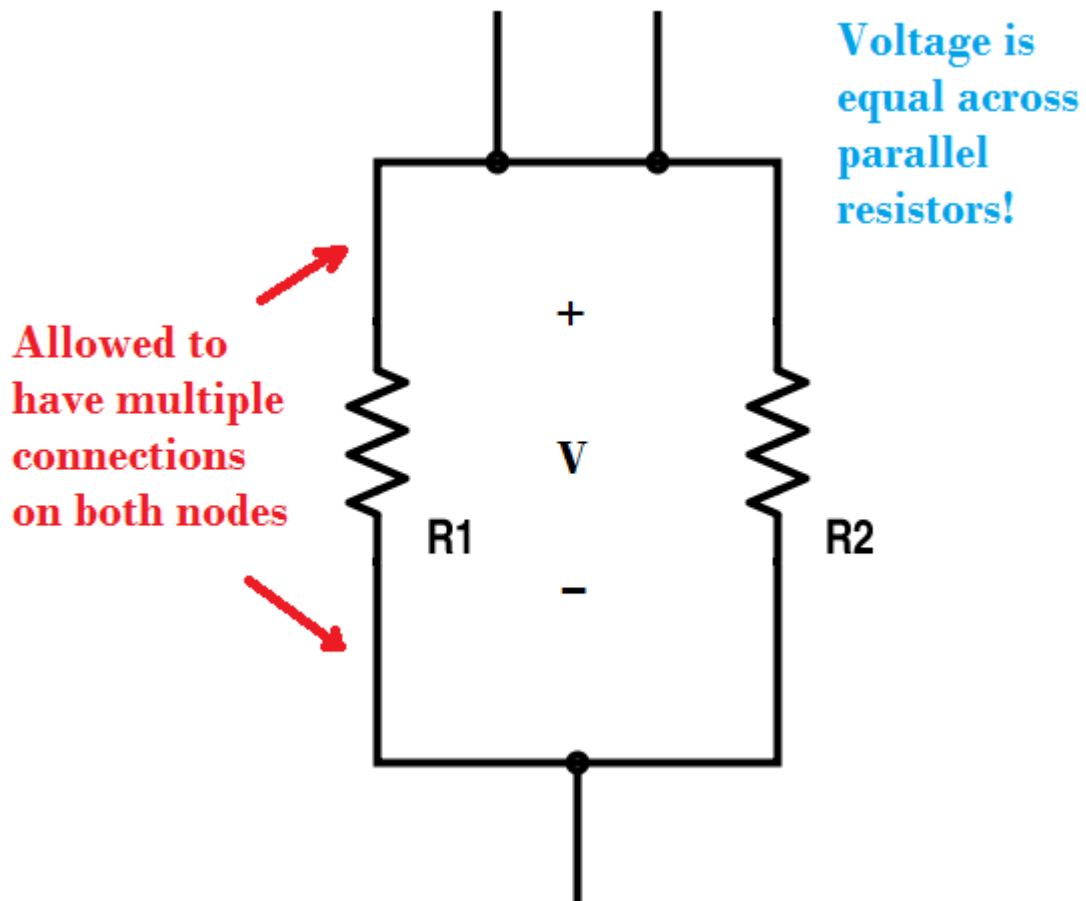


Figure 3: Parallel Combination

Figure 4 shows an example circuit with two resistors in parallel. To determine the current through each resistor in this circuit, we utilize the current division equation:

$$i_1 = \frac{R_2}{R_1 + R_2} i, \quad i_2 = \frac{R_1}{R_1 + R_2} i$$

Notice that the total current  $i$  is shared by the resistors in inverse proportion to their resistances; the larger the resistance, the smaller the current it draws. This is called the principle of current division, and the circuit in Figure 4 is called a current divider.

Knowing that conductance  $G = 1/R$ , in general, if a current divider has  $N$  conductors ( $G_1, G_2, \dots, G_N$ ) in parallel with the source current  $i$ , the  $n$ th conductor ( $G_n$ ) will have a current of:

