Lab 5—ECE 230L

MOSFET Simulation with PSpice

Table of Contents

List of Figures

Objectives of this Laboratory

- Simulate an NMOSFET in PSpice
- Use manufacturer, measured, and empirical average NMOSFET model parameters to evaluate the transistor in PSpice
- Compare the measured threshold voltage, transconductance, and channel-length modulation parameters to those specified by the manufacturer

PSpice Simulation of a MOS Field-Effect Transistor

The circuit used in the electrical characterization of the MOSFET is shown in figure 1. Simulate the drain current-voltage characteristics using PSpice of the BS170 Metal-Oxide Semiconductor Field-Effect Transistor (MOSFET). The PSpice code to simulate and plot the ID (VGS, VDS) drain-current characteristics of this N-channel MOSFET is given below. Use this model to create a PSpice input file for simulating the MOSFET characteristics. Note in particular the model parameters used for the threshold voltage (VTO), the transconductance (KP), and the channel-length modulation parameter (lambda).

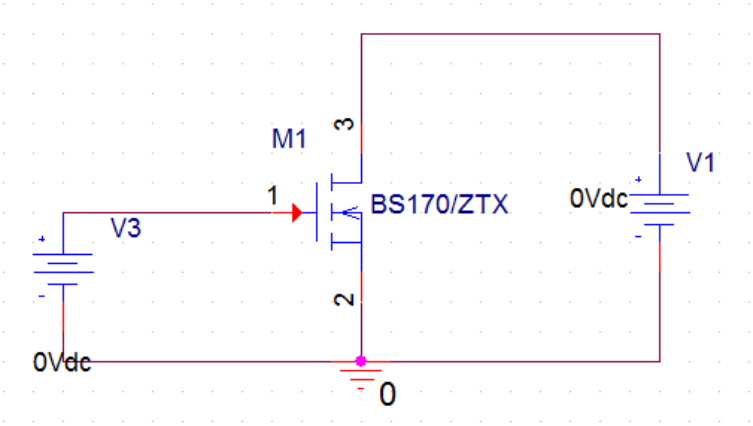


Figure 1: Circuit used to characterize a NMOSFET

Placing a NMOS Transistor

To add a transistor to your schematic select Place in your toolbar and then select Part. Use the 'Add Library' button shown in figure 2 to search the Library -> PSpice folder for the 'ZETEK' library.

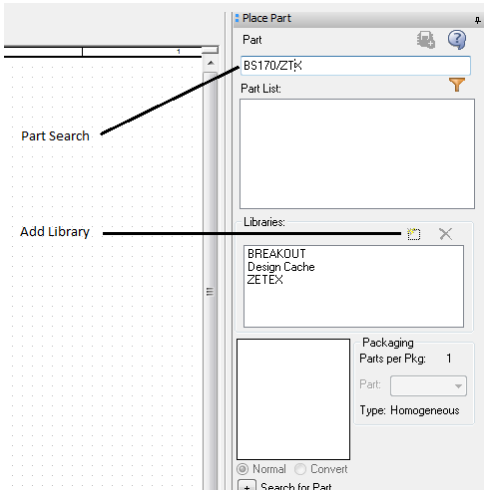


Figure 2: Place part menu

Once you add the 'ZETEK' library, search for BS170/ZTX. Add this part to your schematic. You will need to add voltage sources to your schematic to represent VGS and VDS and wire them to match the circuit in figure 1.

***NOTE:** To add a library while using PSpice LITE, refer to Appendix A. The correct .OLB and .LIB files are on Sakai.

Setting up the Simulation Profile

Use the following parameters to simulate the ZETEK BS170/ZTK MOSFET:

- Analysis Type: DC Sweep
- Primary Sweep
    - Voltage Source: VDS (V1 in figure 1)
    - Sweep Type: Linear
    - Start Value: 0v
    - End Value: 5v
    - Increment: 0.01v
- Secondary Sweep
    - Voltage Source: VGS (V3 in figure 1)
    - Sweep Type: Linear
    - Start Value: 2v
    - End Value: 3v
    - Increment: 0.25v

Run your simulation and save your results. Use I(M1:D) as your trace. The results of the ZETEK BS170/ZTK simulation are shown in figure 3.

