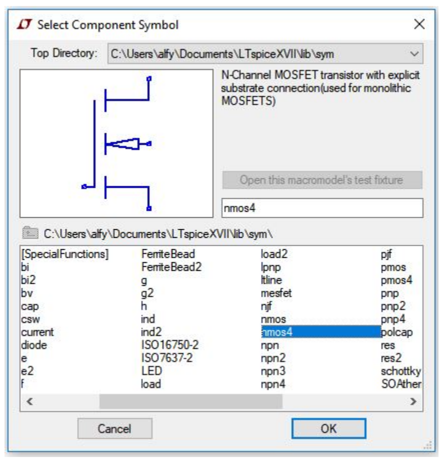
LTspice Basic Simulation Exercises

Alfred Festus Davidson
alfy7.github.io

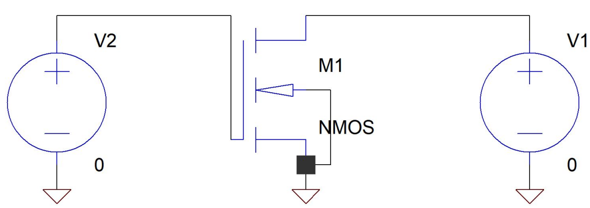
December 6, 2017

MOSFET Model

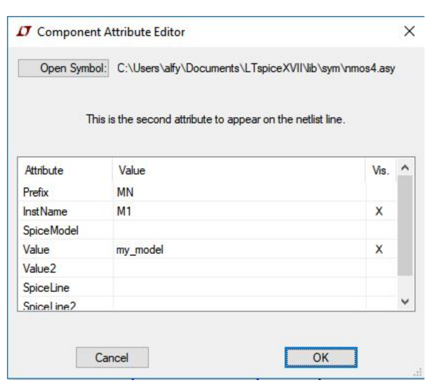
- Use the *Add Component* icon and search for *nmos4* to add an n-channel MOSFET.

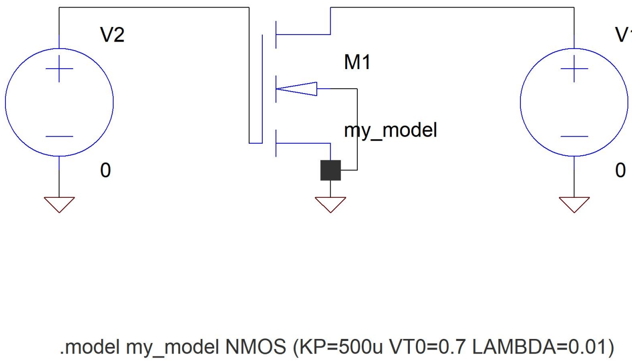


- Draw the following circuit.

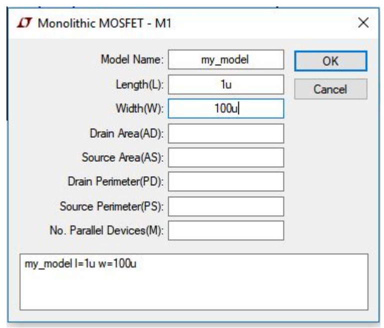


- We will first create a model for a MOSFET and simulate it. Hold *Ctrl* and right-click the MOSFET. Under value, type *my_model*. Now, we define the characteristics of the MOSFET using a spice directive. Add the directive *.model my_model NMOS (KP=500u VT0=0.7 LAMBDA=0.01)*. This defines an NMOS model called *my_model* with $\mu_n C_{ox} = 500\mu A/V^2$, $V_T = 0.7V$ and $\lambda = 0.01$

