

M.2: LTspice Fundamentals – Schematic Capture

LTspice also allows us to submit the circuit description via schematic capture, which we will discuss here in the form of an example.

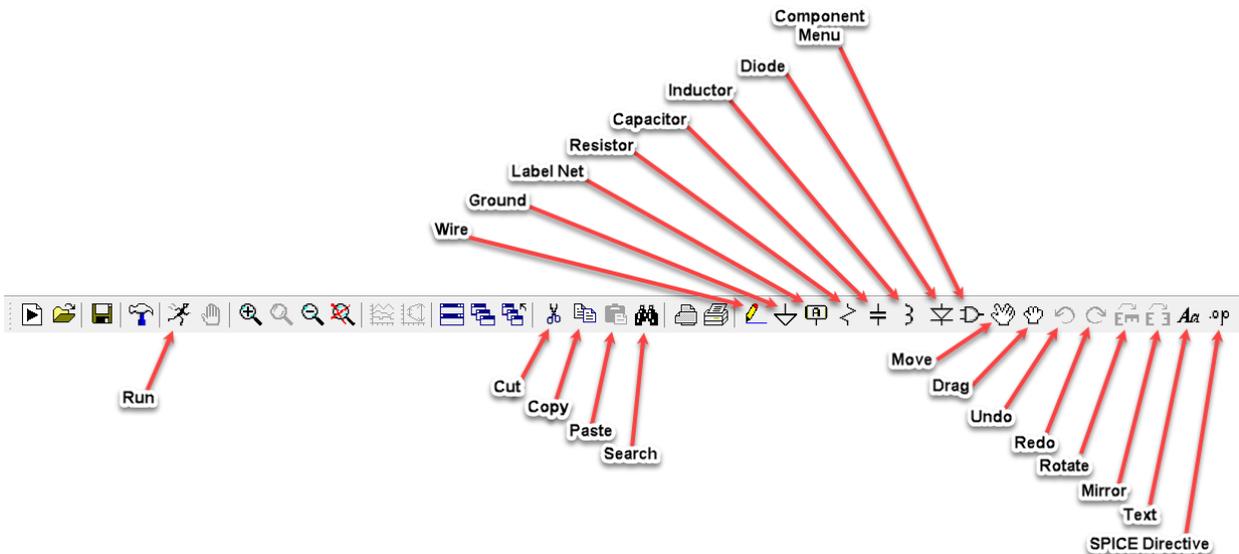
Example: Steady-state DC voltage divider circuit.

