Laboratory 3: Electrical Circuits and Measurements II

3.1 Introduction

This lab focuses on using a schematic editor and circuit simulator. By the end of the lab, you should be able to virtually build a circuit as well as take voltage and current measurements using a simulation system called Multisim Live.

3.2 Resources

The additional resources required for this assignment include:

- Book: Alexander & Sadiku for circuit drawings.
- Pratt Pundit Pages: Resistor Color Codes, Multisim

3.3 Equipment

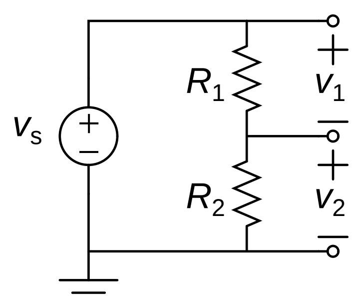
This lab will all be done in a web browser. Later, you will be comparing the simulated results to actual measurements.

3.4 Multisim Live

Multisim (multisim.com) is a project from National Instruments that allows you to use a circuit schematic editor to build and simulate a wide range of devices. Follow the instructions on Pundit on the Multisim page for setting up an account and for completing the circuits tutorial.

3.5 Building Your Voltage Divider Circuit

For part of this lab, we are going to look at a voltage divider. To build this circuit in Multisim, you are going to place your components on the schematic and use a voltage/current probe voltage and current measurements. To begin, click the File icon at the top left and click New file. The schematic for the circuit you are building is:



where v_s will be set by the power supply and R_1 and R_2 will be a matched pair of resistors. Ideally, this voltage divider will split the voltage from the supply exactly in half. You will be measuring the voltage drop across both the top and the bottom resistor (v_1 and v_2, respectively).

Copy a ground node, a DC Voltage source, and two resistors onto the schematic. Connect them in basically the same way as shown in the figure above. Next, add a voltage probe to the top of R_1. This defaults to measuring the voltage at that node relative to ground, but we actually want to measure the voltage relative to the bottom of that resistor. Click on the probe and the click on the **V-** option. This allows you to pick a reference node. Pick the node at the bottom of R_1.

Next, add a "voltage and current" probe to the node between the resistors. Make sure the current probe is measuring the current going from top to bottom. You will not need a voltage reference node here since the bottom of v_2 is the ground node. If the labels overlap each other, that is fine; you can also make your circuit take up more space if you want everything to be "circuitly distanced."[1]

[1] Too soon?

3.5.1 Running the Simulation

Now that you have built the circuit and added the probes, you can run a simulation. Set the simulation type to DC Op and click the play button. You should get a reading of 6 V for each voltage and 6 mA for the current based on the default values for the components. In Lab 2, we used 5 V for the source so go ahead and make that change. Note that when you change values in a circuit the probes will tell you their values are out of date. Re-run the simulation and you should now be reading 2.5 V across each resistor and 2.5 mA.

Multisim has powerful tools for analyzing a circuit as sources values change, but nothing to automatically examine what happens as resistances change. Furthermore, the voltage probe is *not* simulating a multimeter - it does not include an internal resistance. Increase the resistors to 100 GΩ and run the simulation again. You will end up getting a voltage measurement different from the 2.5 V expected, but that is a *computational* issue rather than a feature of the simulation. Go ahead and change your resistors back to 1 kΩ, run the simulation, and take a screenshot. You can also export the schematic image, rather than taking a screenshot by clicking the Export icon (a box with an arrow coming out of it at the top right of the window). Be sure to look at this, however, because sometimes the exported image is scaled differently from the screen and labels get lost or overlap. Once you have an acceptable image, save this circuit.

3.6 Assignment

For this assignment, you are going to present screenshot of the schematic you took from the divider and also use Multisim to solve for values in four other circuits.

3.6.1 Voltage Divider

This part of the assignment is described above.

3.6.2 Based on Problem 2.57

Instead of what is asked in the problem, build the circuit using a 1 A current source instead of a 20 V voltage source. Use Multisim to measure the voltage drop across the 1 A source. The equivalent resistance R_{eq} will be $\frac{v}{1\text{ A}}$. If you had been asked to do this by hand, there would have been two Y-Δ conversions to simplify the network! Take a screenshot showing the voltage measurement and report the equivalent resistance in your lab report.

