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: <u>http://www.seas.upenn.edu/~jan/spice/spice.overview.html</u>. An introduction for the SPICE model for MOSFET is available at: <u>http://www.seas.upenn.edu/~jan/spice/spice.overview.html#MOSFETS</u>. The names, units and default values of parameters in the SPICE model for MOSFET can be found at: <u>http://www.seas.upenn.edu/~jan/spice/spice.MOSparamlist.html</u>.

To run simulations of MOSFETs we need to at least set the values of parameters L (channel length), W (channel width), VT0 (zero-bias threshold voltage), KP (transconductance, $\mu_{n/p}C_{ox}$), and LAMBDA (channel-length modulation coefficient, λ).

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 ^{op} button. In the pop-up window, type-in ".MODEL TestN NMOS (KP=90u VT0=0.7 LAMBDA=0.01)" in the dialogue box to set KP, VT0, 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 VT0=-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.

🗸 Edit Text on the Schemat	ic:		×
How to netlist this text Comment SPICE directive	Justification Left Vertical Text	Font Size 1.5(default)	OK Cancel
.model Testivinmos (kp=300 v	Cor	urtesy of	Ť
Type Ctrl-M to start a new line.	Liı	near Techno	logy

Fig. 1 Setup NMOS model

TEdit Text on the Schemati	ic:	x
How to netlist this text Comment SPICE directive	Justification Font Size OK Left ▼ 1.5(default) Cance Vertical Text Cance	el
model TestP pmos (kp=40u vi	t0=-0.7 lambda=0.01)	*
	Courtesy of	-
Type Ctrl-M to start a new line.	Linear Technology	

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.

17 Select Compo	nent Symbol	×
Top Directory:	C:\Program Files (x8	6)\LTC\LTspiceIV\lib\sym 👻
	μĴ	N-Channel MOSFET transistor with explicit substrate connection(used for monolithic MOSFETS)
		Courtesy of
		Linear Technology
	<u> </u>	Open this macromodel's test fixture
		nmos4
C:\Program	Files (x86)\LTC\LTspi	celWib\sym\
g2	nmos	pnp4
n	nmos4	poicap
ind ind2	npn	res
	npn2	sebettler
load	npn3	sw
load2	Dif	tline
Ipnp	pmos	TVSdiode
Itline	pmos4	varactor
mesfet	pnp	voltage
njf	pnp2	zener
•		4 III
	Cancel	ОК

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 I=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 $1k\Omega$, 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.

Component /	Attribute Editor		x
Open Symbo	c:\Program Files (x86)\LTC\LTspiceIV\lib\sym\	vnmos4.asy	
1	This is the first attribute to appear on the netlist line.		
Attribute	Value	Vis.	
Prefix	MN		
InstName	M1	Х	Ξ
SpiceModel	TestN		
Value	NMOS	Х	
Value2	Courtesy of		-
	Linear Technol	ogy	
	Cancel		

Fig. 5 Choose the self-defined model for NMOS

17 Monolithic MOSFET - M1	— X —
Model Name:	TestN OK
Length(L):	10u Cancel
Width(W):	100u
Drain Area(AD):	
Source Area(AS):	
Drain Perimeter(PD):	
Source Perimeter(PS):	
No. Parallel Devices(M):	
TestN I=10u w=100u Co Li	urtesy of near Technology

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

Independent Voltage Source - V2		
Functions (none)		DC Value
 PULSE(V1 V2 Tdelay Trise Tfall Ton Policy SINE(Voffset Vamp Freq Td Theta Phil 	eriod Ncycles) Ncycles)	Make this information visible on schematic: \fbox
EXP(V1 V2 Td1 Tau1 Td2 Tau2) SEEM0(cff Varia Faar MDI Fair)		Small signal AC analysis(.AC)
PWL(t1v1t2v2)		AC Phase: 0
PWL FILE: Vinitial[V]: Von[V]:	Browse 3 10	Parasitic Properties Series Resistance[Ω]: Parallel Capacitance[F]: Make this information visible on schematic: ☑
Tdelay[s]: Trise[s]: Tfall[s]: Ton[s]: Tperiod[s]:	0 10n 10n 20m 100m	Courtesy of
Ncycles:	10	Linear Technology
Make this information visible	on schematic:	Cancel OK

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 Id of the NMOS transistor M1 is 2.56mA, the gate voltage Vin = 3V, and the drain-to-source voltage Vds = Vout = 7.44V. This simulation result can be verified to be the same as hand calculations.