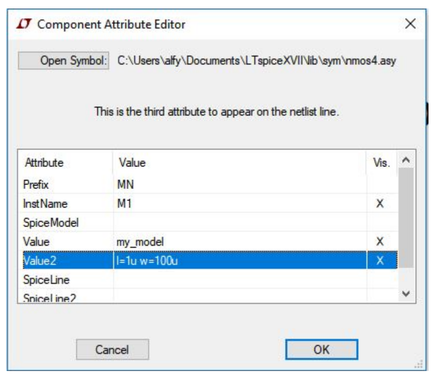




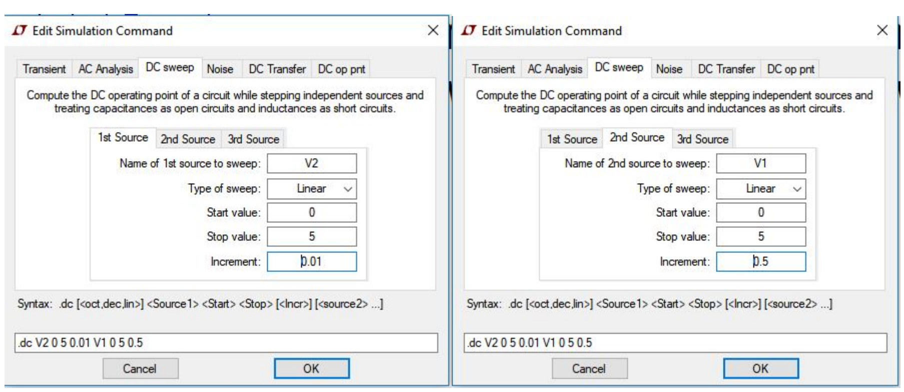
- Right-click the MOSFET and enter its length and width.

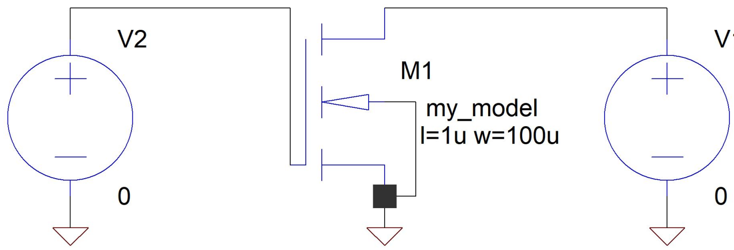


- We would like the width and length information to be visible on the schematic. Hold *Ctrl* and right-click the MOSFET. Double click under the column *Visible* to make *Value2* also visible.



- To probe the $I_D - V_{GS}$ characteristics, we need to sweep both the voltage sources. Enter the following details in the simulation command window.

