

Fall 2018

California State University, Northridge

Department of Electrical & Computer Engineering

Experiments 4 & 5

4 – Computer Simulation

5 – Circuit I

7 October, 2018

ECE 240L

Professor: Franco Mikhailidis

Group 2: Jonathan Roa

Ridge Tejuco

EXPERIMENT 4 & 5

Introduction:

The purpose of these experiments were to be introduced to the PSpice software that was needed not only to build and simulate circuits, but also to confirm data that had been calculated on paper beforehand. In order to reach this goal, simple circuits were constructed using the software with the intention that analyzing these circuits within PSpice would encourage exploration of the functions and commands that are used most often.

Equipment Used:

Type	Model	Serial No.	Calibration Date
Function Generator	Agilent 33220A	MY44017172	N/A
Oscilloscope	Tektronix 2213A	LR37158	N/A
Digital Multimeter	Tektronix CDM250	CDM-250TW52380	N/A

Parts Used (BB portion of experiment 5):

Quantity	Component	Value	Type
1	Resistor	$20 \times 10^2 \Omega \pm 5\%$	Carbon
1	Resistor	$60 \times 10^2 \Omega \pm 5\%$	Carbon
2	Resistor	$30 \times 10^2 \Omega \pm 5\%$	Carbon

Software Used:

- PSPICE
- Microsoft Paint

Theory (Ignore if already familiar with PSPICE Software):

PSpice Basics: Starting a New PSPICE File

1. On the computer's Desktop screen, click on Start, move the cursor to **Programs > Orcad 16.2 > Capture CIS**. Depending on the directory names that you used during the installation these titles may read differently. No license will be found if you are running the demo version. Indicate that you want to start using the demo license.
2. The Orcad Capture window should open. Click on **File > New > Project**.
3. The New Project dialog box should open.

4. Type the name of the circuit in the **"Name"** box, and indicate the path to the directory in which the file is stored under **"Location"**.
5. Click on the radio button next to: **Analog or Mixed A/D**, then click **OK**.
6. The Create PSPICE Project dialog box will appear. Select "Create a blank project", and click OK.
7. A schematic entry window will appear like the one shown below in figure 1 (differences in appearances vary from version to version).

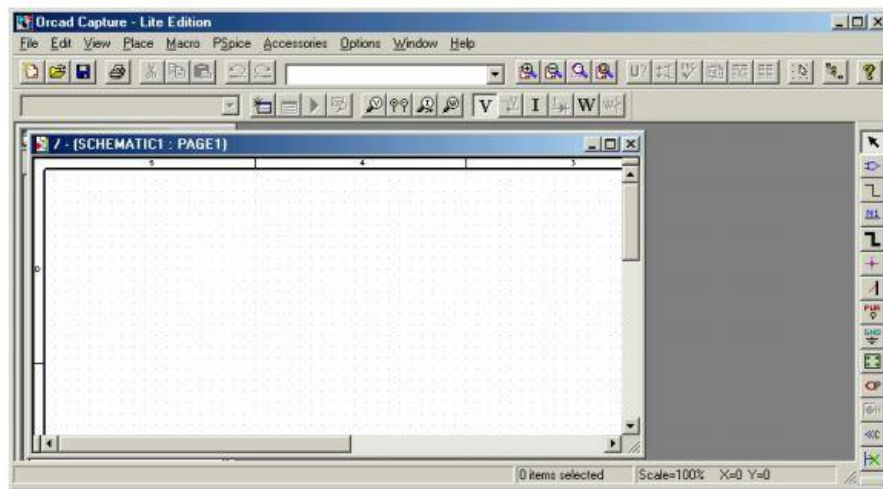


Figure 1

PSPICE Basics: Placing Parts

Select **Place > Part**. The following window shown below in figure 1.2 will appear.

