

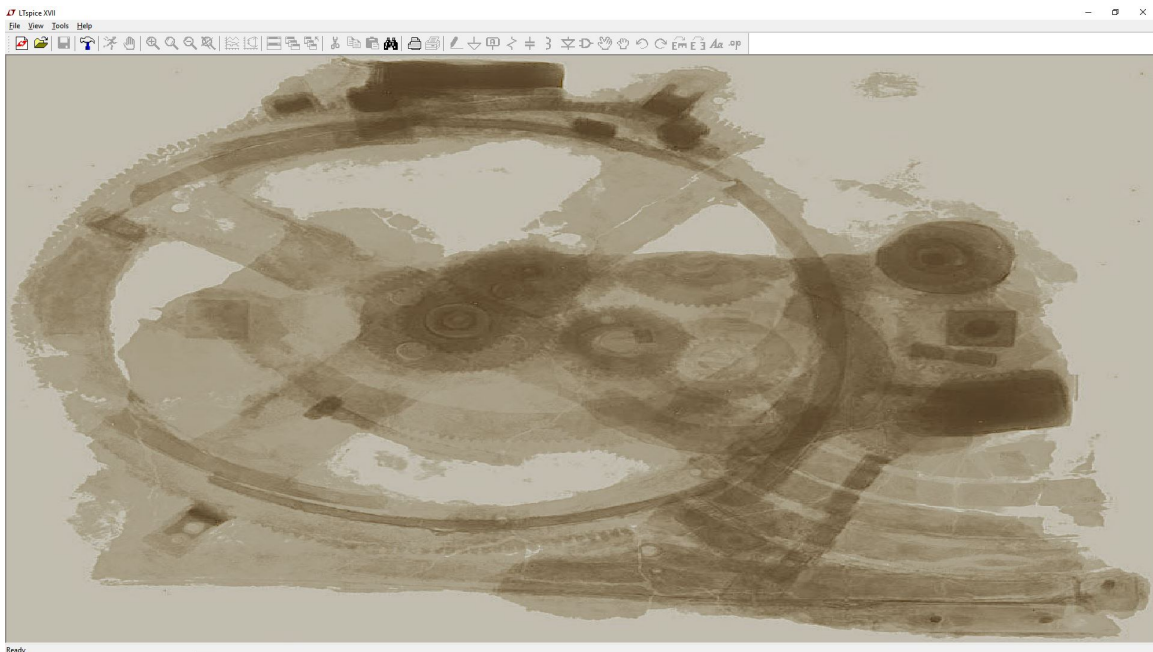
LTspice Basic Simulation Exercises


Alfred Festus Davidson
alfy7.github.io

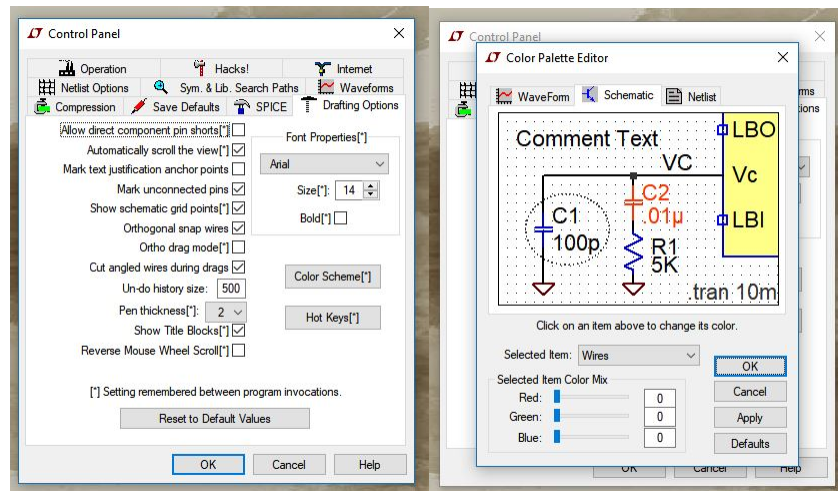
November 30, 2017


R-2R Ladder

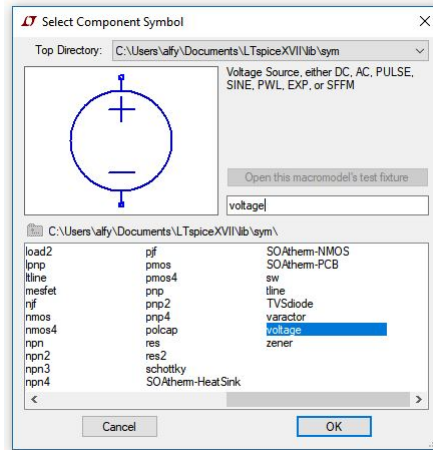
- First, download and install LTspice. Once you open it, you should see a screen like this. I will skip over the basic stuff like icon names/shortcuts/title bar/menu bar for now and revisit them as and when needed.








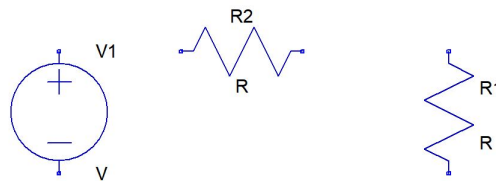
- Lets change the default color options and preferences. Go to *Tools->Control Panel* or click the control panel icon . Go to *Drafting Options* and set preferences as desired. Click on *Color Scheme* and set them as desired.





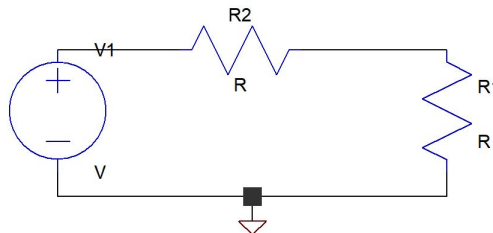
