Introduction to LTspice

Lab 1

Let us start: What is SPICE?

- SPICE stands for Simulation Program with Integrated Circuit Emphasis.
- SPICE is as close to a universally available package for doing numerical network analysis as one can find.

Getting LTspice

- On the handout, is the procedure to access LTspice on ASU’s computers.

- You can download LTspice from the EEE202 Lab Blackboard website or www.linear.com/LTspice

Accessing LTspice Today

- Turn on the computer.

- Use your ASURITE ID and your own Password to log on.

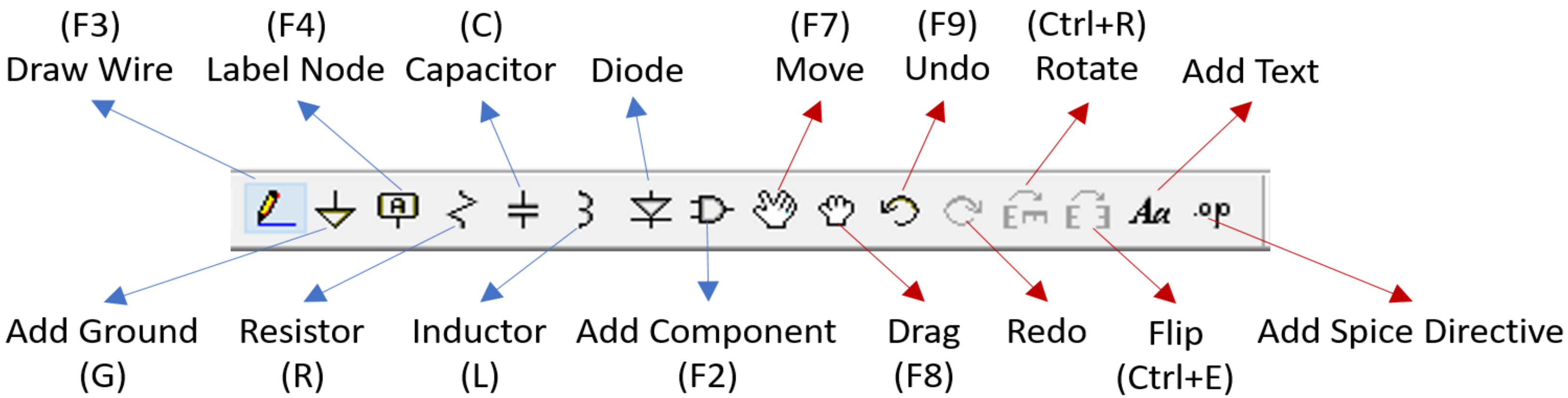
- Click on “Start”, then choose LTspice IV from the list of programs. Alternatively, you can search for “LTspice” in the search bar.

Starting a new schematic

- Go to “File” -> “New Schematic” or use the “New Schematic” button on the toolbar.

Placing components

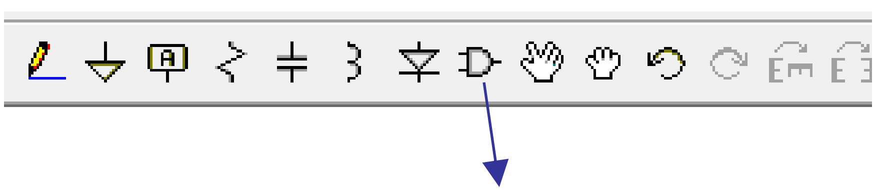
- Go to the Edit menu and click on the component you want to place.
- You can also use the buttons on the toolbar to place components.



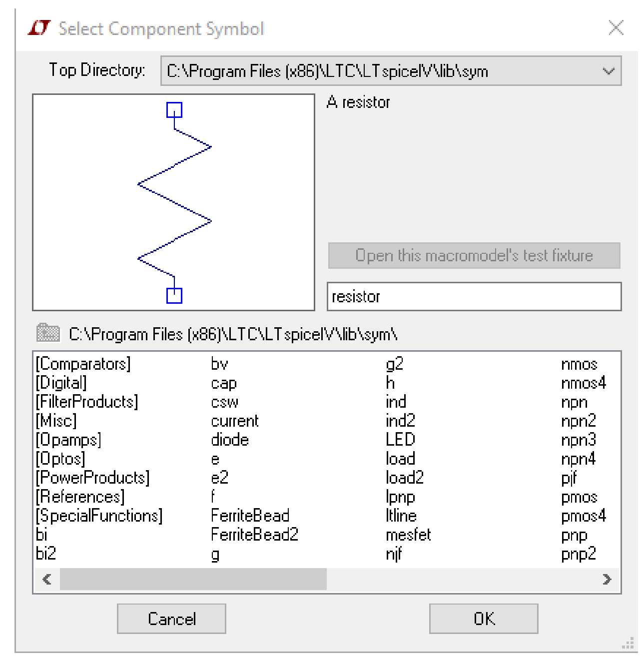
- After placing a component, you can cancel your selection by clicking the right mouse button.

Placing Components (contd.)

- You can also search for a component by clicking on the "Add component" button or pressing 'F2'.

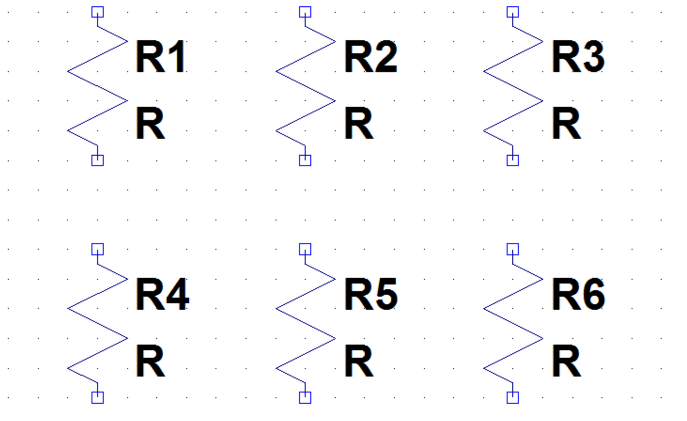


Add Component
(F2)



Placing Components (contd.)

