Lab 4: Resistance Calculations with LTspice

Background:

Being able to calculate resistance between any two points in a given resistor network is a skill that is required for some more advanced circuit analysis techniques, such as finding a circuit's Thevenin Equivalent. It is also a skill that can be surprisingly difficult to people new to circuit analysis.

This lab assignment will have you calculating resistance between various nodes in resistor networks, and then checking if your work with LTspice. LTspice simulations output to basic things: voltage at each node with respect to ground and current through each circuit element. LTspice does not report resistance. In fact, in order for a simulation to run, you must provide LTspice with all resistor values.

Resistance between two arbitrary nodes is neither something that is entered to do an LTspice simulation nor is it something outputted in simulation results. This lab investigates how we can get a these simulated resistance values despite them not being directly provided.

The resistor network below has four nodes. The resistance between each pair of nodes is different.

Figure 1: Resistor Network

The resistance between nodes A and B, denoted as R_{AB}, is simply 120 Ω. The 47 Ω and the 100 Ω resistors do not factor into into R_{AB}. If you were to attach a 1A test source between nodes A and B to make current flow into A and return through B (see below), think about where current would flow.

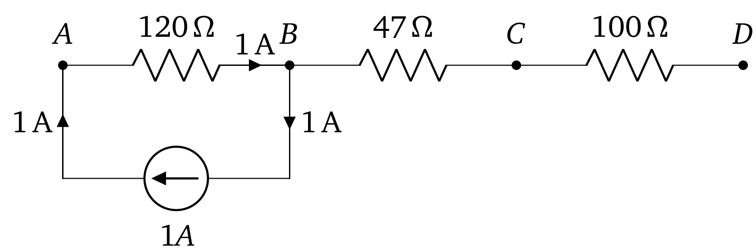

Figure 2: Test Source to Visualize What Resistor Values are Relevant to R_{AB}

Current would only flow through the 120 Ω resistor. From this perspective, the 47 Ω and the 100 Ω resistors are in a dead-ended branch, thus no current would flow through them and they do not factor into R_{AB}.

The resistance between two other nodes is different. R_{BD} is calculated by considering where current flow if a test source were hooked up to nodes B and D, as shown below.

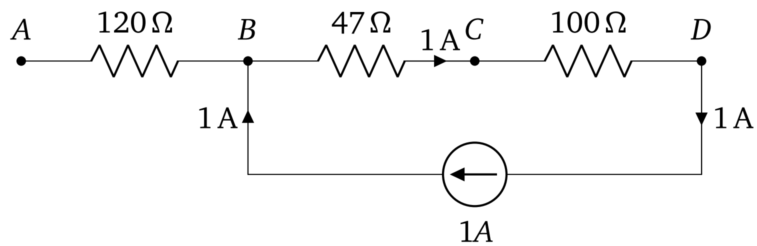

Figure 3: Test Source to Visualize What Resistor Values are Relevant to R_{BD}

Consider a similar resistor network, but with an important difference. Below is the previous network with a wire (short circuit) added between A and D.

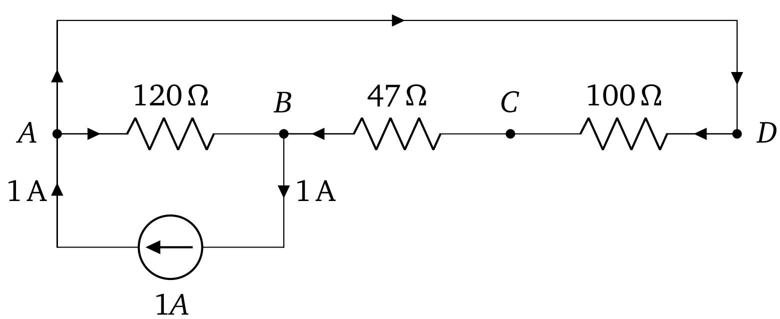

Figure 4: Changed Network with Test Source for R_{AB}

This new wire changes resistance between all points. There is a test source hooked up to visualize how to calculate R_{AB} in this new circuit. Being able to visualize this current flow allows you to see which resistors are in series and which are in parallel.

The 1A from the source is split into two pieces. One piece flows from node A to node B via the 120 Ω resistor, and the second piece flows to the same place via the new short circuit, the 100 Ω resistor, and the 47 Ω resistor. R_{AB} is calculated as follows.

$$R_{AB} = 120//(100+47) = \frac{1}{\frac{1}{120}+\frac{1}{100+47}} = 66.07\Omega$$

With this new circuit, calculating R_{AB} has no dead-ended resistors.

Objectives:

- Obtain simulated values that are not directly provided by the simulation software.
- Calculate resistance between any two points in a resistor network.
- Use Excel formulas to perform calculations.

Required Equipment:

- LTspice XVII
- MS Excel or an equivalent spreadsheet program

1 Prelab:

1. Read through this entire lab assignment.
2. Create all of the required Excel tables that perform the required calculations.