$$i_n = \frac{G_n}{G_1 + G_2 + \dots + G_N} i$$

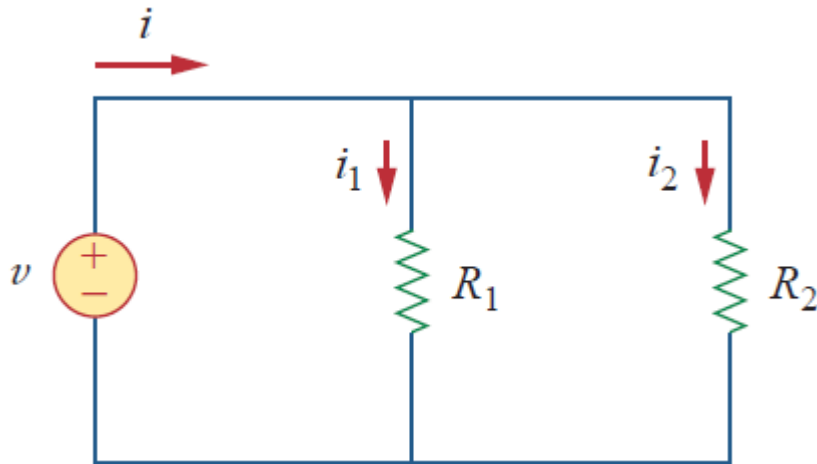


Figure 4: A two-loop circuit with two resistors in parallel

## V. Development

### 5.1 Voltage Divider

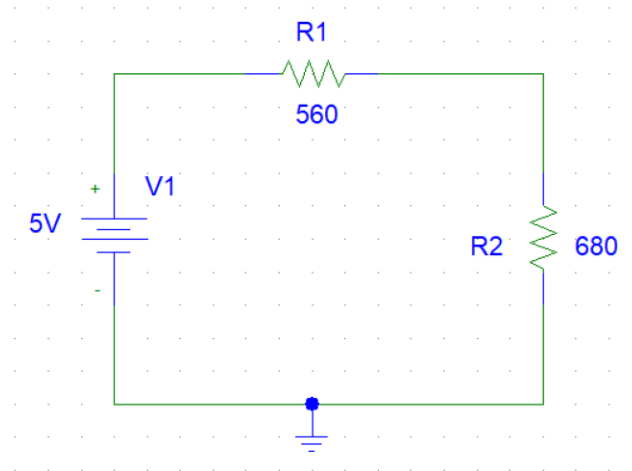


Figure 1.1.- Voltage Divider circuit in pSpice

1. We are going to simulate the circuit in Figure 1.1. On the search box at the taskbar, type in “Schematics” and hit “Enter” to open the Schematics program. The Schematics window, shown partially in Figure 1.2, should appear. Next, we get the necessary parts (dc voltage and current sources and resistors) by selecting Get New Part from the Draw menu. Figure 1.3 shows the Parts Browser window where VDC has been typed into the Part Name field. Selecting Place & Close returns us to the Schematics window with the cursor changed to a battery symbol. Left click once to place a voltage source on the page. Right click to return the cursor to its original shape. Your Schematics page should look something like Figure 1.4.

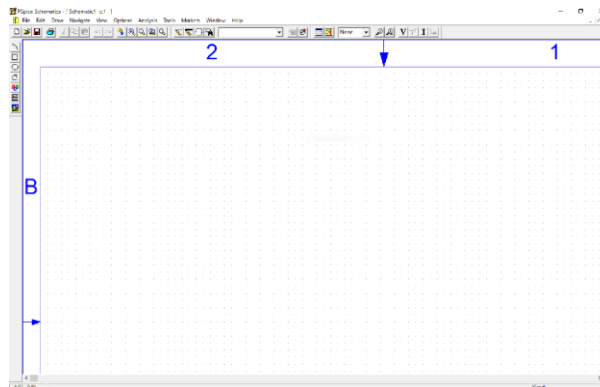


Figure 1.2.- Schematics window



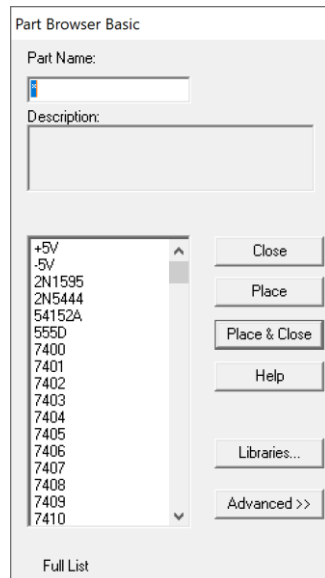


Figure 1.3.- Part Browser window

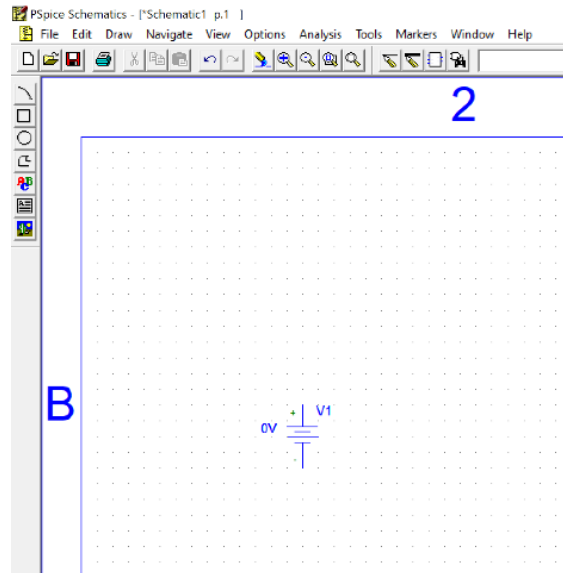


Figure 1.4.- The Schematics window showing the VDC part after placement.

2. Go ahead and open Draw menu, click on Get New Part, then select part name and finally click on Place & Close at the end of the insertion. As shown in the next Figure. Observe there is GROUND-Earth symbol.

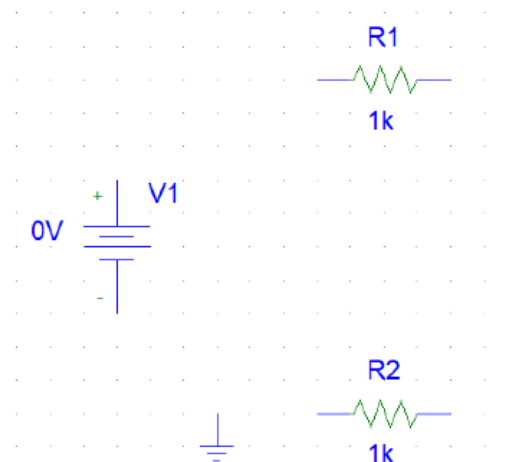


Figure 1.5.- Components placed at the Schematics window.