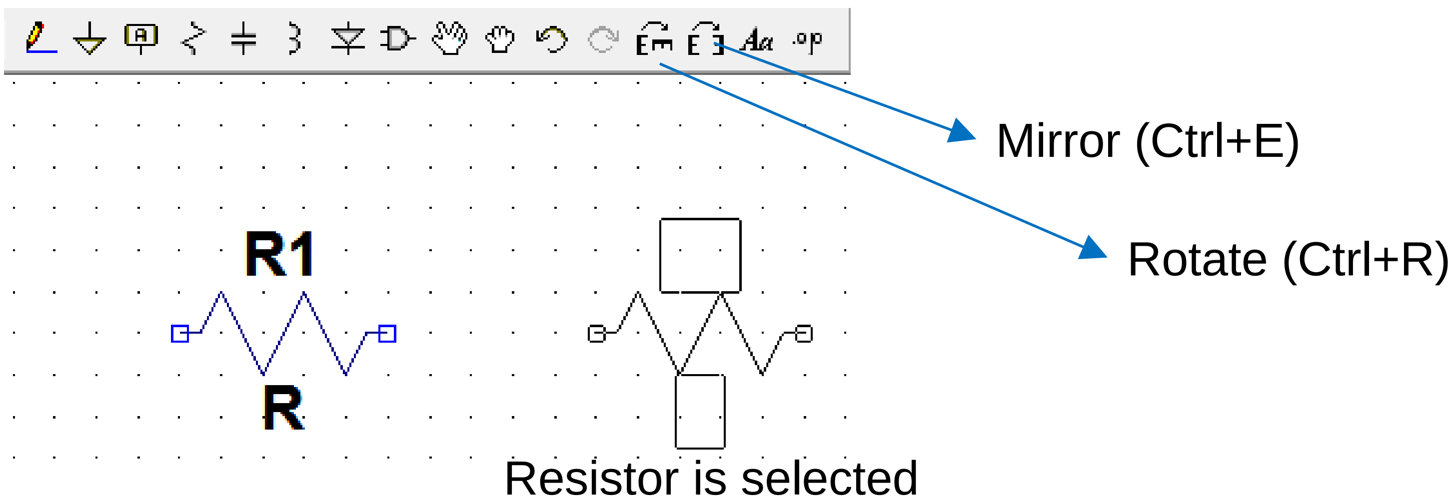
- Place about a half dozen resistors down by moving the mouse and clicking on the left mouse button



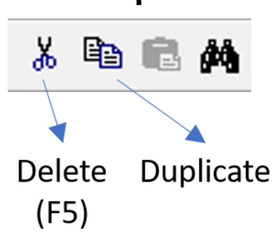
- When done, click the right mouse button or you will keep placing resistors

Manipulating Components

- To move a component, first click on the ‘Move’ button or press ‘F7’. Then select the component and move it around.
- While the component is selected, it can be rotated (Ctrl+R) or mirrored (Ctrl+E).



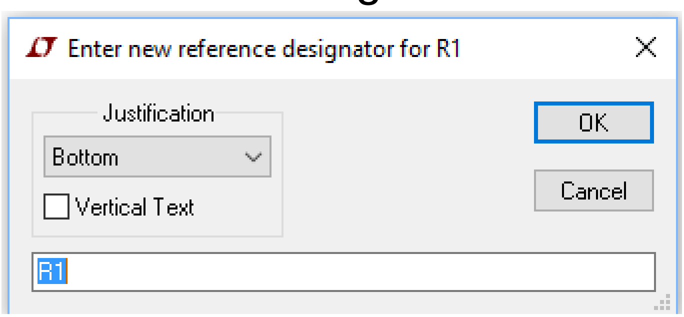
- To delete a component and press the ‘Delete’ key or ‘F5’ on the keyboard.

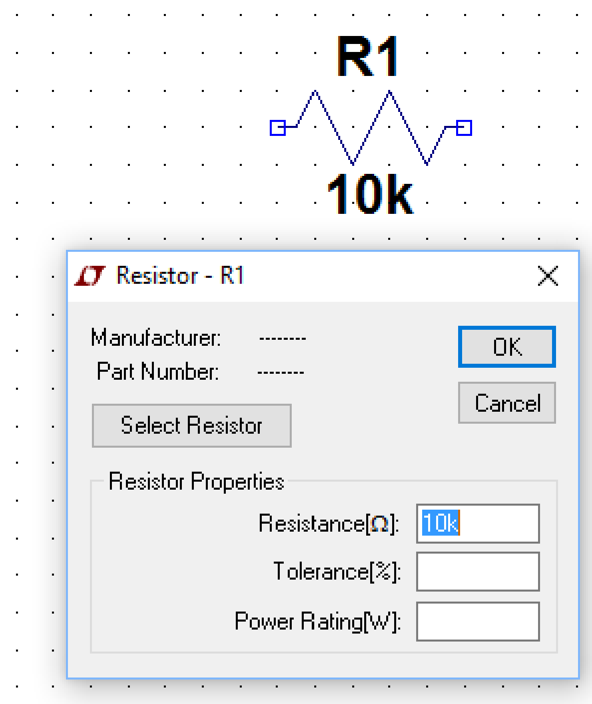


Manipulating Components (contd.)

- Right click on a resistor to see its attributes. New values can be entered in the attributes window.

Right click on the name of the resistor to change its name.

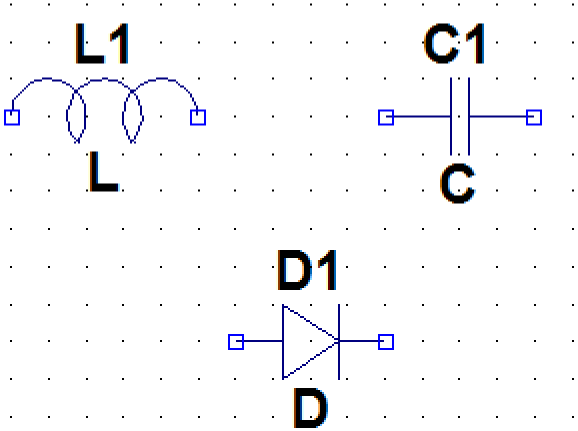




Right click on the resistor to change its attributes.