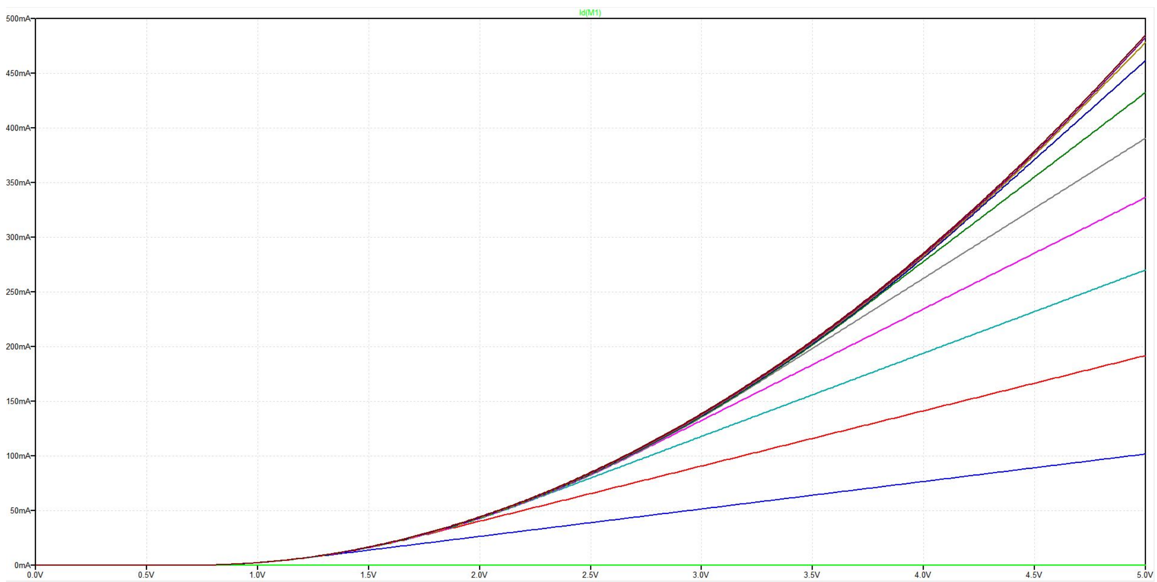




.dc V2 0 5 0.01 V1 0 5 0.5

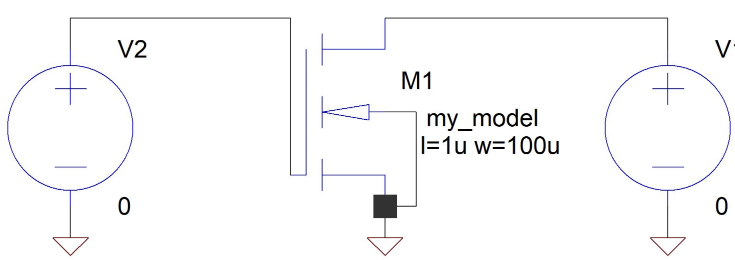
.model my_model NMOS (KP=500u VT0=0.7 LAMBDA=0.01)

- Probe the current through the drain.



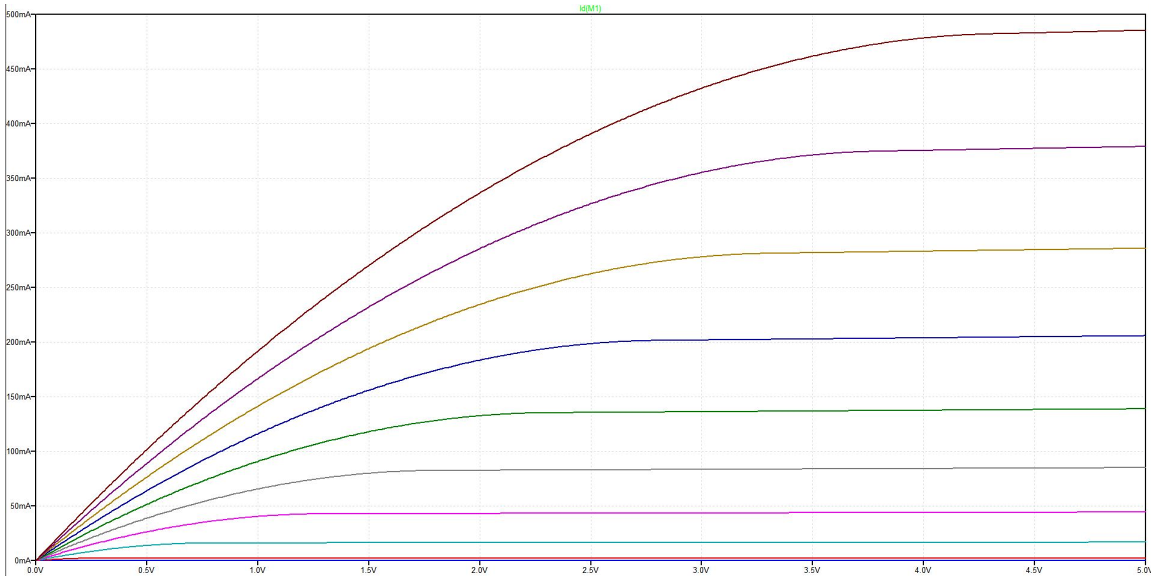
- Interchange the order in which the voltage sources are swept to get the $I_D - V_{DS}$ characteristics. (Notice the change in the $.dc$ directive)

.



.dc V1 0 5 0.01 V2 0 5 0.5

.model my_model NMOS (KP=500u VT0=0.7 LAMBDA=0.01)



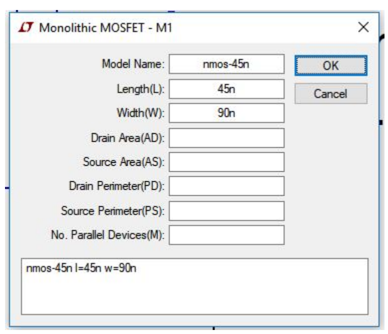
We get the expected curves of a long channel MOSFET
.

