EE 320 L ELECTRONICS I

LABORATORY 9: MOSFET TRANSISTOR CHARACTERIZATIONS

DEPARTMENT OF ELECTRICAL AND COMPUTER ENGINEERING
UNIVERSITY OF NEVADA, LAS VEGAS

1. OBJECTIVE

Get familiar with MOSFETs, enhance the understanding of MOSFET characteristic under various DC bias scenarios, and learn how MOSFET are applied in practical circuits, e.g. working as switches, amplifiers, etc.

2. COMPONENTS & EQUIPMENT

Power Supply
Function Generator
Multimeter
Oscilloscope
Breadboard & Jump wires
Resistors & Capacitors
MOSFET (2N4351, ZVN3306, ZVP3306)

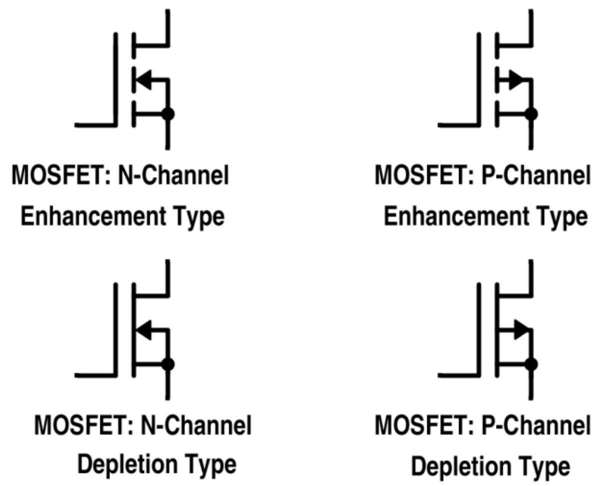
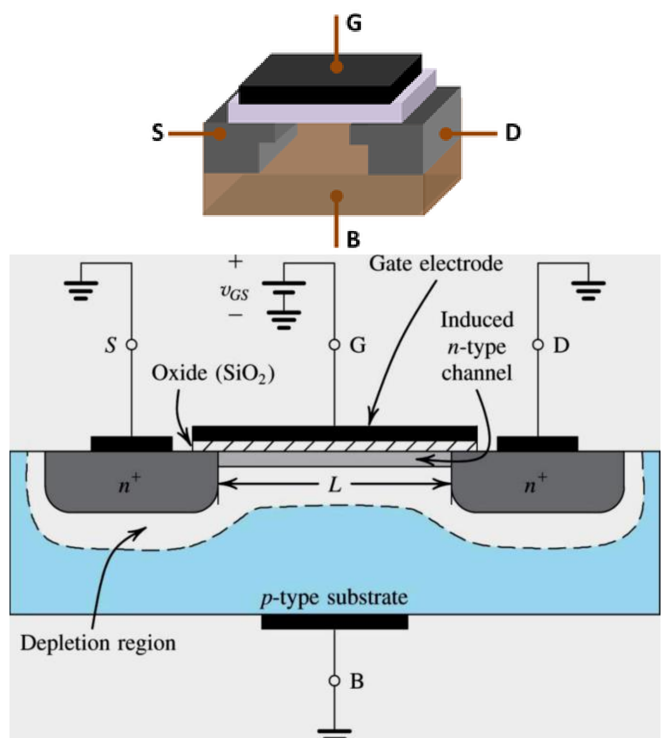
3. BACKGROUND

The metal-oxide-semiconductor field-effect transistor (MOSFET, MOS-FET, or MOS FET) is a type of field-effect transistor (FET), most commonly fabricated by the controlled oxidation of silicon. It has an insulated gate, whose voltage determines the conductivity of the device. This ability to change conductivity with the amount of applied voltage can be used for amplifying or switching electronic signals.