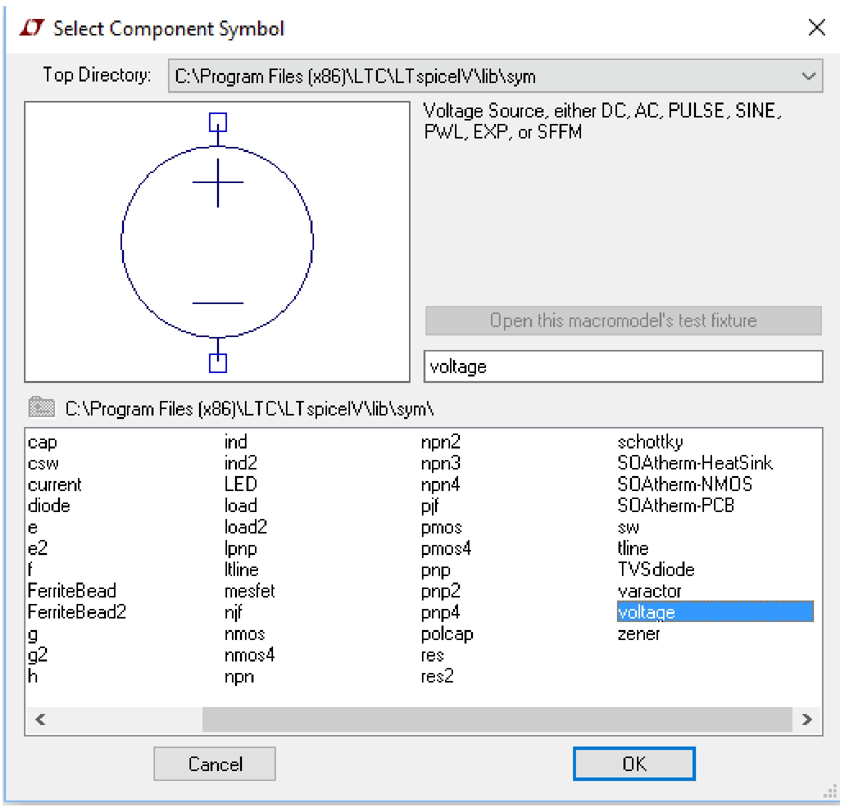
Other Components

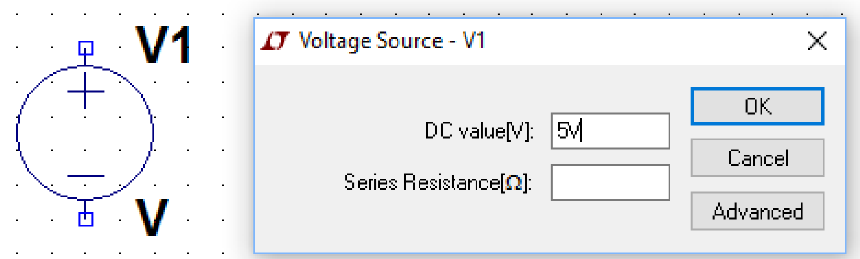
- LTspice includes various other circuit components such as:
  - Inductors (L), capacitors (C) , diodes



Sources

- Most circuits in LTspice require a source of some kind.
- Click ‘Add component’ button or ‘F2’. Then select ‘voltage’.

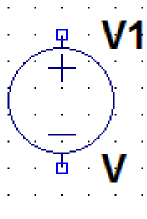


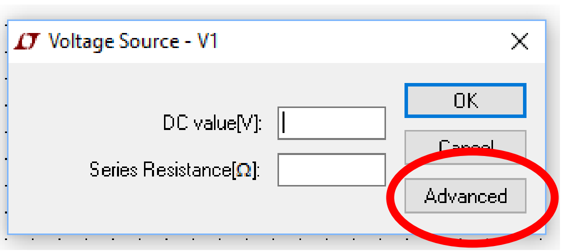


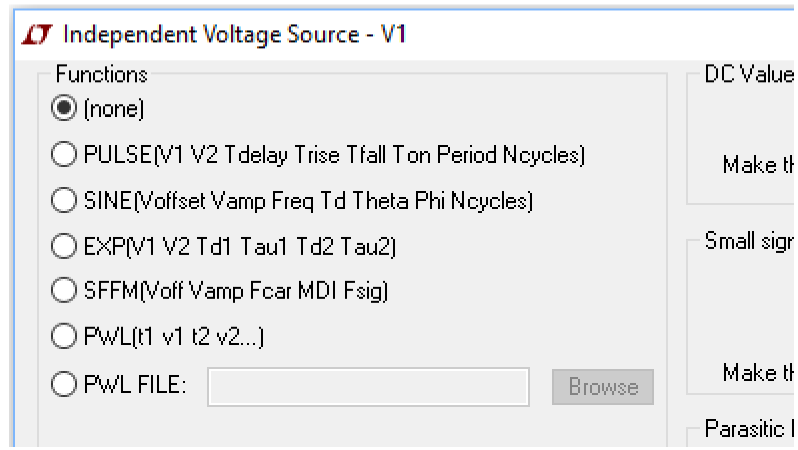
Right click on the voltage source to change the voltage. Note that this is a DC voltage source.

Sources (contd.)

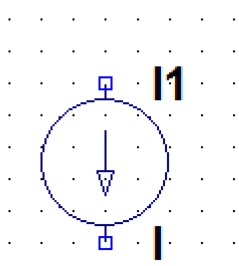
- AC sources can be obtained after clicking ‘Advanced’ in the attributes window.







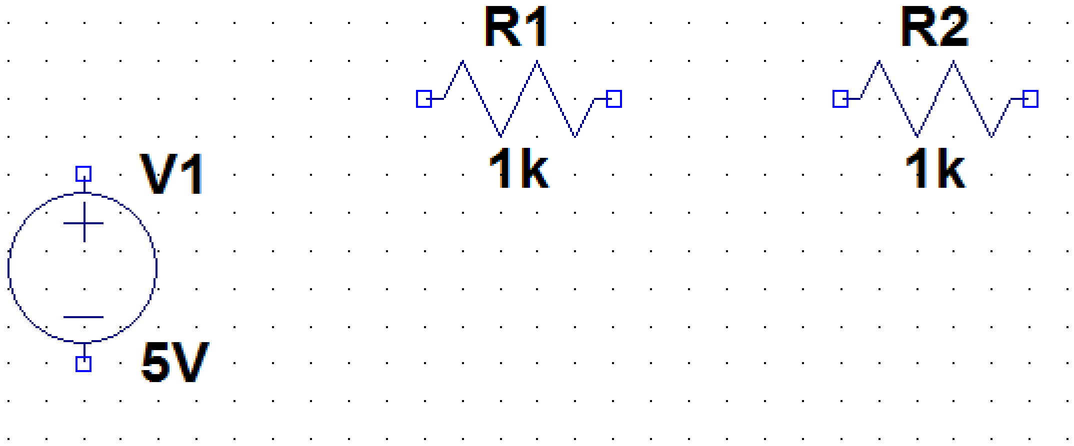
Current sources have the same features just like voltage sources.



Note: the arrow points in the direction of conventionally flowing current.

Basic circuits

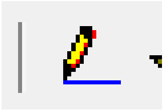
- Build a voltage divider in Ltspice.
- Step 1: Place the components as shown and enter the given values in the schematic.