3. Put them together to look like Figure 1.1. Double click on the numerical values and labels to change their values or names. To add wiring, from the Menu select Draw, then Wire which changes the cursor to a pencil icon.
4. You must now Save your job, otherwise the simulation will not run. After you save it, run the program with Yellow icon or the Menu Analysis > Simulate. To display currents and voltages on the board, use Menu Analysis > Display Results on Schematic. You are done, current and voltages are displayed.

- It is interesting to look at Menu Analysis > Examine Netlist, a text file is opened showing the components' roster, together with their connections. Always generate this file to understand better what pSpice is doing.

## 5.2 Current Divider

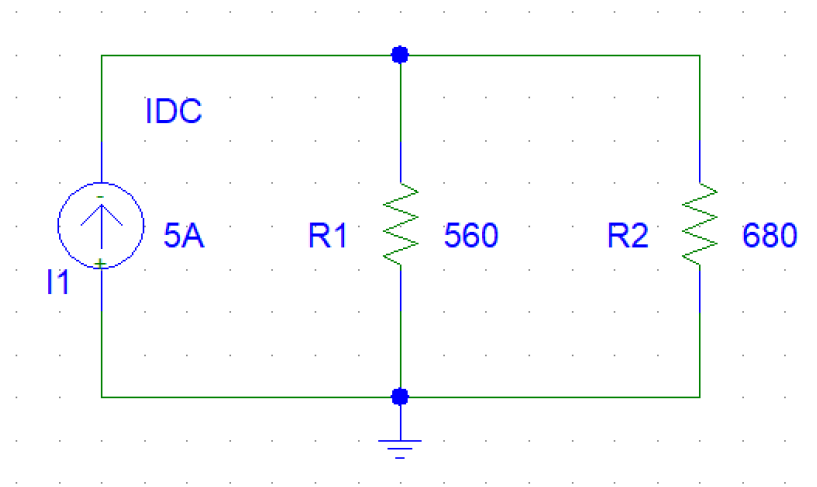


Figure 1.6.- Current Divider circuit at Schematics

- Open a new Schematic with the White page icon at the top left corner. Assemble the circuit of Figure 1.6. Complete all necessary steps until obtaining the displayed currents and voltage. To the left of File on the Main menu there is a yellow Schematics Icon, opened it to toggle between Schematics.
- To verify these results, first calculate the equivalent resistance of  $R_1 || R_2 = \underline{307 \text{ Ohms}}$
- Next use Ohm's law to calculate  $V = IR = (5 \text{ A})(R_1 || R_2) = \underline{1535 \text{ V}}$

$$R_{eq} = \frac{560(680)}{1,240}$$

$$V = (5)(307) = 1,535 \text{ V}$$

### 5.3 pSpice Simulation of AC Networks (DC)

Alternating current networks use sinusoidal voltage sources, such as the one depicted below, Figure 1.7.

1. Insert new parts: VSIN and the other parts familiar to you.

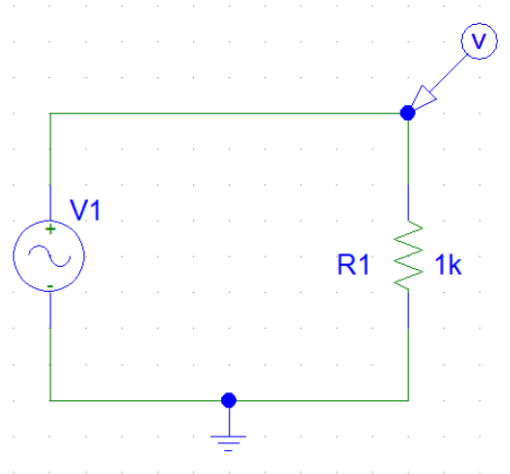


Figure 1.7.- AC circuit at the Schematics

2. Double click on VSIN symbol are fill the drop Menu with the source's parameters, (Fig 1.8).

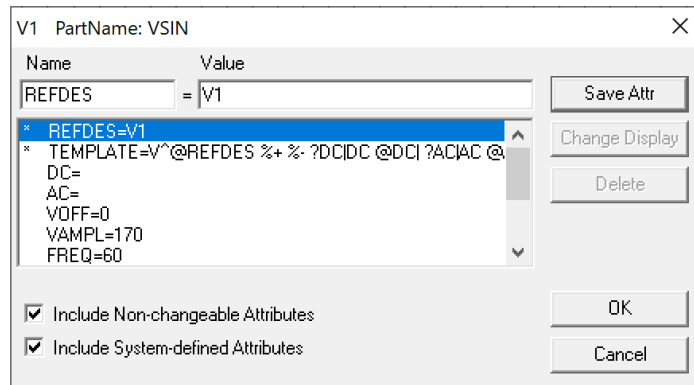


Figure 1.8.- VSIN Properties menu

3. Open Analysis > Setup > Transient and fill in the following parameters

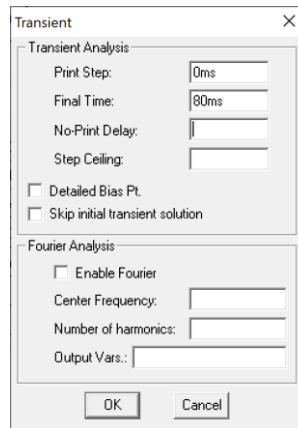


Figure 1.9.- Transient Analysis menu

4. Insert an icon Voltage Level Marker to plot the voltage as showed in Figure 1.7.
5. Click on Simulate and print the sinusoidal voltage. Right Click on the curve: Properties and increase its width and change its color to red.
6. Bring out the cursor and measure the period of the sinusoid.  
 $T = 16.8187 \text{ ms}$  and the frequency  $f = 1/T = 59.81 \text{ Hz}$

