

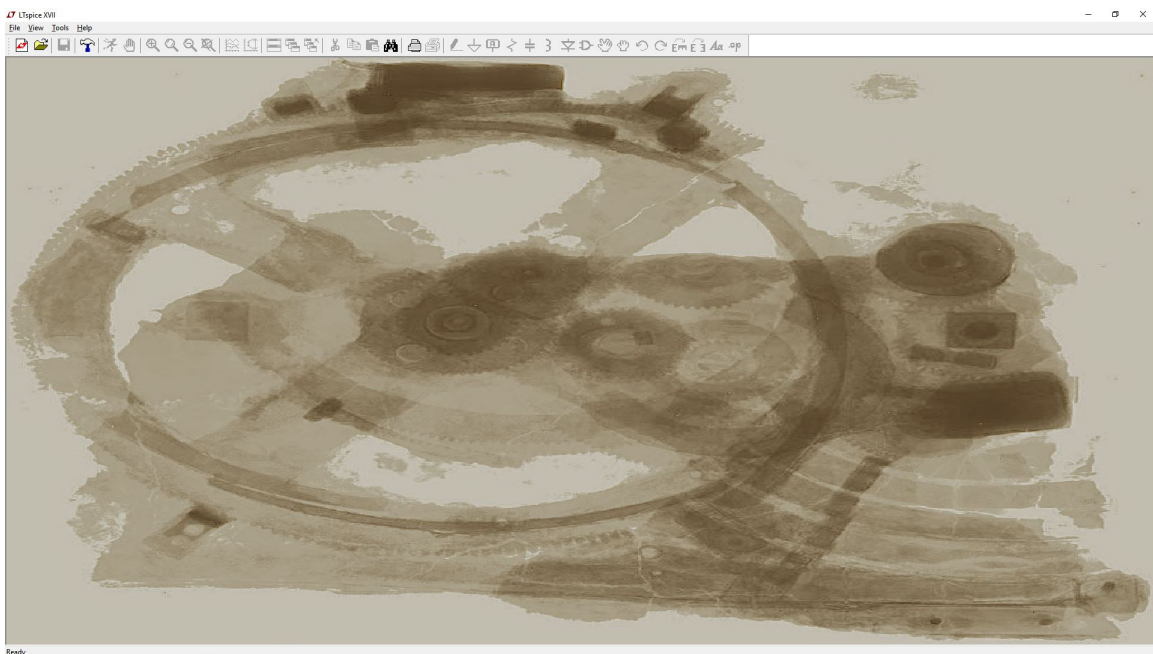
LTspice Basic Simulation Exercises


Alfred Festus Davidson
alfy7.github.io

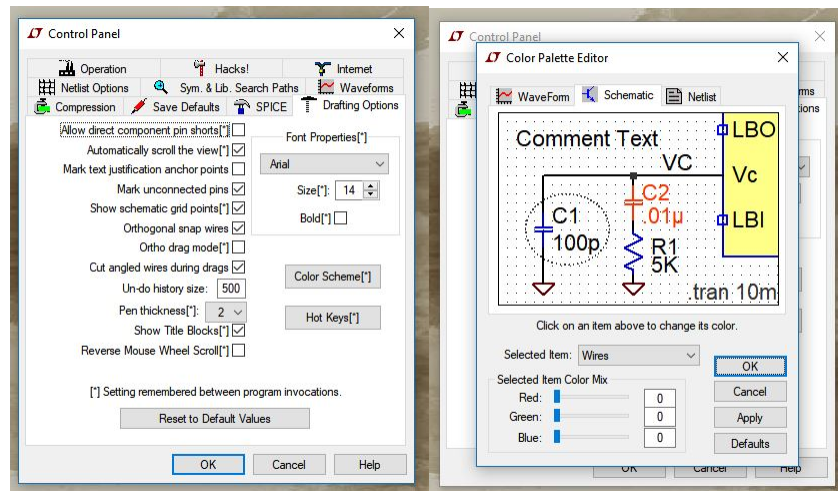
November 30, 2017


R-2R Ladder

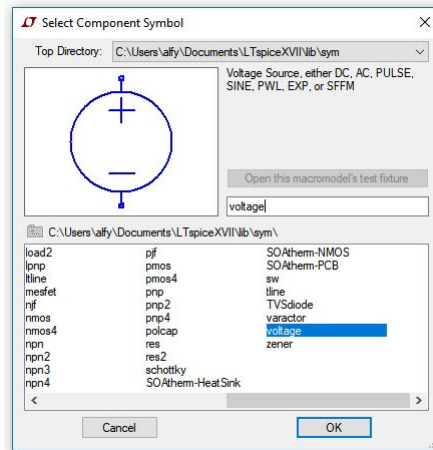
- First, download and install LTspice. Once you open it, you should see a screen like this. I will skip over the basic stuff like icon names/shortcuts/title bar/menu bar for now and revisit them as and when needed.