Procedure

1. Open a new schematic using the menu button, or go to **File > New Schematic** to do so.



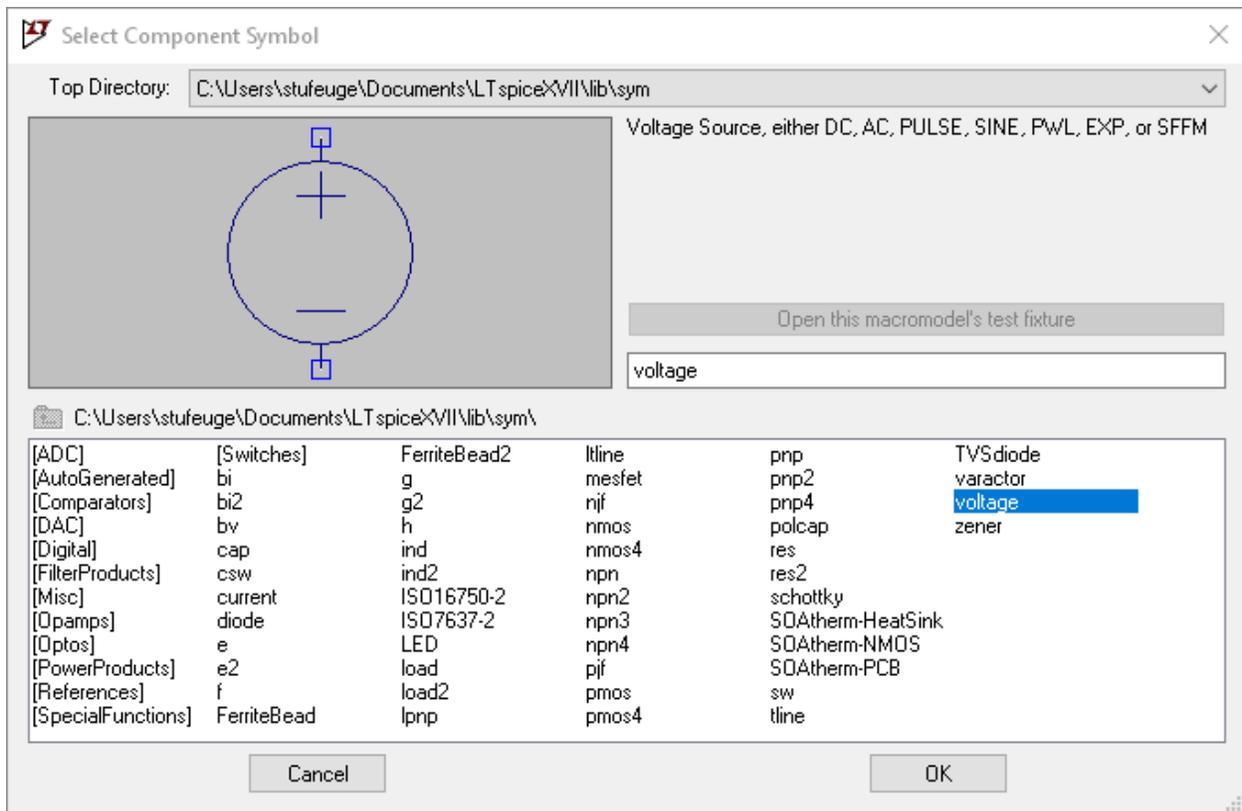
2. Components are placed by choosing the appropriate symbols from the Schematic Editor Toolbar and placing them in the schematic window. Review the toolbar (shown below) to gain an idea of what's there. Also note that components are placed vertically by default, so you may need to rotate some of them when building the schematic diagram.



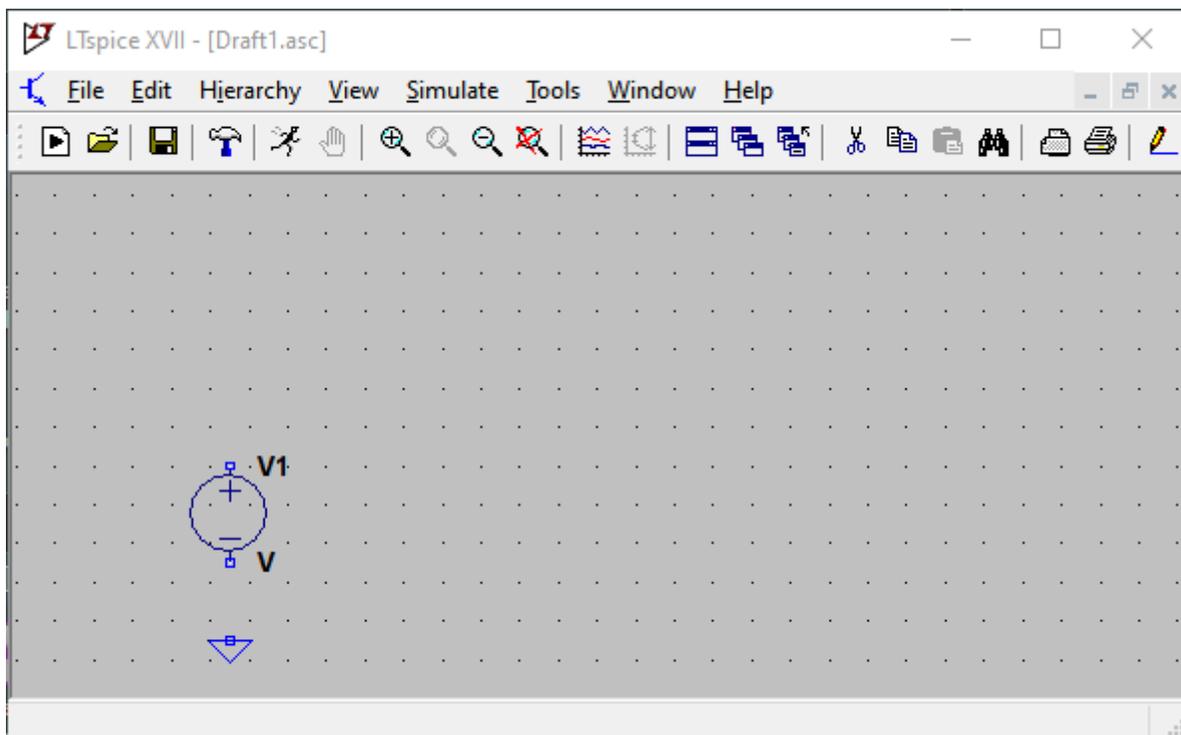
3. Insert a ground symbol as follows:
 - a. Left-Click on the **Ground** icon.
 - b. Left-Click in the schematic window to place the symbol. (Multiple copies of the symbol can be placed one-at-a-time while the symbol is selected.)
 - c. Right-Click or press the Esc key to end the function.