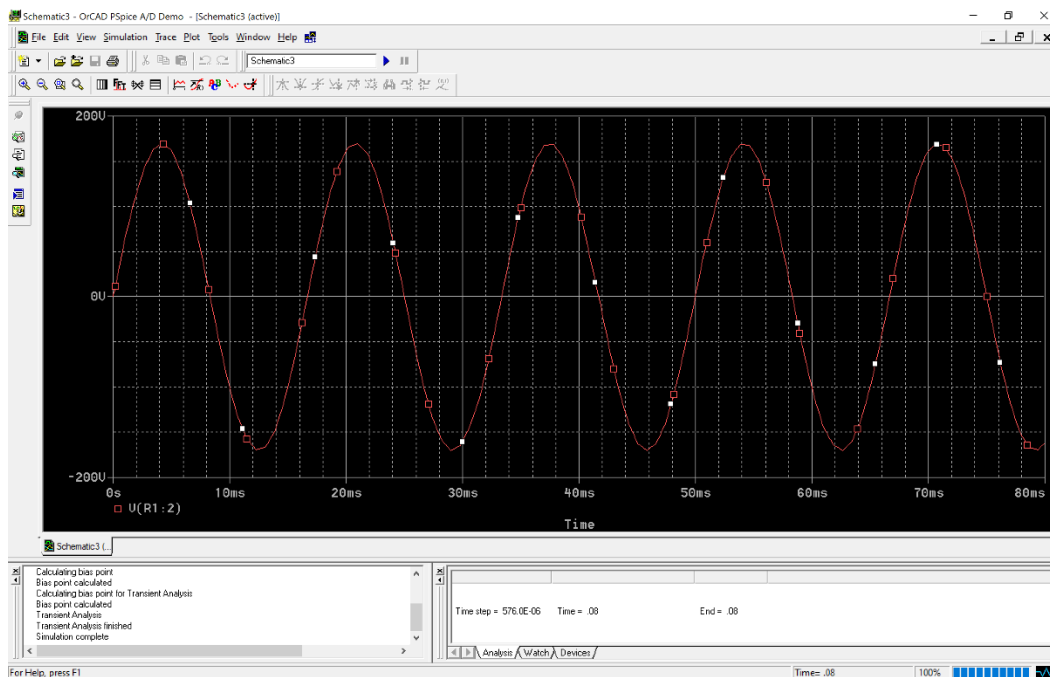


Figure 1.10.- Sinusoidal voltage of AC circuit

### 5.4 Frequency Response with pSpice (Remove)

Use the following components: L, R, C, VAC, Voltage Marker and GND-EARTH to construct the circuit below with the values shown.

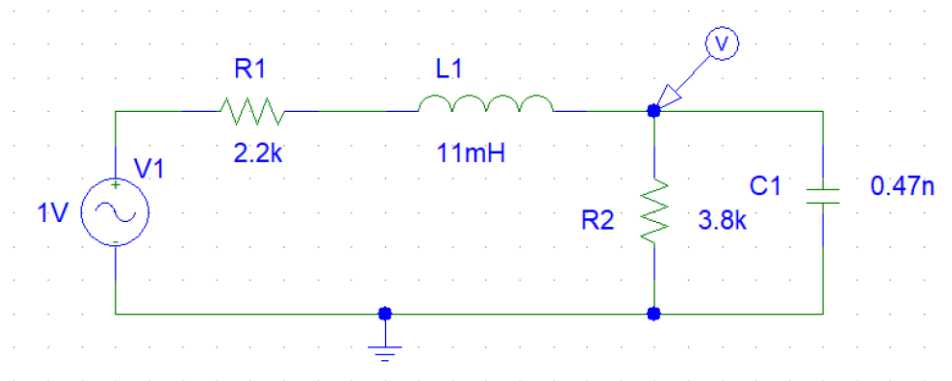


Figure 1.11.-AC circuit for frequency response analysis

To assign values to the AC Source, double click on VAC schematic symbol and the window below will pop up. Uncheck both attribute boxes at the bottom and assign the values shown below. Click Save and OK.

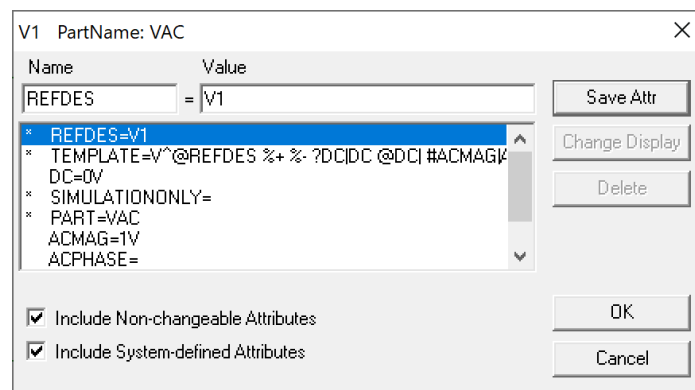


Figure 1.12.- VAC Source properties.