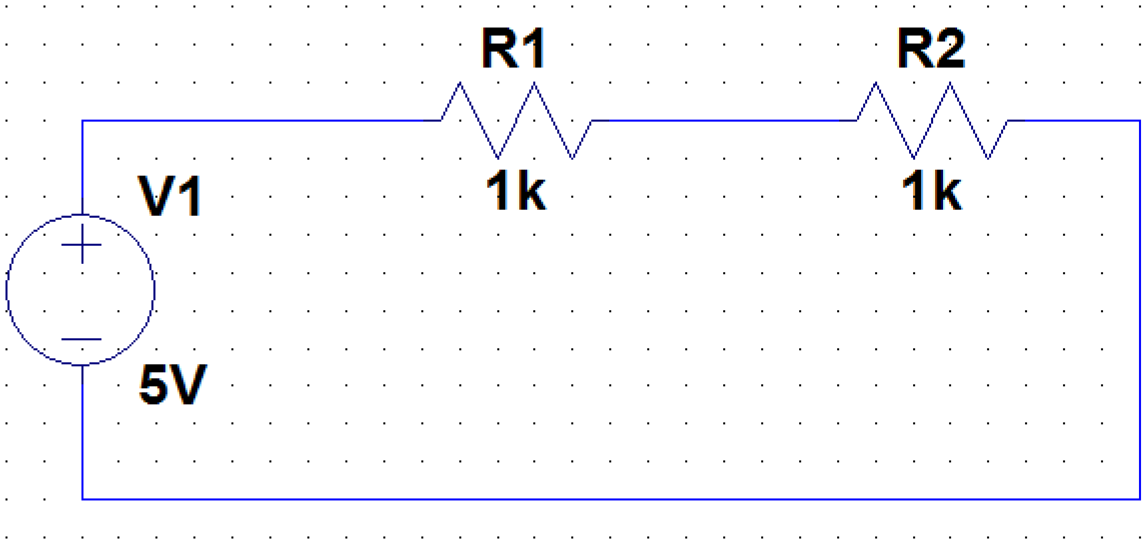


  - To fit the components within the window, in LTspice, press Space on your keyboard.

Wiring components

- Step 2: Wire the components together
  - Click the ‘Wire’ button or press ‘F3’

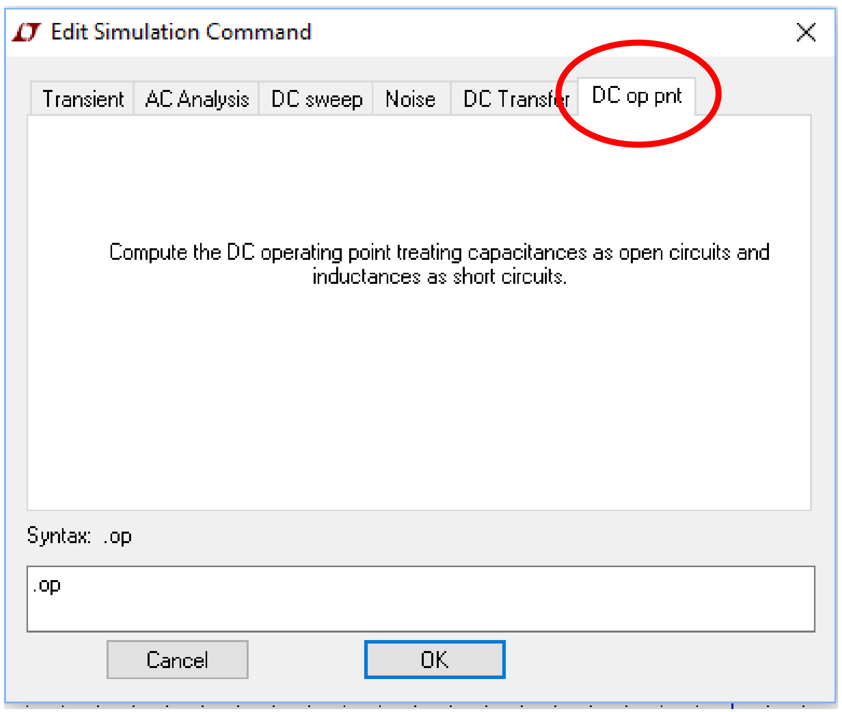


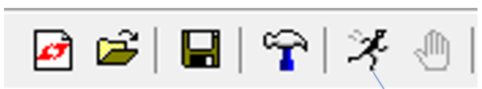


Your circuit should look like this.

Simulating Circuits on LTspice

- Before simulating the circuit, save the schematic.
- To simulate the circuit, click on the *Simulate* menu and click *Run* or click on the ‘Run’ button.



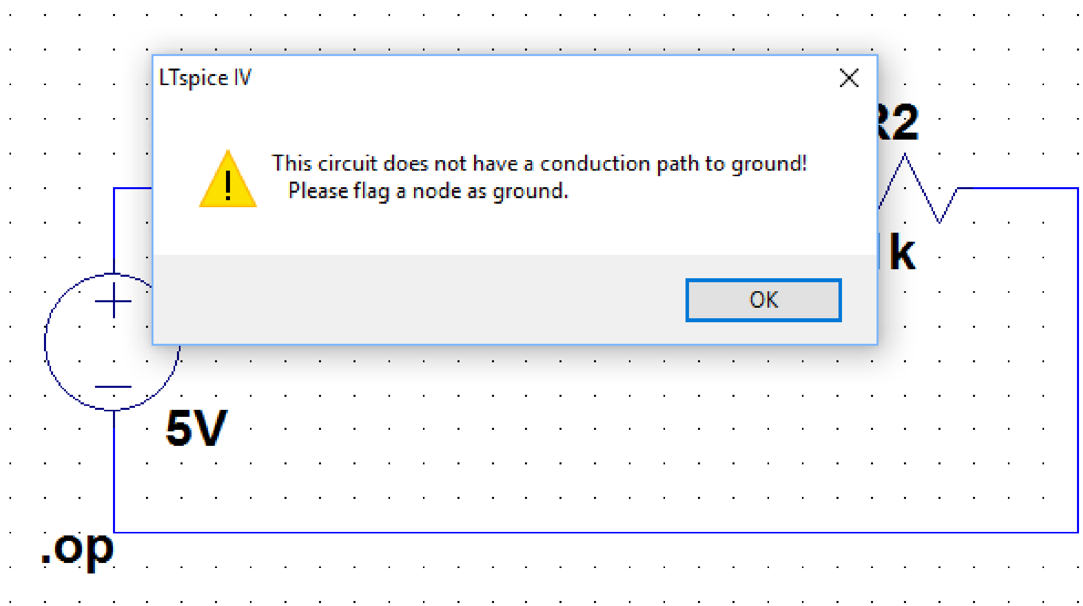


To simulate a DC circuit choose *DC op pnt*

An operating point analysis provides a DC analysis of the circuit.