A ground node is required in all simulations.

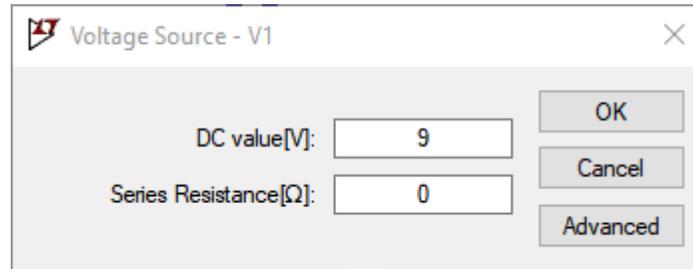
4. Add a voltage source.
 - a. Click on the **Component Menu** icon to open the component library.
 - b. Scroll right to find the voltage source.
 - c. Select voltage and then click OK.



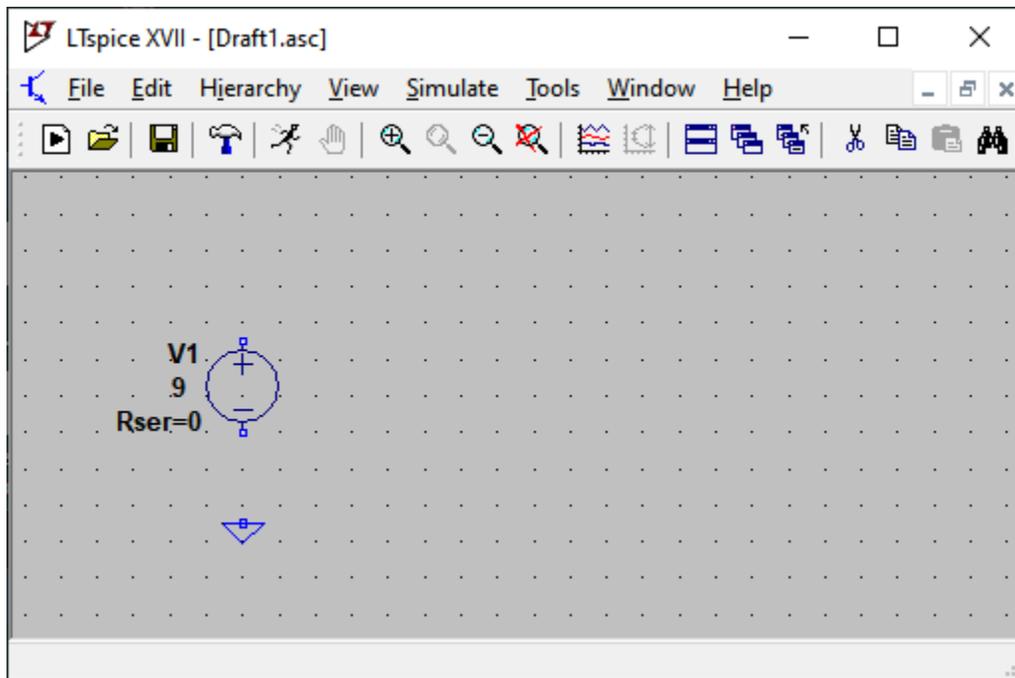
d. Place the source in the schematic window.