- Create a new Schematic using *Ctrl+N* or *File->New*. The usual short cuts and icons work for opening and saving schematics.
- First, lets add a voltage source. Click the *component* icon  and type *voltage*



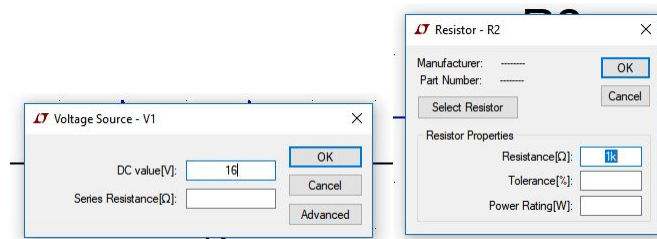
- Now click anywhere in the schematic to add the voltage source. Similarly, to add the resistors, either use the shortcut key *R* or use the resistor icon . To rotate the resistor, use the rotate icon  or the shortcut *Ctrl+R*. Try to place the components like the way shown below. Right-click to get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




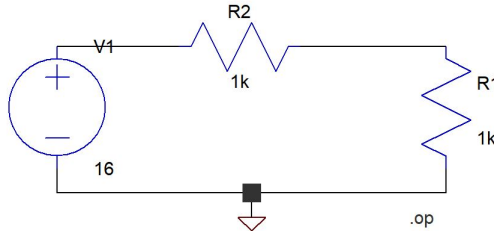
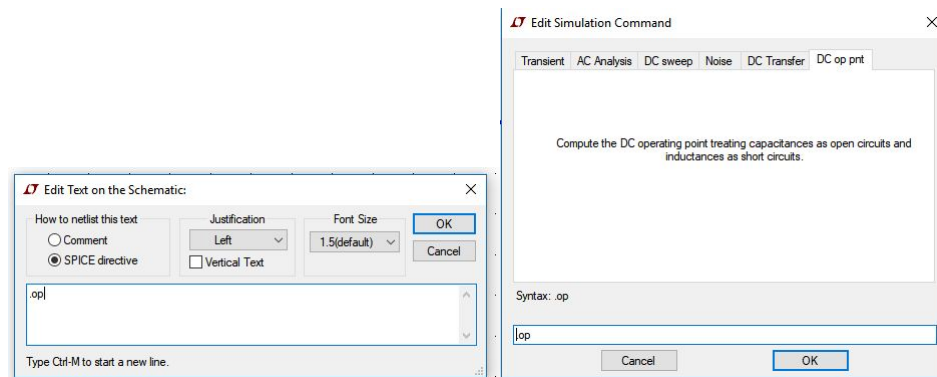
- Use the wire tool  to make connections between the components. All LTspice circuits need a ground connection. Add a ground using the shortcut key *G* or the icon .




