Lab 4: The MOSFET. Simulation with Pspice.

Objectives:

At the end of this lab, the student should:

Know how the enhancement MOSFET transistor works:

- In order to do that, an electronic simulation with Pspice will be done.
- The study will focus on the N-channel MOSFET transistor, which is the most widely used nowadays.

Contents:

1. **MOSFET characteristic curves**
2. **Bias circuit**

Material:

PC and PSpice simulation program for Windows (from Orcad). There is a student version in PoliformaT.

Development:

Log in with your user of the ALUMNO domain and start PSpice by double clicking on the PSPICE icon on the desktop. Save the created files in the folder W:\TCO\Prac4.

VERY IMPORTANT: when you save the circuit, put a **different** file name to schematic1, schematic2, etc.

If you have problems in the libraries of simulator, check that within the schematics environment, in **Analysis / Library and Include Files** library is at least included **nom.lib ***

1. MOSFET characteristic curves

1.1 First we generate the characteristic curves of drain (output curves) I_{DS} = f (V_{DS}) of an NMOS transistor. To do this, we have to start *Schematics* and edit the circuit shown in Figure 1.

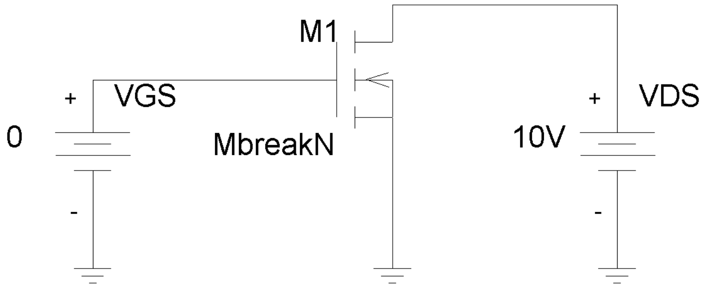

Figure 1. Circuit to draw the characteristic curves of drain.

We will use the command **Draw/Get New Part** to get the various components: **VDC**for power supplies, **EGND** for the ground, and**MbreakN3** for NMOS transistor. Change the names of the power supplies to: $\mathbf{V_{GS}}$ and $\mathbf{V_{DS}}$.
To specify the parameters of the transistor (V_T and K), we first click on the NMOS transistor shape (it becomes red), and then use the command **Edit/Model/Edit Vto=2 Instance Model**. Specify the parameters: **Kp= 2m**, **Vto = 2V** in the editing window (see Figure 2). Then double click on the symbol of the NMOS and write thevalues of the length and width of the channel: **L = 1u, W = 2u**. Remember that *u* indicates *micro* (10^{-6}) and *m* indicates *milli* (10^{-3}) prefix.

Note: The relation between the parameters used in the analytical calculation (V_T and K), and PSpice parameters (Vto, Kp, W, L) is as follows: V_T coincides with Vto, and K is equivalent to: **K = (W/L) Kp/2**.

Question 0: Obtain the value of K for the given parameters.

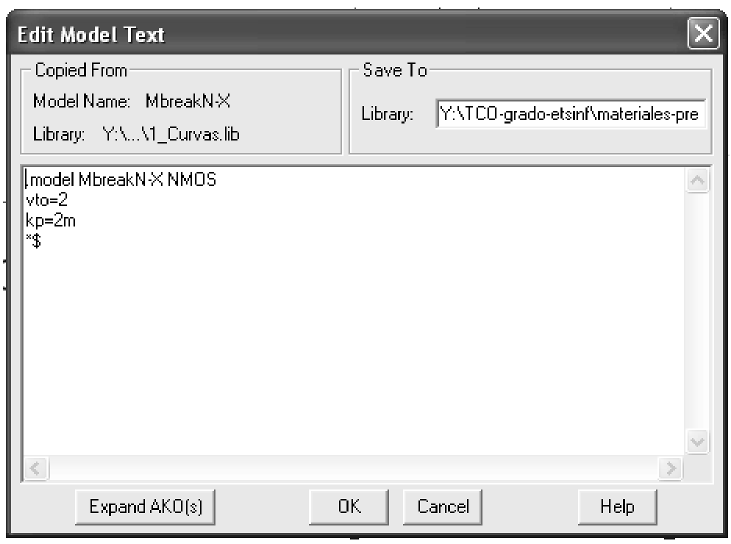

Figure 2. Model edition window.