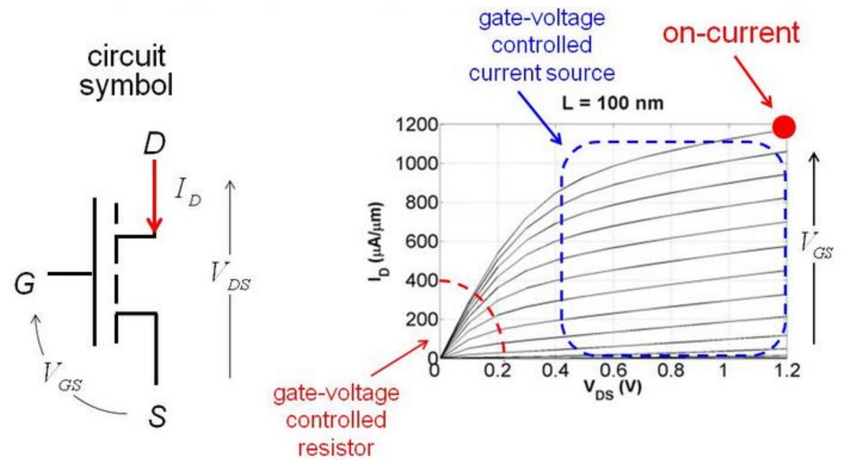
The main advantage of a MOSFET is that it requires almost no input current to control the load current, when compared with bipolar transistors. It is rather a **<u>voltage control current source</u>**. There are two different modes of MOSFETs: 1) in an enhancement mode MOSFET, voltage applied to the gate terminal increases the conductivity of the device; 2) in depletion mode transistors, voltage applied at the gate reduces the conductivity.

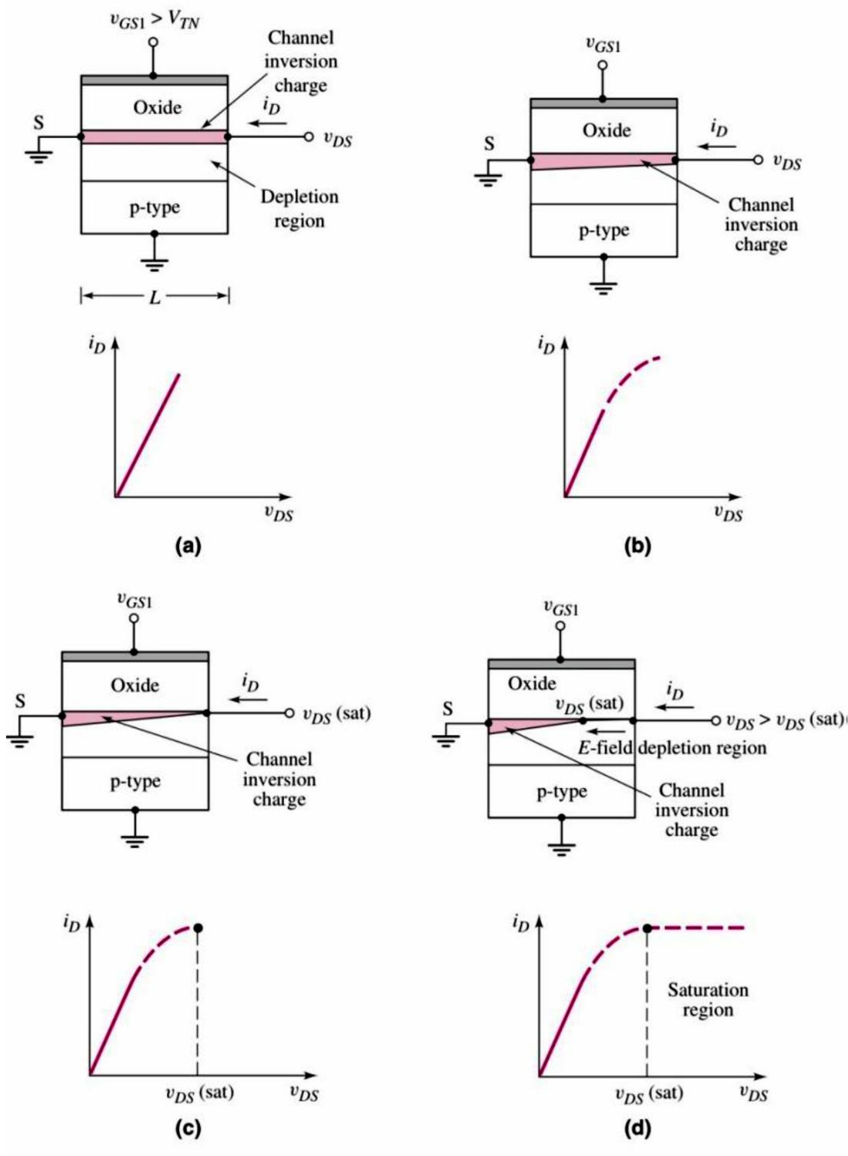
The MOSFET is by far the most common transistor in digital circuits, as millions may be included in a memory chip or microprocessor. Since MOSFETs can be made with either p-type or n-type semiconductors, complementary pairs of MOS transistors can be used to make switching circuits with very low power consumption, in the form of CMOS logic.