- Here, we have used a simplified model for the MOSFET. LTspice can also simulate more complex MOSFET models. Let us use the Predictive Technology Model (PTM) which can be downloaded from `http://ptm.asu.edu/`. Navigate to *Latest Models* and download the *45nm PTM HP model.* Save the file as *45nm_HP.txt*
- The name of the default n-channel MOSFET model is *nmos* (We have used *my_model*). The name of the model in the PTM file is also *nmos*. To avoid confusion, use a text editor to open *45nm_HP.txt* and change the name to *nmos-45n.*

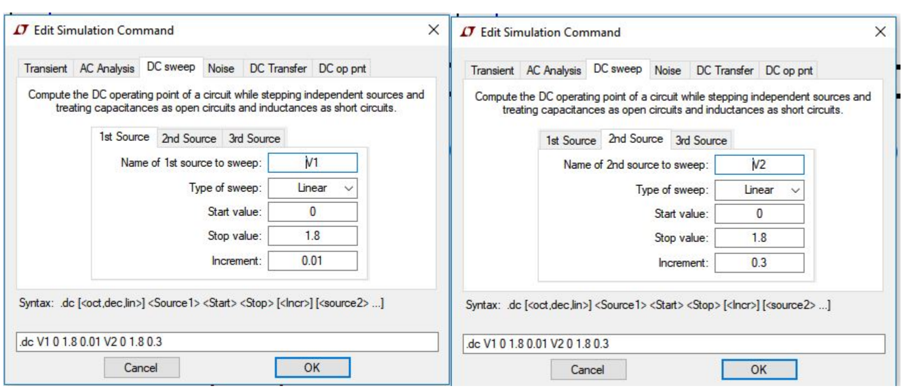
Change `.model nmos nmos level = 54` to `.model nmos-45n nmos level = 54`

Do the same for *pmos* and change it to *pmos-45n.*

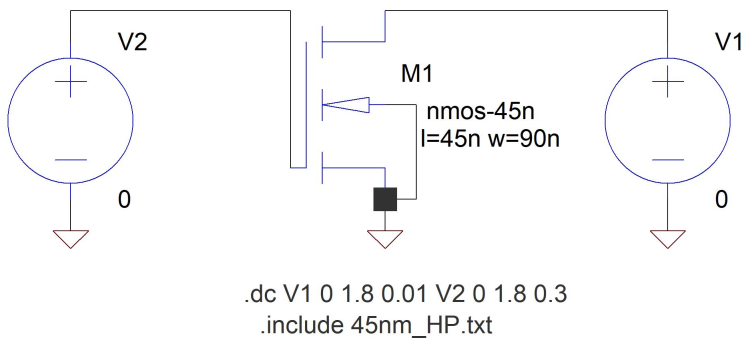
- Now to add the model file to this schematic, first save the model file in the same folder as the schematic. Use the spice directive *.include 45nm_HP.txt* to add the file.
- Right-click the MOSFET in the schematic and change the model name to *nmos-45n.* Change the width and length also.