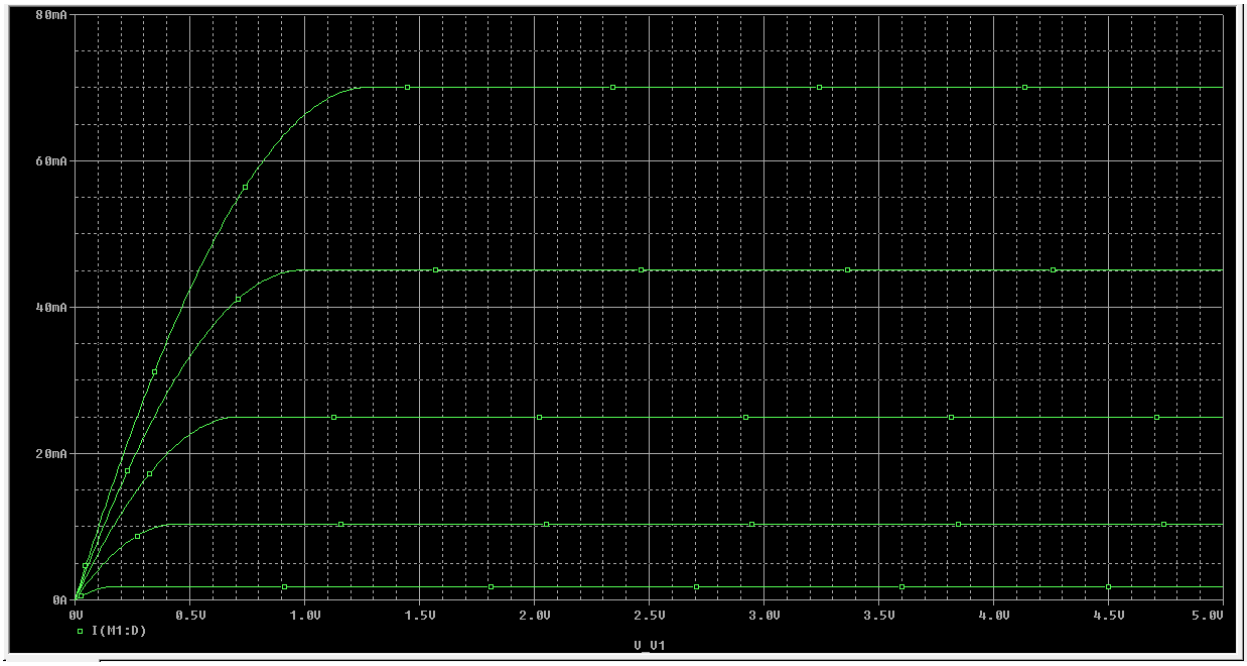


Figure 3: Results from ZETEK BS170/ZTK simulation

Changing Netlist Parameters

To model a MOSFET with your extracted parameters you must edit the BS170/ZTK netlist. Click on the BS170/ZTK symbol and select Edit -> PSpice Model. The netlist is under the 'Model Text' heading. The Model Editor is pictured in figure 3