Key knowledges and formulas related to BJT amplifiers.

MOSFET symbols	
Cross-section view of an n-type MOSFET (NMOS)	

NMOS I-V characteristic

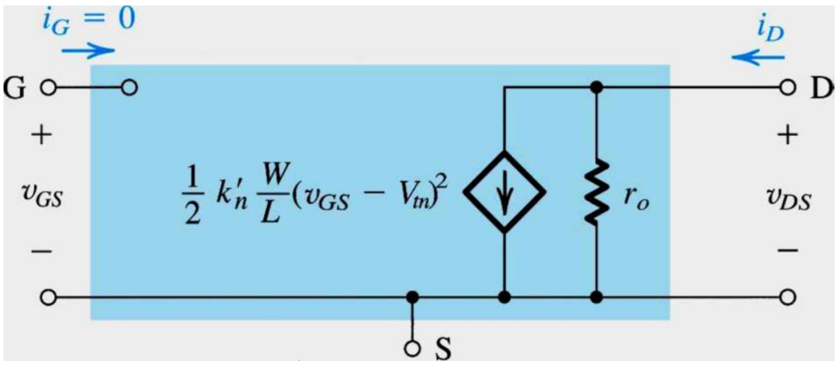




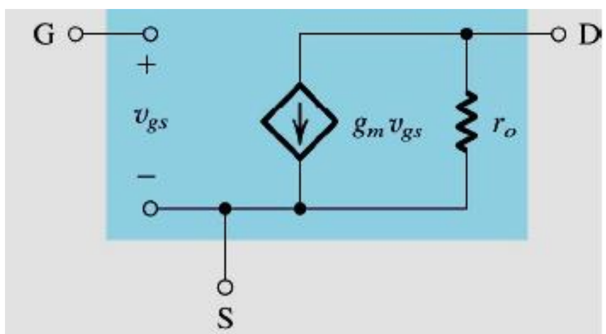
IV status table:

Cut Off	$V_{GS} \le V_T$	$I_{DS} = 0$
Linear	$V_{GS} > V_T$, $V_{DS} \le V_{GS} - V_T$	$I_{DS} = \mu_n C_{ox} \frac{W}{L}\left[(V_{GS} - V_T)V_{DS} - \frac{V_{DS}^{\ 2}}{2}\right](1 + \lambda V_{DS})$
Saturation	$V_{GS} > V_T$, $V_{DS} > V_{GS} - V_T$	$I_{DS} = \frac{1}{2}\mu_n C_{ox} \frac{W}{L}(V_{GS} - V_T)^2(1 + \lambda V_{DS})$

