

# University of Pennsylvania

## Department of Electrical and Systems Engineering


### ESE 216 MOSFET Simulation Guide

LT Spice software allows users to define their own devices and use their own models for simulations. This tutorial will show the steps to add a user-defined model of MOSFET transistors for simulation.

#### 1: MOSFET Spice Model

Spice is the most commonly used circuit simulation tool. You can find a brief overview of SPICE at the link: <http://www.seas.upenn.edu/~jan/spice/spice.overview.html>. An introduction for the SPICE model for MOSFET is available at: <http://www.seas.upenn.edu/~jan/spice/spice.overview.html#MOSFETS>. The names, units and default values of parameters in the SPICE model for MOSFET can be found at: <http://www.seas.upenn.edu/~jan/spice/spice.MOSparamlist.html>.

To run simulations of MOSFETs we need to at least set the values of parameters L (channel length), W (channel width), VTO (zero-bias threshold voltage), KP (transconductance,  $\mu_{n/p}C_{ox}$ ), and LAMBDA (channel-length modulation coefficient,  $\lambda$ ).

In LT Spice, these parameters can be specified by inserting the model into the schematic. Go to “Edit” on the menu bar and choose “Spice Directive”, or just click the  button. In the pop-up window, type-in “.MODEL TestN NMOS (KP=90u VTO=0.7 LAMBDA=0.01)” in the dialogue box to set KP, VTO, and LAMBDA of the NMOS transistor that we will use for this tutorial (as shown in Fig. 1). Click “OK” and place the model sentence onto the schematic. And similarly, create another model for PMOS a transistor by type in “.MODEL TestP PMOS (KP=40u VTO=-0.7 LAMBDA=0.01)”. Notice that TestN and TestP are the model names and can be any name you give to the transistor model. The other names or variables in the .MODEL are standard and cannot be renamed.

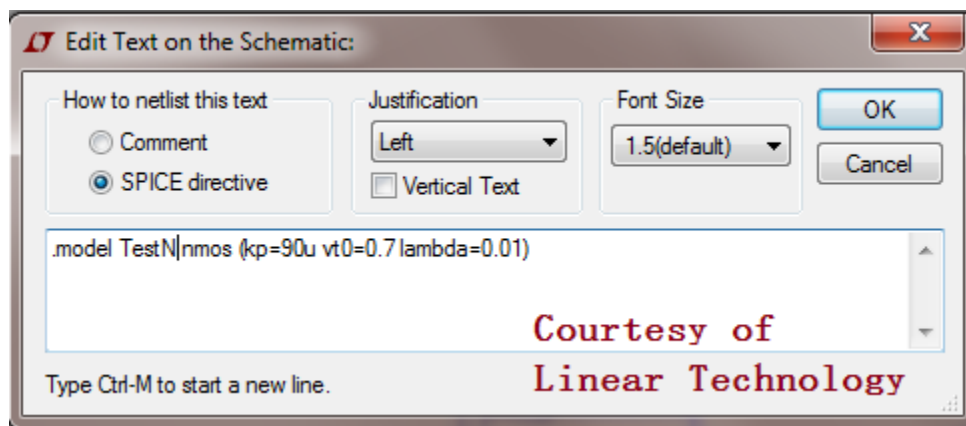


Fig. 1 Setup NMOS model

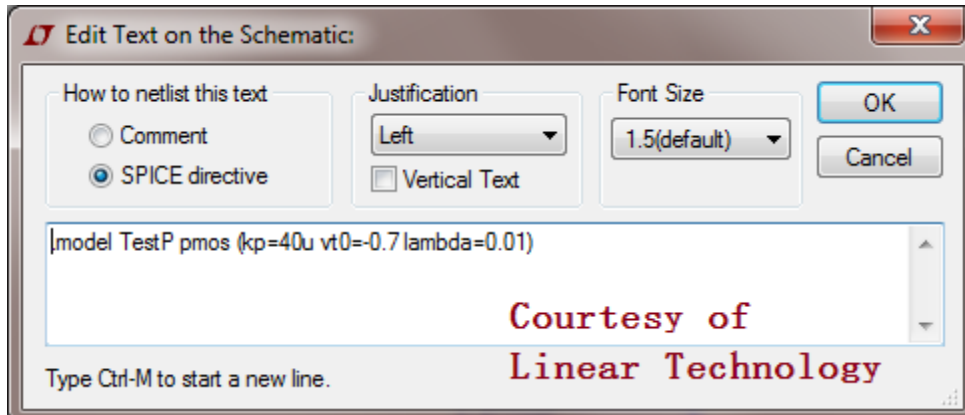


Fig. 2 Setup PMOS model

## 2: Schematic Simulation with Self-Defined MOSFET Model

The MOSFET we need to use in our simulation should be “nmos4” and “pmos4”. In our tutorial, we just use NMOS4 for a demonstration. Insert the components, make connections, and name the input and output nets as shown in Fig. 4.

