Abu Dhabi University

Electric Circuits

Lab Report 3

Simulating Electrical Circuits Using Computer Software

Author:
Muhammad Obaidullah 1030313
Mehdi Ismail 1005689
Saif Mohammed 1003317
Bilal Arshad 1011929

Supervisor:
Dr. Montasir Qasymeh

Section 1

August 10, 2012

Abstract

In this lab we were introduced to National Instrument's circuit simulating software Multisim. We also analysed some basic DC circuits and learnt how to take readings using from the circuit simulation using virtual meters.

1 Introduction

In First Exercise, we were just introduced to all the features of Multisim and we drew a simple circuit.

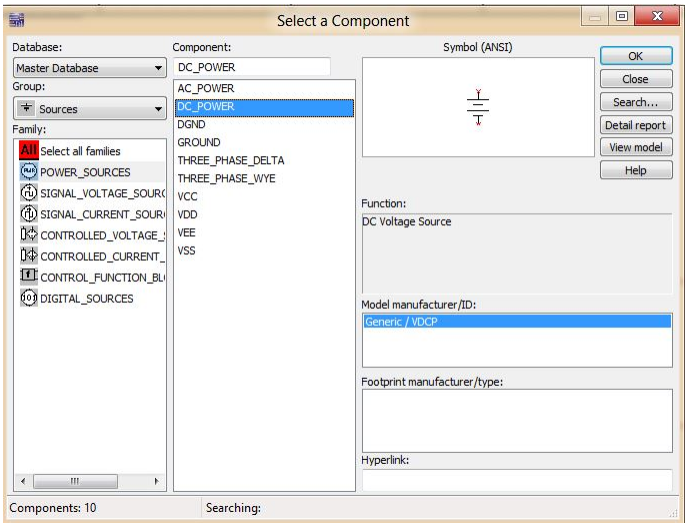

Figure 1: First Simulation Circuit

In Second Exercise, we drew, simulated and calculated the voltages and currents on various parts of the following circuit.

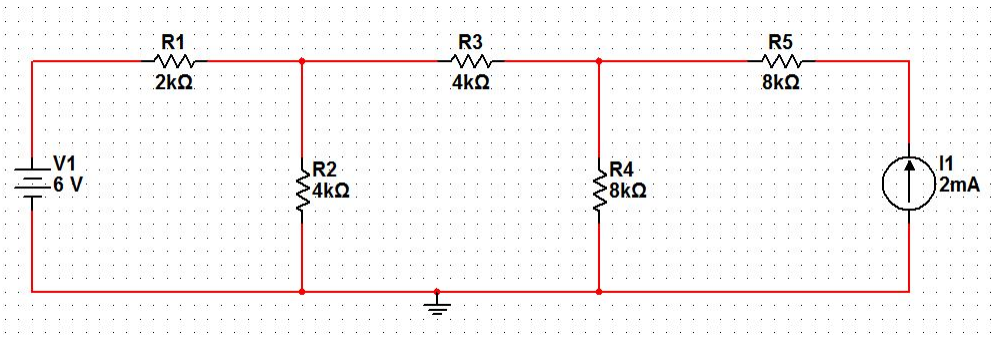

Figure 2: First Simulation Circuit

In Third Exercise, we drew, simulated and calculated the voltages and currents on various parts of the following circuit.

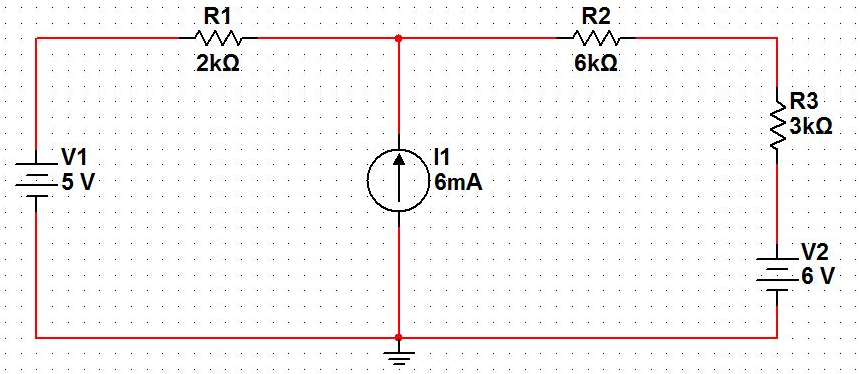

Figure 3: Second Simulation Circuit

2 Experiment Set-up

The components were placed and fixed onto the training board and wires were used to connect the components in the circuit as shown in the figures 2 and 3. A multimeter was used to calculate all the values as the multi-meter can calculate Current, resistance, and voltage.

3 List of Equipment used

- A Computer.
- National Instruments Circuit Design Tool.

4 Procedure

4.1 Method 1 - Using Traditional Ammeter method

- Draw the circuit in the Multisim using place a component button to place a component and simply click to draw wires.
- Place voltmeter across each components in the circuit using the place indicator button.
- Click the simulate button on the top right corner of the window.
- Read and note the values of the voltmeters which were placed across each component of interest.
- Place Ammeter in series with each component in the circuit using the place indicator button.
- Click the simulate button on the top right corner of the window.
- Read and note the values of the ammeter which were placed in series of each component of interest.