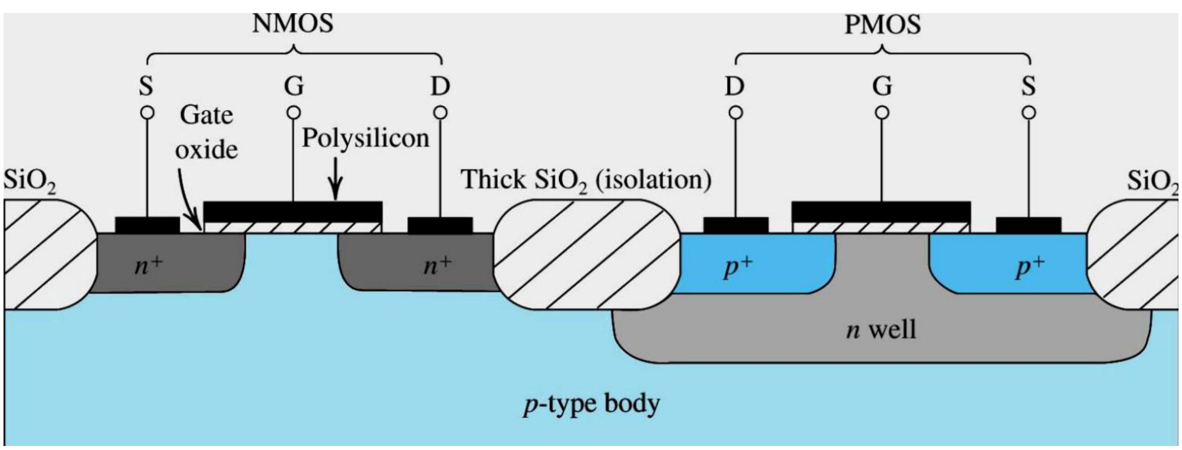
Large-signal equivalent model



Small-signal equivalent model



Complementarty MOS or CMOS:



4. LAB DELIVERIES

PRELAB:

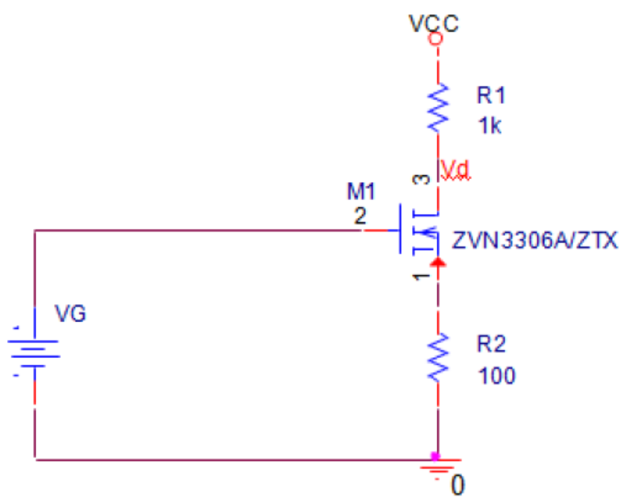
1. **Go over the basic configuration and IV characterization of MOSFETs, partial key knowledge of which is listed in the previous section.**

2. **Overview the key character of diodes in their datasheets.**

2N4351	http://www.calogic.net/pdf/2N4351_Datasheet_Rev_A.pdf
ZVN3306A	http://eelabs.faculty.unlv.edu/docs/datasheets/ZVN3306a.pdf
ZVP3306A	http://eelabs.faculty.unlv.edu/docs/datasheets/ZVP3306a.pdf

