

Appendix I

Circuit Simulation Using SPICE

Throughout the semester we will be using SPICE to simulate circuits. There are numerous versions of SPICE available for free download. Some of the more common ones are PSPICE, LTSPICE, and Multisim. This appendix and other places throughout this lab manual will use “LTspice” (Linear Technologies) but you are free to use whichever version you like. You should be able to install your own free version of the LTspice software at <http://www.linear.com/designtools/software/>.

There are numerous tutorials on line showing you how to simulate circuits using LTspice:

<http://cds.linear.com/docs/en/software-and-simulation/LTspiceGettingStartedGuide.pdf>
<https://www.youtube.com/watch?v=GmzfJa2GS7c>
<https://www.youtube.com/watch?v=lyADW32wi10>

To start with, in this Appendix, we'll go over how to perform simple DC resistive circuit simulation. Open the LTspice program and click on the new schematic  icon on the far left side of the toolbar menu. You should see a blank schematic as shown in Figure I.1.

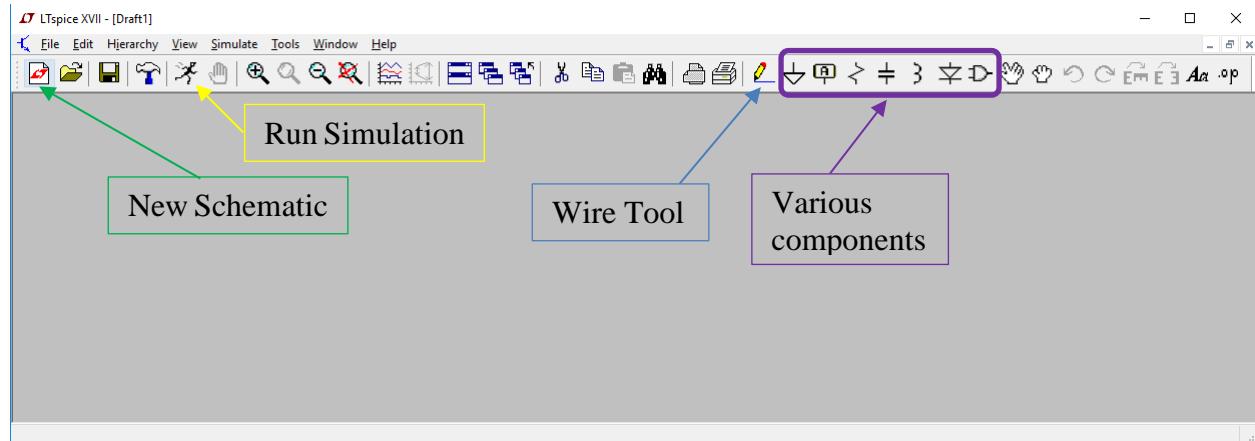


Figure I.1 – Blank LTspice schematic.

To make a voltage divider circuit. Click on the resistor icon on the toolbar and add a couple of resistors to the schematic. You may want to rotate one or more of the resistors. To do so, click on one of the hand  icons (move or drag) to highlight the rotate icon . You can add a

voltage source using the component  icon. This will open a menu of a large number of advanced components. Click on [voltage] to put a voltage source in your schematic. At this point your schematic should look something like the one shown in Figure I.2.

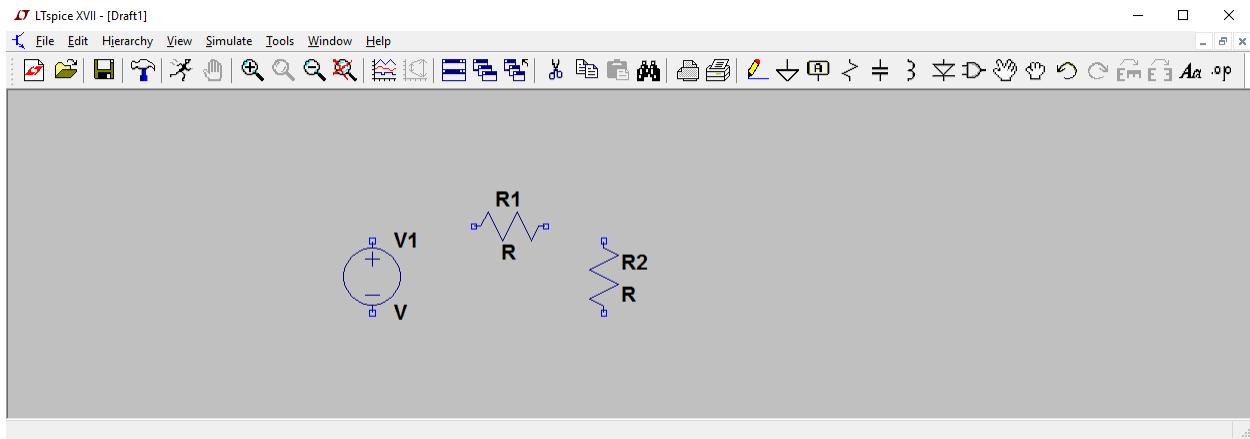


Figure I.2 – Schematic diagram with a voltage source and two resistors.