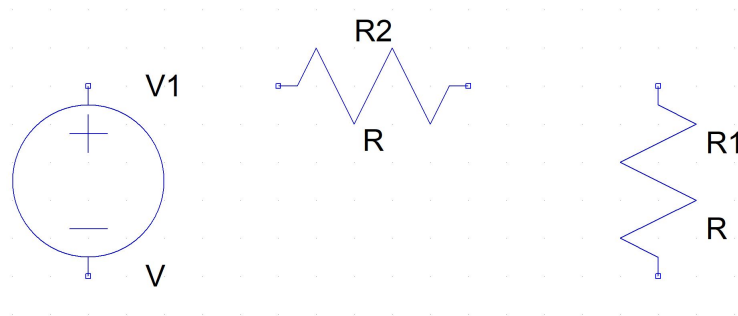
- Lets change the default color options and preferences. Go to *Tools->Control Panel* or click the control panel icon . Go to *Drafting Options* and set preferences as desired. Click on *Color Scheme* and set them as desired.





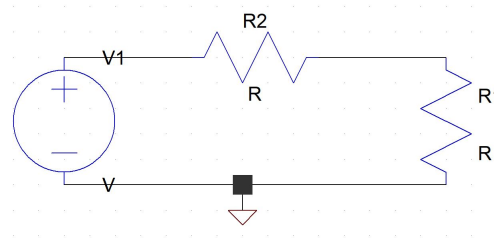
- Create a new Schematic using *Ctrl+N* or *File->New*. The usual shortcuts and icons work for opening and saving schematics.
- First, lets add a voltage source. Click the *component* icon  and type *voltage*



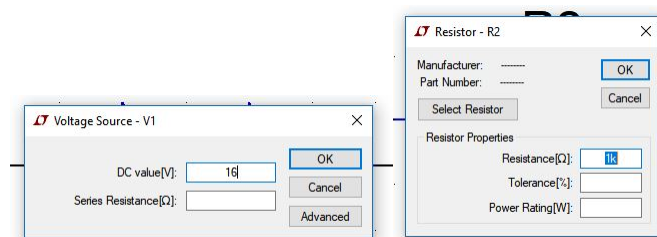
- Now click anywhere in the schematic to add the voltage source. Similarly, to add the resistors, either use the shortcut key *R* or use the resistor icon . To rotate the resistor, use the rotate icon  or the shortcut *Ctrl+R*. Try to place the components like the way shown below. Right-click to get back the original cursor. The mirror tool  can be used to mirror/flip the selected component. Use the undo/redo tools  to make corrections.




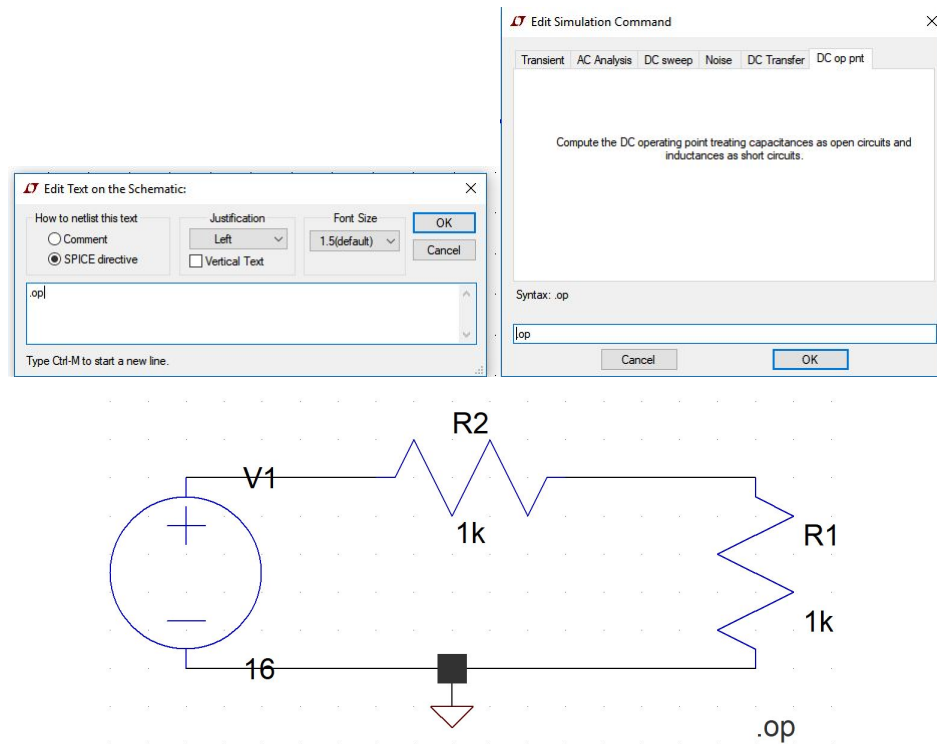
- Use the wire tool  to make connections between the components. All LTSpice circuits need a ground connection. Add a ground using the shortcut key *G* or the icon .