From the Analysis menu click on Setup and the window below will appear. Check the box next to AC Sweep and click on “AC Sweep” box

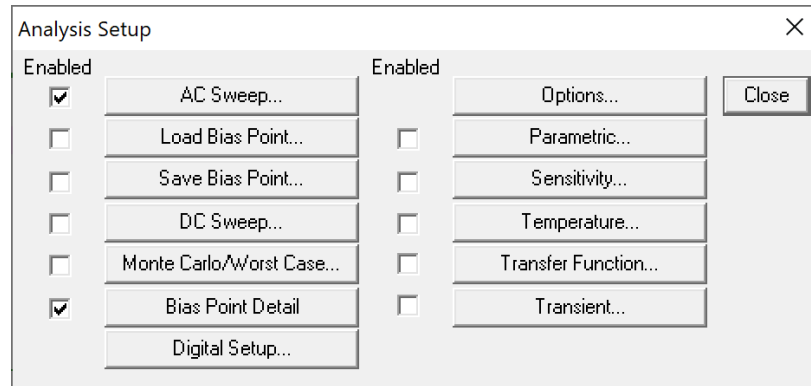


Figure 1.13.- Analysis Setup menu

The window below will appear, populate with the values shown below and check the Decade radial.

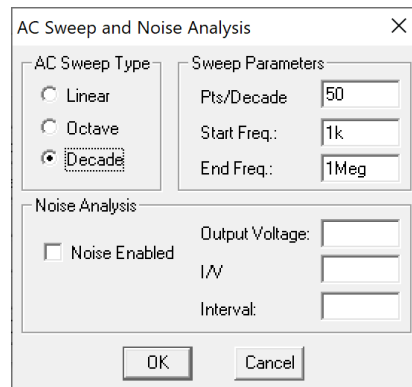


Figure 1.14.- AC Sweep menu

Save the file and simulate by clicking on Analysis > Simulate; or by hitting the F11 Key. After a few seconds, the AC Sweep icon should appear in your Windows Taskbar. Your results should look similar to the screen capture below.

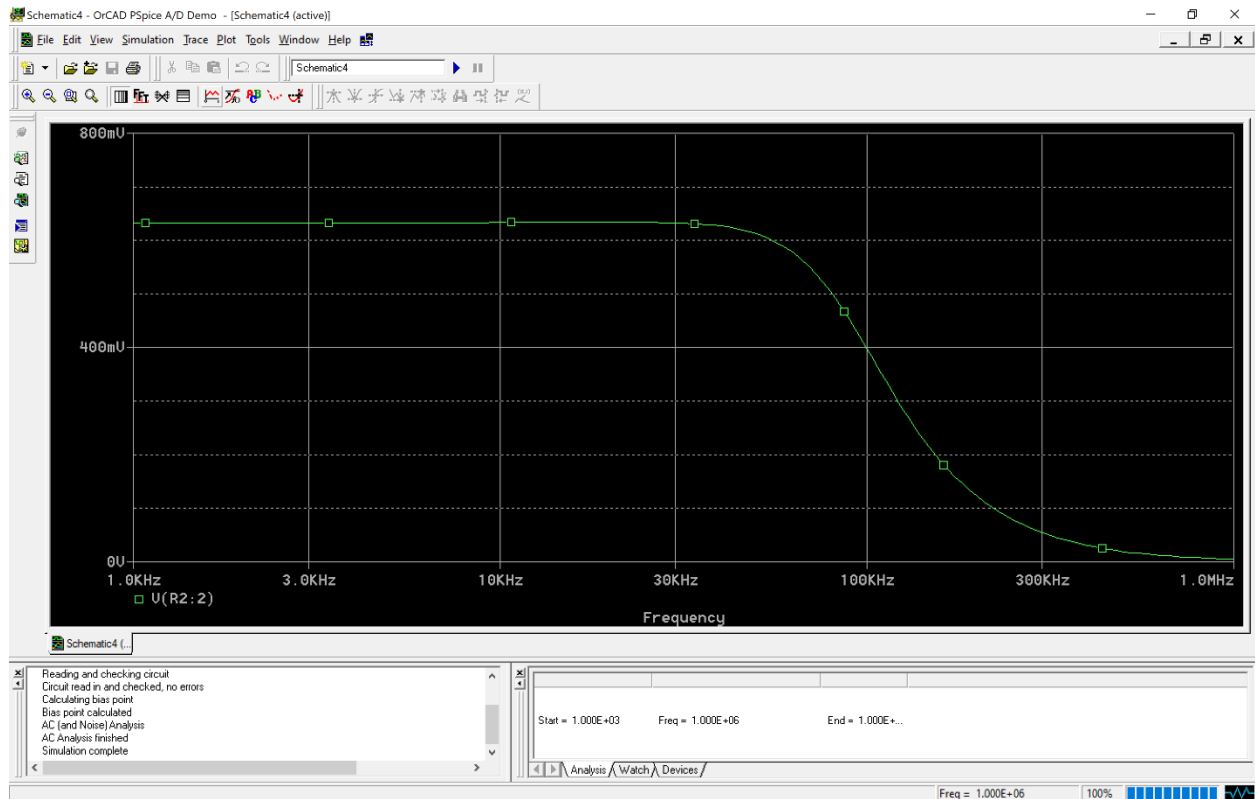
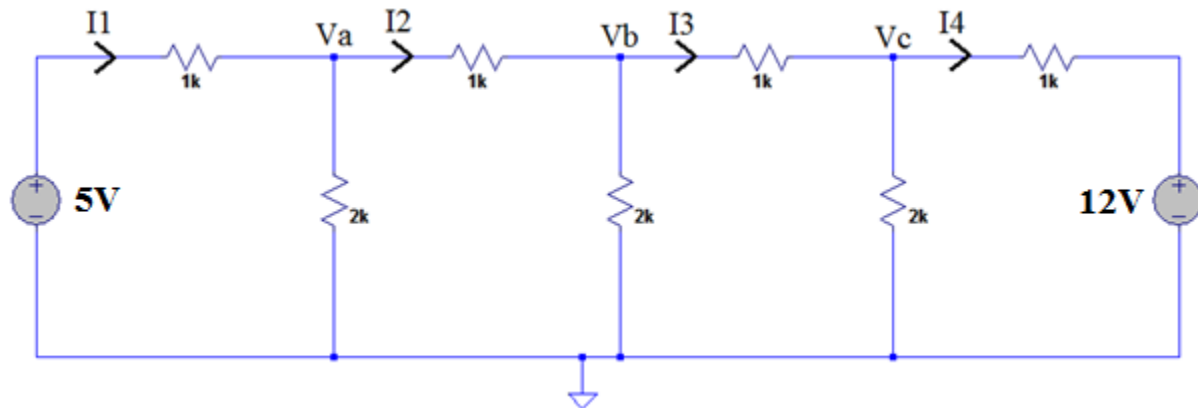