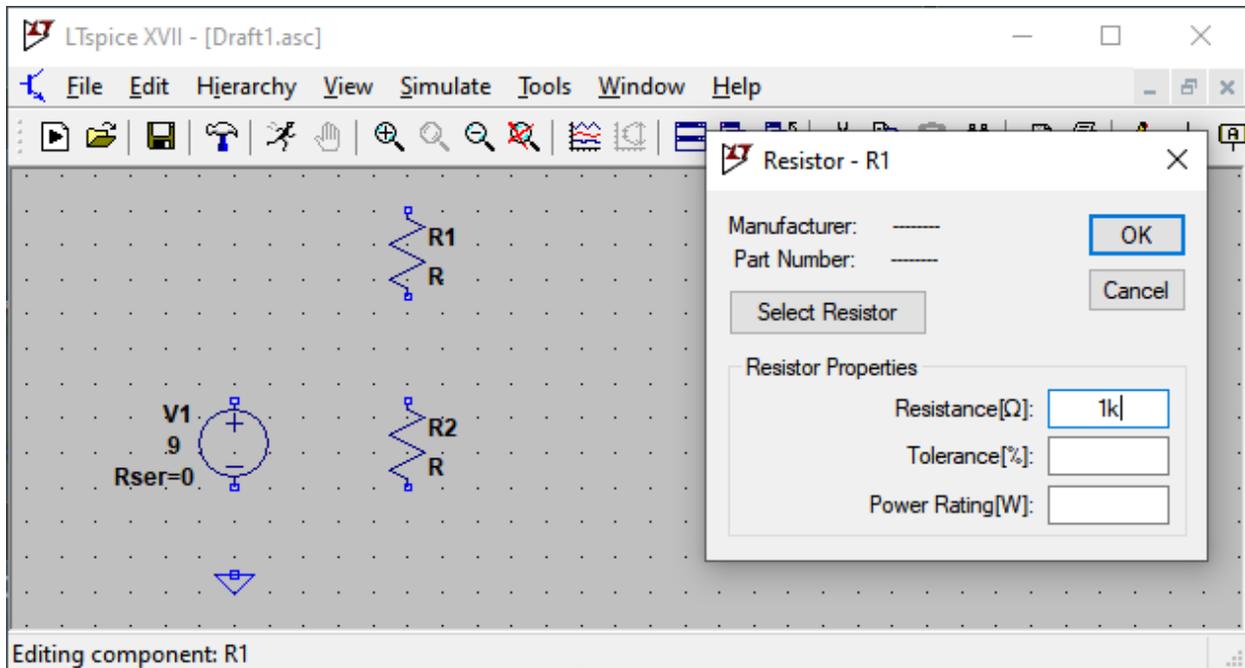
- e. Right-Click on the voltage source to open its Properties window.
- f. Set the DC value to 9V and the Series Resistance to 0Ω.



