

EE 304
MOSFET Characteristics
Graded Spice Assignment – 25 points

You are expected to do the following analysis and simulation problem alone. Obtaining outside assistance from other than the instructor will be considered a breach of academic integrity as well as the ECE Student Code of Conduct.

1. Consider an NMOS transistor with $V_{t+} = .7V$, $k_n' = 100 \mu A/V^2$, $W = 10 \mu m$, and $L = 1 \mu m$. (15pts)

(a) Carefully hand plot I_D versus V_{DS} from 0V to 5V, for V_{GS} ranging from 0V to 3V in steps of .5V increments. For each value of V_{GS} above V_{t+} , give the value of V_{DSsat} and I_D at V_{DSsat} . Show all your calculations and make sure your plot is neat and accurate.

(b) Use SPICE to verify your curves plotted in part (a) by doing primary and parametric DC sweeps. For the NMOS transistor use the MbreakN part from the Breakout Library and edit the SPICE model to conform to the NMOS specifications given above. The SPICE parameters for V_{t+} and k_n' are given in Table 4.7 of your text. The SPICE parameters for W and L are, cleverly, W and L . To edit the SPICE model, first left click on the MOSFET to highlight the part, right click to open a menu and then left click on "Edit PSpice Model" to open the PSpice Model Editor. In the editing window insert appropriate values as PARAMETER=value with spaces between entries. Also, in the schematic be sure to connect the body terminal to the source terminal. Annotate and submit your I_D versus V_{DS} curves found using SPICE as well as the circuit you used to obtain the curves.

(c) Redo part (b) but include the effect of channel length modulation. In addition to the specifications given above, assume $V_A = 50V$ or $\lambda = .02 V^{-1}$. The SPICE parameter for λ is LAMBDA. Annotate and submit your I_D versus V_{DS} curves obtained using SPICE.

2. Repeat parts (a) and (b) for a PMOS transistor (you don't have to repeat (c).) In this case, assume $V_{t+} = -.7V$, $k_p' = 100 \mu A/V^2$, $W = 10 \mu m$, and $L = 1 \mu m$. Also, plot I_D versus V_{DS} from 0V to -5V, for V_{GS} ranging from 0V to -3V in steps of -.5V increments. For the PMOS transistor use the MbreakP part from the Breakout Library and edit the SPICE model to conform to the PMOS specifications given above. Note that the same SPICE parameter symbol is used for k_n' and k_p' . (10pts)