ECE 230L - LAB 6

MOSFET SIMULATION WITH PSPICE

Contents

List of Figures

1   Objectives of this Laboratory

The objectives of this laboratory session are as follows:

- 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

2   PSpice simulation of a MOSFET

The circuit used in the electrical characterization of the MOSFET is shown in Figure 1. Using PSpice, simulate the drain current-voltage characteristics of the BS170 Metal-Oxide Semiconductor Field-Effect Transistor (MOSFET). The PSpice code to simulate and plot the $I_D(V_{GS}, V_{DS})$ 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).
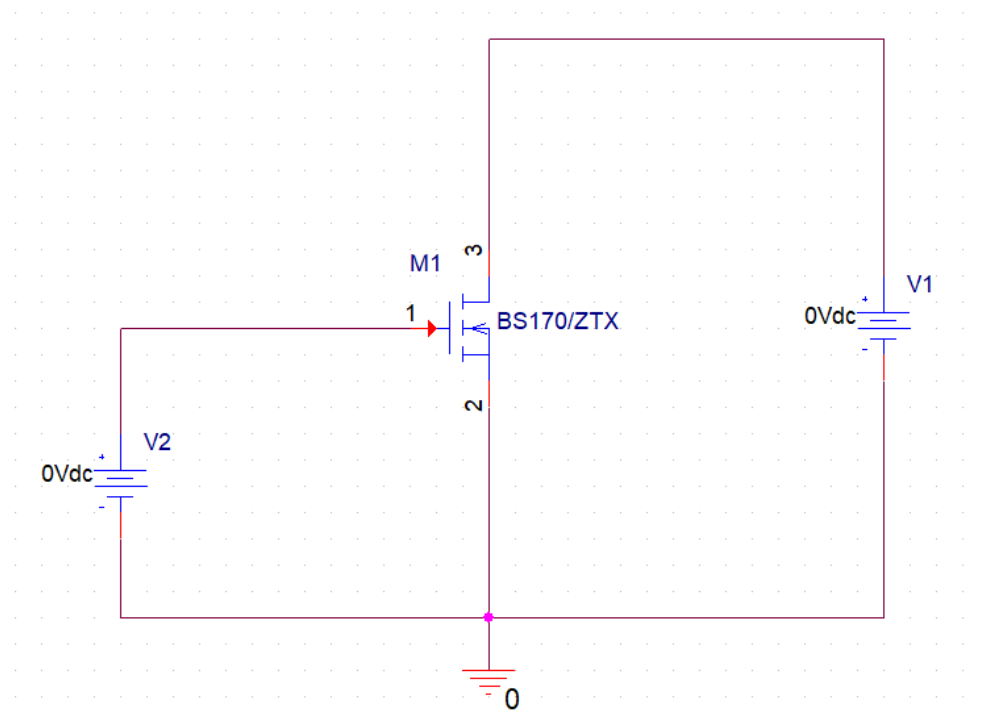

Figure 1: Circuit used to characterize an NMOSFET

2.1   Placing an 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 'ZETEX' library.
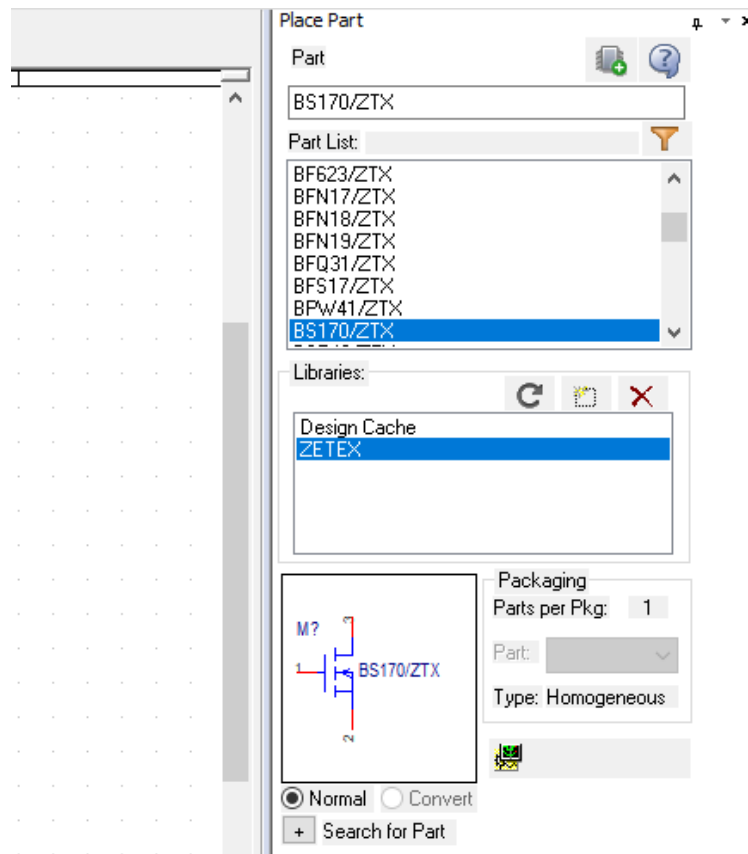

Figure 2: Place part menu

Once you add the 'ZETEX' library, search for BS170/ZTX. Add this part to your schematic. You will need to add voltage sources to your schematic to represent V_{GS} and V_{DS} and wire them to match the circuit in Figure 1.