- 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*

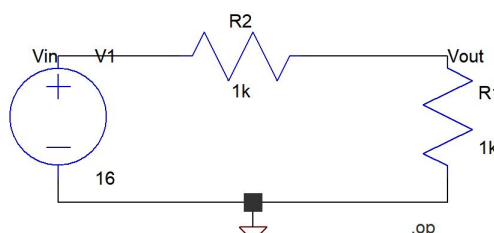
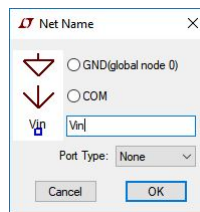
```

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*.

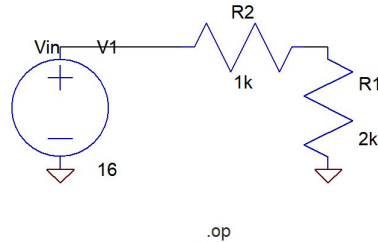
- It will be hard tracking node numbers when there are a lot of nodes. So, we add meaningful labels to each node. Use the label icon  to add names to every node (also called *net*). Click on a net to set its name. Two nets with the same name are connected by default. All ground nets are connected by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




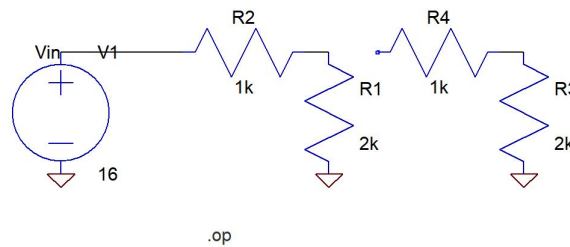
- Now, run the simulation command to see the operating point (Click the run icon). The net names have been updated.





--- 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

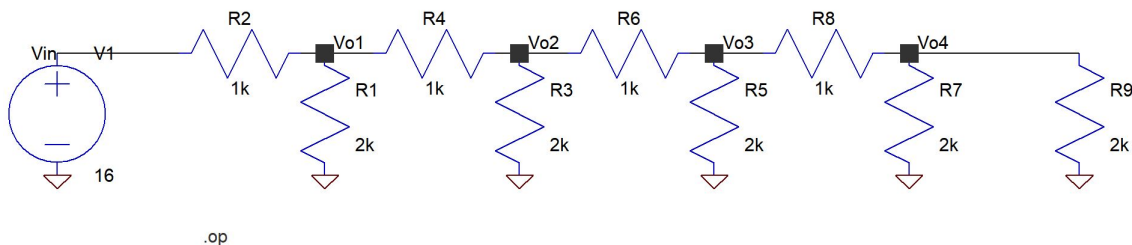
- Use the cut tool (*Ctrl+X* or ) to remove the bottom wire and ground. Also remove the net names we just assigned. Use the drag tool  to move the resistor *R1* along with its connections (connections are intact) nearer to *R2*. Add individual grounds to *R1* and the voltage source. This method of grounding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




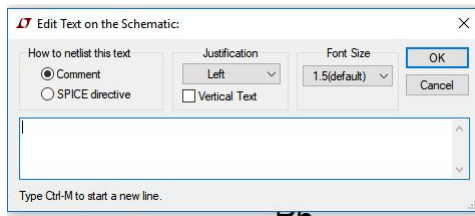
- Now copy this set of 4 resistors and 2 grounds to the right. To complete the R-2R ladder, add another resistor of value *2k*. Now connect all the components and label the nets as follows. Use the *space-bar* to fit the schematic into the screen. Other magnification options can be chosen using the icons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



- 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.