Figure 1.15.- Frequency domain analysis.

Using the cursor, add a marker to locate the cutoff frequency (frequency at which output gain is 70.7 % of the DC value). Write  $f_0 = 90.314$  kHz

## V. MATLAB: Solution of Simultaneous Equations

Use PSpice to draw and simulate the circuit shown in the following picture.



After running the simulation, take note of the simulated nodal voltages and branch currents below.

$$V_a (\text{simulated}) = \underline{3.6 \text{ V}}$$

$$I_1 (\text{simulated}) = \underline{1.4 \text{ mA}}$$

$$V_b (\text{simulated}) = \underline{4 \text{ V}}$$

$$I_2 (\text{simulated}) = \underline{-0.4 \text{ mA}}$$

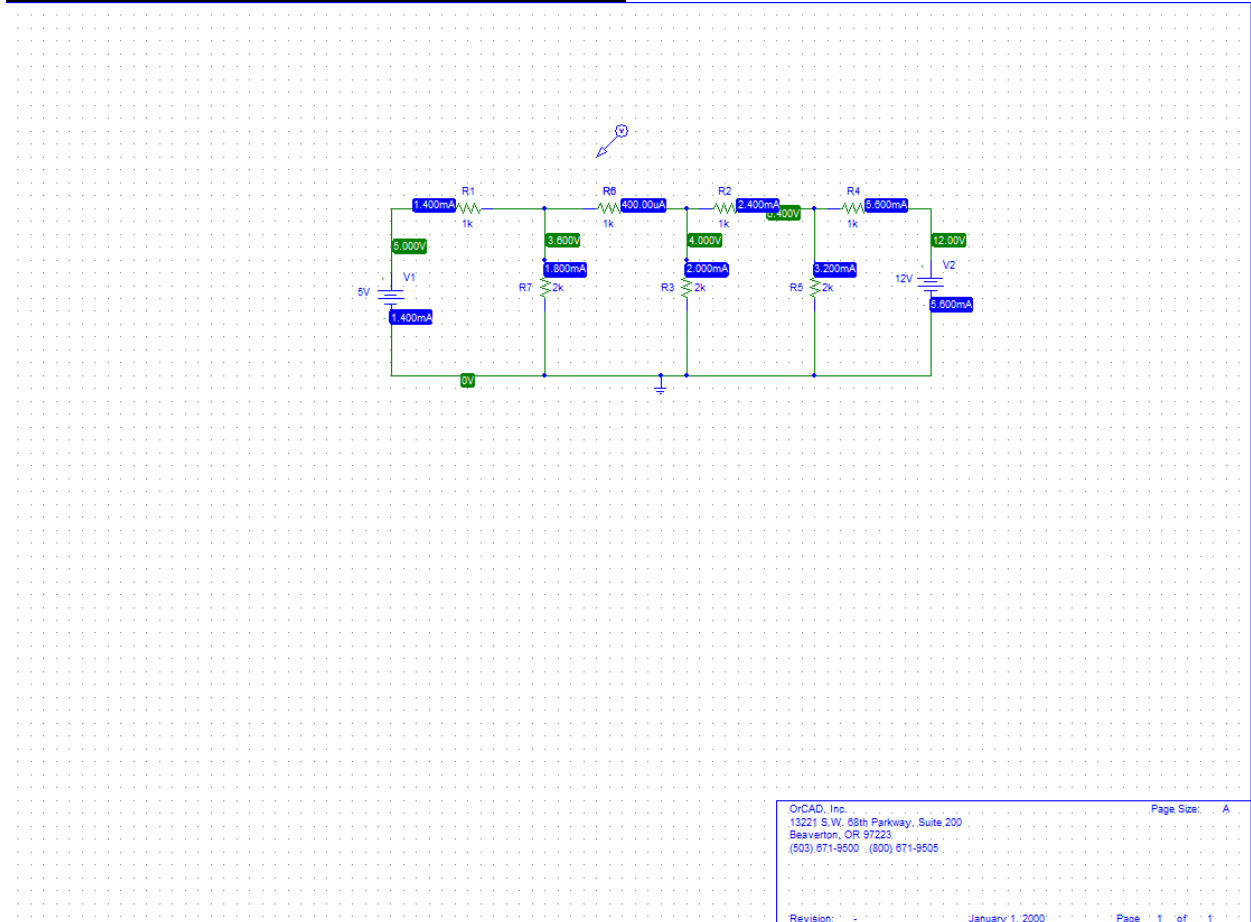
$$V_c (\text{simulated}) = \underline{6.4 \text{ V}}$$