We want that V_{GS} and V_{DS}voltages vary to analyze the behavior of the MOSFET and to reproduce its characteristic curves.
In order to do this, we use the command **Analysis/setup/dc sweep** window to access the **DC Sweep** (DC voltage variation). We chose V_{DS} as the main variable sweep (this name must match the name we have given to the source **VDC**), and establish a sweep from **0** to **10V,** with increments of **0.1V**.
Then we click the **Nested Sweep** button to define a second nesting, where we choose variable V_{GS} (secondary variable) and set an initial value of **0**, a final value of **5V**, and an increment of **1V**. IMPORTANT: make sure you click the **Enable Nested Sweep**,**Voltage Source**and **Linear**.

Finally, we will use the command **analysis/simulate (F11)** to start the simulation, after which it automatically starts PROBE. Use the command **Trace/Add Trace** (or **Insert** key) to select the drain current I_D. After that, the MOSFET characteristic curves should appear on the screen.

Question 1: How many curves are there? What V_{GS} value correspond each one? Justify your answer. Why the curves are spaced unequally?

Question 2: Write down the values (V_{DS} and I_{DS}) of the points that mark the transition from Ohmic to saturation regions in each curve (just between the horizontal line and the elbow of the curve).

Check in all points the condition $V_{DS} = V_{GS} – V_T$

Question 3: Calculate the value of K from one curve at random, using the expression of saturation: $I_{DS} = K\,(V_{GS} - V_T)^2$. Check that roughly coincides with the value of K calculated above in Question 0.

In the ohmic region, we can observe first an "almost" straight line from the coordinate origin, with a certain slope (as a resistor). After this zone, we observe an elbow that connects with the horizontal zone (zone of saturation, in which the MOSFET behaves as current source: $I_D = f\,(V_{GS})$). In thestraight line region, the simplified expression of I_{DS} (from which we can infer R_{ON}) can be used, while in the elbow region; the full ohmic region expression must be used:

$$I_{DS} = K\,[2(V_{GS} – V_T)V_{DS} – V_{DS}^2]$$

Question 4: Obtain the slope of the area "almost" straight of the curve that corresponds to V_{GS} = 5V. Measure the slope very near the origin of coordinates. R_{ON} is the inverse of this slope. For measuring the slope, use Zoom and cursors in the Probe environment:

$$R_{ON} \approx \frac{1}{slope} = \frac{1}{\Delta y / \Delta x} = \frac{x}{y}$$

Where X corresponds to V_{DS} measured in volts, and Y to I_{DS} measured in milliamps. The cursor window indicates us these values A1 (x value) (y value).

Check that the value is roughly the same as we obtained analytically from the expression of R_{ON}:

$$R_{ON} \approx \frac{1}{2 \cdot K(V_{GS} - V_T)}$$

Question 5: Is it the same R_{ON} for other curves? Justify your answer**.**

1.2 Load line. Visualizing the curves, we can plot the load line of any hypothetical circuit. We will plot the load line of circuit we will see in paragraph 2 (see Figure 3). To do this, select **Add Trace** (**Insert**) and then enter the expression of the load line:

(10 - V (VDS:+)) / 1k

Corresponding to:

$$I_{DS} = \frac{V_{DD} - V_{DS}}{R_D}$$

Being V_{DD} = 10V and R_D = 1k.

Question 6: Write down the data for the points we will have for V_{GS} = 3, V_{GS} = 4V and V_{GS}= 5V. What region is the transistor operating in each case?

Question 7: Are the points of intersection with the X and Y axis as expected?

1.3 Modify the value of W/L to: W=1u and L= 1u. Simulate and visualize the curves again.

Question 8: How does the value of W/L modifies the value of the drain current I_{DS}?

Put again the parameters to W=2u and L=1u.

1.4 Now we want to generate the parable of saturation I_D = f (V_{GS}). In this case only the variable V_{GS} is changing. To do this, go first to the *Schematics* and fix V_{DS} = 10V (this situates the transistor in the saturation region). Then sweep V_{GS} from 0 to 5V with 0.1V increments. Remember to turn off before **Nested DC Sweep**, and to change the name of the variable of the first **Sweep** loop.

Question 9: The expression of saturation must be met for any point of the curve with V_{GS}> V_T. Check it for at least two values of V_{GS}.