3. **Use LTspice to simulate Circuit 1.**

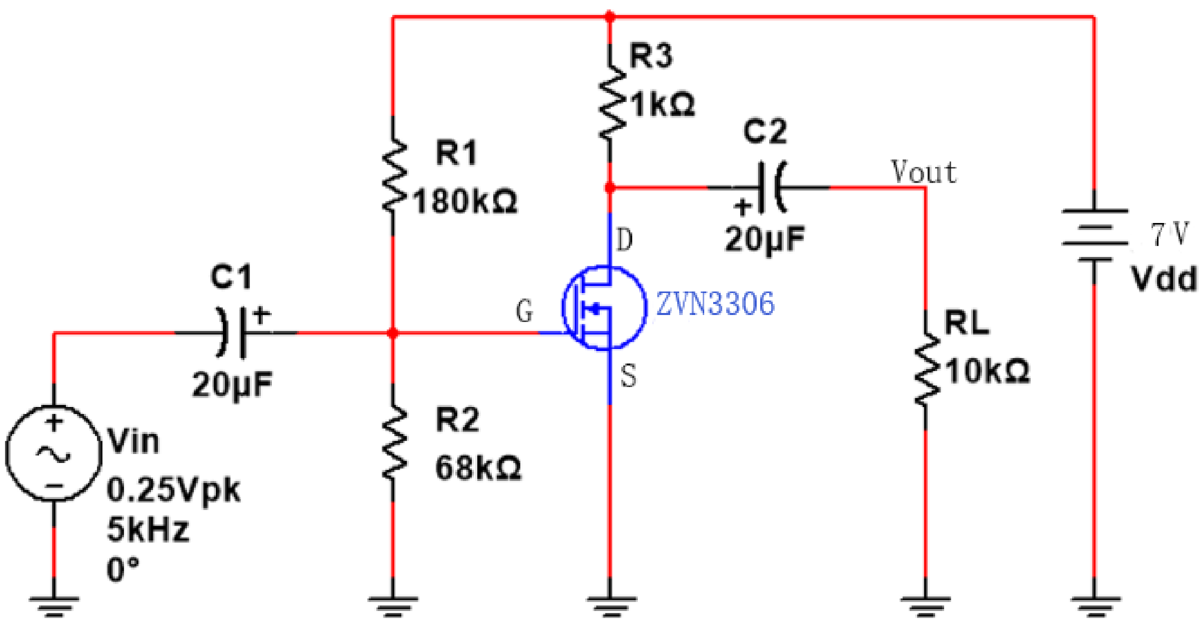
   1) Set VCC = 5V, and VG as a sine wave with amplitude of 5V, frequency of 10Hz. Observe the voltage at node 1 and 3 (i.e. V(s) and V(d)), and the current running through (i.e. I(ds)), respectively. Write down the peak voltage (V_p) and peak-to-peak voltage (V_{pp}) of V(c), and screenshot the output waveforms.
   2) Run ac analysis for the input source (i.e. VG) from 10Hz to 1MHz.
   3) Set VG = 3V, and VCC as a sine wave with amplitude of 5V, frequency of 10Hz. Observe the voltage at node 1 and 3 (i.e. V(s) and V(d)), and their currents (i.e. I(ds)), respectively.
   4) Run ac analysis for the input source (i.e. VCC) from 10Hz to 1MHz.
   5) Change NMOS to PMOS (i.e. ZVP3306A), and repeat 1) ~ 4).



Circuit 1

4. Use LTspice to simulate Circuit 2.

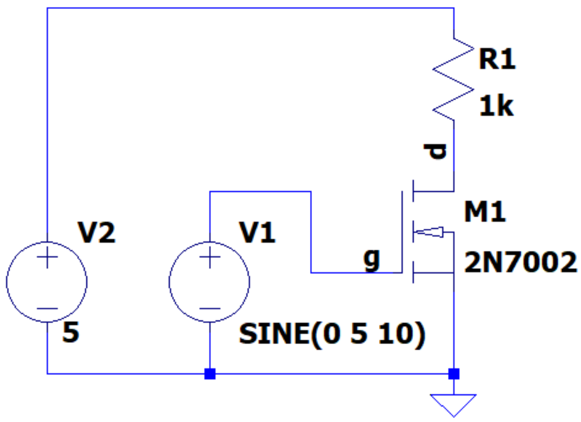
1) Run transient analysis to observe the voltages at Node G, D and S of NMOS. b, c, e nodes of the BJT. Write down the peak voltage (V_p) and peak-to-peak voltage (V_{pp}) of Vout. Computer $A_v = V_{pp}(D)/V_{pp}(G)$ and $A'_v = Vout/Vin$.
2) Compute the input impedance by using $Z_{in} = Vin_{pp}/I_{pp}(C1)$. Note that there may be phase difference between Vin and I(C1).
3) Run the ac analysis for the Bode plot of the AC input from 10Hz to 1MHz.
4) Remove/Short R_L to re-measure and compute $Z_{out} = V_{pp}(D)/I_{pp}(DS)$.