$$I_3 (\text{simulated}) = \underline{-2.4 \text{ mA}}$$

$$I_4 (\text{simulated}) = \underline{-5.6 \text{ mA}}$$



**Take a screenshot of your PSpice circuit schematic, showing voltage and current values, make a printout, and attach it to your report.**



- (a) For larger circuits as the one from the previous picture, it is often necessary to apply more advanced methods of analysis to extract voltage and current values in the network. There are two important methods of analysis which you will cover in more detail at a later lecture/lab session of the course: **Nodal Analysis** and **Mesh Analysis**.

- Nodal Analysis is a systematic application of KCL, and helps find nodal voltages
- Mesh Analysis is a systematic application of KVL, and helps find mesh currents

By applying Nodal Analysis to the electric circuit in Figure 7, one ends up with the system of nodal equations:

$$\begin{aligned} 5V_a - 2V_b &= 10 \\ 2V_a - 5V_b + 2V_c &= 0 \\ 2V_b - 5V_c &= -24 \end{aligned}$$


There are three unknown nodes in the circuit, and therefore three equations with three unknowns are obtained, where the unknowns are the voltages  $V_a$ ,  $V_b$ , and  $V_c$ .

Write the **system of nodal equations** in matrix form below:

$$\begin{bmatrix} 5 & -2 & 0 \\ 2 & -5 & 2 \\ 0 & 2 & -5 \end{bmatrix} \begin{bmatrix} 3.6 \\ 4 \\ 6.4 \end{bmatrix} = \begin{bmatrix} 10 \\ 0 \\ -24 \end{bmatrix}$$

$\mathbf{Z} \qquad \mathbf{V}$

- (b) Now use MATLAB to calculate the unknown voltages from the system of equations derived from the circuit. On your desktop computer, open the MATLAB software. This is

MATLAB's icon:  . It may take some time for the software to load.

- (c) Upon opening MATLAB, you will be greeted by the Command Window, on which you will be working for this part of the laboratory. Your job now is to input the system of equations in matrix form (which you entered above) into the Command Window, and then have MATLAB calculate the results for you. As shown in the matrix, the “z” area covering the coefficients on the left side is defined as the *data matrix*, and the “v” area covering the coefficients on the right side is defined as the *source term vector*. You will need to input a symmetrical matrix “z” and a vertical vector “v” holding their respective coefficients in MATLAB.

Before entering the matrix into MATLAB, type the following in the Command Window:

*format long*

This will allow you to retrieve solutions with enough decimal places.

To input the data matrix “z”

$$z = \begin{bmatrix} z_{11} & z_{12} & z_{13} \\ z_{21} & z_{22} & z_{23} \\ z_{31} & z_{32} & z_{33} \end{bmatrix}$$

where  $z_{11}$ ,  $z_{12}$ , ...,  $z_{33}$  are your coefficient values, type in the Command Window:

$$z = [z_{11}, z_{12}, z_{13}; z_{21}, z_{22}, z_{23}; z_{31}, z_{32}, z_{33}]$$

To input the source term vector “v”

$$v = \begin{bmatrix} v_{11} \\ v_{21} \\ v_{31} \end{bmatrix}$$

where  $v_{11}, v_{21}, \dots, v_{41}$  are your coefficient values, type in the Command Window:

$$v = [v_{11}; v_{21}; v_{31}]$$

As you might notice, in MATLAB the column entries are separated by commas, and rows are separated by semicolons.

To find the solution, type in the Command Window:

$$I = \text{inv}(z)*v$$

If everything was typed correctly, you should get the solution for all the unknown voltages in a vertical vector. The first element of the vector is  $V_a$ , and the last is  $V_c$ .

- (d) On MATLAB, perform exactly the same steps to calculate currents  $I_1, I_2, I_3$ , and  $I_4$  by using the system of mesh equations derived through Mesh Analysis:

$$\begin{aligned} 3kI_1 - 2kI_2 &= 5 \\ -2kI_1 + 5kI_2 - 2kI_3 &= 0 \\ -2kI_2 + 5kI_3 - 2kI_4 &= 0 \\ -2kI_3 + 3kI_4 &= -12 \end{aligned}$$

where  $k = \times 10^3$ . Write the **system of mesh equations** in matrix form below:

$$\underbrace{\begin{bmatrix} 3k & -2k & 0 & 0 \\ -2k & 5k & -2k & 0 \\ 0 & -2k & 5k & -2k \\ 0 & 0 & -3k & 3k \end{bmatrix}}_{\mathbf{Z}} \underbrace{\begin{bmatrix} 1.4 \\ -0.4 \\ -2.4 \\ -5.6 \end{bmatrix}}_{\mathbf{V}} = \begin{bmatrix} 5 \\ 0 \\ 0 \\ -12 \end{bmatrix}$$

There are four unknown mesh/loop currents in the circuit, and therefore four equations with four unknowns are obtained, where the unknowns are the currents  $I_1$ ,  $I_2$ ,  $I_3$ , and  $I_4$ .

