Circuit Simulation

Introduction

Circuit simulators are used to model various circuits. They are generally very powerful programs that are able to handle voltage and current calculations for thousands of devices. We are not going to build any circuit this big. However, even with our small ten-element circuits, simulators can help reduce our workload. We should also become comfortable using simulators to back up and confirm our ideas or calculations.

There are many different circuit simulators on the market today. Most are based off of SPICE, an open-source analog circuit simulator. We will be using LTspice from Analog Devices. It is free simulator tool that is straightforward to use. You should download and install the program on your home computer. A link to download it can be found with a quick internet search.

The analysis for this experiment requires computation of series and parallel resistors. Combining resistors in this way allows us to reduce the number of resistors in our circuit. In this lab, the circuit will go from four resistors down to one resistor.

Resistors in Series

Resistors in series can be replaced by an equivalent resistance found by adding the resistor values. For example, a 10Ω resistor in series with a 20Ω resistor can be replaced by an equivalent resistance of 30Ω.

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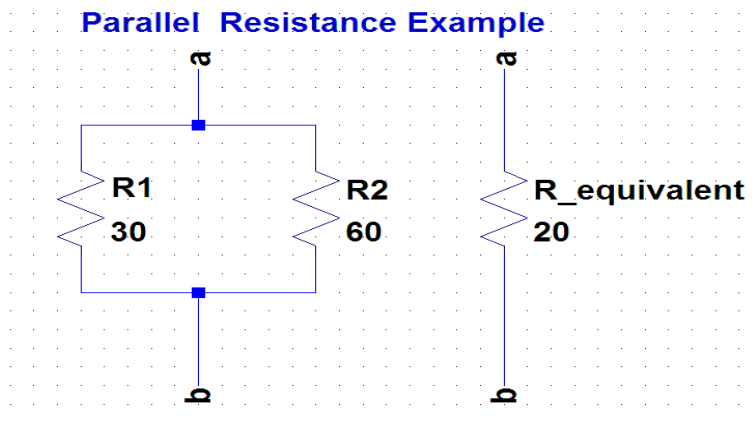

Series Resistance Example
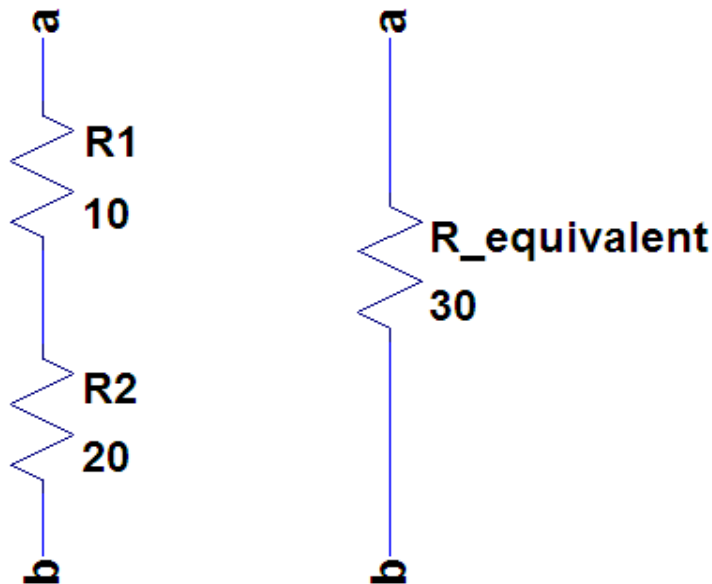
Circuit Simulation

Resistors in Parallel

Resistors in parallel combine using the formula shown below.   For example, a 30Ω resistor in parallel with a 60Ω resistor can be replaced by an equivalent resistance of 20Ω.