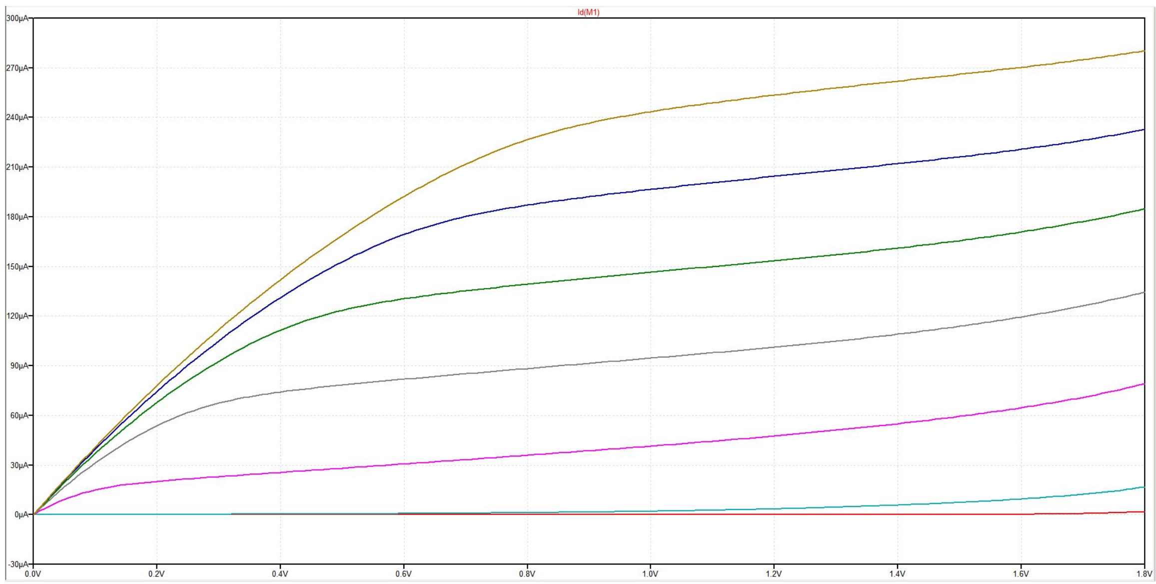


- Change the sweep parameters to lower maximum voltages.



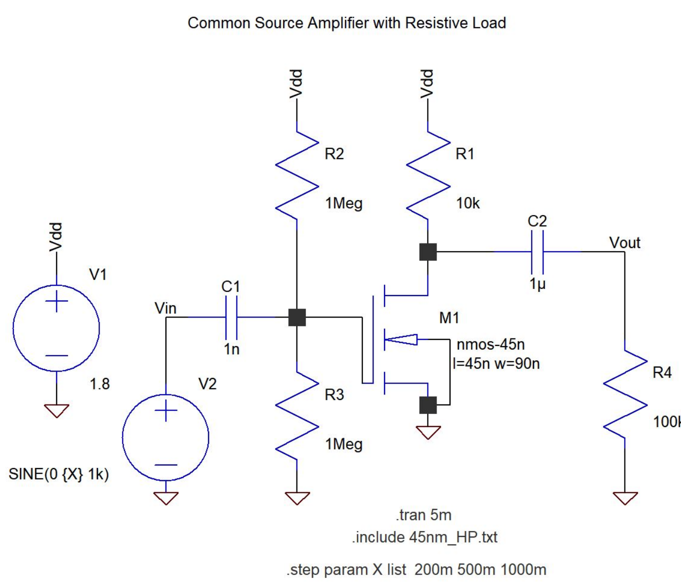
- Run the simulation and probe the drain current



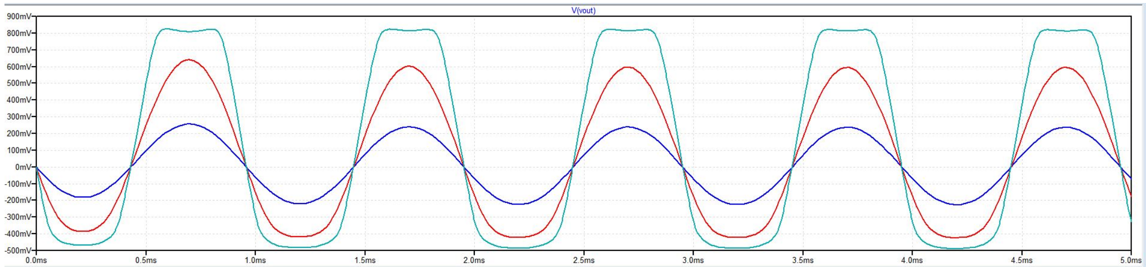


We see the effects of small geometry in the $I_D - V_{DS}$ curves.

- Now, draw the following circuit. This is a common source amplifier with resistive load. Do a transient simulation for different amplitudes of input voltages.



- Probe *Vout*



Notice that the input with amplitude $200mV$ is not distorted a lot. However, when the amplitude is $500mV$, the MOSFET goes into triode and we see that the gain is much lesser than the gain in saturation (for the voltage range the device is in triode). At $1000mV$, the device also goes into cutoff. Notice that at cutoff, the output is effectively clipped, while at triode there is a relatively soft clipping.

- Now draw the following circuit. This is a cmos inverter. Run a DC sweep simulation to get the input output characteristics. Orient the p-channel MOSFET properly.