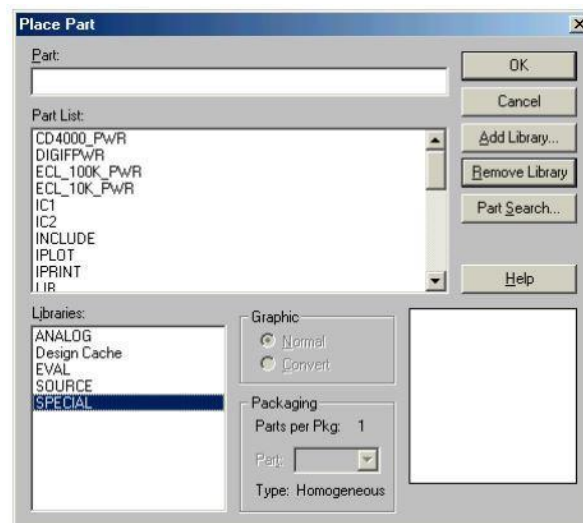


Figure 1.2

At minimum, the following libraries should be installed: analog, eval, source, and special. To add a Library, click on the **Add Library** icon and add each of the libraries mentioned above. From this window, circuit parts can be placed and wired together in the schematic window shown above in figure 1. But note that **every circuit simulated in PSPICE must have a ground node indicated or else the simulation will not run.**

PSPICE Basics: Specifying the Type of Analysis

1. Choose the menu option: **PSPICE > New Simulation Profile**. A dialog box will open.
2. Type in the name of the simulation and click **Create**.
3. The Simulations Settings dialog box opens. Under Analysis Type, select **Bias Point** and click **OK**.
4. Save the circuit.

PSPICE Basics: Running the Simulation

1. Select the menu option: **PSPICE > Run**. The simulation will run and the simulation window will open as shown below in figure 1.3.

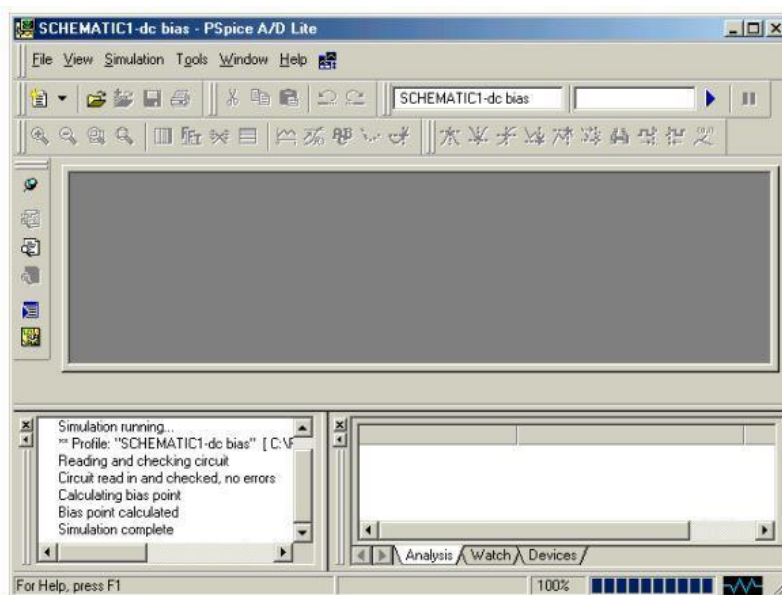


Figure 1.3

2. In the lower left corner of the window is an output text section that displays the progress of the simulation and any errors that were encountered.

3. Simulation results are viewed as a SPICE output file. Select the menu option: **View > Output File** to see results of simulation.

Procedure and Results:

In the first “demonstration” circuit, a $3.3\text{k}\Omega$ resistor and a $6.8\text{k}\Omega$ resistor were placed in series to a 9V power source and PSpice was used to simulate the circuit and find what the voltages were at every node of the circuit, the voltage across the resistors, as well as the current traveling throughout. The results can be observed in Figure 4.0.

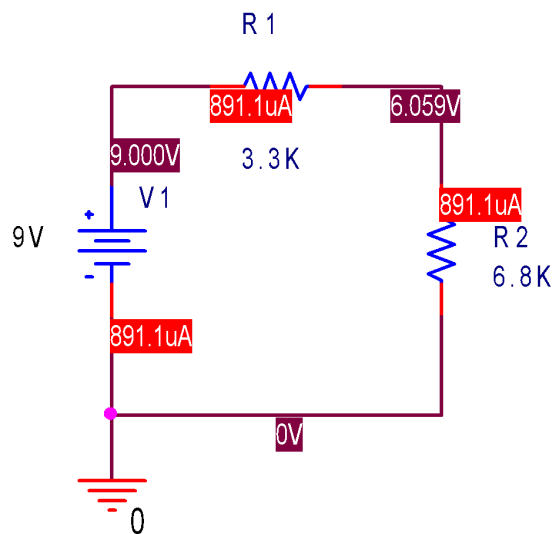


Figure 4.0

The results of calculating the voltages across the resistors by hand can be seen in Table 4.0