Simulation Error

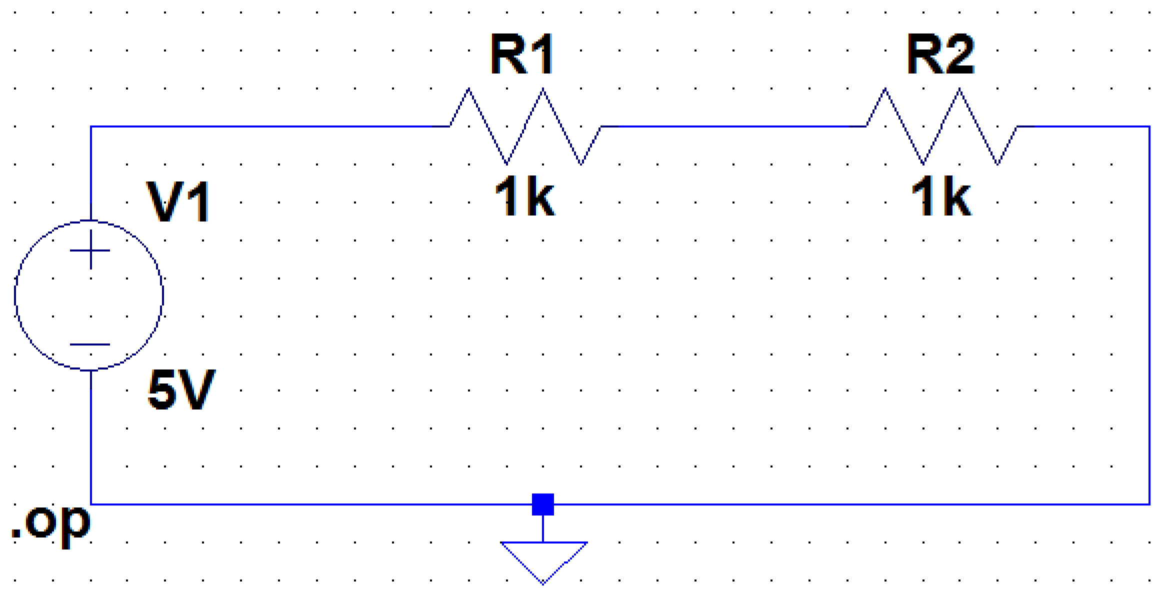
You probably got an error as shown below:



Every circuit in LTspice MUST have a ground!

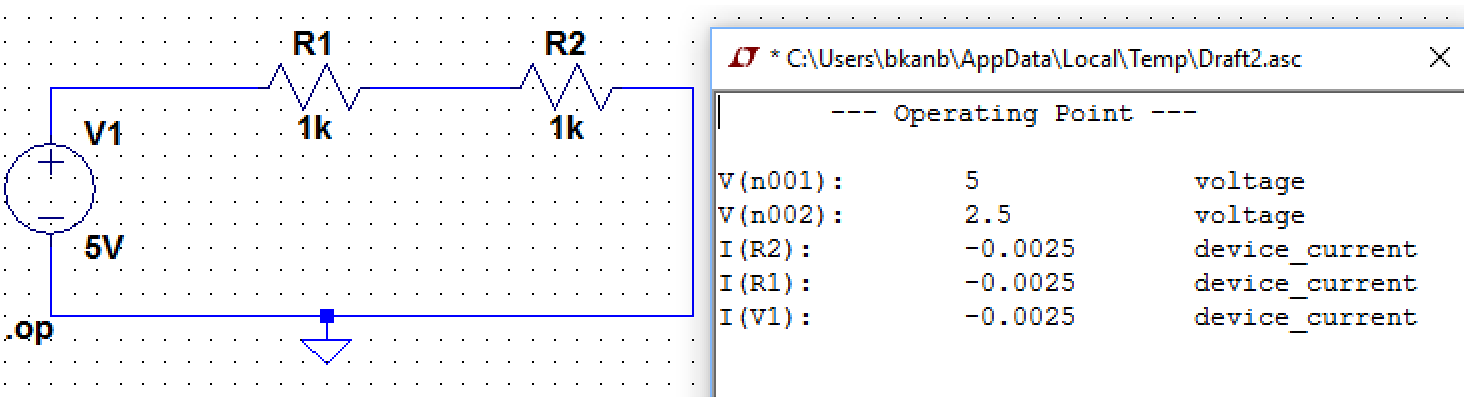
Simulating Circuits on LTspice

- Connect a ground to the circuit as shown and simulate your circuit as described earlier. File->Save



Simulation Result

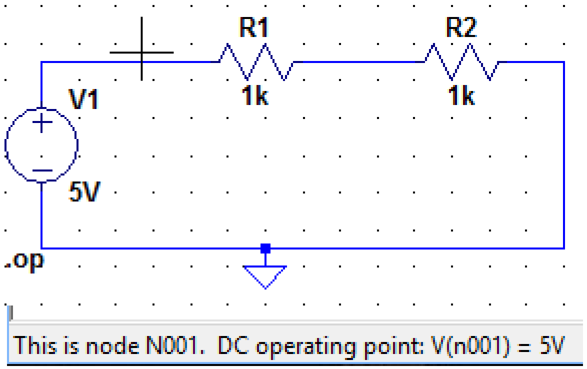
- You should see something similar:



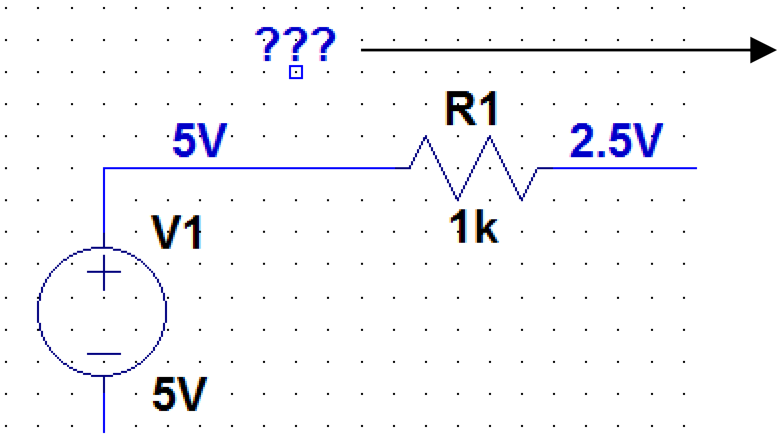
  – Simulation is complete, but there are no meaningful results displayed.

Displaying Results

- After the simulation is complete, move the cursor over wires and components to see information in the status bar.