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

$$I_d = \frac{1}{2}\mu_n C_{ox} \left(\frac{W}{L}\right) (V_{GS} - V_T)^2 (1 + \lambda \cdot V_{DS}) = \frac{1}{2}KP \left(\frac{W}{L}\right) (V_{IN} - V_{T0})^2 (1 + LAMBDA \cdot V_{OUT})$$

$$= \frac{1}{2} \times 90u \times \left(\frac{100u}{10u}\right) \times (3 - 0.7)^2 \times (1 + 0.01 \times 7.44) = 2.6mA$$



Fig. 8 Full schematic for simulations



Fig. 9 DC operating point simulation menu

	(x86)\LTC\LTspiceIV\E)raft6.asc
Oper	ating Point	-
V(n001): V(vin): V(vout): Id(M1): Ig(M1): Ib(M1): Is(M1): I(C1): I(C1): I(R1): I(V2): I(V1):	10 3 7.44234 0.00255766 0 -7.45234e-012 -0.00255766 7.44234e-018 0.00255766 0 -0.00255766	<pre>voltage voltage device_current device_current device_current device_current device_current device_current device_current device_current device_current</pre>
		Courtesy of
	I	Linear Technology

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.

WaveForm Schematic Netlist V(1) V(2) V(3) V(4) V(5) V(6) V(7) V(8) V(9) V(10) V(11) V(12) Courtesy of Courtesy of Linear ATechnology Click on an item above to change its color. OK Selected item: Background OK Red: 255 Apply Blue: 255 Defaults	🖉 Color Palette Editor
V(1) V(2) V(3) V(4) V(5) V(6) V(7) V(8) V(9) V(10) V(11) V(12) Courtesy of Linear ATechnology Click on an item above to change its color. Selected Item: Background OK Red: 255 Apply Blue: 255 Defaults	WaveForm 🤾 Schematic 📄 Netlist
Courtesy of Linear ATechnology Click on an item above to change its color. Selected Item: Background Selected Item: Color Mix Red: 255 Green: 255 Blue: 255 Defaults	V(1) V(2) V(3) V(4) V(5) V(6) V(7) V(8) V(9) V(10) V(11) V(12)
Linear Arechnology Click on an item above to change its color. Selected Item: Background Selected Item Color Mix OK Red: 255 Green: 255 Blue: 255 Defaults	Courtesy of
Click on an item above to change its color. Selected item: Background Selected item Color Mix Red: Cancel Green: 255 Blue: 255 Defaults	Linear Technology
Selected Item: Background Selected Item Color Mix Red: Green: Selected Item Color Mix Cancel Cancel Blue: Cancel Cancel Cancel Defaults Cancel Cancel Cancel Cancel Cancel Cancel Cancel Canc	Click on an item above to change its color.
Selected Item Color Mix Red: 255 Green: 255 Blue: 255 Defaults	Selected Item: Background
Green: 255 Apply Blue: 255 Defaults	Selected Item Color Mix Cancel
	Green: 255 Apply Blue: 255 Defaults

Fig. 11 Setup the background color

🖉 Control Panel
Operation Composition Composition Composition Image: Netlist Options Image: Netlist Options Image: Netlist Options Image: Netlist Options Image: Netlist Options Image: Netlist Options Image: Netlist Options Image: Netlist Options Image: Netlist Options Image: Netlist Options Image: Netlist Options Image: Netlist Options Image: Netlist Options Image: Netlist Options Image: Netlist Options
ASCII data files: Only compress transient analyses; Enable 1st Order Compression:
Window Size(No. of Points): 300 Relative Tolerance: 0.0025 Absolute Voltage tolerance[V]: 1e-005
(These settings are not remembered between program invocations.) Courtesy of
Linear Technology Reset to Default Values
OK Cancel Help

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