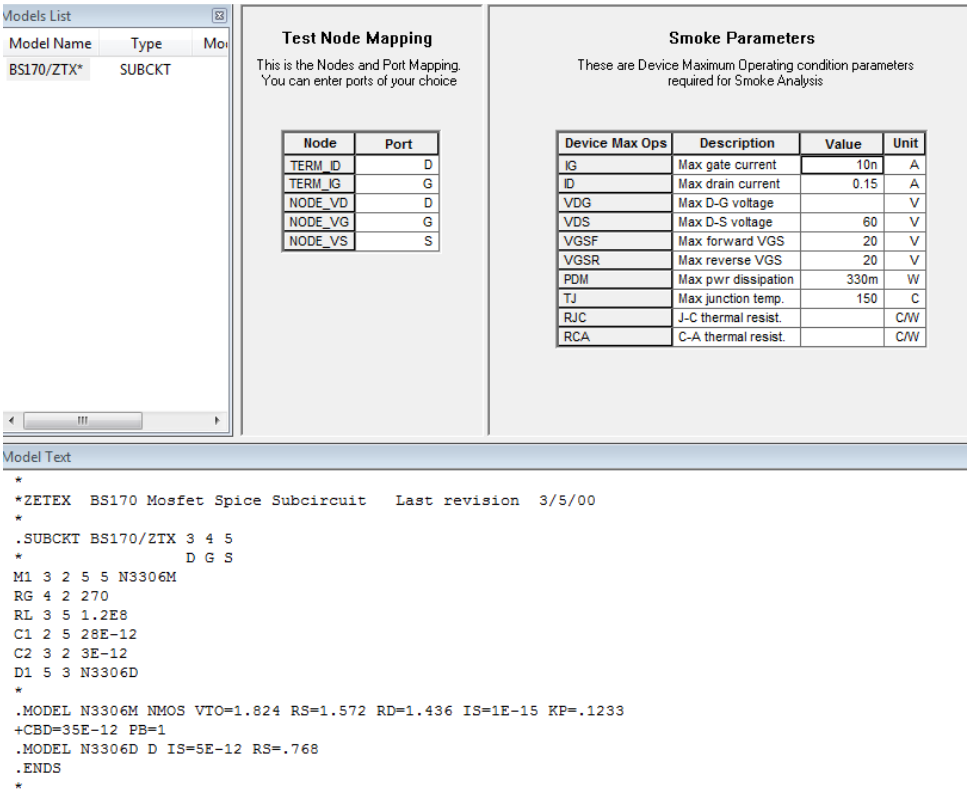


Figure 4: Model editor with original BS170/ZTK netlist

Simulating Your Extracted BS170 Parameters

Replace the parameters in the ".MODEL…" text with your own extracted parameters. Save the new netlist. Run your simulation and save your results. You should have obtained these parameters in Lab 4: MOSFET Characterization.

Simulating Class Average BS170 Parameters

Replace the parameters in the ".MODEL…" text with the average extracted parameters. Save the new netlist. Run your simulation and save your results.

Simulating Manufacturer Specified BS170 Parameters

Replace the parameters in the ".MODEL…" text with the parameters found in the Fairchild Semiconductor netlist. Save the new netlist. Run your simulation and save your results.

Note: The BS170 used in Lab 4 was made by Fairchild Semiconductor. Remember that parameters can vary by manufacturer. The ZETEK and Fairchild Semiconductor parameters may be different.

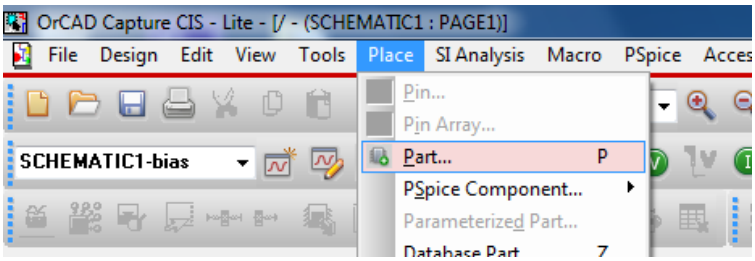
Include in the simulation section of your lab the following:

- A schematic of the circuit used to simulate the MOSFET. Be sure to include all voltage sources in your circuit diagram. This is the PSpice file used to simulate this NMOSFET drain-current characteristics.
- Three separate plots of the simulation results of ID vs. VDS for different VGS values for manufacturer, measured, and empirical average.
- Compare the measured threshold voltage, transconductance, and channel-length modulation parameters for manufacturer, measured, and empirical average. Comment on how these three parameters affect the PSpice simulated output plots.