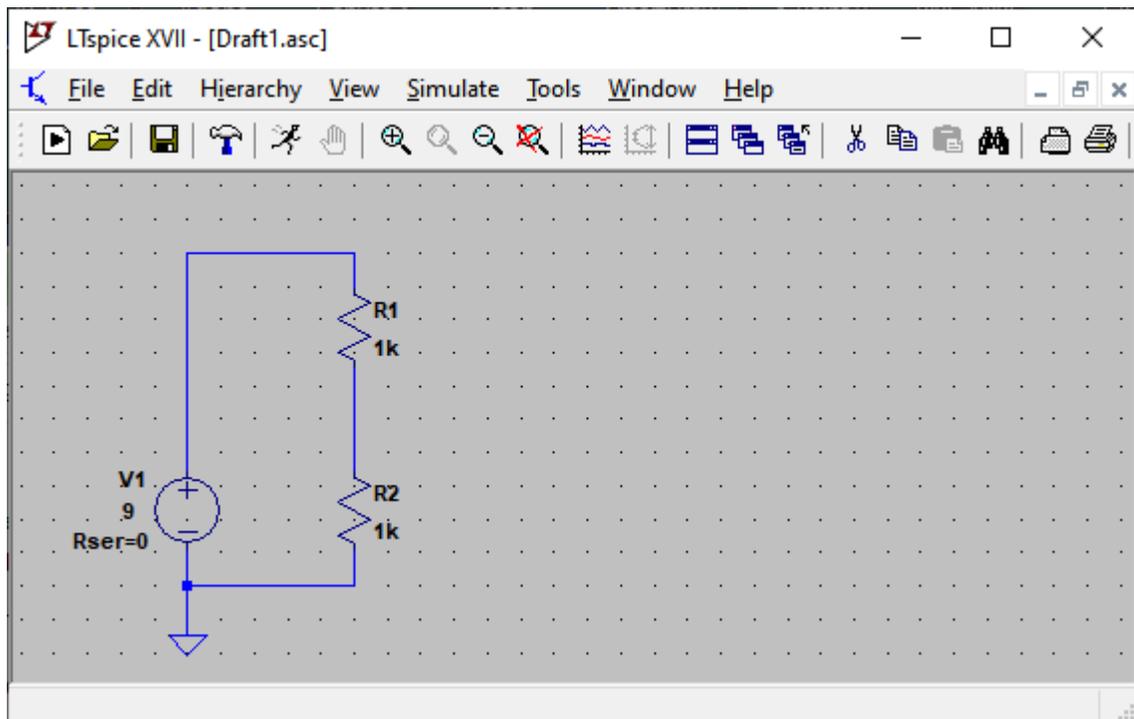
- 5. Select the Move tool.
 - a. You can click on any component or component label and move it to another location if desired.



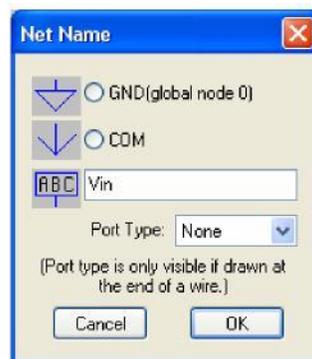
- b. Right-Click on an empty area of the page to cancel the move function.
- 6. Place two resistors in the schematic window using the Resistor tool. Right-Click on each resistor and set its value to 1kΩ.



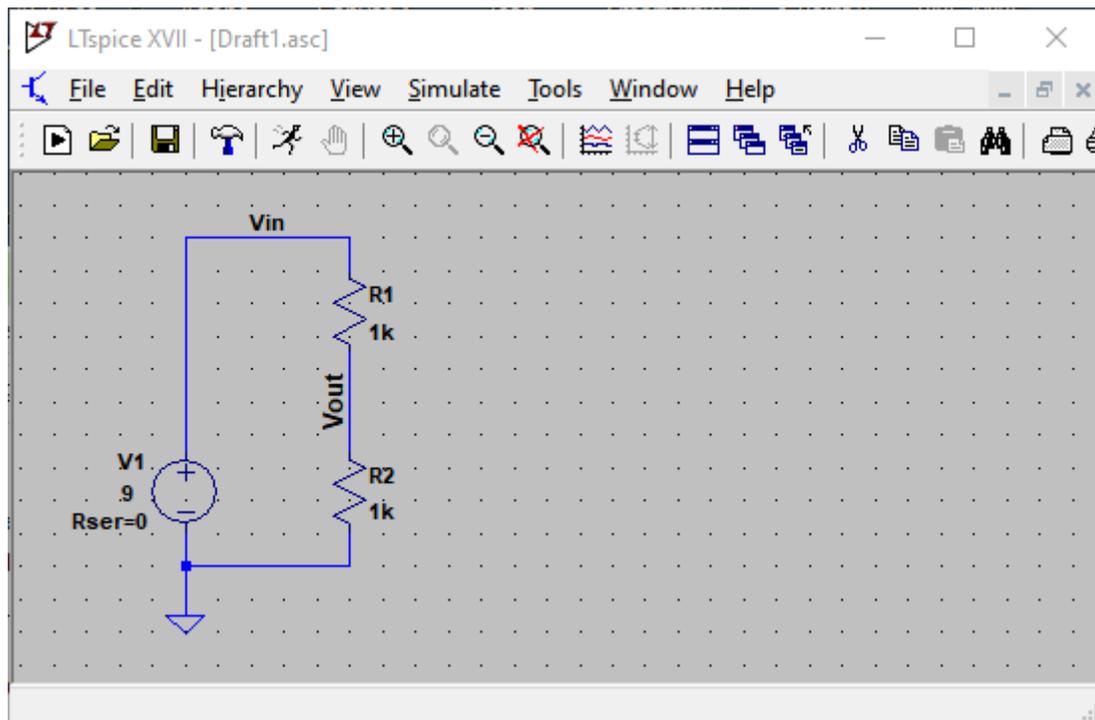
7. If you have accidentally placed any extra components, use the Cut (Delete) tool to remove them.
8. Use the Wire tool to connect the components together in the schematic.
 - a. Left-Click to start/end or to place corners.
 - b. Wires will snap to other wires and to component terminals.
 - c. Right-Click to terminate the wire tool.



