R·I·T
Kate Gleason
College of ENGINEERING

EEEE 281: Experiment 1
Voltage Division

From: Andrei Tumbar [Computer Engineering]
To: Section 2 TA: LJ Boone, Harrison Keats
Date: Performed: 02/04/2020 Due: 02/11/2020
Subject: Lab 1
Lab Partner(s): N/A

Component	Percentage of Grade	Score	Comment
Report Formatting	20 %		
Hand Calculations: Circuit 1	10 %		
PSPICE Setup:	5 %		
PSPICE Simulation:	10 %		
PSPICE Discussion:	10 %		
Experimental Setup:	5 %		
Experimental Data:	20 %		
Experimental Discussion:	20 %		
Total Score:			
Graded By:			

Abstract

In the first part of this exercise, the software portion, a circuit with three resistors (two 5.6 kΩ, 1 kΩ) in series was designed in PSPICE, a circuit design, and simulation program. The resistors were listed with tolerances. A simulation was run to display a histogram of likely outcomes given the tolerance and resistance rating of each resistor. A Monte Carlo simulation displayed a statistical analysis of the likely outcomes of testing the circuit. In the hardware portion of the exercise, the circuit was implemented on a breadboard. The voltage was measured across each resistor given a supplied voltage of 5V. An effective resistance was measured along with resistance from each resistor. Finally, the voltage of each node relative to ground was measured and recorded. These experimentally determined values were compared to those calculated using the voltage division laws to analyze the circuit.

1 Introduction and Theory

The objective of this lab was to show how resistors in series will add overall resistance and show voltage division. Also, the Monte Carlo simulation showed how resistance tolerance can alter data.

1.1 Theory: Circuits Investigated

This circuit had a $5\,V$ power source followed by two $5.6\,k\Omega$ resistors followed by a $1\,k\Omega$ resistor. The node between the final $1\,k\Omega$ and the voltage source was grounded.

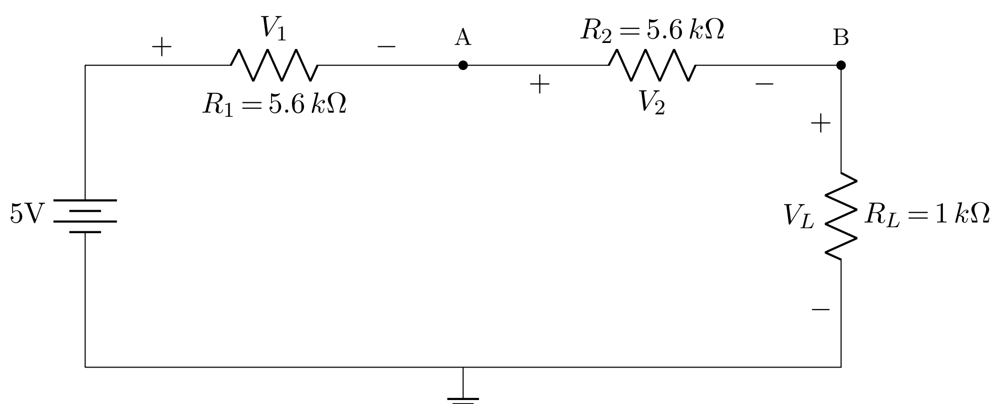

Figure 1.1.1: Circuit configuration given R_1, R_2, and R_3 with ground and voltage supply.

This is a simple series circuit with three resistors. The negative pole on the voltage supply is connected to ground. The voltage is set to $5\,V$. The first two resistors are both $5.6\,k\Omega$ resistors. The third resistor is a $1\,k\Omega$.

1.1.1 Theory: Kirchhoff's Voltage Law and Voltage Division

Kirchoff's Voltage law is the concept that the sum of the voltage drops in any loop equals zero. A loop is defined as a path where none of the nodes are visited more than once. The path must reach the original point. The idea of Voltage Divison states that in series, the voltage drop across each resistor is equal to the total voltage drop multiplied by the ratio of the current resistance to total resistance.

$$V_1 = 5\,V \frac{5.6\,k\Omega}{5.6\,k\Omega + 5.6\,k\Omega + 1\,k\Omega} = 2.295\,V \tag{1.1.1}$$

$$V_2 = 5\,V \frac{5.6\,k\Omega}{5.6\,k\Omega + 5.6\,k\Omega + 1\,k\Omega} = 2.295\,V \tag{1.1.2}$$

$$V_L = 5\,V \frac{1\,k\Omega}{5.6\,k\Omega + 5.6\,k\Omega + 1\,k\Omega} = 0.410\,V \tag{1.1.3}$$

V_A can be found by moving from the bottom node connected to ground to the node labeled A. There is a $5\,V$ gain from the voltage source, then there is a $2.295\,V$ drop across R_1. Therefore the voltage V_A is 2.705 relative to ground. The same method was used to find the voltage V_B which evaluated to be $0.410\,V$.