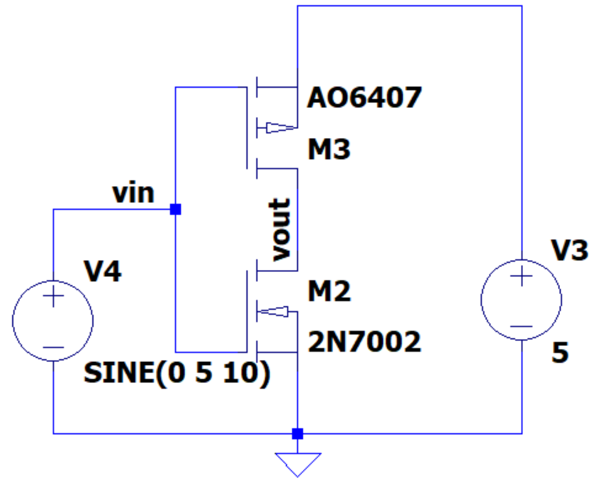
Circuit 2

5. Use LTspice to simulate Circuit 3 and Circuit 4.

1) Set V1 and V4 sine waves of $V_{pp} = 5V$ and frequency of 10Hz. Observe the voltages and currents at Node d in Circuit 3 (pull down NMOS circuit), and Node vout in Circuit 4 (i.e. CMOS circuit), respectively.
2) Run AC analysis for the Bode plot both circuits, respectively, with input frequencies form 10Hz to 1MHz.
3) Run DC sweep for V1 and V4, from 0V to 5V with increment of 0.1V, respectively. See the VTC between each input and output pair.



Circuit 3



Circuit 4

LAB EXPERIMENTS:

1. **Implement and measure Circuit 1 in Prelab Experiment 3 on breadboard, and compare with LTspice results.**
   - Use frequencies of 10Hz, 100Hz, 1KHz, 10KHz, 100KHz, and 1MHz for Bode plot.

2. **Implement and measure Circuit 2 in Prelab Experiment 4 on breadboard, and compare with LTspice results.**
   - Use frequencies of 10Hz, 100Hz, 1KHz, 10KHz, 100KHz, and 1MHz for Bode plot.

3. **Implement and measure Circuit 3 and Circuit 4 in Prelab Experiment 5 on breadboard, and compare with LTspice results.**
   - Use ZVN3306 and ZVP3306 instead.
   - Use frequencies of 10Hz, 100Hz, 1KHz, 10KHz, 100KHz, and 1MHz for Bode plot.

POSTLAB REPORT:

Include the following elements in the report document:

<table>
<tr><th>Section</th><th colspan="2">Element</th></tr>
<tr><td>1</td><td colspan="2">Theory of operation
Include a brief description of every element and phenomenon that appear during the experiments.</td></tr>
<tr><td>2</td><td colspan="2">Prelab report
1. Hand calculation results of Prelab Experiment 3~5.
2. LTspice schematics and simulation results of Prelab Experiment 3~5.</td></tr>
<tr><td rowspan="5">3</td><td colspan="2">Results of the experiments</td></tr>
<tr><th>Experiments</th><th>Experiment Results</th></tr>
<tr><td>1</td><td>Screenshots of LTspice simulations and oscilloscope waveforms.</td></tr>
<tr><td>2</td><td>Screenshots of LTspice simulations and oscilloscope waveforms.</td></tr>
<tr><td>3</td><td>Screenshots of LTspice simulations and oscilloscope waveforms.</td></tr>
<tr><td rowspan="4">4</td><td colspan="2">Answer the questions</td></tr>
<tr><th>Questions</th><th>Questions</th></tr>
<tr><td>1</td><td>What are the major differences between BJTs and MOSFETs?</td></tr>
<tr><td>2</td><td>Which do you prefer in digital integrated circuit (IC) applications? BJTs or MOSFETs, and why?</td></tr>
<tr><td>5</td><td colspan="2">Conclusions
Write down your conclusions, things learned, problems encountered during the lab and how they were solved, etc.</td></tr>
<tr><td>6</td><td colspan="2">Images
Paste images (e.g. scratches, drafts, screenshots, photos, etc.) in Postlab report document (only .docx, .doc or .pdf format is accepted). If the sizes of images are too large, convert them to jpg/jpeg format first, and then paste them in the document.

Attachments (If needed)
Zip your projects. Send through WebCampus as attachments, or provide link to the zip file on Google Drive / Dropbox, etc.</td></tr>
</table>

5. REFERENCES & ACKNOWLEDGEMENT

1. Adel S. Sedra & Kenneth C. Smith, “Microelectronic Circuit”, 6th Ed.
2. Previous lab instructions.
3. Related circuit component datasheets.

I appreciate the help from faculty members and TAs during the composing of this instruction manual. I would also thank students who provide valuable feedback so that we can offer better higher education to the students.