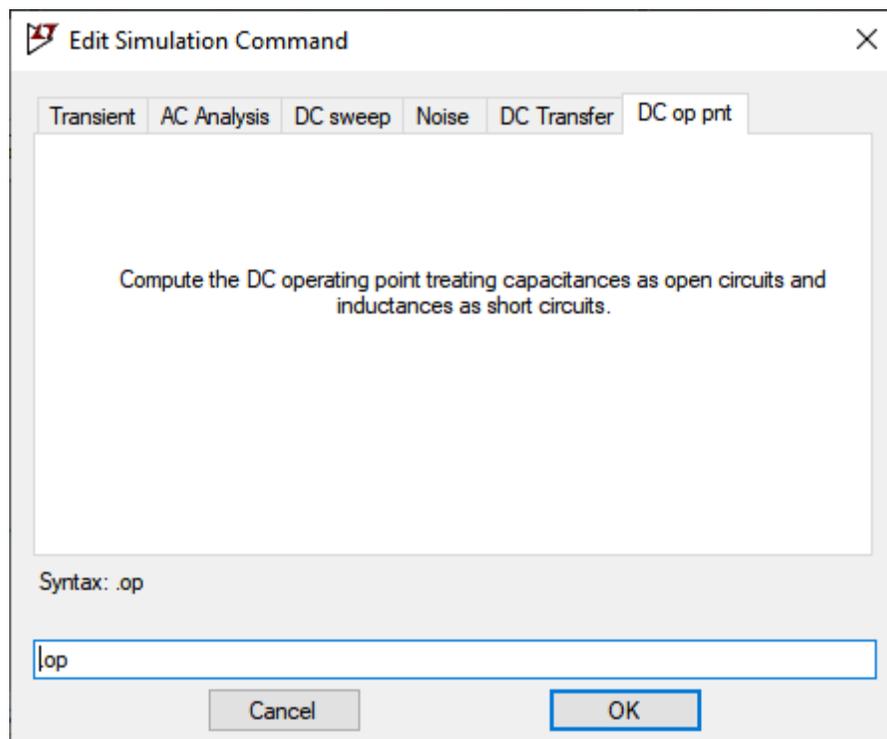
9. To move components after they are connected, use the Drag tool. This will allow you to move a component and simultaneously drag the wiring along with it.
10. Note that you can zoom in/out by scrolling the mouse wheel.
10. Save the file. Use Save As to assign a name and folder location of your choice.
11. Use the **Label Net** tool to label the nodes on the schematic. This will make it easier to interpret the simulation results.
 - a. Select the **Label Net** icon and in the Net Name dialog, enter “Vin”.
 - b. Place the net name on the wire between the voltage source and the top resistor.
 - c. Similarly label the node between the two resistors as “Vout”.