Table 4.0 Voltage Across Resistors and Total Current

I_{total}	V_1	V_2
0.8910 mA	2.9405 V	6.0588 V

The hand calculated values are a very close approximation to what is shown in the simulation, so it was assumed the circuit was correctly constructed in PSpice. Next, the first step in the procedure of the Experiment 4 lab called for the construction of the circuit shown in Figure 4.1.

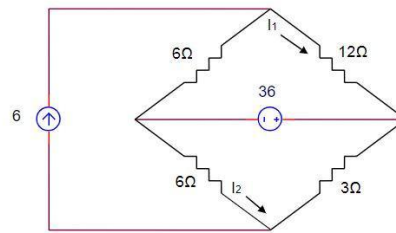


Figure 4.1

Figure 4.1

This circuit was to be analyzed using PSpice to calculate the node voltages as well as currents I_1 and I_2 . The results of the simulation can be seen in figure 4.2

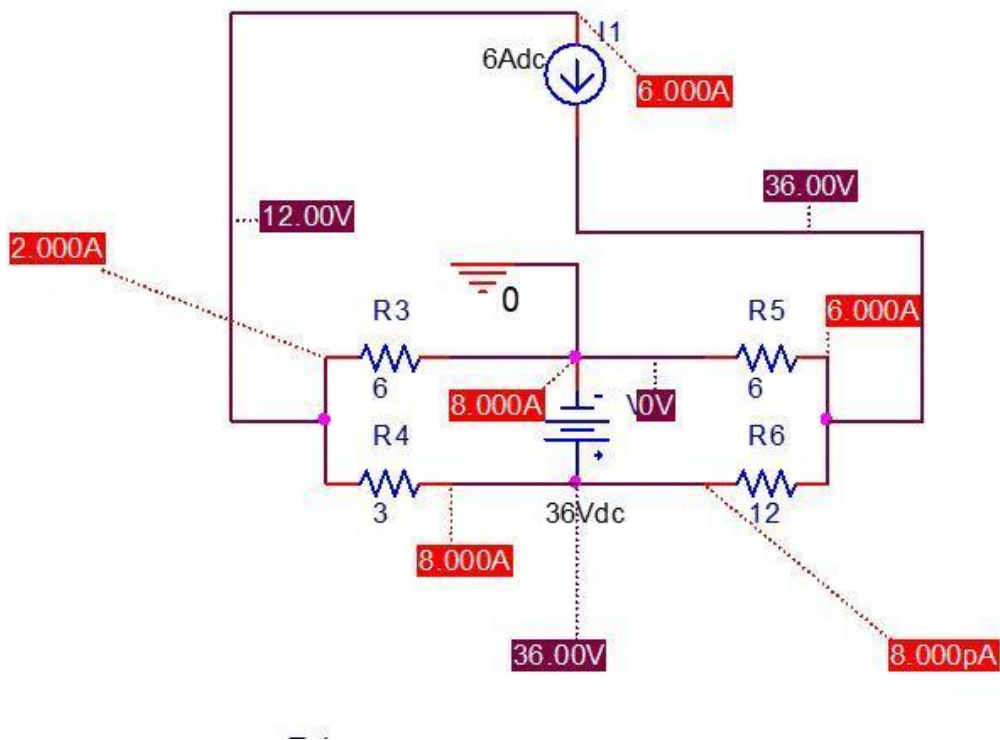


Figure 4.2

Letting V_1 be the node that the 6A current travels into first, V_2 the node connected to the positive terminal of the 36 V power supply, V_3 the node connected to the negative terminal of the power supply, and V_4 the node that I_2 travels into, and ground point being connected to the negative terminal of the power supply voltage, the results of the simulation are plotted into Table 4.2

Table 4.2 Node Voltages & Currents

Voltages	Currents
----------	----------

$V_1 = 36 \text{ V}$	$I_1 = 8.000 \text{ pA}$
$V_2 = 36 \text{ V}$	$I_2 = 8 \text{ A}$
$V_3 = 0 \text{ V}$	
$V_4 = 0 \text{ V}$	

It can be seen that there is almost no current flowing through the 12Ω resistor, which is reasonable considering that there is no difference of voltage between V_1 and V_2 . All of these results were confirmed by hand calculation except for I_2 which can more than likely just be attributed to human error considering that everything else was confirmed. The next part of the experiment called for Transient Analysis of the circuit shown in Figure 4.2.