Note that this curve indicates the saturation current (maximum) that can flow through the transistor for a given V_{GS}. The actual current in a circuit might be less than this value if the transistor is in ohmic region. We can not see this case in this parable, but in the output (drain) characteristic curves.

2. Bias circuit

We want to analyze the operation of bias circuit of Figure 3. Transistor parameters are the same than those of paragraph 1: **Kp = 2m, Vto = 2V, L = 1u W = 2u**.

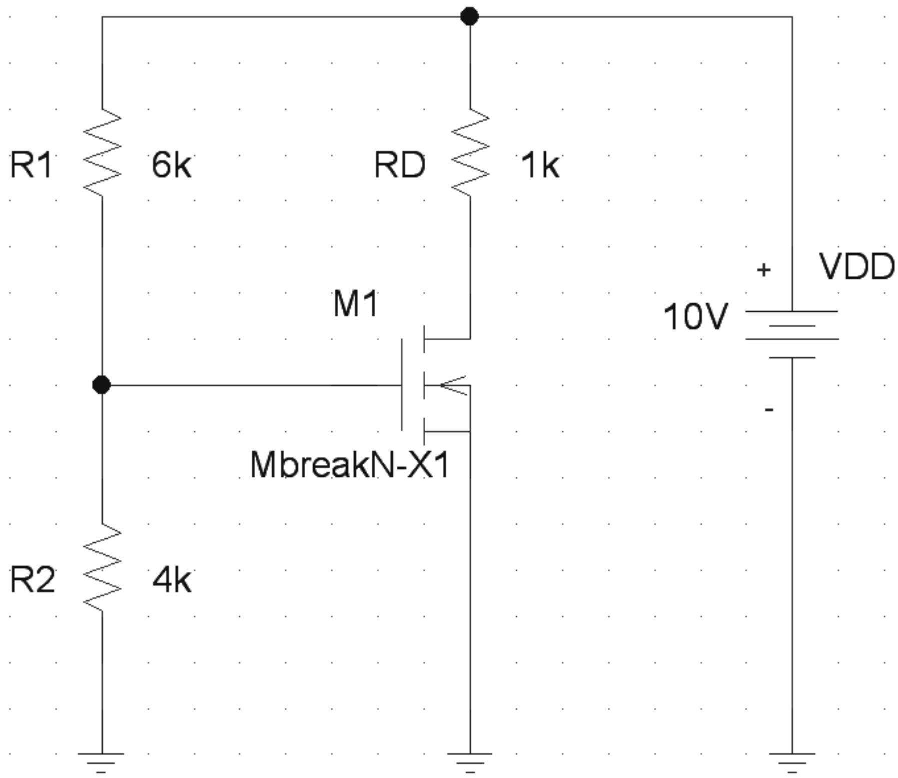

Figure 3. Bias circuit with voltage divider biasing the transistor.

2.1 Edit the circuit using *Schematics*. Specify the parameters following the steps indicated in paragraph 1.1. The symbol of the resistor is **R**(symbols can be rotated with **CTRL-R** or **Edit/rotate)**. The voltage source is a **VDC** component. The components that have already been used before can be obtained directly from the drop-down buttons on the toolbar.

2.2 As it is a DC circuit, the simulation will involve calculating the voltage and current of operating (bias) point Q. For this simulation specificity **Bias Point Detail** in **Analysis Setup**.

2.3 Activate **Analysis/Simulate**. In the lower left window of PROBE, we should see a message indicating the success (or failure!) of simulation.

2.4 We cannot see the results in this case as a graph, since the variables are not time dependent. Voltages and currents are constant. The way to visualize them is to return to *Schematics* and activate the **V and I buttons**. In this case, the values for the various circuit nodes will appear. An alternative way to do this is to use the **Analysis/Examine Output** command. On this way, we just visualize the output file with the result of the simulation. Scrolling down, we can find the parameters of the transistor and the various voltages and currents we need to know.

Question 10: Indicate the operating point of transistor Q: V_{GS}, V_{DS} and I_{DS}.
Check that matches with the point Q we determined graphically for V_{GS} = 4V. What region is the transistor working on?

Question 11: What is the $I_G$current value? What is the current through the resistive divider at the input of the gate?

Question 12: Compare the simulation with the manual calculation. To make the calculations, use the same parameters of NMOS: V_T = 2V, K = 2 mA/V^2.

Note that doing the same, we could simulate any bias circuit seen in the lectures,to better understand its operation.