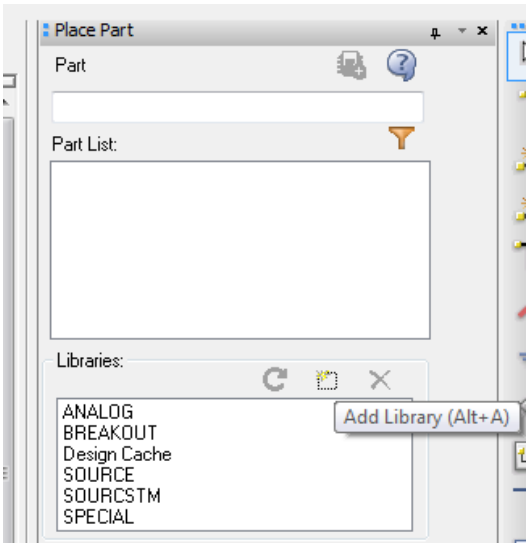
Appendix A: Adding a new library in PSpice LITE

Adding new parts in PSpice requires two types of files: .olb and .lib. These files provide the part's symbol and model information for PSpice simulation. Follow the steps below to import a new part into PSpice.

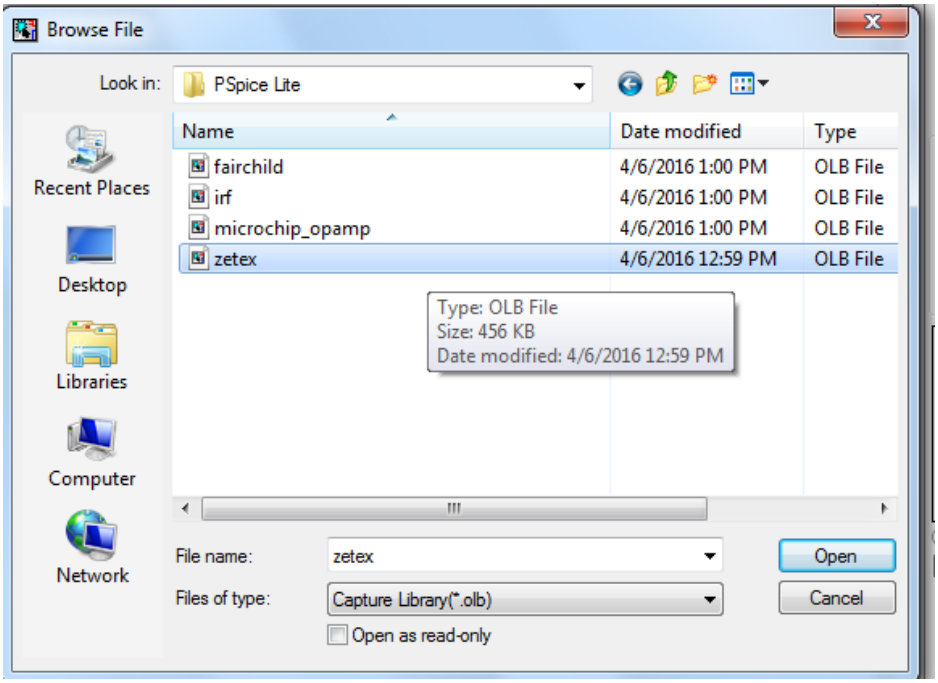
Navigate to the Place – Part menu



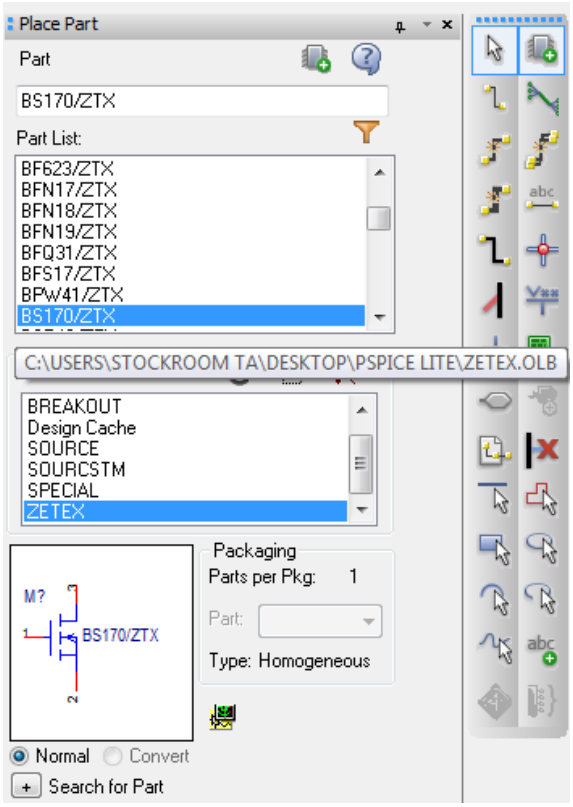
Press "Add Library" in the Place Part menu (under Libraries)