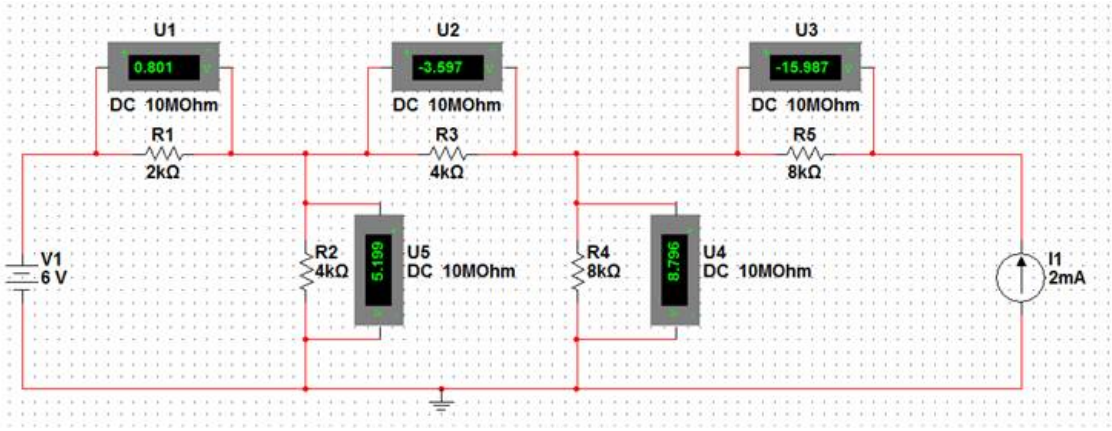

Figure 4: Measuring Voltages across resistors

4.2 Method 2 - Using Probe method

- Draw the circuit in the Multisim using place a component button to place a component and simply click to draw wires.
- Place voltmeter across each components in the circuit using the place indicator button.
- Click the simulate button on the top right corner of the window.
- Read and note the values of the voltmeters which were placed across each component of interest.
- Click on the measurement probe button on the right side of the window.
- Place the probe on each branch of interest in the circuit.
- Read and note the values of the probe which were placed.

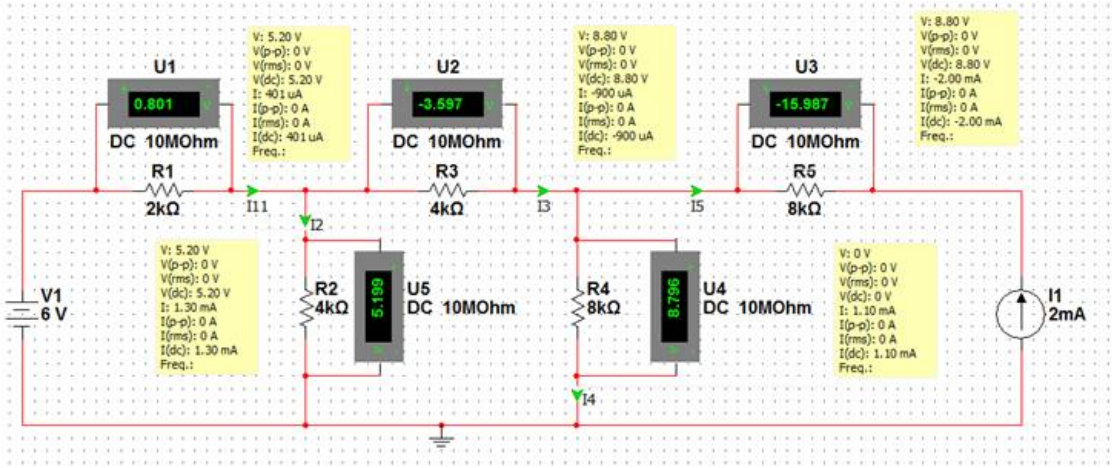

Figure 5: Measuring current in each branch using probe

5 Results and Discussions

At the end of these exercises we got the following results:-

- Multisim values and calculated values were different from each other.
- Error Percentages were very low.
- Usually we applied Mesh Analysis to calculate the current in the circuit.

Branch Values	Multisim Results	Calculation Results	Error Percentage
V1	0.801V	0.8V	0.13%
V2	5.199V	5.2V	0.02%
V3	-3.597V	-3.6V	0.08%
V4	8.796V	8.8V	0.05%
V5	-15.987V	-16V	0.08%
I1	401µA	400µA	0.25%
I2	1.3mA	1.3mA	0%
I3	-900µA	-900µA	0%
I4	1.10mA	1.10mA	0%
I5	-2.00mA	-2.00mA	0%

Figure 6: Results for Exercise 1

Branch Values	Multisim Results	Calculation Results	Error Percentage
V1	-9.998V	-10V	0.02%
V2	5.998V	6V	0.03%
V3	3.000V	3V	0.00%
I1	-5.999mA	-6mA	0.02%
I2	-1.000mA	-1mA	0%
I3	5.000mA	5mA	0%

Figure 7: Results for Exercise 2

6 Conclusion

- Ohm's Law is valid for both experiments. Thus V=IR. But we used Kirchoff's Rules for easier circuit analysis.
- The calculated and multisim values are different because multisim assumes some small resistance of its ammeter and not assume not large resistance for volt meter.
- Multisim results will match the real life circuit much more closely than a calculated value.

7 Questions

Is the simulation exactly equal to the theoretical results? Why?

No, The simulation result are slightly different than the theoretical results. Because when we calculate results theoretically, we assume the resistance of voltmeter to be infinite, which is impossible in real life, and we assume the resistance of Ammeter to be exactly 0 Ohm, which is also impossible in real life. On the Other Hand Multisim assumes some very small resistance for the ammeter and a very large resistance for voltmeter. Thus,calculates its results according to its assumptions which are close to real life circuit.