CSE 30342 LTspice Tutorial

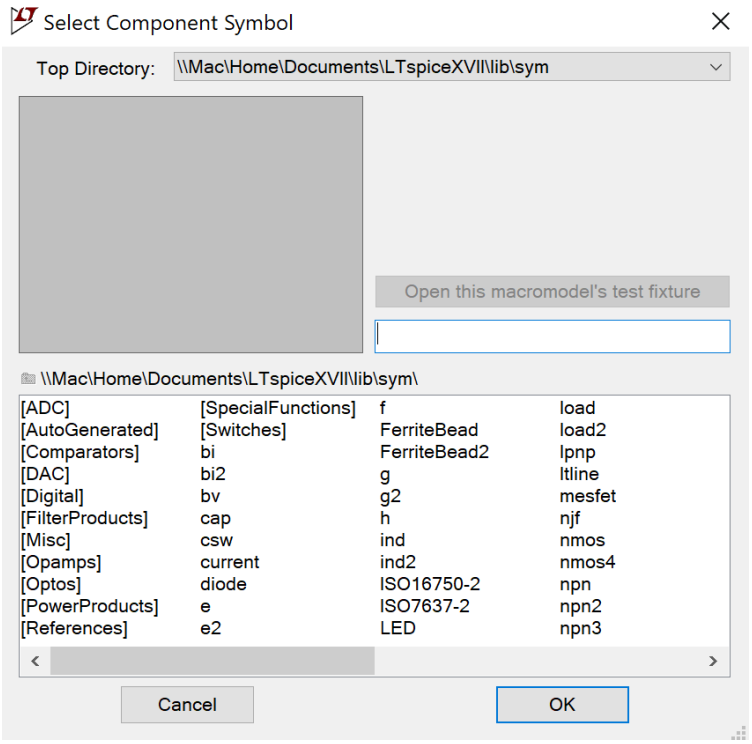
1 Installing LTspice

Download and install LTspice here: https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html

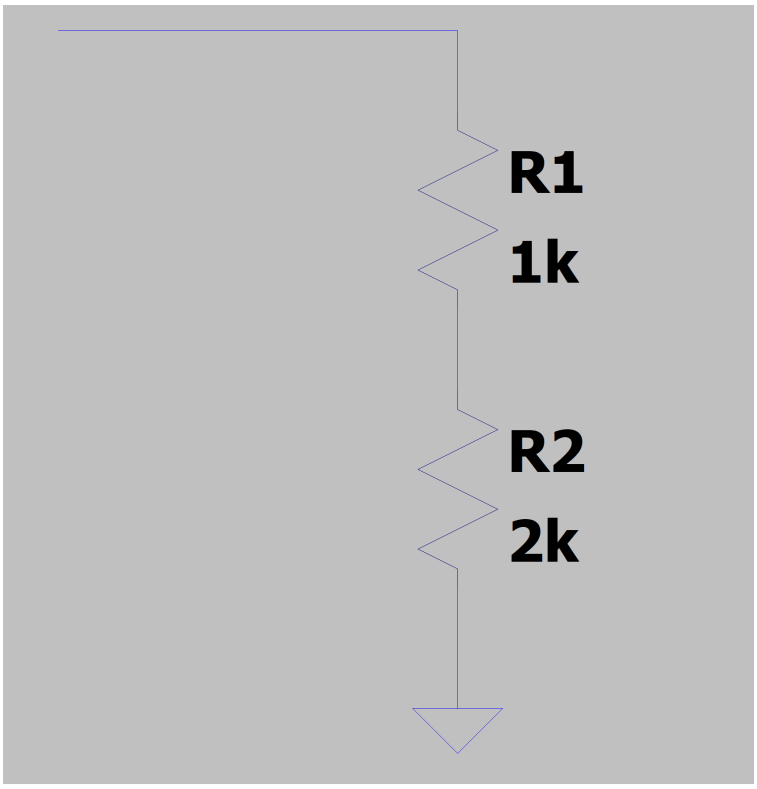
2 Creating a Schematic

Start a new blank schematic from Files > New > New Schematic or Ctrl-N/Cmd-N.

For the tutorial let's build a simple resistive voltage divider. You can add a component by right-clicking and going to Draft > Component. LTspice will bring up a menu that looks like the following:

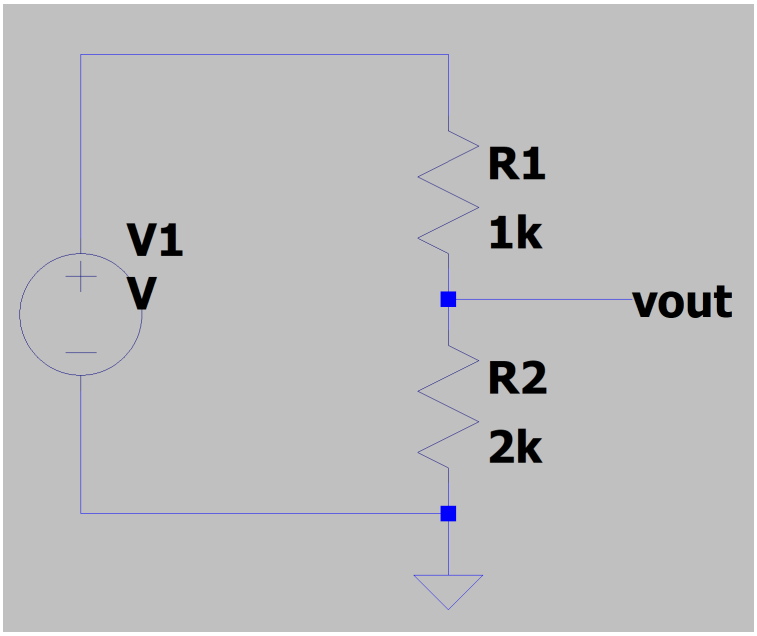


You can use the search bar to search for resistors. Now place two resistors on your schematic and right-click on the resistors to assign values to them. For the tutorial let's use 1 kΩ and 2 kΩ resistors for R1 and R2. Right-click and go to Draft > Draw Wire to connect the two resistors. Press G to place ground on R2. It may be helpful to map the Component and Draw Wire to hot-keys for quicker access or use the default keyboard shortcuts. Your schematic should look like the following:



If you need to move, drag, duplicate, or delete wires or components, you can select these commands by right-clicking and going to Edit. Again, you can map these commands or use the default keyboard shortcuts.