1.2 Theory: PSPICE Simulation Summary

The PSPICE simulation began with a design of the circuit. A voltage supply along with three resistors were placed. A 10% tolerance was added to each of the resistors. The resistance was set for each element. The nodes A and B were named as such. A ground was placed according to the circuit diagram in figure. The PSPICE setup used the libraries imported from the Circuits 1 library from the lab computer. The libraries were called **analog.olb**, **opamp.olb**, **source.olb**, and **special.olb**. The components used were “R/ANALOG” from the **analog.olb** for the resistors, “VDC/SOURCE” from the **source.olb** for the voltage source and “O/CAPSYM” from the **special.olb** for ground.

1.2.1 PSPICE: DC Simulation of Circuit 1

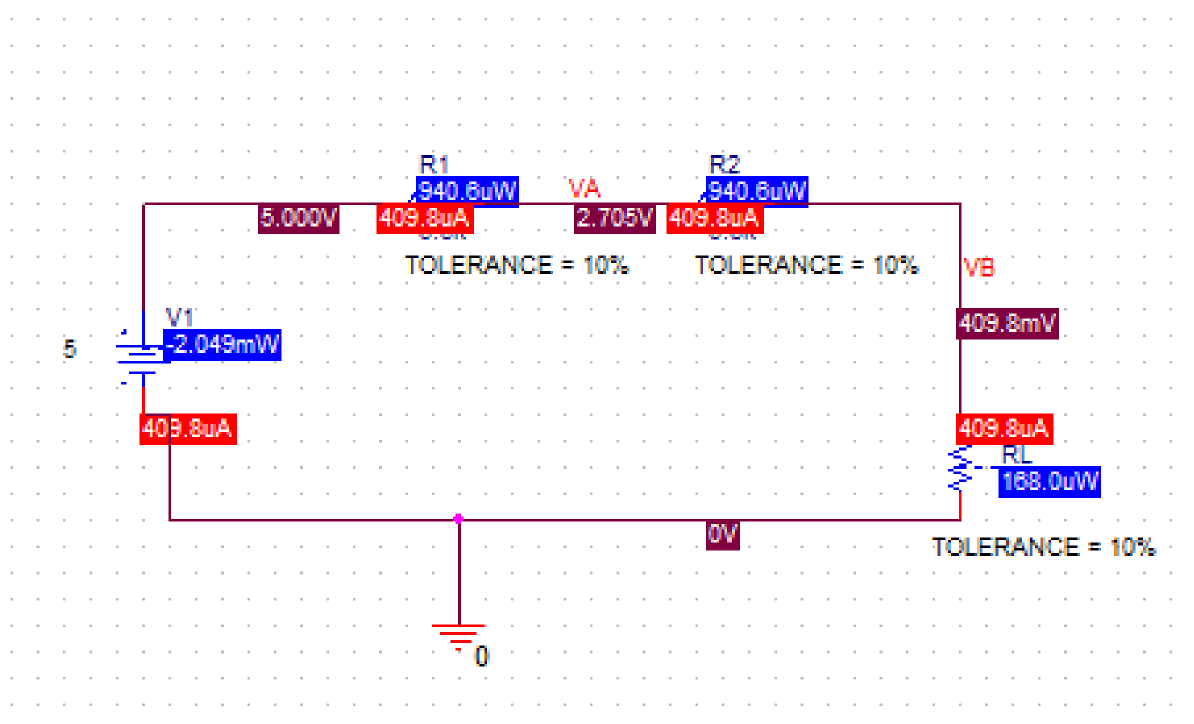

Figure 1.2.1: Circuit 1 with voltage markers.

Table 1: Simulation results for Circuit 1.

Node	Voltage (V)
Supply	5
Node A	2.708
Node B	0.411

The results in Table 1 agree with the hand calculations in equations 1.1.1 - 1.1.3.