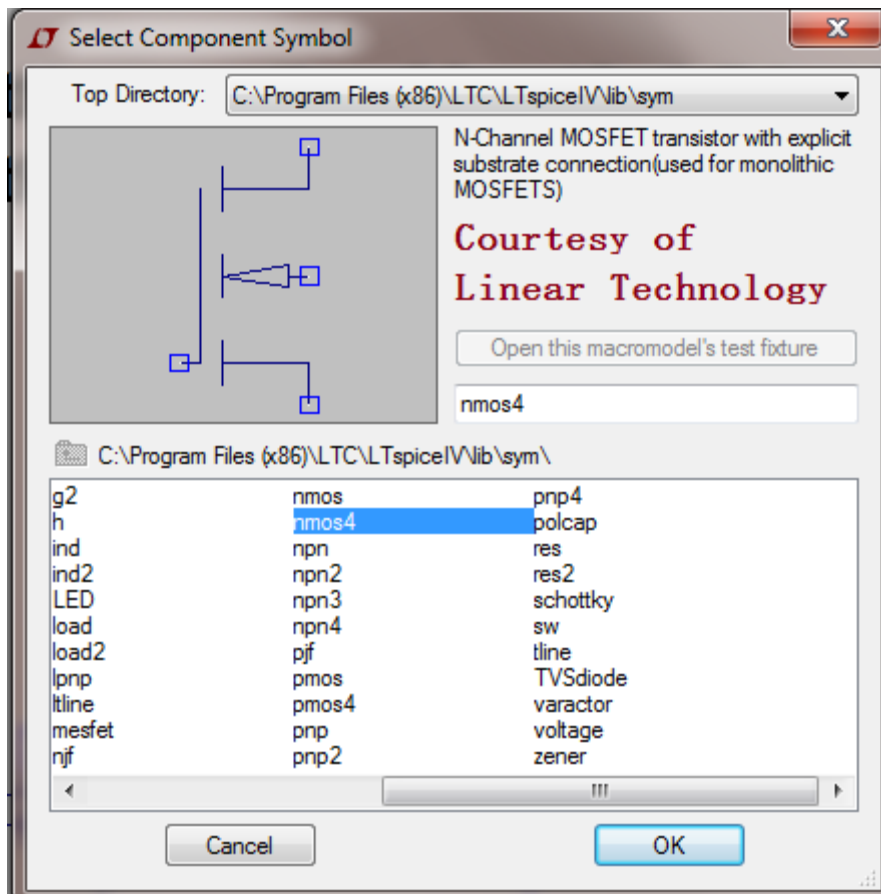


Fig. 3 Selecting MOSFET for simulation

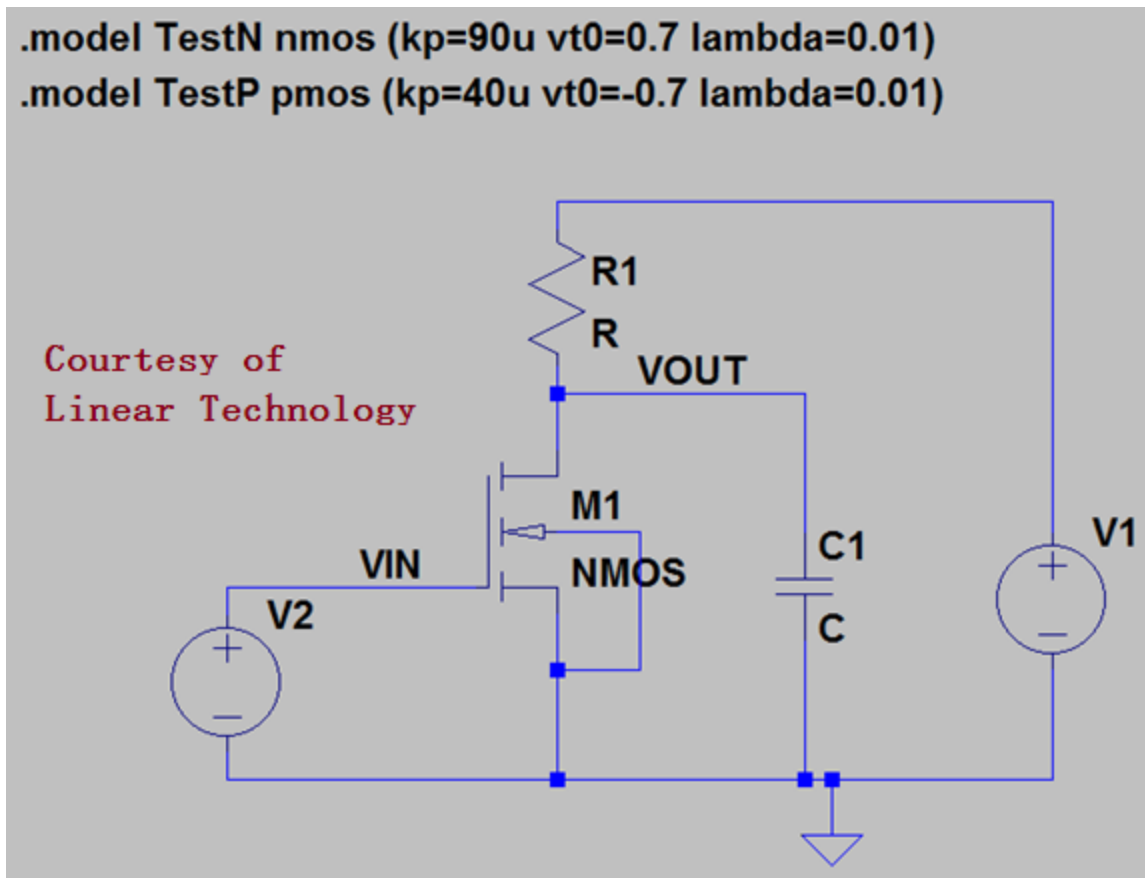


Fig. 4 Full simulation schematic

Then we need to set the values of each component.

First, we set the NMOS transistor. Hold the “Ctrl” key and right-click on the NMOS symbol, then the “Component Attribute Editor” window will pop-up. Double click on the value of “SpiceModel” and type in the name of our self-defined NMOS transistor, “TestN” (as shown Fig. 5). Click “OK” to confirm.

Then, WITHOUT holding the “Ctrl” key, right click on the NMOS model. In the pop-up window, input the Length (L) and Width (W) values as L=10u and W=100u (as shown Fig. 6). Note that the bottom dialogue box shows “TestN l=10u w=100u”, which means we have successfully set the NMOS model to our self-defined model, TestN and we have set the channel length and width to 10um and 100um, respectively.

After that, set the resistor R1 to 1kΩ, the capacitor C to 1uF, and the DC supply voltage V1 to 10V. At last, we need to set the input voltage source as shown in Fig. 7.

