

ECE 2100 Spice Project #S3

Due: Friday 4/18/03

A. Stolp
4/18/02

I want you to do one more PSpice project. but since you're all getting a bit strapped for time here at the end of the semester, I'll talk you through it. Try to learn something despite that. This help was prepared with PSpice versions 9.2. I will try to make sure 9.2 is installed on all the lab computers soon. If you run version 9.1 at home, please update if you want this document to be much help.

Basics

For basic installation and program help, see the previous handout entitled *Introduction to PSpice 9.2*, or *Introduction to PSpice 9.1*. Close all other programs— PSpice doesn't seem to share well.

Assignment

1. Make some i_{DS} vs v_{DS} curves like you got from the curve tracer in lab 11.

Create a new directory for your new project. Start a new project, give it a name, and make the location your new directory. Create a blank project.

Place an n-channel MOSFET. The only model we have is the IRF150, (in EVAL lib) which is a power device, so you'll have to change some parameters after you have it on your schematic.

Highlight the FET.

Edit -> PSpice Model or right-click on the highlighted part and select "Edit PSpice Model".

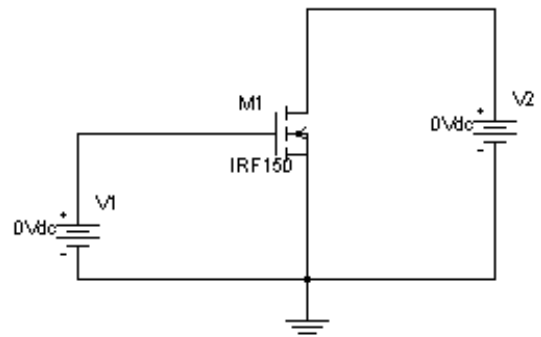
All lines starting with * are comments. The important lines are the .model and the "+" lines that follow it.

1. Change the Level to Level=1
2. Type in Lambda=0.014 right after the Level=1
3. Change W to 225u (or value calculated from lab 11, see below)
4. Change L to 8u (or value calculated from lab 11)
5. Change Vto to 1.3 (or your V_t value from lab 11)

Ideally, you should use values that you found in Lab 11 to modify these numbers. If you have your lab notebook, look at the Ohmic Measurements section. Find the values of V_t and $k_n'W/L$ that you found from the plot of $1/r_{DS}$ vs V_{GS} . Enter your value of V_t instead of the Vto given above. Adjust the W in the model until the model's K_pW/L is equal to your $k_n'W/L$. Furthermore, you can find lambda (λ) from the slopes of the i_{DS} vs v_{DS} curves from the curve tracer, see Fig. 5.16, p 371 in your textbook. If you don't have the time to do all this, I understand, Just use my values above, but you won't learn quite as much from this assignment.

File --> Save, to save your new model.

Draw a schematic like the one shown. When you place the ground, change its name to 0 before you place it. My computer crashed twice when I tried to change its name later. I don't know why, I'm just giving you a "heads up".



Setup the Simulation

PSpice -> New Simulation profile -> give the simulation a name, say "curves" -> Create.
Analysis -> DC Sweep. Primary sweep, Voltage Source, Name V2, Linear, Start Value 0, End Value 20, Increment 0.2.

Secondary Sweep, Voltage Source, Name V1, Linear, Start Value 0, End Value 10, Increment 1. OK

Save.

Run.

Trace, Add Trace, ID(M1). You should get something that looks like the curves you got from the curve tracer. If you want them to look closer to what you saw in the lab, adjust the W value in the model and try again.

Problems? Check model changes. Check Simulation profile and read the steps above carefully.

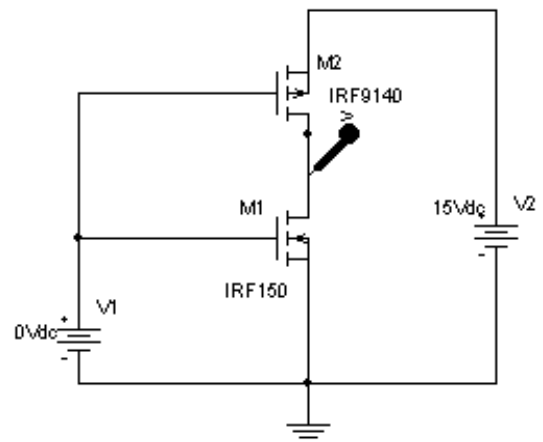
Print these curves.

2. Make a transfer curve like you did in Experiment 3 of lab 11.

Modify your circuit to the one shown at right. When you place the IRF9140 part it will be upside down at first. To flip it over, click on it to highlight it, right-click, Mirror Vertically.

Highlight the new P-FET. Edit -> PSpice Model or right-click on the highlighted part and select "Edit PSpice Model".

1. Change the Level to Level=1
2. Type in $\Lambda=0.014$ right after the Level=1
3. Change W to 450u
4. Change L to 8u
5. Change Vto to -1.3



Ideally again, you'd should use values that you found in Lab 11 instead of these numbers.

File --> Save, to save your new model.

PSpice -> Edit Simulation profile ->

Turn off the secondary sweep, if it's still on. Primary sweep, Voltage Source, Name V1, Linear, Start Value 0, End Value 15, Increment 0.2 OK

Save.

Run.

You should get something that looks like the transfer curve you got in lab 11. If you want it to look closer to what you saw in the lab, adjust the model values and try again.

Print these curves.

3. Finish

Print your schematic, label the nodes like you've done before, using the netlist node names. Staple the three sheets together, write your name on them and hand them in— you're done.

P.S. Did you learn something from this??? Please say "yes".