Now, add a voltage source by going to the component menu and search for "voltage". Right-click the voltage source and assign it a default value of 0 V. Place it in your schematic and connect it to R1 and ground. Place a wire between R1 and R2 and label the net as vout by going to Draft > Label Net or Draft > Net Name (for Mac). Your schematic should look like the following:



3 Running a DC Analysis

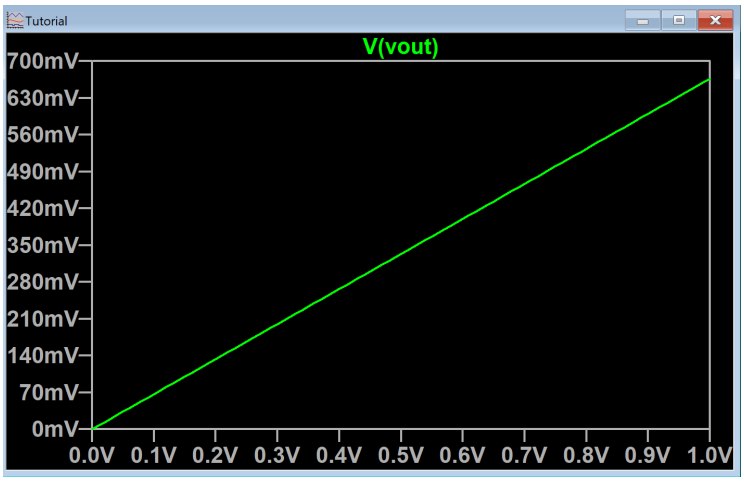
To run a DC Analysis, you can enter in the spice directive manually by pressing S or right-clicking and going to Draft > SPICE directive. The command for a DC Analysis is formatted as follows.

.dc <srcnam> <Vstart> <Vstop> <Vincr> [<srcnam2> <Vstart2> <Vstop2> <Vincr2>]

For our case, let's sweep V1 from 0 to 1 V with 0.01 V increments. So the command will be

.dc V1 0 1 0.01

Place the spice directive somewhere on your schematic. To run the simulation, right-click and select Run. An empty graph will pop up. You can see the voltage for vout by clicking on its net or right-click and select Add Trace and select V(vout). You should see something like the following:



This looks right, since vout is 2/3 of the applied voltage, which is the correct resistor divider ratio from the given resistor values.

4 Importing the 16nm Transistor Model

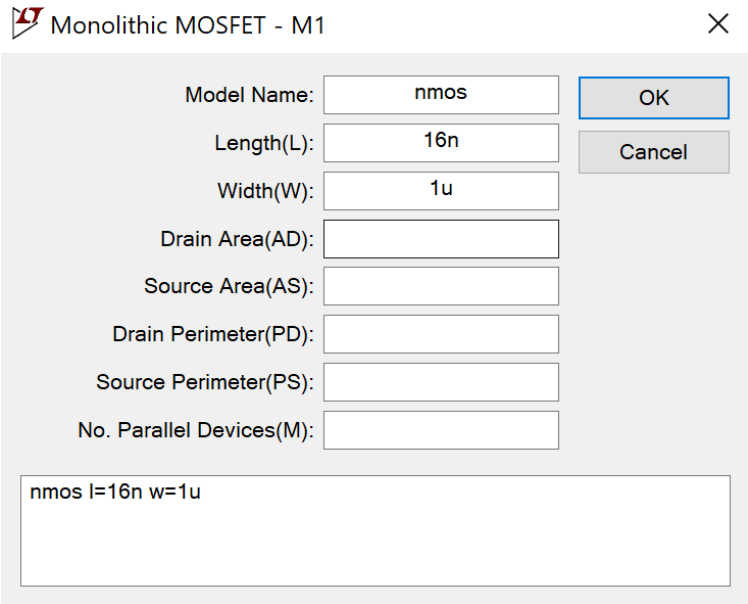
Create a new schematic. Download the 16nm process technology provided on http://ptm.asu.edu/modelcard/LP/16nm_LP.pm. Your browser may save it as 16nm_LP.pm.txt, which is fine, just make sure to include the full file name in the command. To use the model, create a spice directive with the following command:

.include <path to 16nm_LP.pm>

If you placed the model in the same directory as your schematic, your command will look like

.include ./16nm_LP.pm

To place an NMOS or PMOS transistor onto your schematic, bring up the component menu and search for nmos4 and pmos4. After placing it, right click on the component and you will get a menu that looks like the following. For NMOS devices, put in nmos, and for PMOS devices put in pmos. This is because the 16nm model file defines the model as such. You will have to assign a Length and Width for these devices. We will provide you the values to use in homework problems. For example, to create an NMOS device with a Length of 16 nm and Width of 1 µm, your menu should look like the following:



Note: Because MOSFETs are 4-terminal devices, you will need to tie the body of NMOS's to GND and the body of PMOS's to VDD.

5 Acknowledgements

16nm Process Technology Model courtesy of the PTM group at ASU. http://ptm.asu.edu/