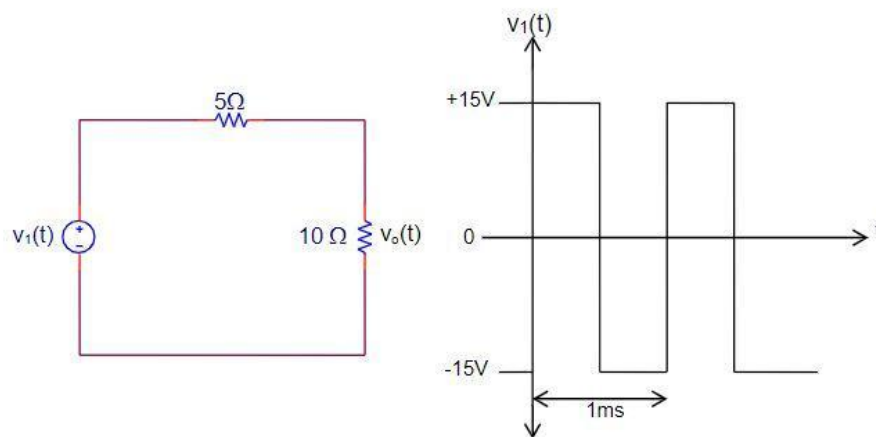
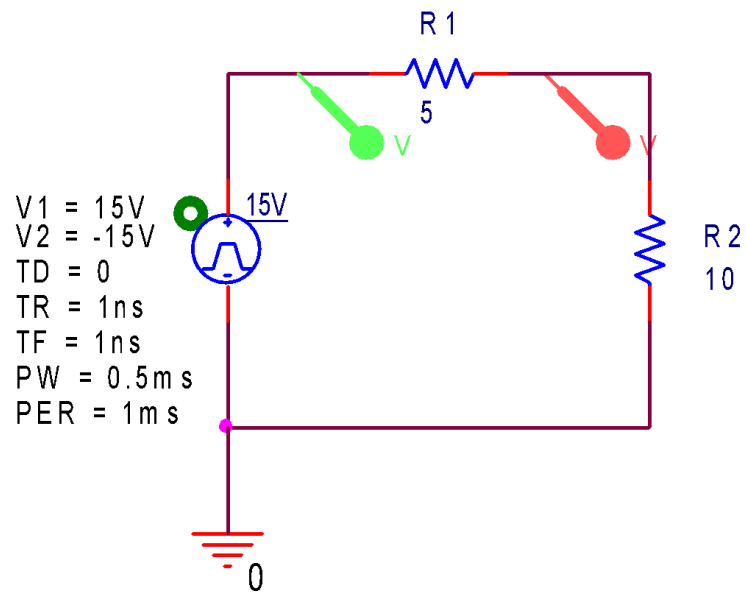


Figure 4.3

It was assumed that $V_1(t)$ was a 1KHz square wave (15 V peak-to-peak) as shown. The objective was to construct PSpice program to plot the first 2 cycles of $V_o(t)$. The circuit constructed in PSpice for the simulation can be seen in figure 4.32.



The result of reaching this objective is shown in the PSpice simulation Figure 4.4, where the green square wave corresponds to the voltage measured across the first resistor, and the red square wave corresponds to the voltage measured across the second resistor.