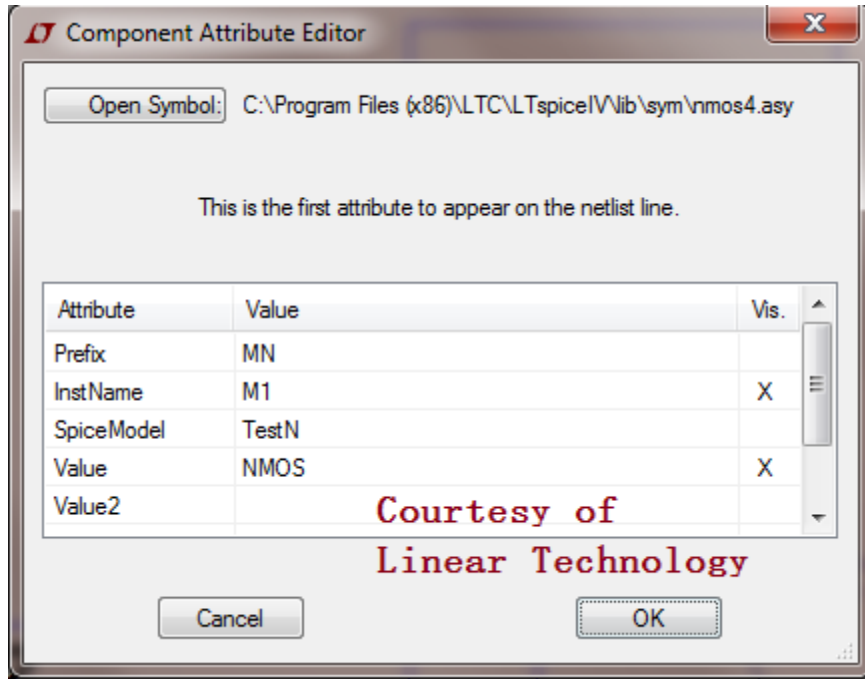


Fig. 5 Choose the self-defined model for NMOS

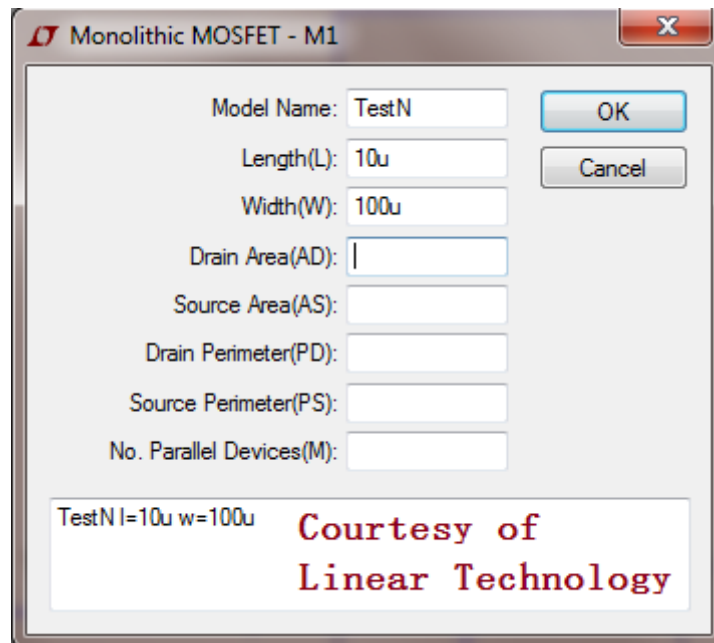


Fig. 6 Setup the channel length and width for the NMOS transistor

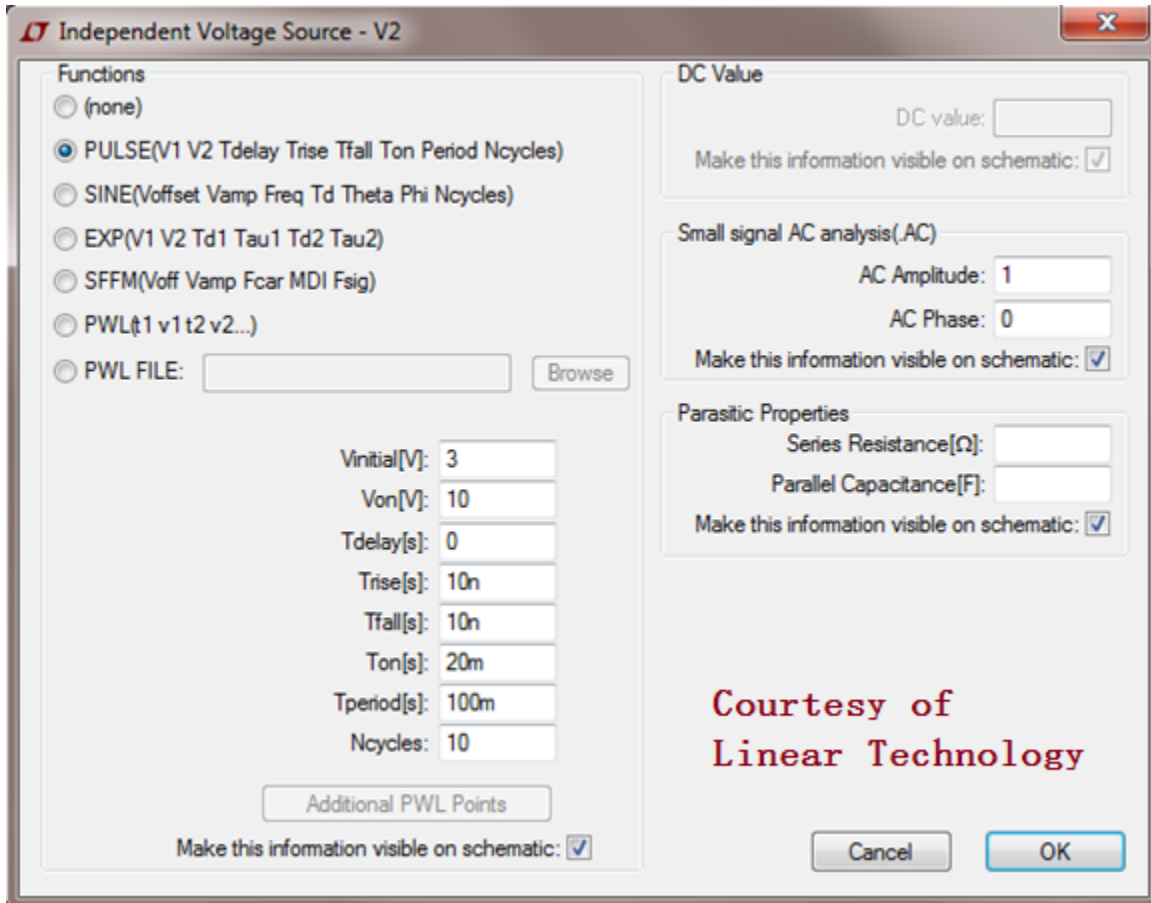


Fig. 7 Setup the input voltage source

After setting all the values, the schematic should look like in Fig. 8.