- Now lets add values to the components. Right-click the voltage source and set it to be 16V DC. Similarly, set the resistors to be 1k each. The dimensions/units are inferred by LTSpice from context.



- Now, to finally simulate this circuit. The only features of interest in this circuit are the nodal voltages and currents through each device. So we do a *DC Operating Point Analysis*. We direct LTspice to do so by using a *SPICE Directive*. Click on  and type `.op` to direct LTspice to find out the DC operating point of the circuit. Alternatively, you can also go to *Simulate->Edit Simulation Command* and click *DC Operating Point*. Click anywhere on the schematic to add the directive.



- Click *Simulate->Run* or use the run icon  to run the simulation.

```

--- Operating Point ---
V(n001):      16      voltage
V(n002):       8      voltage
I(R2):       -0.008    device_current
I(R1):        0.008    device_current
I(V1):       -0.008    device_current

```


- Here, we see the voltages at node *n001* and *n002*, and the currents through *R1*, *R2* and the voltage source *V1*. It is evident here that the node on the top left is *n001* and the node on the top right is *n002*. The signs of current are due to the way LTspice has interpreted the circuit diagram into the SPICE netlist.
- To understand this, go to *View->SPICE netlist*

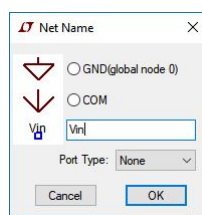
```

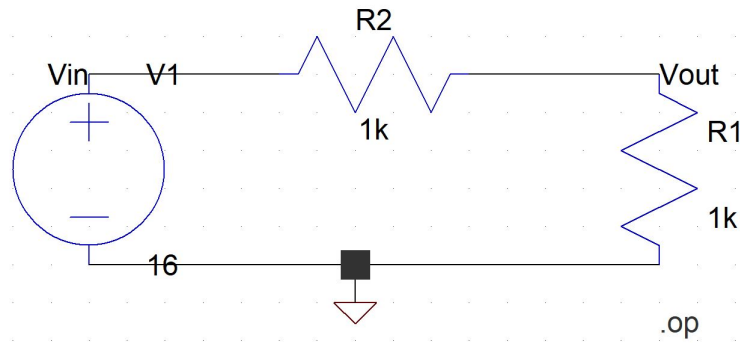
V1 N001 0 16
R1 N002 0 1k
R2 N002 N001 1k
.op
.backanno
.end

```

Here *R1* is connected between *n002* and *0* (ground). So a positive current through *R1* implies current is flowing from *n002* to *0*. *R2* is connected between *n002* and *n001*. Here, a positive current implies current is flowing from *n002* to *n001*. Hence, we find positive current flowing through *R1* and negative current through *R2*.

- It will be hard tracking node numbers when there are a lot of nodes. So, we add meaningful labels to each node. Use the label icon  to add names to every node (also called *net*). Click on a net to set its name. Two nets with the same name are connected by default. All ground nets are connected by default.