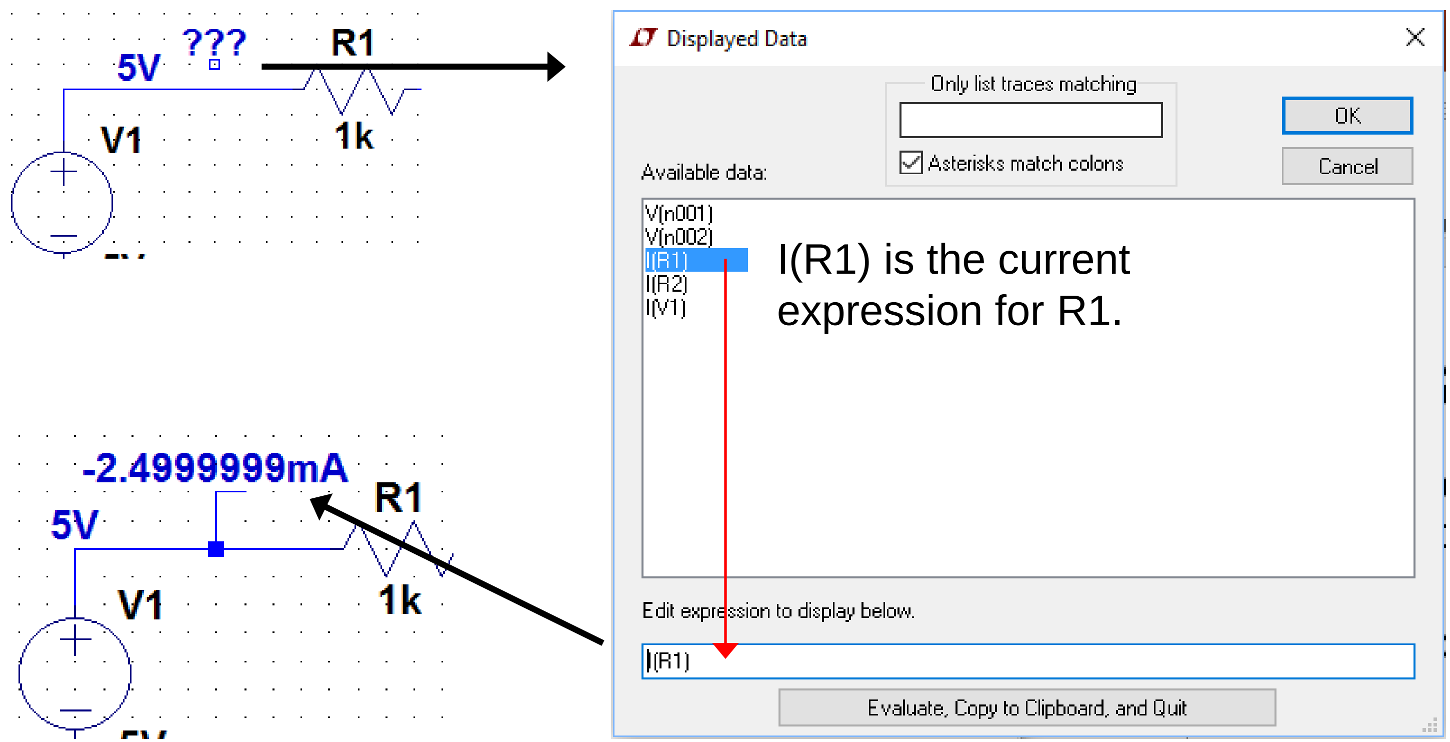
- To display the voltage values at the nodes, left-click on the wire for that node and place the empty label box on the wire. The label will display the voltage.



If the box isn't connected to the wire, it won't display the voltage value.

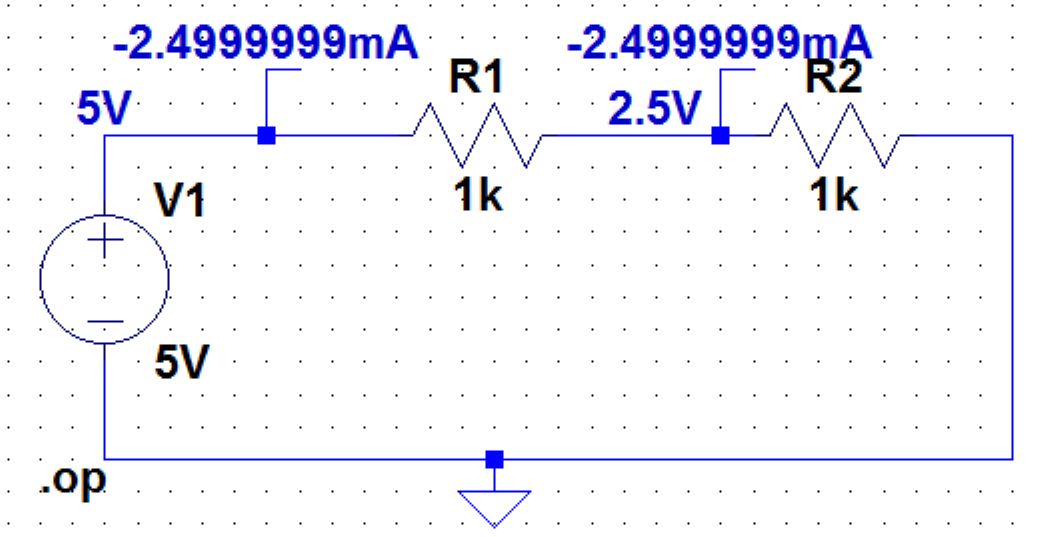
Displaying Results

- To display the value of the current passing through a component, left-click on the wire that is connected to the component and place the label box near the component.
- Right-click on the label '???' and *Displayed Data* window will open.



Displaying Results

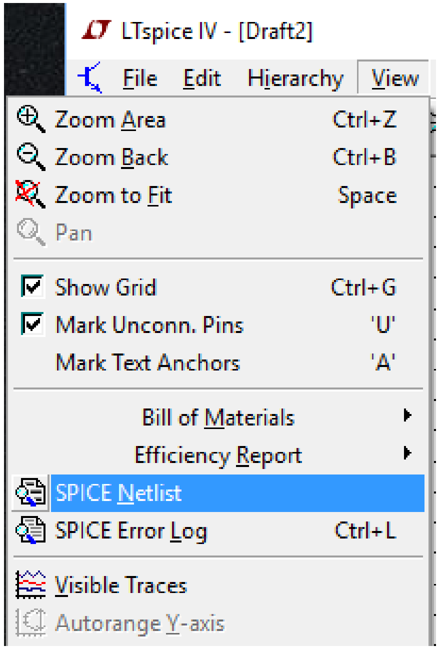
- All the values should be displayed on the circuit clearly.



Note that the current values are negative.

FYI: Netlists

- When you simulate your circuit, the internal software automatically generates something called a netlist that drives the actual number crunching
  - The Netlist describes your circuit textually
- **<u>In the View menu, click on "SPICE Netlist"</u>**



FYI: Understanding the netlist

- Below is a sample netlist for the circuit you built previously

```
V1 N001 0 5v
R1 N002 N001 1k
R2 0 N002 1k
```

- There are four columns:
  - The first column denotes the circuit component.
  - The second and third columns specify the nodes between which the component is connected.
  - The last column provides the value of the component

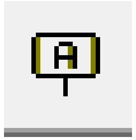
FYI: Examine Your Netlist

- For this circuit, the Netlist is easy to deduce. You can figure out which point on the circuit is denoted by nodes $N_0001, $N_0002 and 0.