Finally, we can move on to simulations. We can first do a DC operating point simulation by selecting “DC op pnt” option in the “Edit Simulation Command” window (as shown in Fig. 9). Place the simulation sentence on the schematic. Run the simulation and we can get a pop-up window showing the DC operating point of all devices on the schematic (Fig. 10). We can read that  $I_d$  of the NMOS transistor M1 is 2.56mA, the gate voltage  $V_{in} = 3V$ , and the drain-to-source voltage  $V_{ds} = V_{out} = 7.44V$ . This simulation result can be verified to be the same as hand calculations.

$$V_{ds} = V_{out} = V_{DD} - R \cdot I_d = 10V - 1k\Omega \times 2.56mA = 10V - 2.56V = 7.44V$$

$$\begin{aligned} I_d &= \frac{1}{2} \mu_n C_{ox} \left( \frac{W}{L} \right) (V_{GS} - V_T)^2 (1 + \lambda \cdot V_{DS}) = \frac{1}{2} KP \left( \frac{W}{L} \right) (V_{IN} - V_{T0})^2 (1 + LAMBDA \cdot V_{OUT}) \\ &= \frac{1}{2} \times 90u \times \left( \frac{100u}{10u} \right) \times (3 - 0.7)^2 \times (1 + 0.01 \times 7.44) = 2.6mA \end{aligned}$$