Insert the electrical parameter values obtained through MATLAB below.

$$V_a (\text{MATLAB}) = \underline{3.6 \text{ V}}$$

$$I_1 (\text{MATLAB}) = \underline{1.4 \text{ mA}}$$

$$V_b (\text{MATLAB}) = \underline{4 \text{ V}}$$

$$I_2 (\text{MATLAB}) = \underline{-0.4 \text{ mA}}$$

$$V_c (\text{MATLAB}) = \underline{6.4 \text{ V}}$$

$$I_3 (\text{MATLAB}) = \underline{-2.4 \text{ mA}}$$

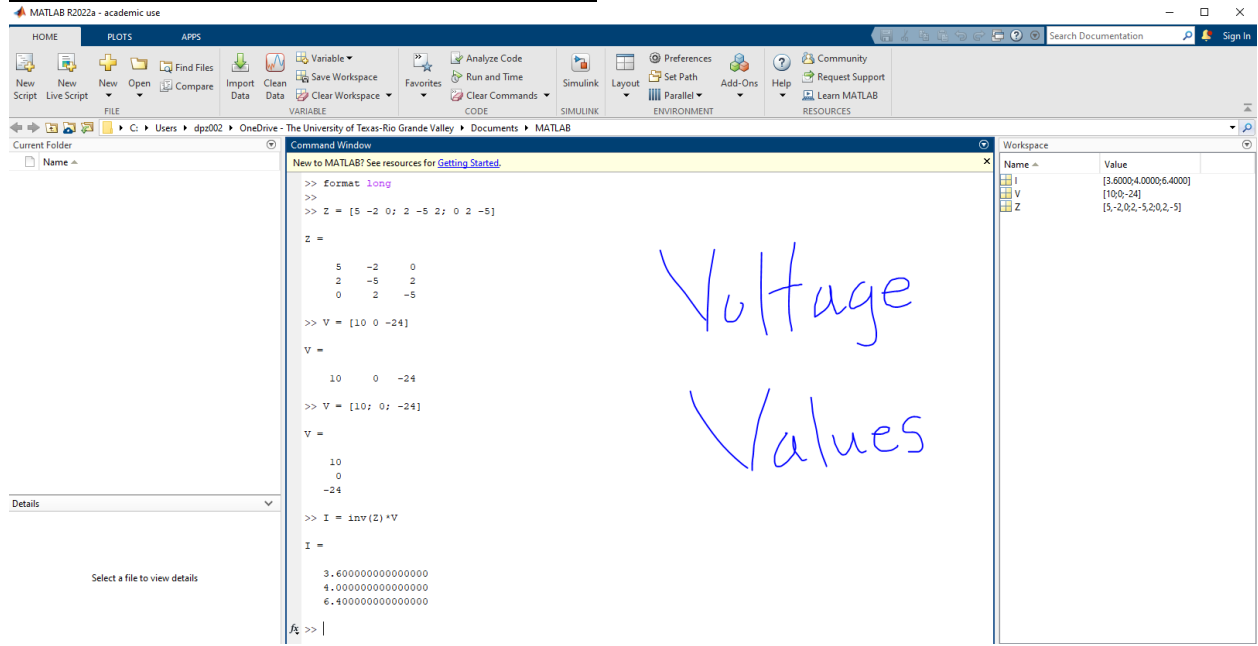
$$I_4 (\text{MATLAB}) = \underline{-5.6 \text{ mA}}$$

(e) Do your MATLAB results agree with the ones obtained through simulation in step (b)?

**YES**

**NO**

**Take a screenshot of MATLAB's Command Window, showing your lines of code and output results, and attach it to your report.**



```

>> Z = [3000 -2000 0 0; -2000 5000 -2000 0; 0 -2000 5000 -2000; 0 0 -2000 3000;]

Z =

    3000    -2000         0         0
   -2000     5000    -2000         0
         0    -2000     5000   -2000
         0         0    -2000     3000

>> V = [5; 0; 0; -12;]

V =

     5
     0
     0
    -12

>> I = inv(Z)*V

I =

    0.001400000000000
   -0.000400000000000
   -0.002400000000000
   -0.005600000000000

>>
  
```

## VI. After the Lab

- 1) Do you think PSpice is helpful for circuit analysis? Why?

I believe PSpice is helpful for circuit analysis as it allows you to simulate any circuit while also being able to view the data of that circuit. This saves time and allows for optimization of circuits all from simulations.

- 2) How can MATLAB help you for circuit analysis?

In this lab, we found MATLAB to be extremely useful for calculating matrix equations. These matrix equations are fundamental when working with circuits, making MATLAB a very useful tool for analyzing circuits.

## VII. Conclusions

Do you have any conclusions? How can you use this software to your advantage during this course? Be open minded!

The two different software used in this lab, PSpice and MATLAB, will be used in the rest of my academic and professional career. From creating circuits, to analyzing them, PSpice will be a helpful tool even as a study tool. MATLAB also has its usefulness in calculating equations that are time consuming, allowing you to focus on analyzing the circuit rather than these long basic equations.