Currently the components in our schematic are disconnected. We will use the wire tool  to connect them. To connect two components, left click on the wire tool, then left click on the points on the schematic where you want to connect the two ends of the wire. When you are done connecting to elements, right click to release the wire tool. Repeat this procedure to all elements you want connected.

When you are done wiring your circuit, you will need to specify a ground point in your circuit. Click on the ground icon  and then place it on the schematic and wire to the point in the circuit you wish to be your ground reference. Your wired circuit will then look like the one shown in Figure I.3.

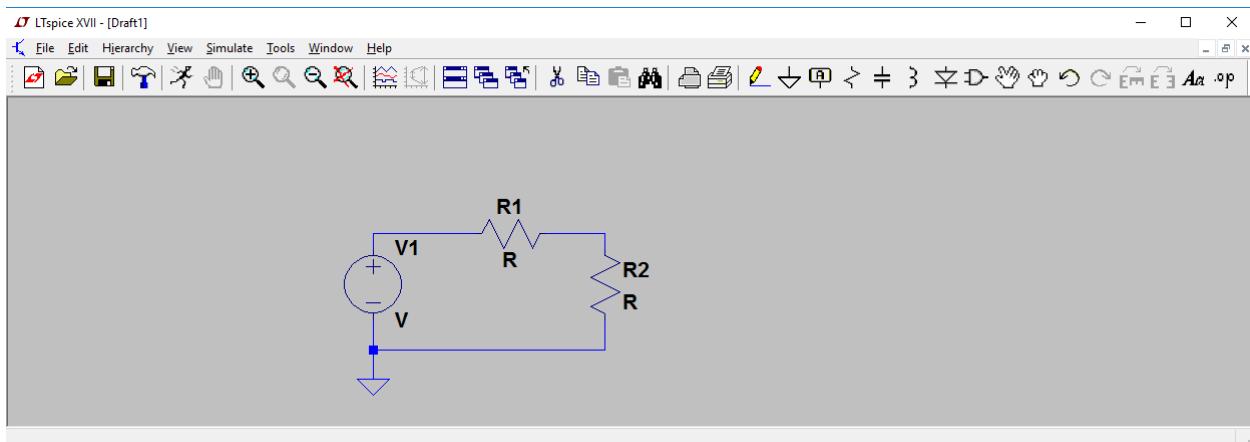


Figure I.3 – Wired circuit with ground.

Next we will need to add component values to each element in the circuit. Right click on each element to open a pop-up window which will allow you to specify values of each component. For resistors, enter the resistance in Ohms, for voltage sources enter the voltage in Volts, etc. Your final circuit schematic might look something like the one in Figure I.4.

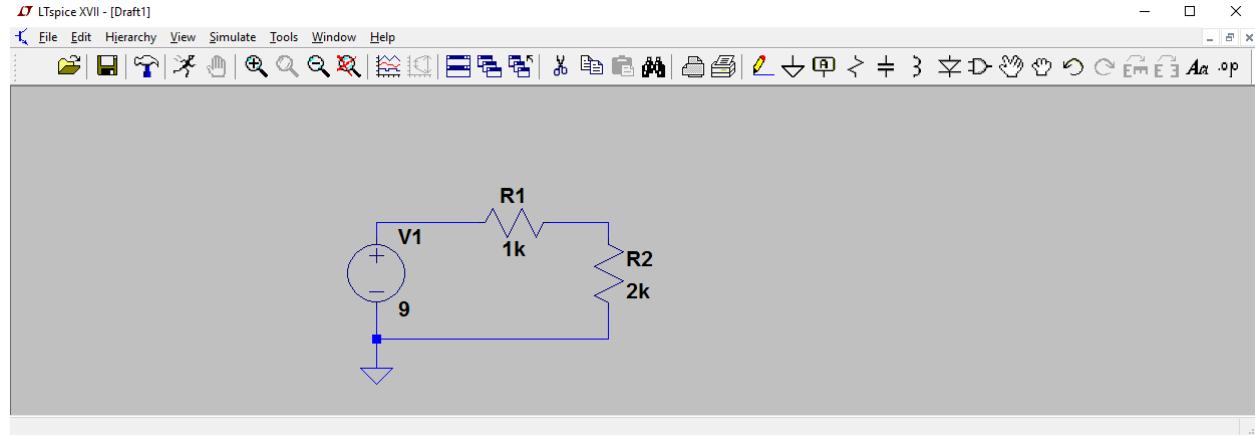


Figure I.4 – Completed schematic for voltage divider circuit.