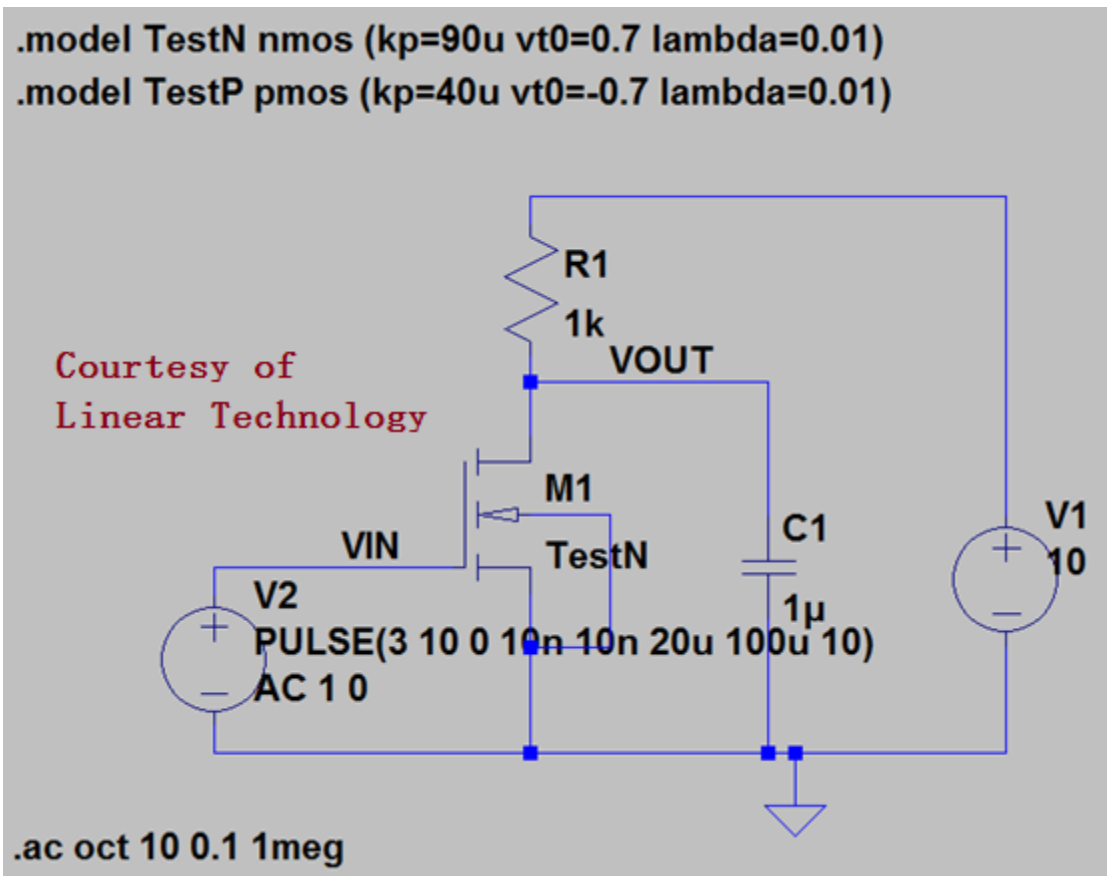


Fig. 8 Full schematic for simulations

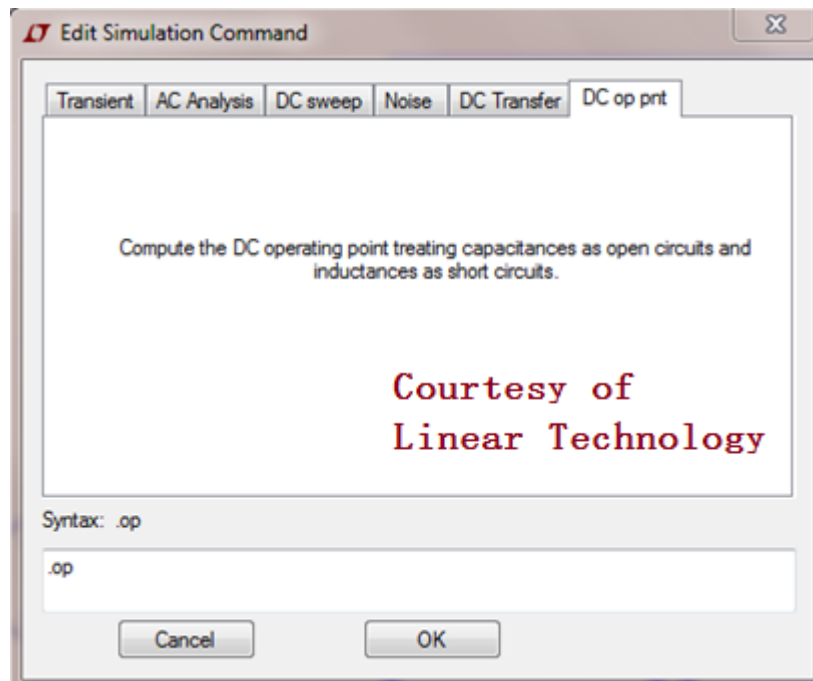


Fig. 9 DC operating point simulation menu

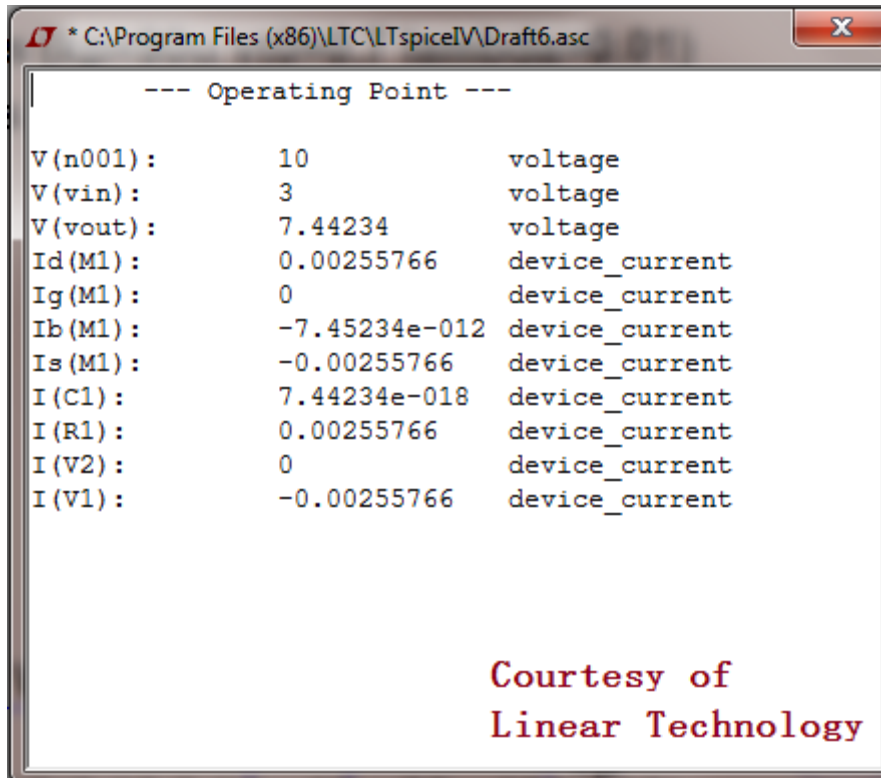


Fig. 10 DC operating point simulation results

Then we can try to run AC simulation to see the frequency response of the gain of the (common-source amplifier) circuit we built. Set the “start frequency” of the AC analysis to 0.1Hz and the “stop frequency” to 1 Mega Hz. Run the simulation and plot “VOUT”.

The plot might come up with black background color. *We can change the background color of the plot by going to “Tools” on the menu bar and choosing “Color Preferences”.* Choose the “WaveForm” page on the pop-up window. Select “Background” in the pull-down menu. We can set the background color to WHITE by setting all the three colors to 255 (Fig. 11). You can also change colors of the plotting traces as you want here.

We can also change the thickness of the plotted traces by going to “Tools” and choosing “Control Panel”. Check the “Plot data with thick lines” option in the “Waveforms” page in the pop-up window (Fig. 12). So the final plot should be like Fig. 13.