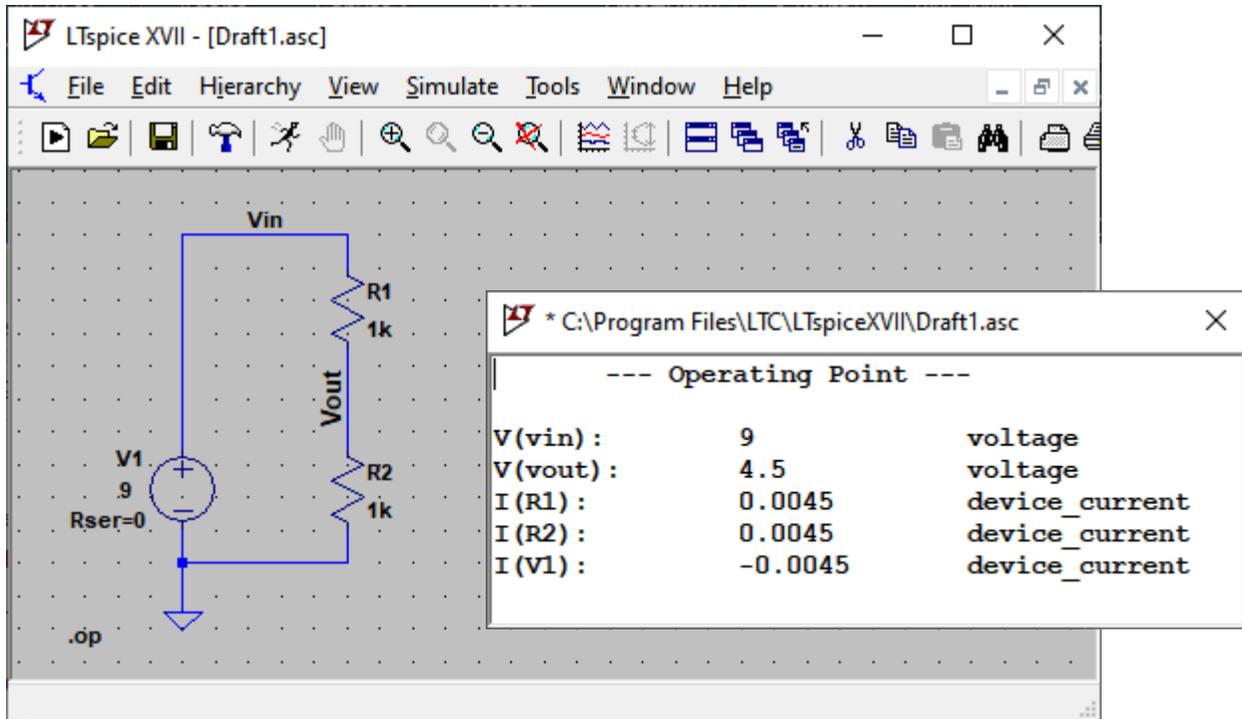
12. Your schematic should now look something like this:



13. On the menu bar, open the Edit menu and look at the keyboard shortcuts for common functions. Familiarization with this list can save you time.
14. Run the simulation. This is a DC circuit and we are interested in the steady state voltages and currents. In SPICE language this is a “DC operating point” simulation.
 - a. Select the **Run** button or choose **Simulate > Run** from the Menu bar. The Edit Simulation Command dialog will appear because this is the first run of the simulation. (To edit simulation commands after the first run, select **Simulate > Edit Simulation Cmd** from the Menu bar.
 - b. In the dialog, select the **DC op pnt** tab.
 - c. Confirm that “.op” is in the bottom text box.
 - d. Select OK.