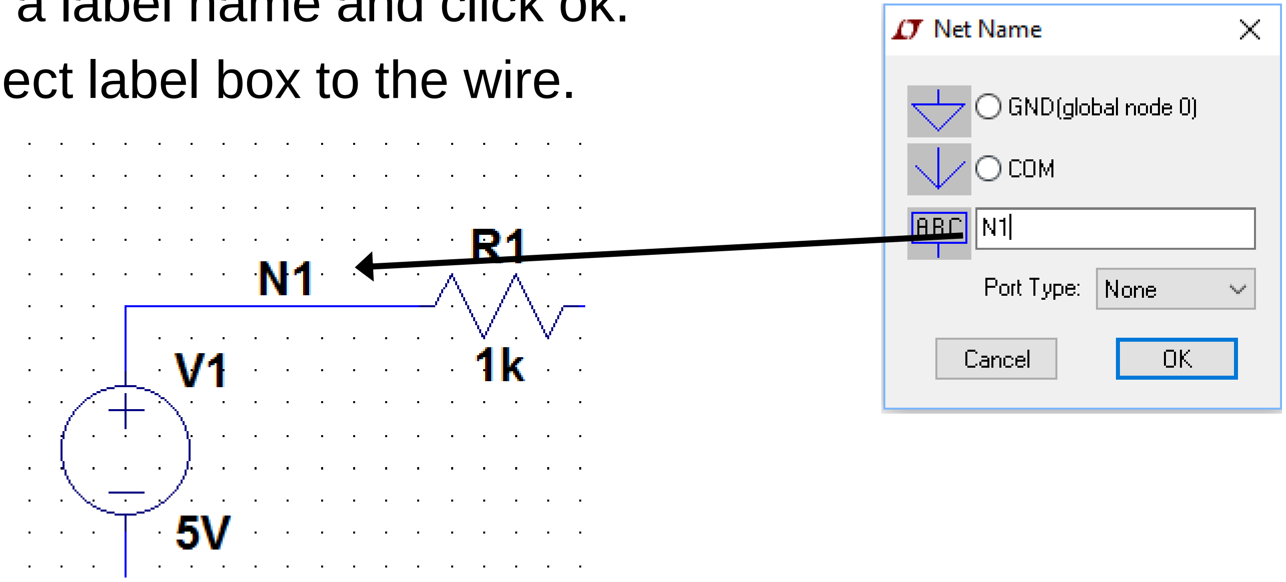
- However, for a more complex circuit, deducing the Netlist can be very hard. Also, in future simulations, plotting the correct waveform can also prove challenging.

Naming the Nodes

- To make the task of understanding Netlists, and especially plotting waveforms in the future easier, we can name the nodes in LTspice.
- <u>Click ‘Label Net’ button or press ‘F4’</u>
  - *Net Name* pop up window will open.
  - Enter a label name and click ok.
  - Connect label box to the wire.

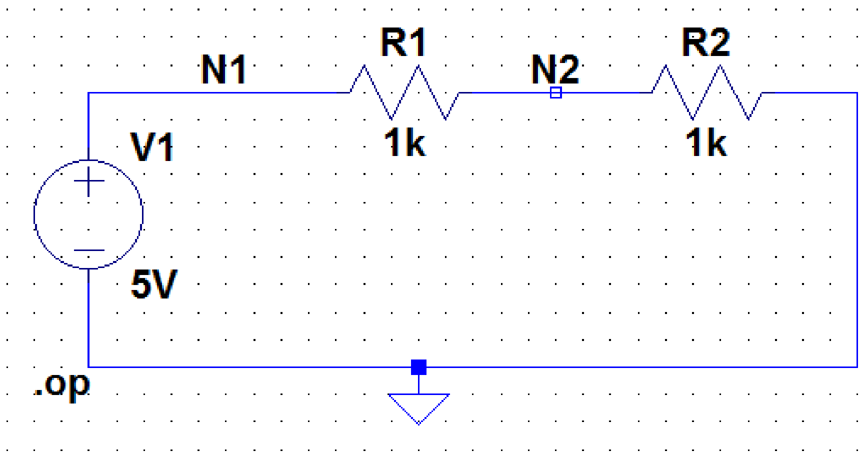


Label net button



FYI: Naming the nodes

- Once you name all the nodes, your circuit should look like this:



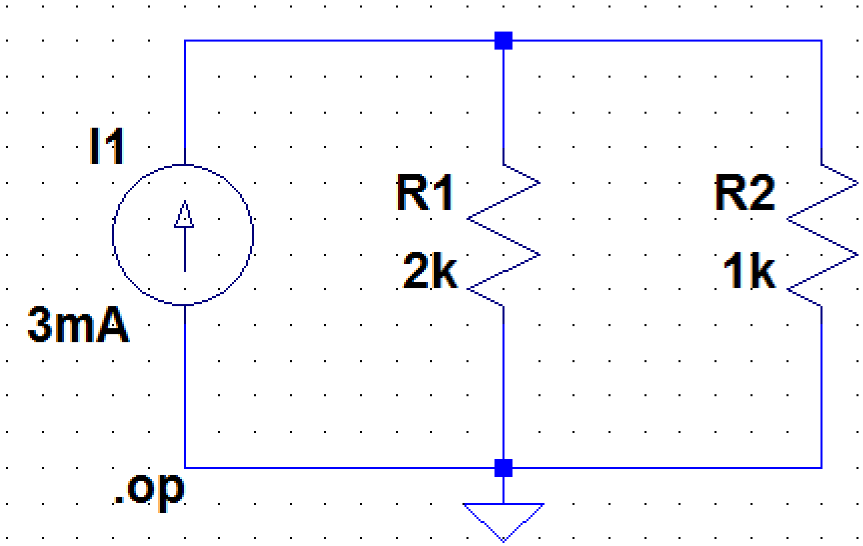
- Your new netlist should look like this:

