

Lab Assignment: MOS/SPICE

1 SPICE PARAMETER EXTRACTION (SEE P-127, SECTION 3.5.1 IN [HODGES])

Problem 1:

- (a) Extract V_{T0} , μ , γ for both NMOS ($W/L = 5\mu/1\mu$) and PMOS ($W/L = 5\mu/1\mu$) [Use $t_{ox} = 7.6\text{nm}$]

The square-root of saturation current can be expressed as

$$\sqrt{I_D} = \sqrt{\frac{\mu C_{OX} W}{2 L}} (V_{GS} - V_{T0})$$

As shown in figure below V_{T0} & μ can be extracted from the intercept and slope of the $\sqrt{I_D} - V_{GS}$ plot.

In order to extract the body-effect coefficient γ , $\sqrt{I_D}$ vs V_{GS} is plotted for a different source-bulk voltage (V_{SB}) and γ can be extracted as

$$\gamma = \frac{V_T(V_{SB}) - V_{T0}}{\sqrt{|2\phi_F| + V_{SB}} - \sqrt{|2\phi_F|}}$$

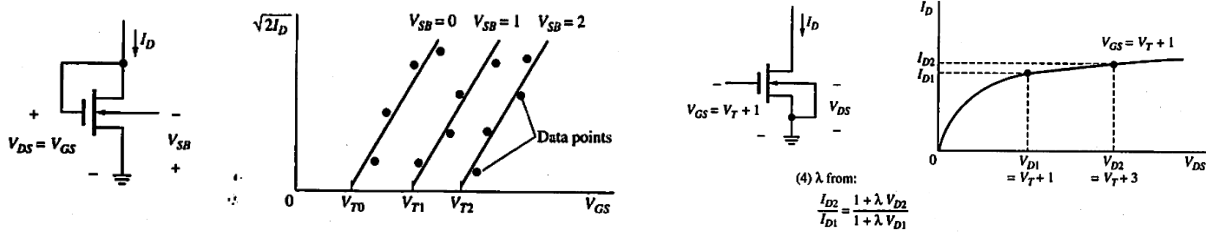


Figure 1: Test setup and measurement method for parameter extraction

- (b) Extract λ for both NMOS ($W/L = 5\mu/1\mu$) and PMOS ($W/L = 5\mu/1\mu$) using the following test setup and extracting the parameter using the expression:

$$\frac{I_{D2}}{I_{D1}} = \frac{1 + \lambda \cdot V_{DS2}}{1 + \lambda \cdot V_{DS1}}$$

2 PERFORMANCE COMPARISON BETWEEN LEVEL-1 AND BSIM3v1 LEVEL-49 MODEL

Using $NSUB=1.7E17$ (to calculate Φ for both NMOS & PMOS), and V_{T0} , K_P , $LAMBDA$, Φ , $GAMMA$ extracted from the above problem assignment, create a LEVEL 1 SPICE Model for both NMOS and PMOS with the extracted parameters. For the model syntax, please refer [Hodges] Example 3.3 P-128 or [Kang-2ed] (page: 117-123, Chapter-4).

1. Using the created LEVEL 1 model, plot I_D vs V_{GS} and I_D vs V_{DS} for both NMOS and PMOS.
2. Compare the LEVEL-1 plot results with the BSIM3v1 LEVEL-49 plot results by overlaying the plots in a single graph. Do this for below two cases.
 - a. Case 1: $L=1\mu m$ and $W=5\mu m$
 - b. Case 2: $L=0.4\mu m$ and $W=0.6\mu m$

Comment on the differences between the two levels of models.

In the above two cases the parameters (V_{T0} , K_P , $LAMBDA$, Φ and $GAMMA$) values in the LEVEL-1 model will be same, what you have extracted in Q-1

Instruction: For doing the above exercise, modify your SPICE NETLIST file according to the requirements and run the simulations in command prompt. If you have doubt on model file definition in the SPICE NETLIST, please go through any SPICE NETLIST FILE and understand thoroughly. To define LEVEL-1 model file in SPICE NETLIST please refer [Hodges] Example 3.3 P-128 or [Kang-2ed] (page: 117-123, Chapter-4).