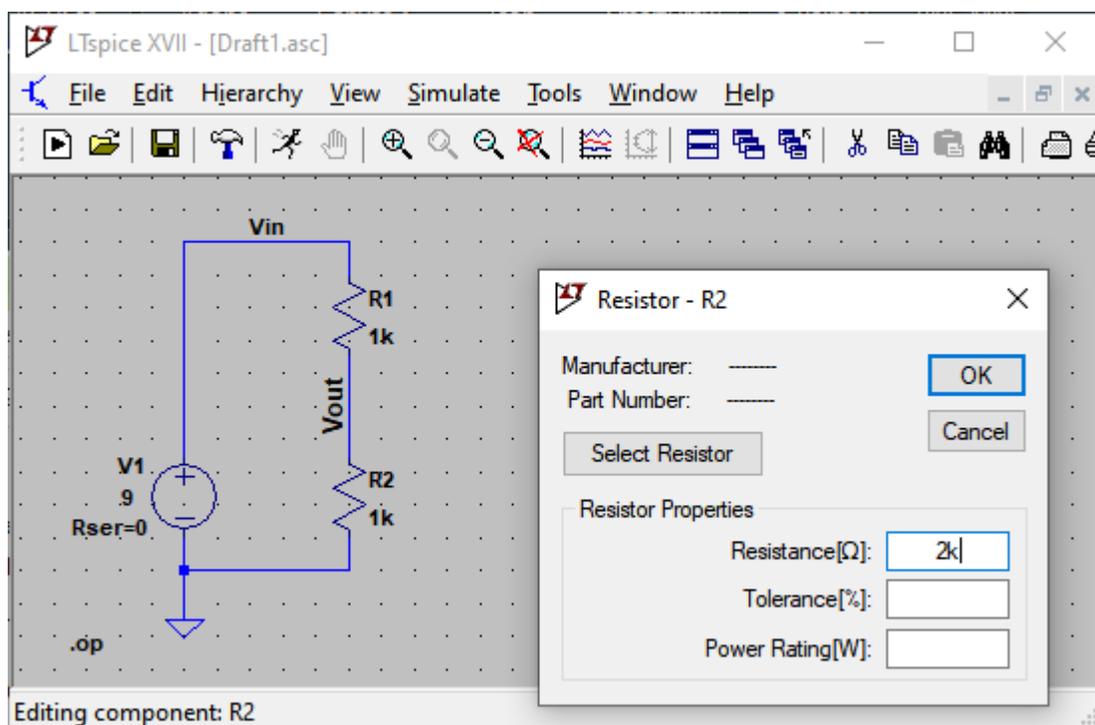
The “.op” SPICE command will be inserted into the schematic and a window with the Operating Point simulation results will appear.



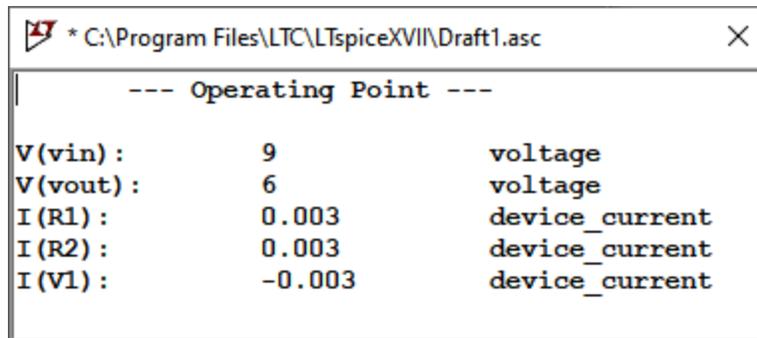
The results show that the input voltage source is 9 V, the output of the voltage divider is 4.5 V and the current through each resistor is 4.5 mA. The current through the voltage source is negative because positive current is defined as going from the + side to the – side of the element (Every element is assumed to satisfy the Passive Sign Convention in SPICE.).

15. Close the results window to return to the simulation.

16. Right-Click on R2 and change its value to 2kΩ.



17. Redo the simulation.
 - a. Click the Run button again.
 - b. The results window will change to the following:



The screenshot shows a window titled '* C:\Program Files\LTC\LTspiceXVII\Draft1.asc' with a close button (X) in the top right corner. The window content displays the following text:

```
--- Operating Point ---  
V(vin) :      9          voltage  
V(vout) :      6          voltage  
I(R1) :      0.003       device_current  
I(R2) :      0.003       device_current  
I(V1) :     -0.003       device_current
```

- c. Notice how the output voltage is now 6 V and the current has dropped to 3 mA. The ease of making such a change and conducting a new analysis shows the power of circuit simulation over actual breadboarding during the initial stages of circuit design.
 - d. Close the results window.
18. Hover the mouse over R2 without selecting anything. The message area at the bottom of the schematic lists the steady-state current through the component (3 mA) and the power being dissipated by the component (18 mW).
19. Now find the current and power associated with R1. This is a convenient way to determine the necessary power rating when you prepare to purchase components to build the hardware version of the circuit.