$$\frac{1}{R_{equivalent}} = \frac{1}{R_1} + \frac{1}{R_2}$$

Parallel Resistance Example

R1	R2	R_equivalent
30	60	20

Objectives

- Learn how Matlab can be used to analyze simple circuits
- Understand how LTspice can be used to simulate electrical circuits
- Develop methodology for analyzing and reporting data collected from a variety of methods.
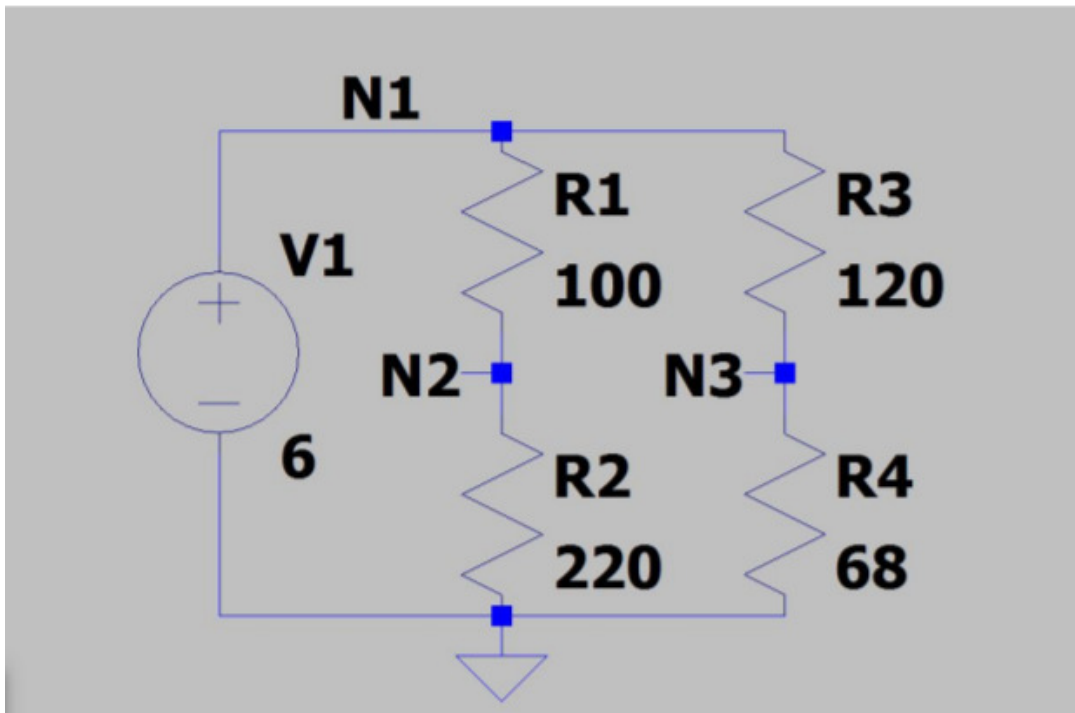
Resources

There are many resources for both Matlab and LTspice available on the web.   Your instructor may provide you with some as well to provide a starting place for you. There are many more resources out there.

Prelab

1. View at least one of the videos explaining how to get started in Matlab.

    - https://www.youtube.com/watch?v=jTS5ZmrrzMs#t=13

    - https://www.youtube.com/watch?v=6HIvXWEbRLk

    -

2. Review the instructor-provided ENGR101 Matlab programs.

3. Install LTspice on your personal computer.

4. Watch the first video on how to build a circuit in LTspice. The subsequent videos will come in handy for later labs: https://learn.sparkfun.com/tutorials/getting-started-with-ltspice/all

5. Review this introduction to LTspice from the file **EET111-Intro_to_LTspice.pdf**.

Look at the circuit below. N1, N2, and N2 are labels attached to those nodes.

For this circuit, hand-calculate the node voltages (N1, N2 and N3) and currents I(V1), I(R1), I(R2), I(R3), I(R4) using KVL and KCL.

Lab Procedures

Create a table for your data. Fill in with the above values. You will be collecting additional data from simulations in the next steps. Rows will be the values. Four columns for hand calculated, spice voltage supply, spice current supply. Leave space for additional columns for accuracy errors.

Simulate and Record Data/Graphs:
   a) Using LTspice, draw and simulate the circuit shown above. Make sure you label the N1, N2, N3 nodes. This helps in reading the Spice output information. You will do just a DC operating point simulation. Record your data in the table. **Also record the voltage across R1 and across R3 (this is a calculated value from your spice results).**

b) Using a screen capture utility, capture spice schematic and the pop-up window that contains the operating point values for nodes and currents.

Put all of your hand calculations and LTspice data into tables in MS Excel.   Use Excel to perform/demonstrate your hand calculations. Include your screen captures from your LTspice simulation.

Compute the difference error (not percent error) between the hand-calculated numbers and the numbers from step **a)** above. Nearby, in your spreadsheet, try to explain any errors.

Calculate the R_total from steps a and b above. Compare and discuss any differences.

Submit your Excel file (**.xlsx**) with your data, screen captures, and hand calculations (performed by the spreadsheet) to the Lab 2 folder in D2L.