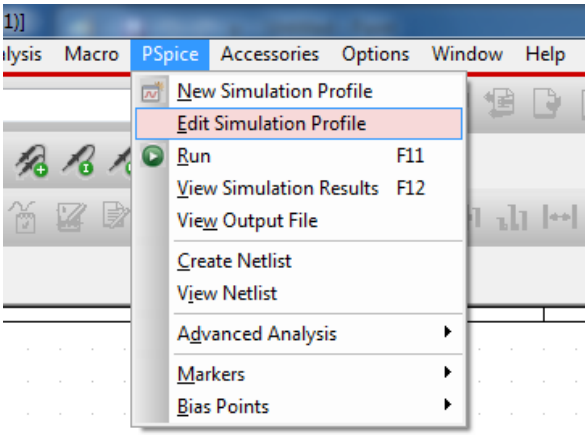
Browse to the location where you installed the .olb file from Sakai



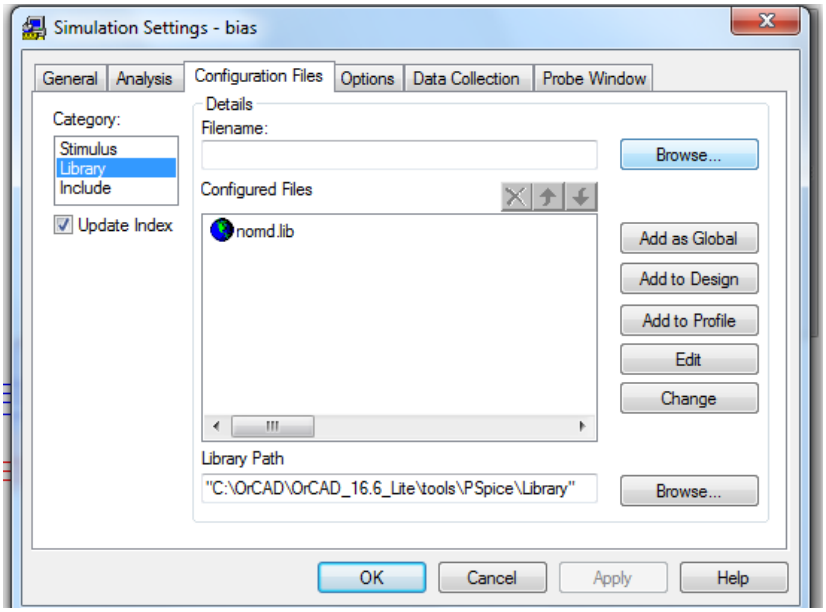
Select the new library and find the part you want to include in your circuit