3.6.3 Problem 2.35

Use Multisim to model the circuit given in Problem 2.35 on p. 69. As in Lab 2, use a 20 V source instead of 200 V so that you can directly compare values with Lab 2. Note that you cannot draw a wire to a wire - you will need to draw from a terminal of an element to another terminal or wire. This will be important when you make the connection between the tops of the two bottom resistors. Unlike Tinkercad, where you needed to have that branch run through the ammeter, you will simply need to put a current probe on the wire. Note that there is an arrow that indicates the direction of the current - make sure it is measuring from left to right. For the voltage probe, note that it will be measuring a node voltage. In this case, as long as the ground is the bottom node, the voltage in that middle right node will be the same as the voltage drop we want to measure.

Once built and running, take a screenshot of the circuit being simulated showing the values of the current and the voltage. After documenting this circuit, go through and change all the resistors to be in MΩ instead of Ω and take a screenshot of the new values. Note that for Multisim, you actually get a finite (though *very* small) current measurement. The voltage measurement will not change at all, and there will be a little bit of roundoff error in the current measurement.

3.6.4 Problem 2.25

Multisim can also model controlled sources, though the schematics tend to get messy because you actually have to connect the controlling variable to the controlled source. There are some example screenshots available on Sakai - look at those first before continuing.

To build this circuit, start a new circuit and drag a DC current source, three resistors, ground, and a voltage controlled current source. The latter is in the Modeling blocks group, which has an icon that looks like a square with two terminals on the left and two terminals on the right. Remember that to rotate the resistors you will need to select them and click the rotate button (or type shift-R).

You will need a combination of rotations and flips to get the VCCS to have the voltage measuring ports on the left and the current flowing from bottom to top. Unfortunately there is no way to adjust the voltage measurement direction relative to the current direction, so your wiring is going to look complicated. First connect all the elements (except for the control voltage) to the ground wire. Next, connect the tops of the relevant elements as needed. Finally, connect the positive measurement of the controlling voltage measurement to node 1 and the negative measurement to ground. You will have to have wires cross over each other. Note that in Multisim wires that cross are assumed to be *unconnected* unless there is a dot at the crossing. Change the values to those given in the problem, add the necessary probes to figure out what is going on with the far right resistor, do a DC simulation of the circuit, and take a screenshot while the simulation is running. Be sure to save the circuit for future use.

3.6.5 Problem 3.31

Create this circuit on Multisim, include probes for each node voltage and for the current through the resistor between node 3 and ground, run the DC simulation, then take a screenshot. This circuit has two controlled sources and the controlling variable in one case is a current. That means you will actually have to have the wire through which the current is measured flow through the current measurement of the CCVS. Those particular connection are, in two words, not pretty.

3.6.6 Figures to include

Include a screenshot of your voltage divider with 1 kΩ resistors, a screenshot for Problem 2.57, screenshots for Problem 2.35 with the two different resistor sizes, a screenshot of Problem 2.25, and a screenshot of 3.31.

3.6.7 Questions, Calculations, and Conclusions

You will not be writing a formal lab report but rather will be presenting your screenshots and then answering some questions. Your report does not have to be in LaTeX but it does need to look professional. As noted above, the screenshots can either be a part of the document or apart from the document.

(1) What are three major differences between a package like Tinkercad and a package like Multisim? Compare and contrast the ease of use for the circuits we have looked at so far, especially the voltage divider circuit with different resistance values.

(2) What is the equivalent resistance of the network in Problem 2.57?

(3) What are the differences between the Multisim answers for Problem 2.35 with different sizes of resistor? How do those differences compare with what happened in Tinkercad?

(4) What is the power absorbed by the 20 kΩ resistor in Problem 2.25? Since power does not show up on the schematic, you will need to calculate that and enter it in your lab report.

(5) What is the power delivered by the VCCS in Problem 3.21? Since power does not show up on the schematic, you will need to calculate that and enter it in your lab report.

(6) If you could make any three improvements to Multisim, what would those be?

Concluding Remarks

Labs 2 and 3 have really just broken the surface on the powerful analysis tools in both Tinkercad and Multisim. It may seem at the moment that there is a lot of overhead for very little reward. Once we get into microcontrollers, Tinkercad will really shine. Once we get into reactive circuits and time series, Multisim will too.