2.2   Setting up the Simulation Profile

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

- Analysis Type: DC Sweep

- Primary Sweep

    - Voltage Source: V_{DS} (V_1 in Figure 1)

    - Sweep Type: Linear

    - Start Value: 0 V

    - End Value: 5 V

    - Increment: 0.01 V

- Secondary Sweep

    - Voltage Source: V_{GS} (V_2 in Figure 1)

    - Sweep Type: Linear

    - Start Value: 2 V

    - End Value: 3 V

    - Increment: 0.25 V

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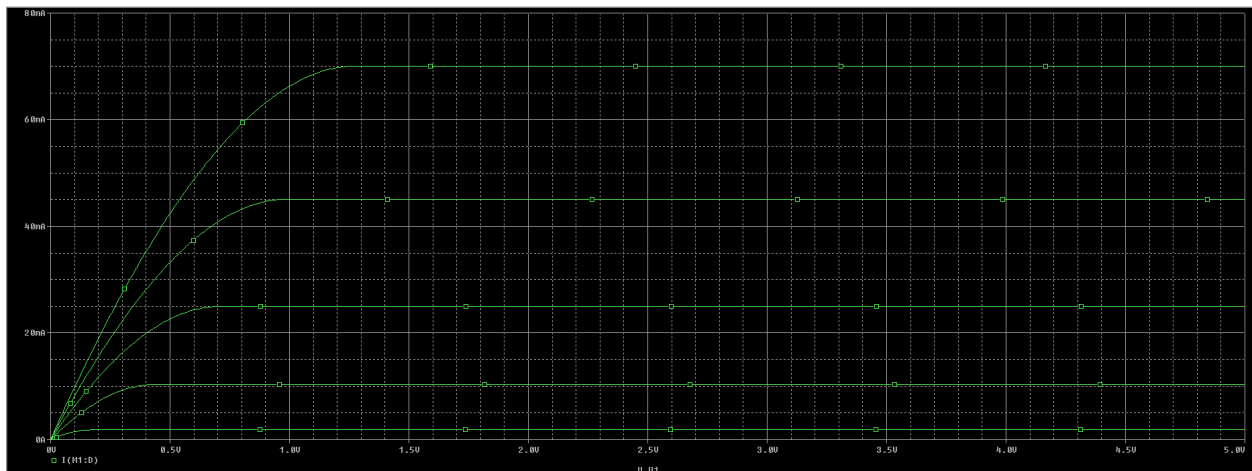

Figure 3: Results from ZETEX BS170/ZTX simulation

2.3   Changing Netlist Parameters

To model a MOSFET with your extracted parameters you must edit the BS170/ZTX netlist. Click on the BS170/ZTX symbol and select **Edit → PSpice Model**. The netlist is under the 'Model Text' heading. The Model Editor is pictured in Figure 4.
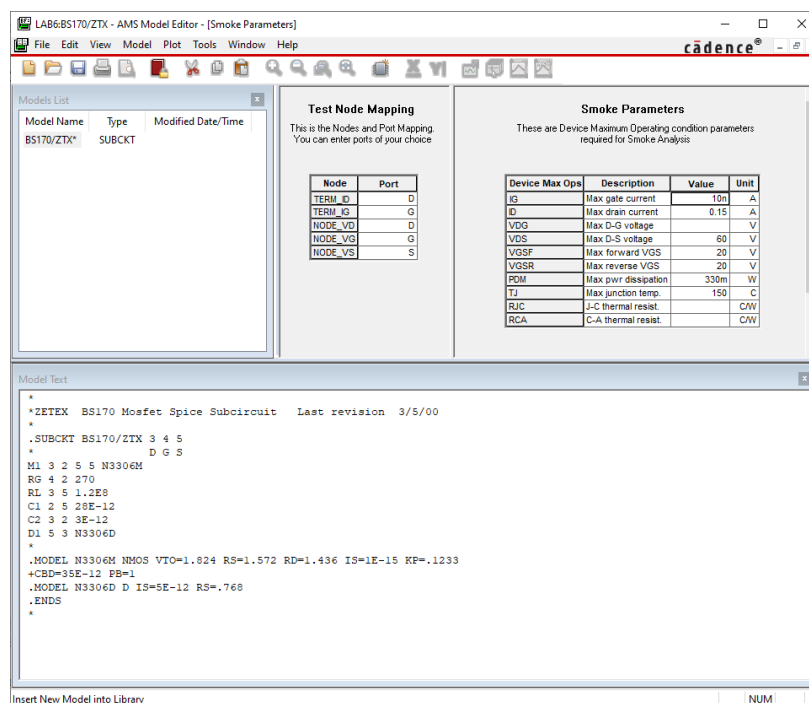

Figure 4: Model editor with original BS170/ZTX netlist

2.3.1   Simulating Your Extracted BS170 Parameters

Replace the parameters in the ".MODEL. . ." text with your own extracted parameters. You will need to add a `lambda` parameter. Save the new netlist. Run your simulation and save your results. You should have obtained these parameters in Lab 5: MOSFET Characterization.

2.3.2   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.

2.3.3   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 5 was made by Fairchild Semiconductor. Remember that parameters can vary by manufacturer. The ZETEX 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 I_D vs. V_{DS} for different V_{GS} 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.

Table 1: ECE 230L Laboratory 6 Grading Rubric

Criteria	Points Possible
Circuit Schematic used to simulate MOSFET	**10**
Verification of ZETEK BS170 Plots	**8**
Extracted BS170 parameter simulation	**15**
Plot of I_{DS} vs $V_{DS}(V_{GS})$	15
Class average BS170 parameter simulation	**20**
Average Calculation	5
Plot of I_{DS} vs $V_{DS}(V_{GS})$	15
Manufacturer specified BS170 parameter simulation	**15**
Plot of I_{DS} vs $V_{DS}(V_{GS})$	15
Compare values of V_{TH}, K_p, and λ for all three simulations	**15**
Comment on the effects of V_{TH}, K_p, and λ on the graphs	**10**
Extension	**7**
Total	**100**