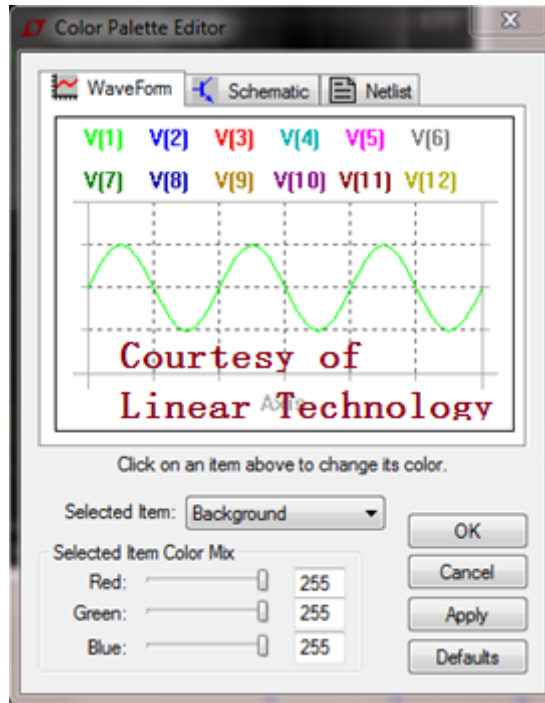


Fig. 11 Setup the background color

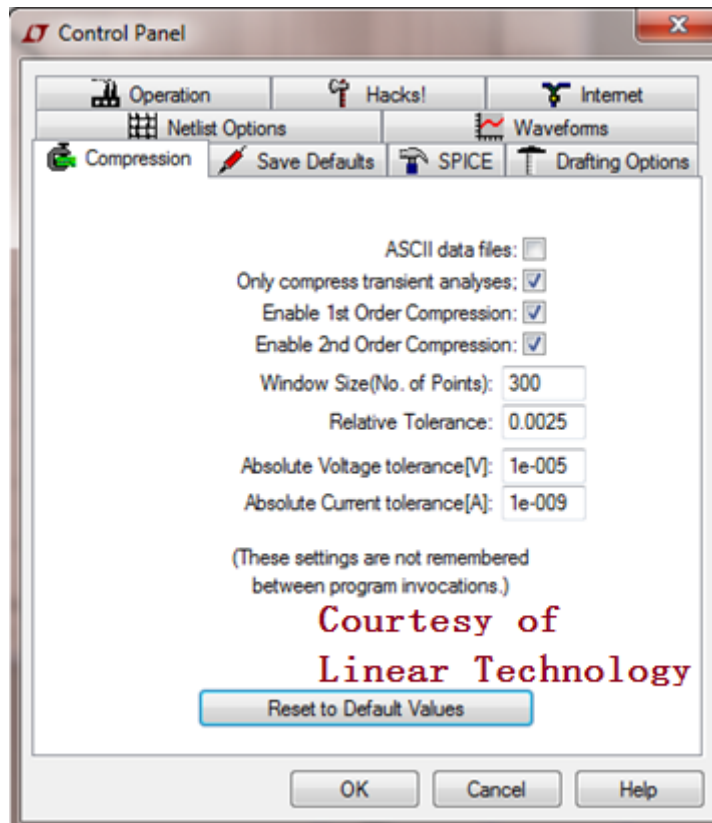


Fig. 12 Setup thick plotted traces



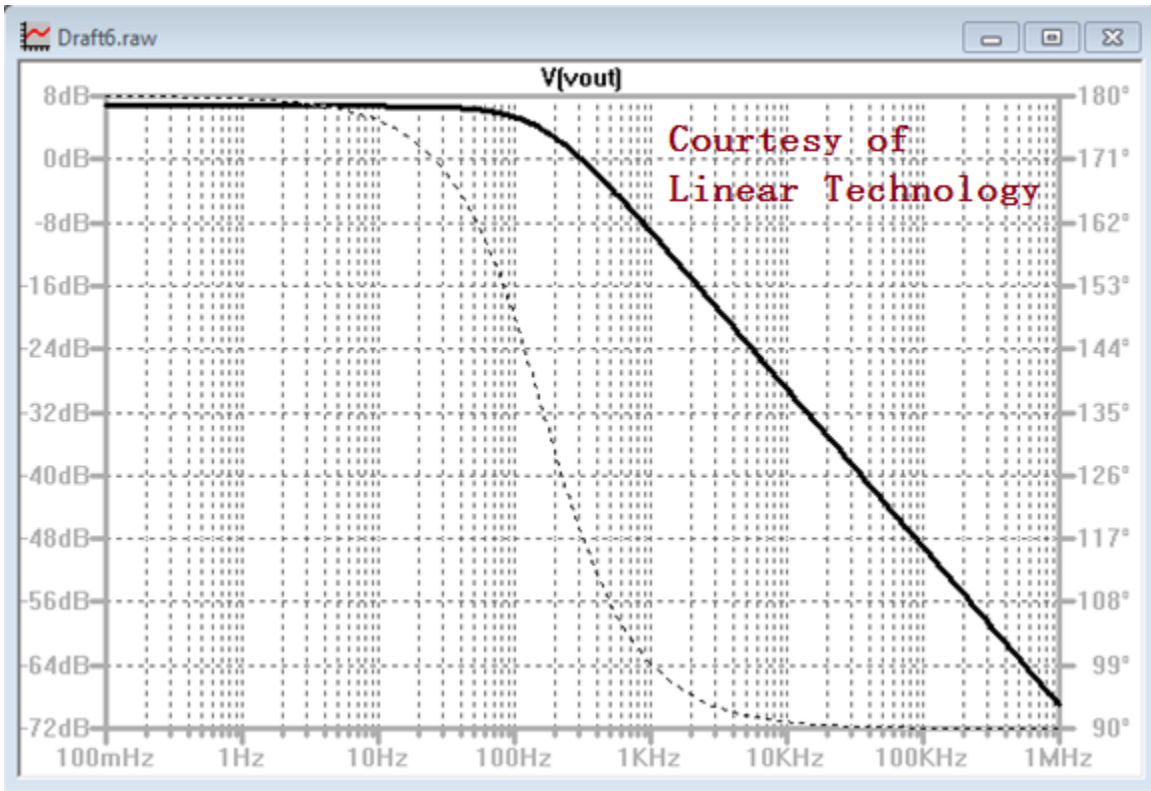


Fig. 13 Frequency response of the gain