* Schematics Netlist *

```
V1 N1 0 5v
R1 N2 N1 1k
R2 0 N2 1k
```

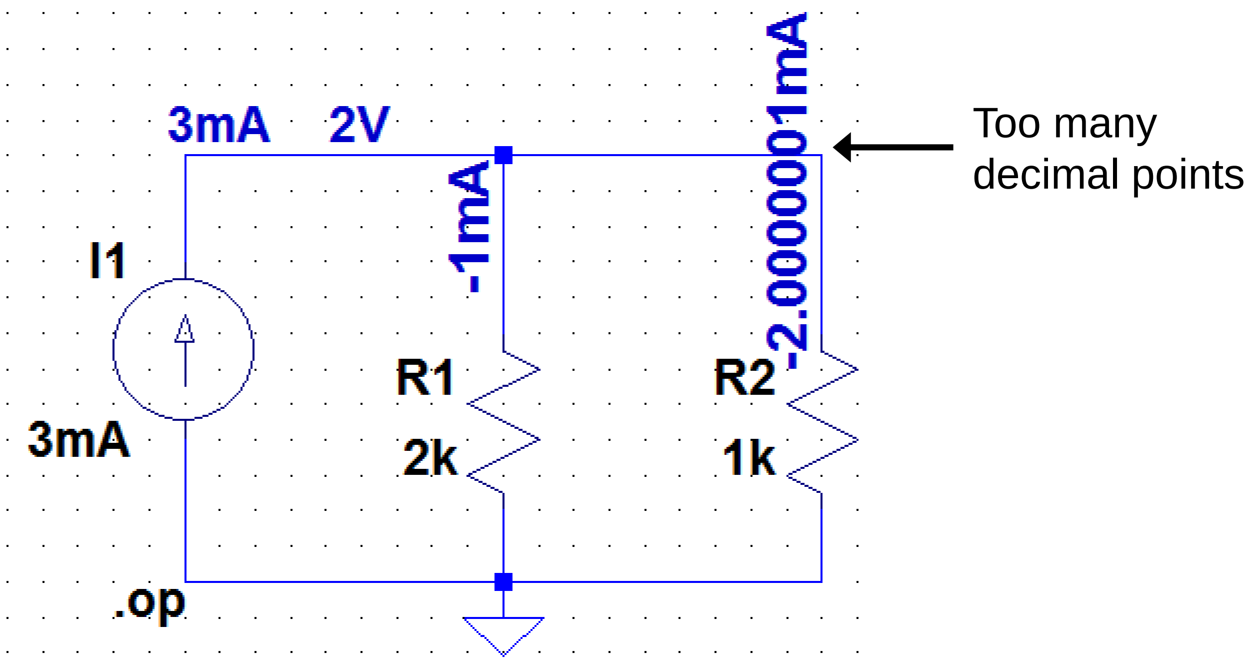
Build another circuit

- Clear the voltage divider and build a current divider as shown on LTspice:
- *Add Component* (F2) -> current (this will get a current source)
- File->Save



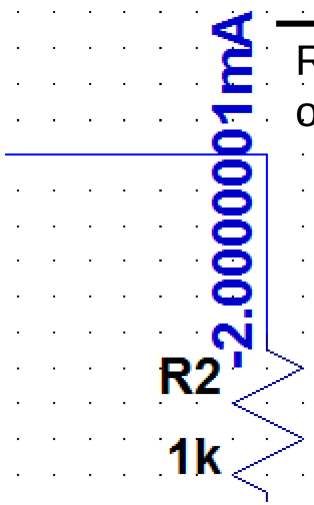
Simulate the circuit

- Simulate the circuit and display only currents in the circuit. You should end up with something similar to this:

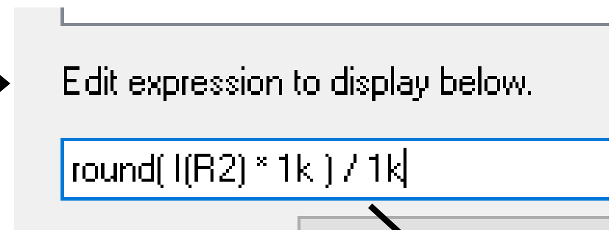


Rounding values

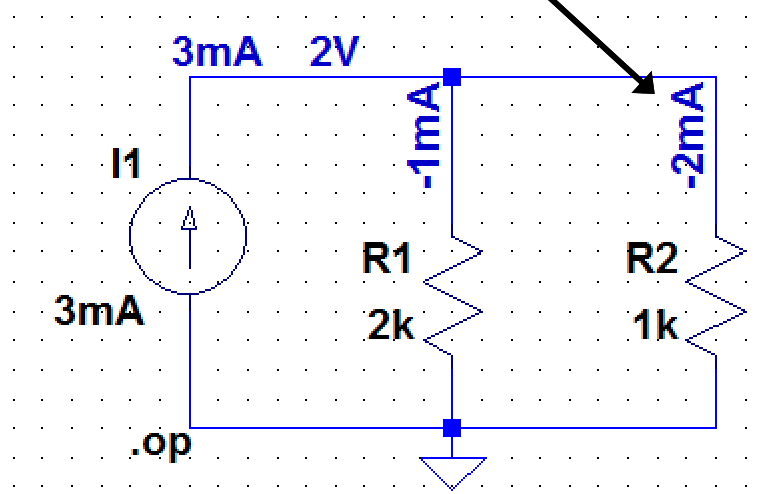
- Sometimes voltage and current labels might show too many decimal points for our purposes. It makes the circuit look crowded. To fix this issue, the voltage/current expression need to be rounded.



Right-click on the value

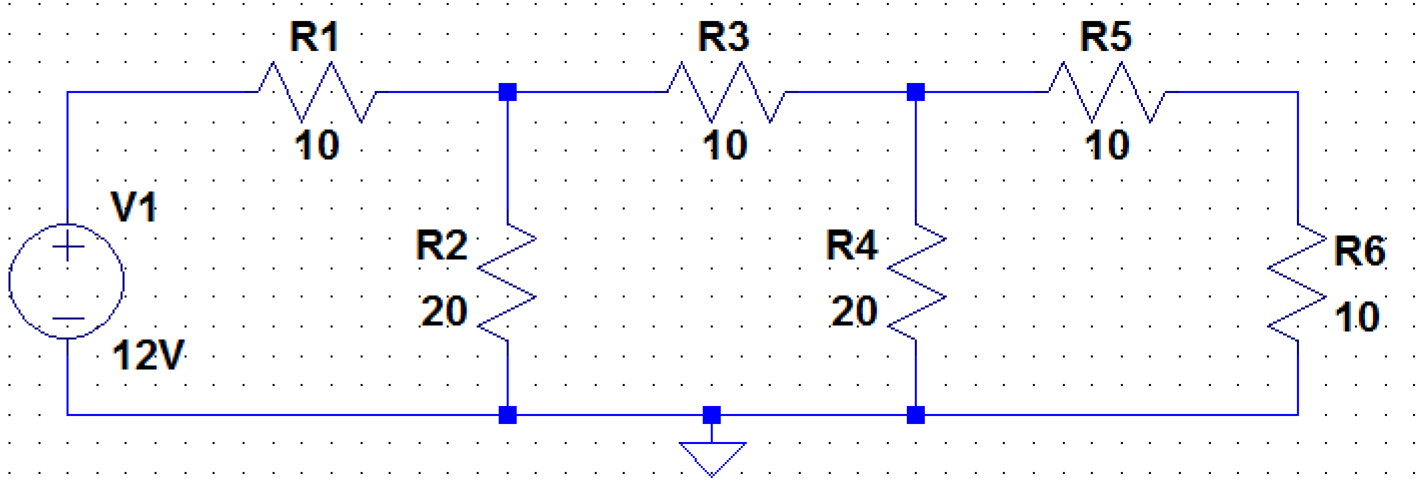


Multiplying with 1000 reduces the number of decimal points to 3.



Circuit Analysis

- Consider the following circuit. Analyze the circuit using techniques learnt in class. Find the current in and voltage across R6
- Show all your work in your data sheet. Write neatly and legibly



Simulate

- Build and simulate the circuit shown in the previous slide. Does LTspice give you the same answers that you calculated?