Laboratory Electronics 2

Task 2: Field Effect Transistor Applications

Manuel Campana

October 31, 2024

Learning Objectives

The objectives of the second lab task are the understanding of three different applications for a MOSFET and the training of measuring those kind of circuits. In the first task, we use the MOSFET as a simple switch, in the second task to realize a current source and in the third task as a single-ended amplifier. All applications are pretty common when it comes to a circuit design.

This time is it not required for you to work on all of the tasks. You can decide whether you would like to work on task 1 or task 2. Task 3 is mandatory for all teams. During the peer review in the end of the lab you have time to discuss about all tasks. The intention is that you explain your application to a group with the other one and vice versa.

- Understanding the MOSFET applications as switch, current source and single-ended amplifier
- