Electronics Laboratory

Winter semester 2025

Lab 3 – MOSFETs

Name	______________________________	Reg. Nr.	______________
Name	______________________________	Reg. Nr.	______________

Score and comments (only for tutors, please leave blank)

Please fill out this cover sheet and submit it with your lab report.

3 Lab 3 – MOSFETs

Introduction

We will simulate and measure three MOSFET circuits: a common-source circuit, with which we can generate the typical input and output characteristics of a MOSFET; a simple inverter, for which the rise and fall times are of interest; and finally a dc step-up converter in which the MOSFET acts as a switch to allow generation of high dc voltage levels from low-voltage inputs.

Notes

- Download the SPICE models `FDD86102.txt` and `10BQ040.txt` for the power MOSFET and the Schottky diode, respectively, used in the simulation of Section 3.4.1.
- The IRFML8244 used in Experiments 3.2 and 3.3 is a power MOSFET usually employed to switch over 5 A of current. We are using it here at atypically low current levels. To see its typical characteristics, model it in SPICE using the `.dc VDD 0 5 50m Vi 1 4 0.5` directive and $R_D = 0$.
- As you should recall from *Einführung in die Elektrotechnik*, rise and fall times are calculated using the 10% – 90% values. Thus the rise time for a signal which increases from 0 to 5 V is determined from the time required for it to rise from 0.5 to 4.5 V (10% of 5.0 V to 90% of 5.0 V); conversely, the fall time is then the time required for the signal to fall from 4.5 to 0.5 V.
- The propagation delay between two signals t_{pd}, for example between an input voltage $v_i(t)$ and an output voltage $v_o(t)$, is defined using the 50% – 50% values: the time between when the input signal reaches 50% of its final value and when the output signal reaches 50% of its final value.

3.1 Preparation

Please answer the following questions *before* beginning with the simulations or experiments. Then complete the preparation quiz "Preparation 3.1 – MOSFETs" on Ilias using your results. The points for this preparation section will only be awarded via Ilias.

3.1.1 MOSFET parameters

You are given a n-channel FET with $V_t = 1\,\text{V}$, $\mu_n = 600\,\text{cm}^2/\text{Vs}$, $t_{ox} = 30\,\text{nm}$ and $\epsilon_{ox} = 35\,\text{pF/m}$. You know that the channel width is $15\,\mu\text{m}$ and its length $1\,\mu\text{m}$. Determine K_n. Pay attention to units.

3.1.2 Input characteristic

For the transistor given in Question 3.1.1, plot the I_D vs. V_{GS} characteristic for values for which the MOSFET is in saturation. Determine I_D for $V_{GS} = 3\,\text{V}$.

3.1.3 Output current

Consider the circuit of Figure 3.2.1. Assuming that $V_{DD} = 5\,\text{V}$ and $R_D = 200\,\Omega$, what is the maximum value for I_D (assuming that $R_{DS(on)} \to 0$)? Write down an equation for the output load line of this circuit. Using this equation, determine the values for V_{DS} when $I_D = 3.3\,\text{mA}$, $6.6\,\text{mA}$ and $10.0\,\text{mA}$.

3.1.4 Transfer characteristic

Think about which operating regions a MOSFET inverter with a resistive load (such as that shown in Figure 3.3.1) goes through when then output switches from high to low ($1 \to 0$).

3.1.5 Amplifying inverter

Consider using the circuit of Figure 3.3.1 as an amplifier; take $R_D = 1\,\text{k}\Omega$. Given that the FET is made using the same technology as that of Question 3.1.1 except now $W/L = 10/1$, think about what conditions are necessary so that the small-signal model would apply and calculate the small-signal amplification around an operating point of $I_{DS} = 2\,\text{mA}$.

3.2 MOSFET characteristics

In this first experiment, we will simulate and experimentally characterize the I_D/V_{GS} and I_D/V_{DS} characteristics of a MOSFET.

3.2.1 Simulation

Consider the common source MOSFET circuit shown in Figure 3.2.1 below.

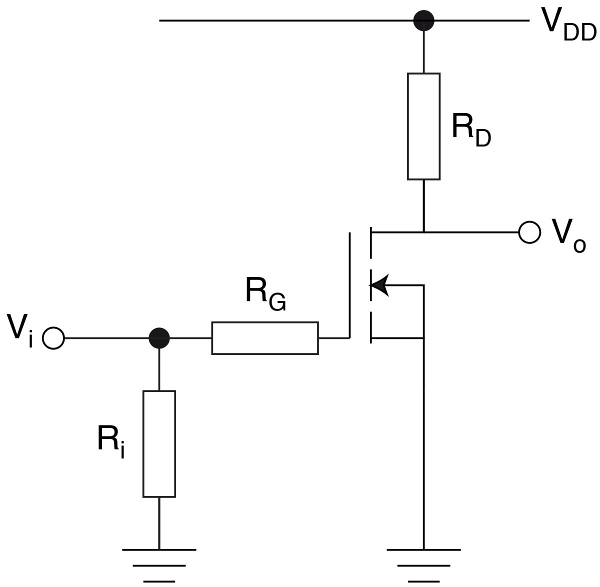

Figure 3.2.1

I_D/V_{GS} characteristic

(a) Set up a SPICE simulation of this circuit with an IRFML8244 MOSFET and using the values $R_i = 100\,\text{k}\Omega$, $R_G = 1\,\text{k}\Omega$, $R_D = 200\,\Omega$ and $V_{DD} = 5\,\text{V}$.

(b) Apply a voltage ramp at V_{in} of 1.8 V to 2.4 V in 1 mV steps.

(c) Plot I_D and V_o as a function of V_{GS}. Explain the form of the characteristic.

(d) Determine $I_{D(max)}$ and compare this value with the calculated value from the preparation (Question 3.1.3).

(e) Determine the V_o values for which $I_D = 3.3\,\text{mA}$, $6.6\,\text{mA}$ and $10.0\,\text{mA}$. Compare with the calculated values from the preparation (Question 3.1.3).

(f) Determine and note down the V_i values for which $I_D = 3.3\,\text{mA}$, $6.6\,\text{mA}$ and $10.0\,\text{mA}$.

I_D/V_{DS} characteristic

(g) Using the same circuit, remove the voltage ramp at V_{in} and replace it with a parameter sweep for V_i (using the `.step param` directive) with a `list` consisting of the three V_i values you determined in Step (f) above.

(h) Define a voltage sweep $0 \leq V_{DD} \leq 5\,\mathrm{V}$ in $50\,\mathrm{mV}$ steps, using the `.dc` directive.

(i) Plot I_D as a function of V_{DS} in the range $0 \leq V_{DS} \leq 5\,\mathrm{V}$.

(j) In this plot, fit a linear characteristic to the three points where $V_{DD} = 5\,\mathrm{V}$ (i.e., the ends of the I_{DS}/V_{DS} curves). Extend to the V_{DS} and I_D axes. From the slope, determine R_D (assuming that $R_{DS(on)} \to 0$).

(k) Explain how and possibly why the simulated I_D/V_{DS} characteristic looks a bit different to the ones we considered in the lecture (Hint: look at the "Notes" at the beginning of this lab again).

3.2.2 Measurement

In the "MOSFET" section of the electronics board, we will use the *MOSFET characteristics* circuit, shown in Figure 3.2.2.

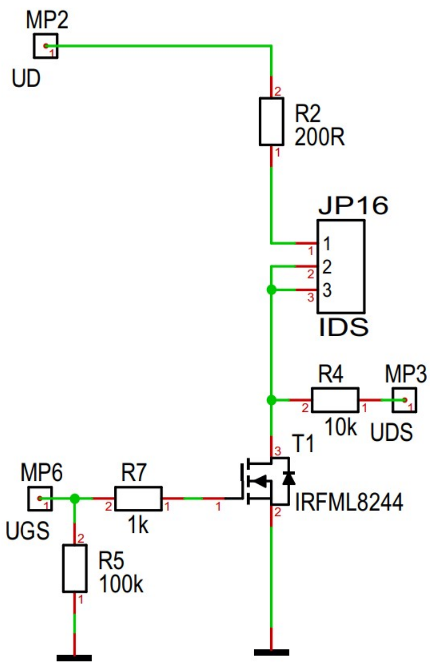

Figure 3.2.2: Schematic of the *MOSFET characteristics* circuit

I_D/V_{GS} characteristic

(a) Determine the actual value of R_2 using the handheld multimeter.

(b) Short-circuit *pin 1* and *pin 2* of jumper JP16.

(c) Connect measurement points *MP2* and *MP3* to the differential amplifier[1] and connect it to *Channel 2* (*MP75*) to measure the voltage across R_2.

(d) Connect the measurement point *MP6* to *Channel 1* (*MP72*) to measure V_{GS}.

(e) Connect the DC input (*MP52* or *MP32*) with $V_{cc} = 5\,\mathrm{V}$ to the measurement point *MP2*.

(f) Connect the first wave generator *W1* (*MP69*) to the measurement point *MP6* (V_{GS}).

(g) Create new channels (using *Add Channel* in *WaveForms - Scope*): 1) with the name *R*, the unit Ω and the value you measured for R2 (decimal point); 2) with the name *Vg*, the unit V and the command *C1*; 3) with the name *Vr*, the unit V and the command *C2*; and 4) with the name *Id*, the unit A and the command *Vr/R*.

(h) Set a triangular voltage (Triangle, $1\,\mathrm{V} - 4\,\mathrm{V}$, 50 %, 50 Hz) as V_{GS}.

(i) Measure and plot I_D as a function of V_{GS}.

(j) Determine the voltages $V_{GS,3.3\,\mathrm{mA}}$, $V_{GS,6.6\,\mathrm{mA}}$ and $V_{GS,10\,\mathrm{mA}}$ where the current reaches $I_{D,1} = 3.3\,\mathrm{mA}$, $I_{D,2} = 6.6\,\mathrm{mA}$ and $I_{D,3} = 10.0\,\mathrm{mA}$ respectively.

I_D/V_{DS} characteristic

(k) Measure and note the values to be set for the voltage generator *W1* (*WaveGen - DC, MP69*) so that the voltages $V_{GS,3.3\,\mathrm{mA}}$, $V_{GS,6.6\,\mathrm{mA}}$ and $V_{GS,10.0\,\mathrm{mA}}$ (from the previous task) are actually applied to the board.

(l) Short-circuit *pin 1* and *pin 2* of jumper JP16.

(m) Connect measurement points *MP2* and *MP3* to the differential amplifier and connect it to *Channel 2* (*MP75*) to measure the voltage across R_2.

(n) Connect the measurement point *MP3* to *Channel 1* (*MP72*) to measure V_{DS}.

(o) Connect the DC input (*MP52* or *MP32*) with $V_{cc} = 5\,\mathrm{V}$ to the measurement point *MP2* and connect the first wave generator *W1* (*MP69*) to the measurement point *MP6* (V_{GS}).

(p) Create new channels (using *Add Channel* in *WaveForms - Scope*): 1) with the name *R*, the unit Ω and the value you measured for R2 (decimal point); 2) with the name *Vds*, the unit V and the command *C1*; 3) with the name *Vr*, the unit V and the command *C2*; and 4) with the name *Id*, the unit A and the command *Vr/R*.

(q) Set $V_{cc} = 5\,\mathrm{V}$ and set $V_{GS,10.0\,\mathrm{mA}}$ such that $I_D \cong 10.0\,\mathrm{mA}$. Start your search at the

[1] Recall that the input of the differential amp is between *MP55* & *MP65* and its output is between *MP66* and ground.

value you measured earlier for *W1*. Note the final value and justify any deviations.

(r) Keeping V_{GS} constant, vary V_{cc} (0, 5, 0.5) Record V_{DS} and I_D for each measurement point.

(s) Repeat the last three steps for $V_{GS,3.3\,\mathrm{mA}}(I_D = 3.3\,\mathrm{mA})$ and $V_{GS,6.6\,\mathrm{mA}}(I_D = 6.6\,\mathrm{mA})$.

(t) Plot the characteristic curves (for each V_{GS}) in a single diagram.

(u) Determine the Early voltage of each trace.

(v) Fit a linear trend line to the three measurement points for $V_{cc} = 5\,\mathrm{V}$. This fit represents the load line for R_2 and the voltage V_{cc}. Calculate from this data the value for R_2 assuming that $R_{DS(on)} \rightarrow 0$.

3.3 MOS logic gates

We will analyze the time-dependent turn-on and turn-off behavior of a simple MOSFET inverter by simulation and measurement.

3.3.1 Simulation

Consider the simple MOSFET inverter shown in Figure 3.3.1 below.

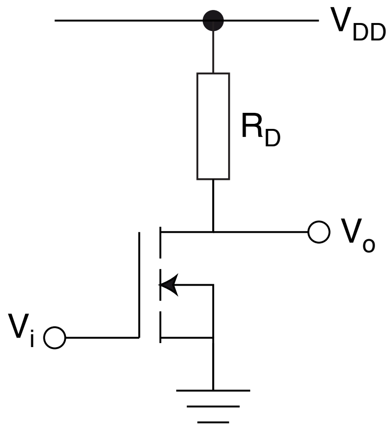

Figure 3.3.1

(a) Set up a SPICE simulation of this circuit using $R_D = 200\,\Omega$ and an IRFML8244 MOSFET.

(b) Define the voltage source V_i as an asymmetric square wave signal from 0 V to 5 V at 50 Hz using the source definition `pulse (0 5 10m 1u 1m 10m 20m)`.

(c) Set the voltage source $V_{DD} = 5\,\text{V}$.

(d) Place a terminal for the output V_o at the drain.

(e) Perform a time-based simulation (using the `.tran` directive), and analyze V_i and V_o. Fiddle with the parameters of the `.tran` directive and perform the simulation numerous times to look at the low → high and high → low transitions in detail.

(f) Determine the maximum and minimum voltages of both signals at their steady state.

(g) Using an appropriate magnification on the time axis, make a plot of the rising edge transitions of the input and output voltages, together in a single plot. Make a second plot of the falling edge transitions. Explain what you see.

(h) Determine the rise and fall times for V_i and V_o. Use the appropriate definitions.

(i) Try to determine the propagation delays ($t_{pd(rise)}$ and $t_{pd(fall)}$) for the rising and falling edges of V_i. Why might this determination be difficult?

3.3.2 Measurement

In the “MOSFET” section of the electronics board, we will continue to use the *MOSFET characteristics* circuit, shown in Figure 3.2.2 above.

(a) Short-circuit *pin 1* and *pin 2* of jumper JP16.

(b) Transfer the potential from the measurement point *MP32* to the measurement point *MP2* and measure the resulting supply voltage there.

(c) Connect a square wave signal (0 V - 5 V, 50 Hz, 50 %) to *MP6*.

(d) Measure the voltage V_i at *MP6* and V_o at *MP3*.

(e) Determine the maximum and minimum voltages of both signals in their steady states.[2]

(f) Using an appropriate magnification on the time axis, make a plot of the rising edge transitions of V_i and V_o, together in a single plot. Make a second plot of the falling edge transitions. Explain what you see.

(g) Determine the propagation delays ($t_{pd(rise)}$ and $t_{pd(fall)}$) for the rising and falling edges of V_i.

(h) Compare your measurement results with the simulation results of Section 3.3.1. Discuss and explain any significant differences.

[2] To obtain enough samples, you can use the following measurement settings: 1) use repeated measurement mode; 2) for the rising edge transition use Source: Channel 1; Condition: Rising; Time axis scale: 1 µs/div, position: 4 µs; Level: 0.5 V (adjust if necessary); 3) for the falling edge transition use Source: Channel 1; Condition: Falling; Time axis scale: 1 µs/div, position: 4 µs; Level: 4 V (adjust if necessary).

3.4 Step-up converter

A step-up (or boost) converter is a circuit which generates a higher dc voltage from a lower dc voltage. The transformer, which you know from *Einführung in die Elektrotechnik*, only increases the voltage of **ac** signals, but the step-up converter is a useful circuit for generating higher **dc** voltages.

A simple step-up voltage converter circuit is shown in Figure 3.4.1. The MOSFET is essentially a switch which alternately generates a short or an open circuit. When the MOSFET is on (a short circuit), a current from the input flows through the coil and the transistor, establishing a magnetic field in the inductor. When the MOSFET switches off (an open circuit), the stored energy in the coil results in a current flow through the Schottky diode, charging the capacitor. When the MOSFET switches on again, the cycle repeats. The increasing charge on the capacitor generates the higher output voltage V_{dc-out}.

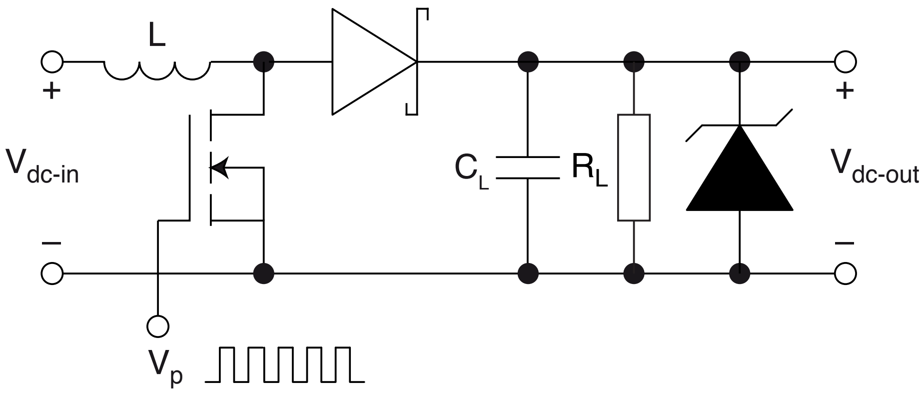

Figure 3.4.1: A basic step-up converter

As an interesting application for MOSFETs, we will now simulate and measure an MOS-based step-up converter.

3.4.1 Simulation

(a) Set up a SPICE simulation of the step-up converter circuit of Figure 3.4.1, which corresponds to the circuit on the electronics board (Figure 3.4.2). Use the FDD86102 power MOSFET (download the model file). Take $L = 100\,\mu H$, $C_L = 22\,\mu F$ and $R_L = 5\,k\Omega$ in parallel to the capacitor. For the Schottky diode, use a 10BQ040 (download the model file) and for the Zener diode a BZX84B36VL (with $V_Z = 36\,V$, in the LTSpice library).

(b) Apply a dc bias of 5 V to V_{dc-in}.

(c) For V_p, use the `pulse` source definition and apply a 10 kHz square wave with a duty cycle of 50% and a magnitude of 5 V.

(d) Using the `.tran` directive, plot V_{dc-out} for 320 ms (the simulation may take a few minutes). Describe the form of the characteristic and determine the output voltage increase factor.

(e) Vary the duty cycle (5%, 20%, 60%) and note what changes for V_{dc-out}. Adapt the simulation conditions if necessary.

(f) Finally, apply square waves for the frequencies 2 kHz, 5 kHz, 20 kHz, 50 kHz, and 100 kHz, all with a duty cycle of 50 %, and describe the resulting output.

3.4.2 Measurement

In the "MOSFET" section of the electronics board, we will use the *step-up converter* circuit, shown in Figure 3.4.2.

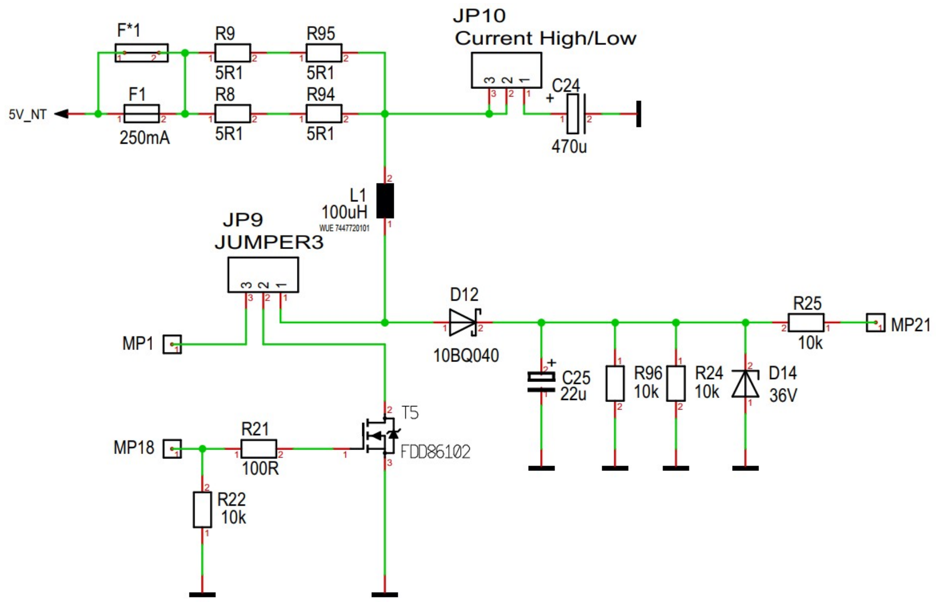

Figure 3.4.2: Schematic of the *step-up converter* circuit

(a) Short-circuit *pin 1* and *pin 2* of the jumper *JP9/JUMPER3* (this is labeled *BOOST* on the board). Open *pin 1* and *pin 2* of the jumper *JP10*.

(b) Connect the input signal of the signal generator to the measurement point *MP18*.

(c) Attach a PWM signal (Square, 0 – 5 V, 50 %) to *MP18* and describe the output signal you measure at *MP21*.

(d) Measure the average value for V_{out} for each of the frequencies: 100 Hz, 200 Hz, 500 Hz, 1 kHz, 2 kHz, 5 kHz, 10 kHz, 20 kHz, 50 kHz.

(e) Plot V_{out} as a function of f in a graph with a logarithmic abscissa.

(f) Set up another PWM signal (Square, 0 – 5 V, 10 kHz).

(g) Measure the average value for V_{out} in each case for the duty cycle D (0, 100, 10%)[3].

(h) Plot V_{out} as a function of D.

[3] You can set the duty cycle by changing the symmetry of a square wave in the *WaveForms* software.