When you create your schematic SPICE keeps track of your circuit by creating a series of nodes and a list of circuit elements specifying what they are, what their component values are, and what nodes they are connected to. This is called a netlist. From the [View] menu on the toolbar, click on [SPICE Netlist]. You will see a window pop up with the net list of the circuit you created. Yours should look something like the one shown in Figure I.5. From the netlist, you can infer what SPICE has labelled each node. Note that it labels the ground node as 0.

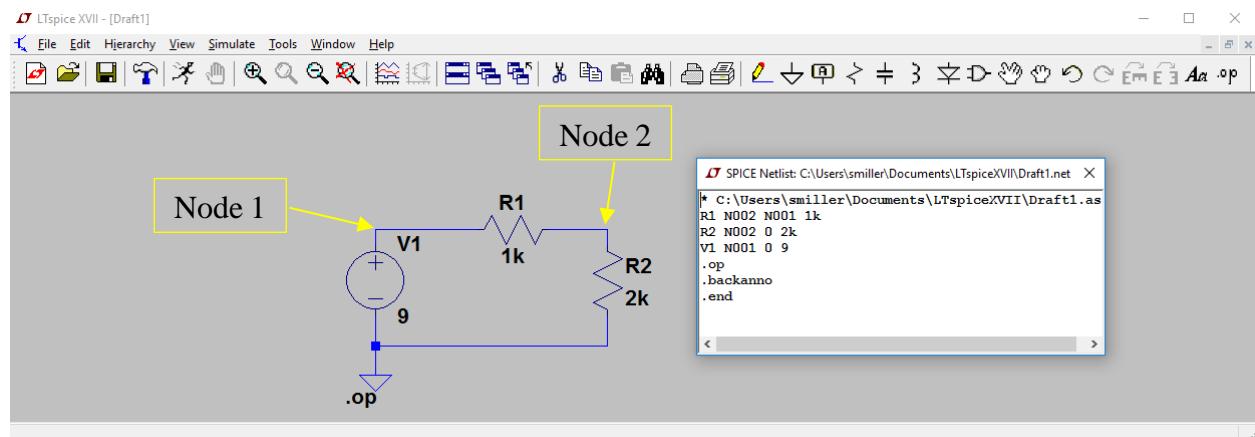


Figure I.5 – Circuit Schematic with Netlist.

Finally, you can simulate your circuit by clicking the run simulation icon. A window will pop up giving you several options for the type of simulation to run. For this circuit, choose the DC operating point simulation. A window will pop up showing you the voltages at each node and the currents through each device. Can you infer from the output shown in Figure I.6 what the convention is for reporting current direction?

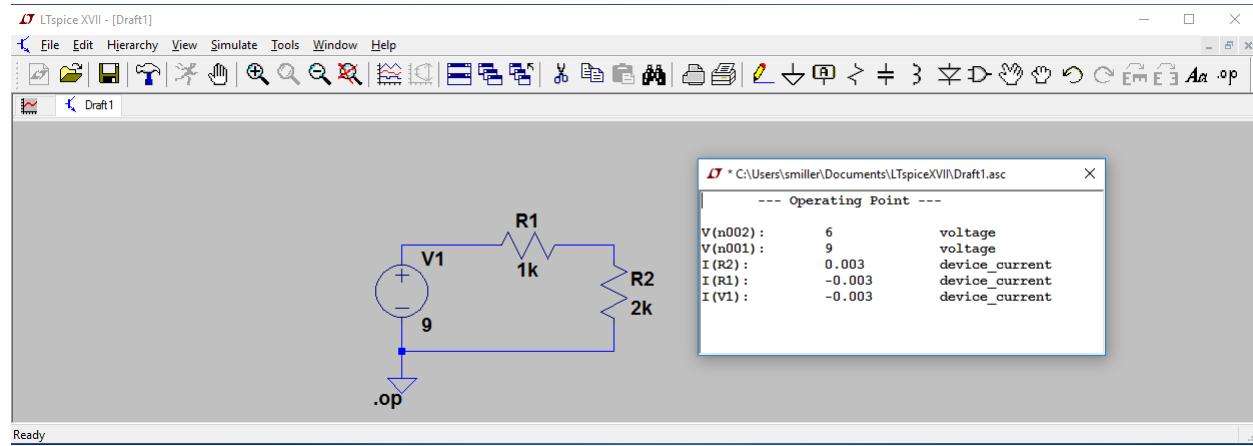


Figure I.6 – Result of DC Operating Point Simulation.