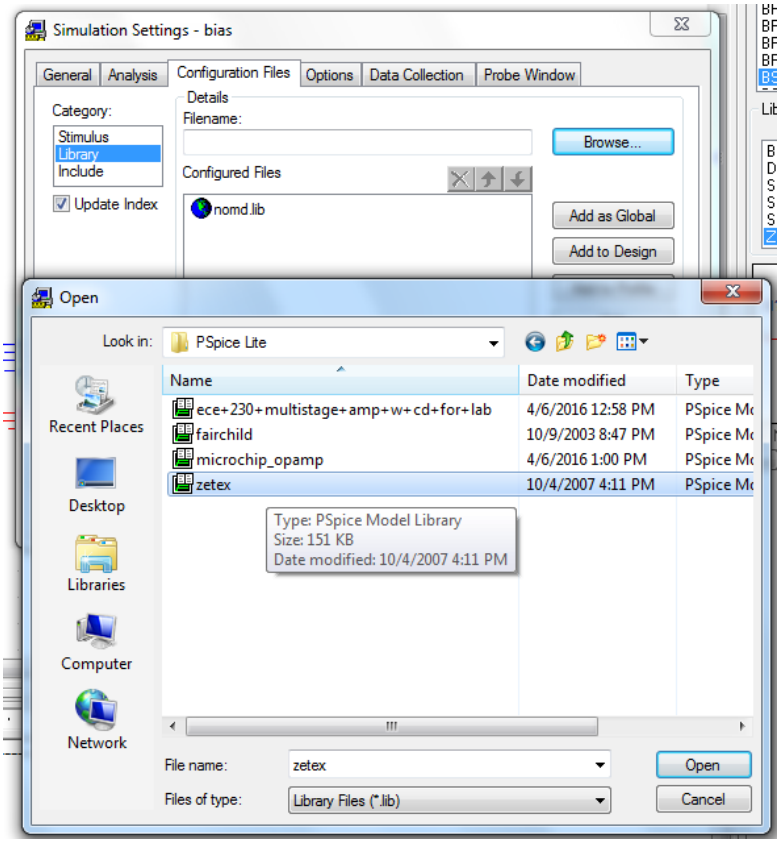
Open the simulation profile editor



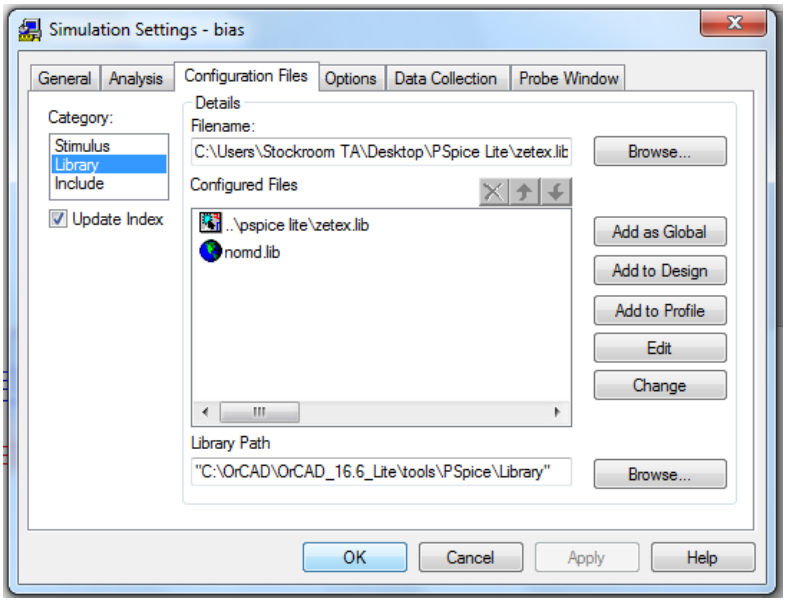
Open the configuration files tab and choose the library category



Browse to the location where you installed the .lib file from Sakai



Open the file, add to design, click Apply, and OK



You are now ready to run your model and view the results

ECE 230L Lab 5 MOSFET Simulation with PSpice Grading Guide

	Points Possible
Circuit Schematic used to simulate MOSFET	10 points
Verification of ZETEK BS170 Plots	10 points
Extracted BS170 parameter simulation - **I_{DS} vs. V_{DS}(V_{GS}) plotted**	15 points
Class average BS170 parameter simulation - **Average calculation (remove outliers)** - **I_{DS} vs. V_{DS}(V_{GS}) plotted**	15 points
Manufacturer specified BS170 parameter simulation - **I_{DS} vs. V_{DS}(V_{GS}) plotted**	15 points
Compare relative values of V_{TH}, K_P, and λ for all three simulations - **Include percent errors**	15 points
Comments on the effects of V_{TH}, K_P, and λ on the graphs	10 points
Quality of thought/analysis	10 points