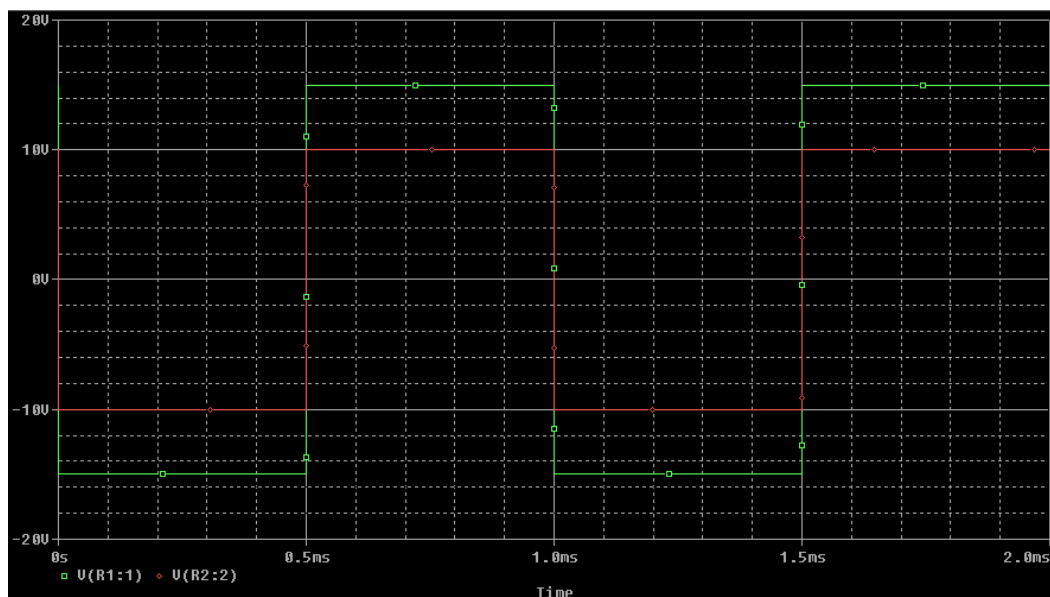


Figure 4.4

The procedure was repeated for the 10th and 11th cycles as well, which can be shown in figure 4.5. It can be seen that there is little to no discrepancy between the first two cycles and the 10th and 11th cycles.

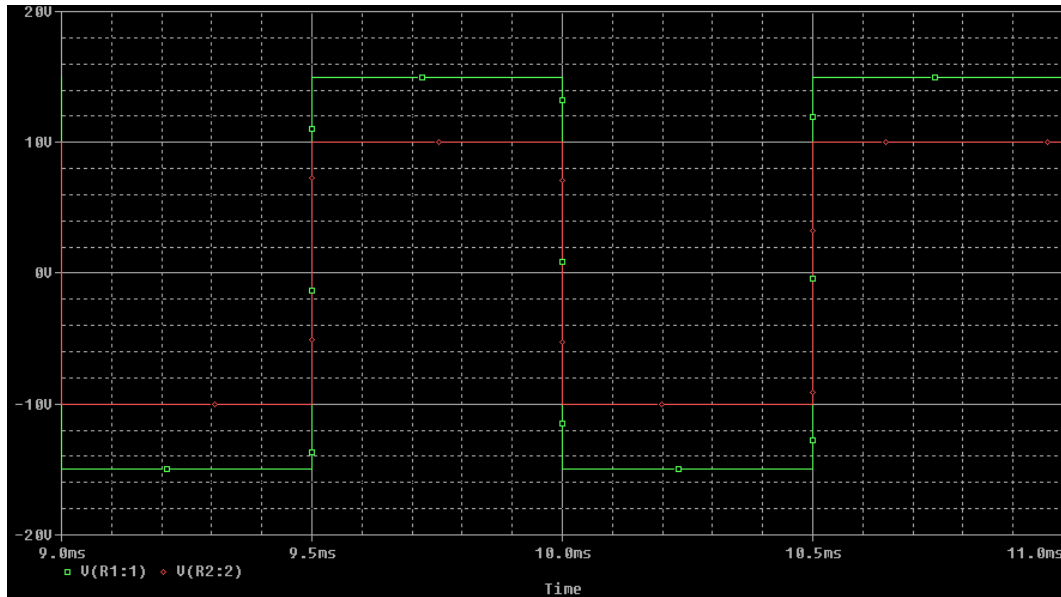


Figure 4.5

Additionally, this part of the experiment was repeated when $V_1(t)$ was a sine wave of peak voltage 15 V and a frequency of 1kHz. The results of constructing the circuits with the sine wave instead of the square wave are shown in figures 4.6 and 4.7. The circuit constructed in PSpice to be simulated is shown in figure 4.52 below:

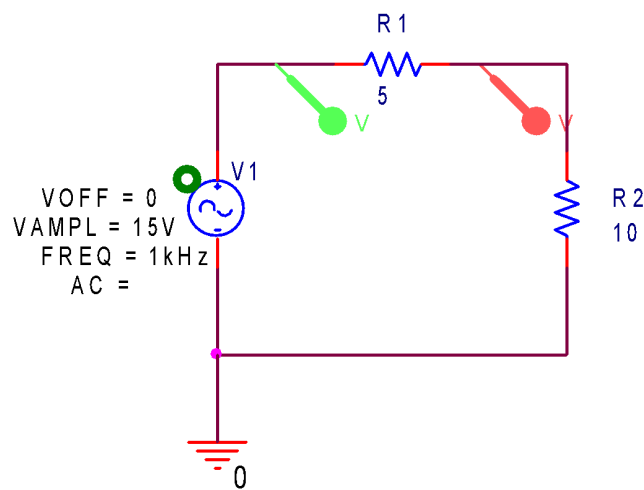


Figure 4.52

Similar to the previous iteration of this exercise, the green sine wave corresponds to the voltage measured across the first resistor, and the red sine wave corresponds to the voltage measured across the second resistor.

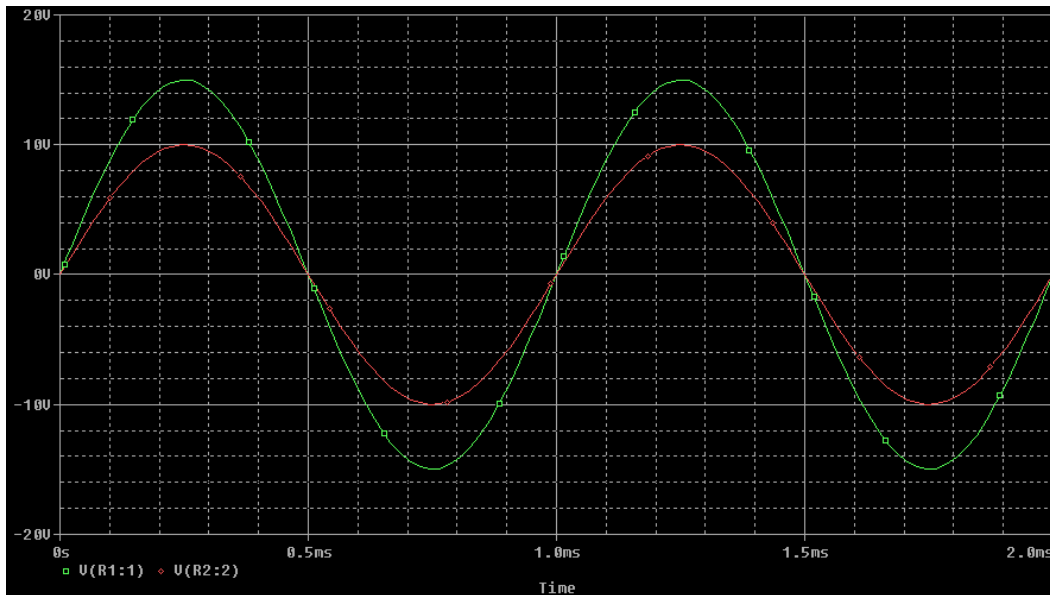


Figure 4.6 (First Two Cycles)

Just like the procedure with the square wave, there is little to no discrepancy between the first two cycles and the 10th and 11th cycles (observed in figure 4.7 below).