2 Procedure A:

1. Create a new Excel document and examine the schematic in Figure 5.

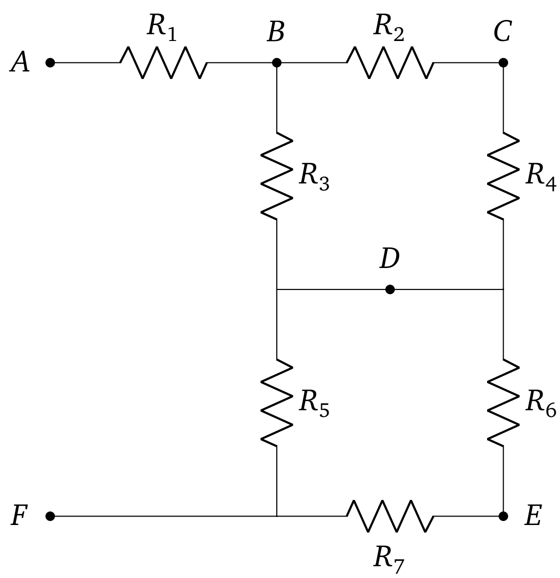

Figure 5: Procedure A Resistor Network

Use the resistor values below for Figure 5.

(a) $R_1 = 100\,\Omega$
(b) $R_2 = 20\,\Omega$
(c) $R_3 = 30\,\Omega$
(d) $R_4 = 50\,\Omega$
(e) $R_5 = 200\,\Omega$
(f) $R_6 = 25\,\Omega$
(g) $R_7 = 125\,\Omega$

2. In your Excel file, create a table listing the resistor values above. Recall that inserting any text (units, engineering prefixes, or otherwise) causes Excel to evaluate your cells numerically as zero. Doing so is not desired.

3. Create a second table within the same Excel worksheet listing calculations for the following resistances.

   (a) R_{AF}
   (b) R_{CD}
   (c) R_{DF}

Calculation requirements and suggestions:

(a) All of you calculations should be done in Excel formulas.

(b) You will not have to do any $\Delta - Y$ or $Y - \Delta$ conversions in this lab assignment.

(c) The entire calculation does *not* have to be in a single cell/formula. You can create a separate table for intermediate results. For example, you may want to calculate the following two values (and probably others) to make calculating R_{AF} easier.

$$R_2 + R_4$$
$$R_3 // \left(R_2 + R_4\right)$$

These results can be stored in intermediate cells for use in your Excel formula to calculate R_{AF}.

In case that you are an advanced Excel user, it is possible to write a VBA function to calculate parallel resistors. This is wonderful, but please do not do this for this assignment. It involves enabling macros which is a security issue.

4. Use LTspice to obtain simulated values for the above resistances. Add columns to your latest Excel table for these values. You should have enough columns to enter the data directly obtained from LTspice plus one column for simulated resistance, which is a calculation based on the directly-obtained data.

5. Add another column to the same table to compare calculated vs simulated resistances using percent error. We will consider simulated resistance to be measured resistance.

6. Change the circuit in Figure 5 so that it has a short circuit between nodes A and B. Repeat steps 3-5 with this new circuit.

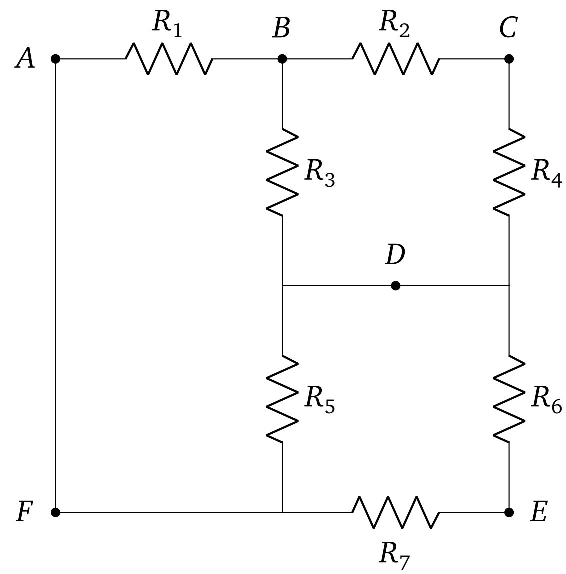

Figure 6: Changed Resistor Network

7. Take a screenshot of your LTspice with your test source in place for obtaining R_{DF}. Add this screenshot to your Excel file.

3 Deliverables:

Submit your Excel file (extension of **.xlsx**) to this lab's folder.

1. Do not include any Excel/VBA macros. If you do not know what these are, then you will not have to worry about it.
2. All calculations must be done in formulas
3. All tables must be labeled clearly. For instance, it should be obvious which tables are for Figure 5 vs Figure 6.
4. Include the LTspice screenshot within your Excel file in accordance with the last step of the Procedure.