1.2.2 PSPICE: Monte Carlo Simulation for Circuit 1

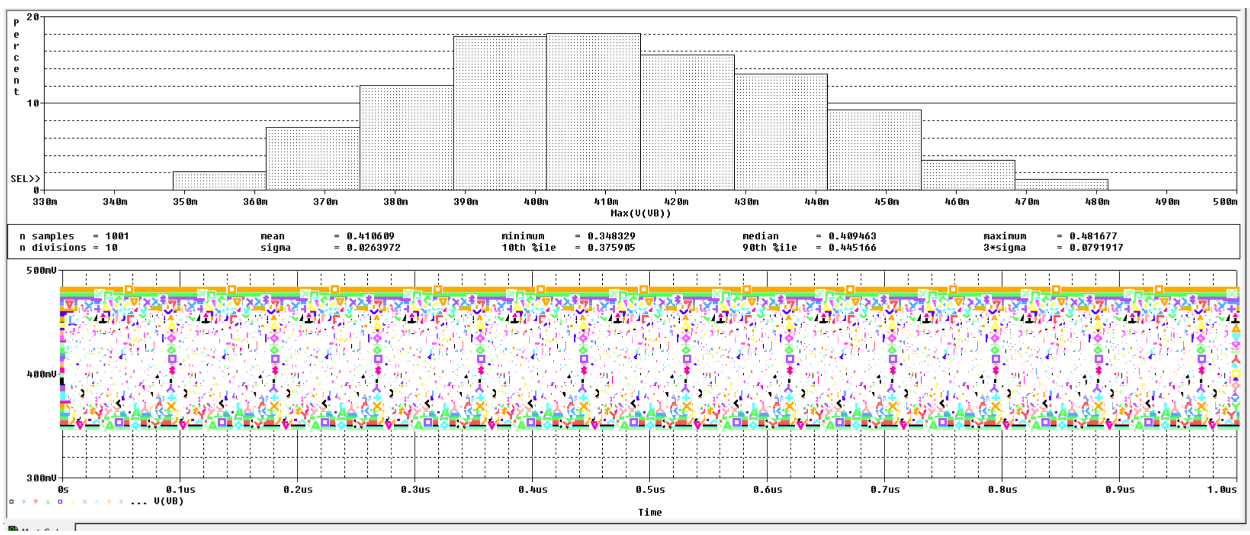

Figure 1.2.2: Monte Carlo simulation of Circuit 1.

The Monte Carlo simulation shown in 1.2.2 shows a probalility curve for likely measurements from Circuit 1. The highest probalility can be seen in the highest column at around 0.407 V This is very close to the calculated value of 0.410 V. The bars around the center show other likely measurements based on the tolerance set. The mean value found through the Monte Carlo simulation was 0.411 V and the sigma was 0.026.

2 Hardware Experiment: Results and Discussion

The circuit board was set up by connecting the power rail on the board to the 6V source on the DC power supply. The positive terminal on the rail was connected to the first 5.6 $k\Omega$ resistor. This first resistor was then put in series with the second 5.6 $k\Omega$ resistor and finally the 1 $k\Omega$ resistor. The node after the 1 $k\Omega$ resistor was connected to the negative terminal on the power rail.
The power supply was configured to 5 V and the current was limited to 0.1 A. This was done to prevent overheating in the case of a short circuit.

2.1 Equipment Used in the Laboratory

Table 2: Equipment/Software required for Lab 1.

Item	Tool	Room
Simulation	OrCAD Capture CIS	09-3200
DC Power Supply	Agilent E3631A	09-3200
Multimeter	Agilent 34401A	09-3200

OrCAD was used to create the circuit and the perform the Monte Carlo simulation. The DC Power Supply shows in Table 2 was used in the hardware portion to power the circuit. The multimeter shown was used to measure the voltage and resistances across each of the resistors.

2.2 Hardware Results/Discussion Circuit 1

The low error from PSPICE indicate that the resistors did not vary too far from the listed resistances. The Node B voltage had the highest error because that resistor had the lowest value. Therefore the likelihood of error is higher than the other two.

$$E_{percent} = \frac{|V_{pspice} - V_{measured}|}{V_{pspice}} \tag{2.2.1}$$

Equation 2.2.1 was used to calculate the error each of the measured voltages in Table 3.

Table 3: Experimental nodal voltages for Circuit 1.

Node	Voltage (V)	Percent Error With PSPICE
Supply	5.002	0.04%
Node A	2.708	0.1%
Node B	0.405	1.5%

Table 4: Differential voltages and measured resistances for Circuit 1.

Element	Component Value	Measured Resistance (Ω)	Voltage (V)
Supply	5 V	N/A	5.002
R_1	5600 Ω	5560	2.292
R_2	5600 Ω	5576	2.303
R_L	1000 Ω	1005	0.405

When Kirchoff's Voltage Law is applied to the measured value, the following is observered:

$$2.292\,\mathrm{V} + 2.303\,\mathrm{V} + 0.405\,\mathrm{V} - 5.002\,\mathrm{V} = -0.002\,\mathrm{V} \tag{2.2.2}$$

Equation 2.2.2 shows that Kirchoff's Voltage law holds for this circuit.

3 Conclusion

The purpose of this exercise was to show the use of voltage division and kirchoff's law as well as show error analysis with PSPICE. A circuit was designed with three resistors in series and run through a Monte Carlo simulation to show likely measurements. The circuit was then built on a breadboard and measurements were compared to PSPICE. Percent error was analysed to be within the tolerance rated on the resistor.

4 Acknowledgments

- Petre Tumbar helped through the PSPICE Monte Carlo setup.
- wikibooks.org helped with the LaTeX code.

References

[1] C.K. Alexander, and M.K.O. Sadiku, *Fundamentals of Electric Circuits, 4th Edition*, McGraw Hill, pp. xx-yy(EDIT), 2009.

[2] S. Rommel, *EEEE 281 Lab 1 Lecture notes*, Spring 2015.