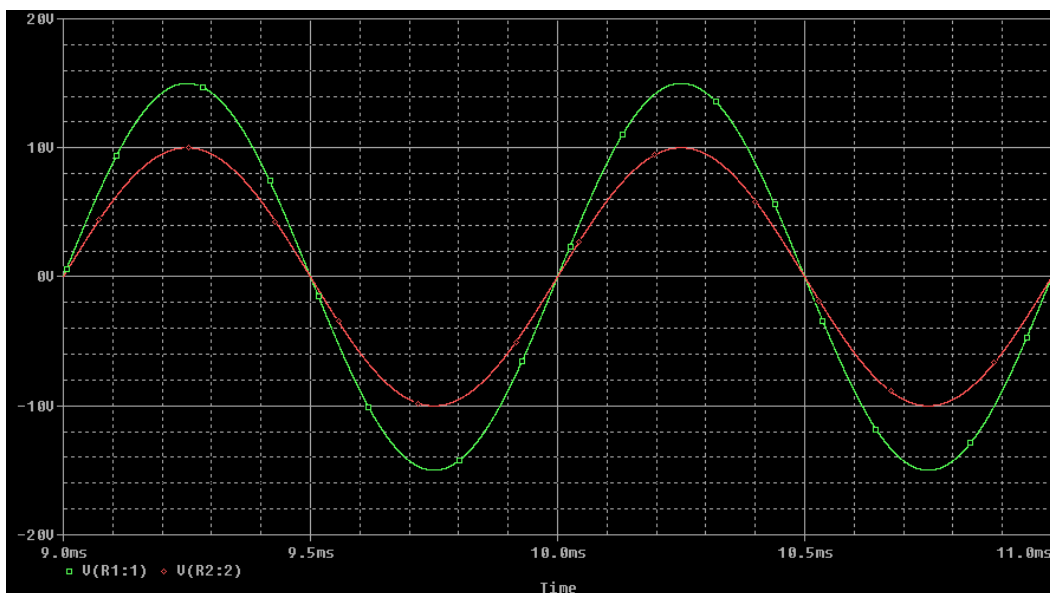


Figure 4.7

Moving forward to Experiment 5, Circuit I called for the physical construction of the circuit shown in the figure 5.1 schematic below

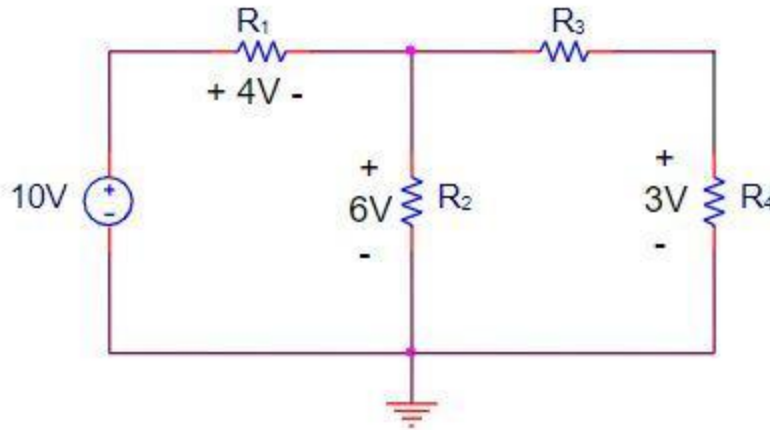


Figure 5.1

In this circuit, resistors R_3 and R_4 are placed in series. This series of resistors is in parallel with resistor R_2 , which is in series with R_1 . For this exercise, resistor values were chosen in such a way that $V_{R1} = 4\text{ V}$, $V_{R2} = 6\text{ V}$, and $V_{R3} = V_{R4} = 3\text{ V}$. These resistor values were calculated by hand and the circuit was simulated in PSpice. After verifying the hand calculations were correct using PSpice, the circuit was physically constructed using the breadboard and voltage values across each resistor were measured. These results of these calculations and measurements - as well as their percent errors, are shown in table 5.12

Table 5.12 Voltage Values & Percent Error

	Calculation	PSpice	Measured	% Error
V_1	4 V	4 V	3.93 V	1.7%
V_2	6 V	6 V	6.14 V	2.3%
V_3	3 V	3 V	3.08 V	2.6%
V_4	3 V	3 V	3.09 V	3%

Since the measured values are very closely approximated to the calculated voltage values and PSpice values, it was assumed that the measured values were void of error. Their differences to the pre-calculated values were attributed to the tolerances and variations in true values of the lab equipment and parts used.

Conclusion:

The objective of the lab was to practice using the PSPICE Software and become accustomed to the software controls as well as how to interpret data taken from them. Proper application of the PSPICE software is not only critical to future circuit analysis, but the software itself is also an indispensable tool as a circuit being analyzed increases in complexity. Looking at the results obtained from using the software, the objective of becoming familiar with the software (at least on a fundamental level) was met as the procedures were correctly executed.