At last, we can run a transient simulation to see the temporal signals in our circuit. Set the “stop time” to 1 second and run the simulation. Then we can plot both the input and output signals on the same graph (Fig. 14).

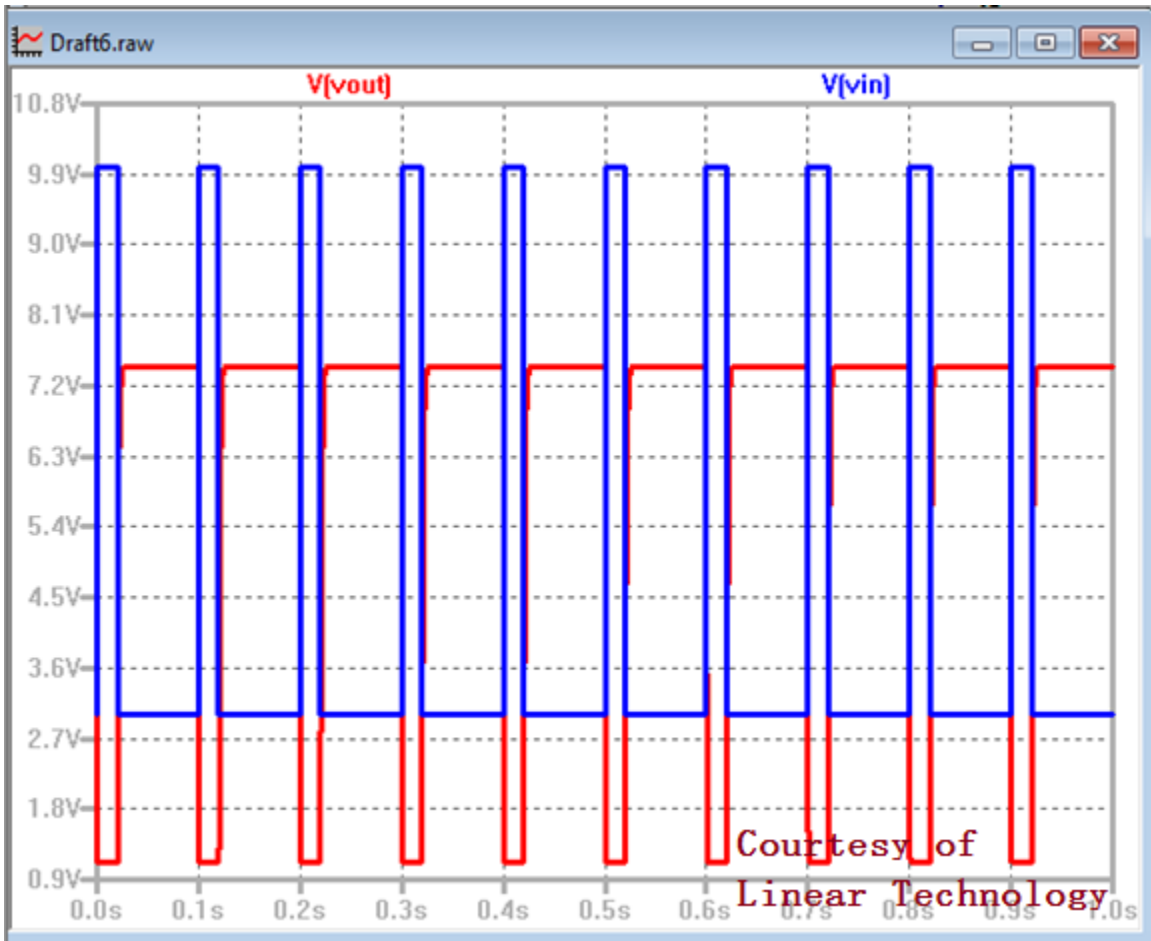


Fig. 14 Transient simulation results