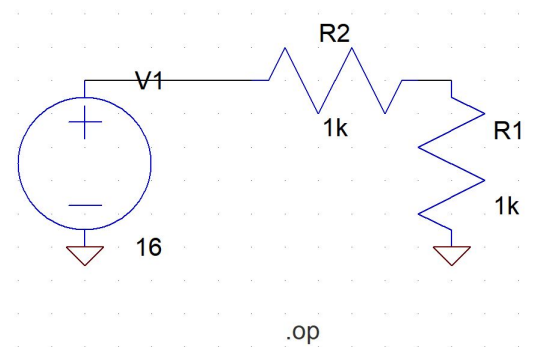
- Now, run the simulation command to see the operating point (Click the run icon). The net names have been updated.


```

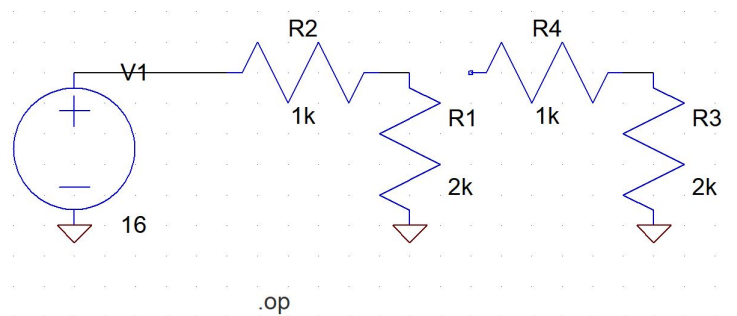
--- Operating Point ---
V(vin):      16      voltage
V(vout):      8      voltage
I(R2):      -0.008   device_current
I(R1):       0.008   device_current
I(V1):      -0.008   device_current

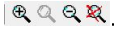
```

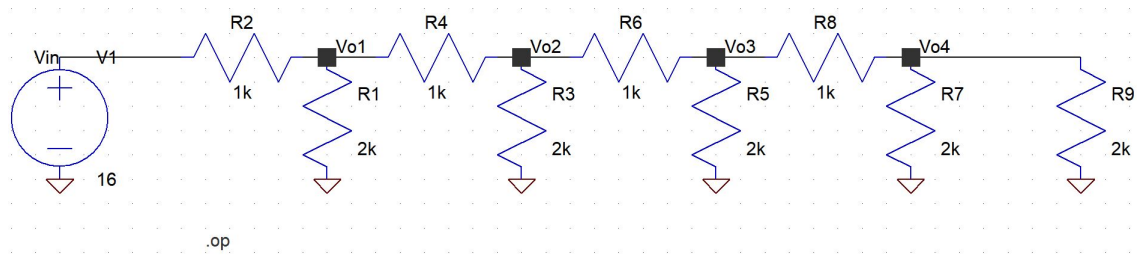
- Use the cut tool (*Ctrl+X* or ) to remove the bottom wire and ground. Also remove the net names we just assigned. Use the drag tool  to move the resistor *R1* along with its connections (connections are intact) nearer to *R2*. Add individual grounds to *R1* and the voltage source. This method of grounding is equivalent to the other method. The move tool  can be used to move components independently of the connections (connections are broken).



- Change the value of *R1* to *2k*. Now, use the copy tool (*Ctrl+C* or ) to copy *R1*, *R2* and ground (left-click and drag to draw a rectangle around it) and paste it to the right. As you can see LTspice copies the components and appropriately numbers them.



- Now copy this set of 4 resistors and 2 grounds to the right. To complete the R-2R ladder, add another resistor of value *2k*. Now connect all the components and label the nets as follows. Use the *space-bar* to fit the schematic into the screen. Other magnification options can be chosen using the icons .




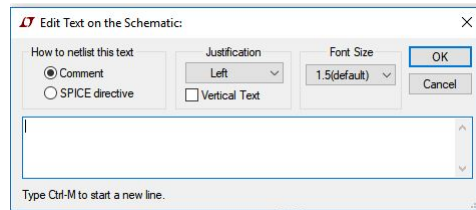
- Now, run the simulation to see the output. The nodal voltages are as expected.

```

--- Operating Point ---
V(vin):      16      voltage
V(vo1):       8      voltage
V(vo2):       4      voltage
V(vo3):       2      voltage
V(vo4):       1      voltage
I(R9):    0.0005    device_current
I(R8):   -0.001    device_current
I(R7):    0.0005    device_current
I(R6):   -0.002    device_current
I(R5):    0.001    device_current
I(R4):   -0.004    device_current
I(R3):    0.002    device_current
I(R2):   -0.008    device_current
I(R1):    0.004    device_current
I(V1):   -0.008    device_current

```

- We can also add comments or a title to the schematic. Use the comment icon  for this.



- Our final schematic of the R-2R ladder is shown below.

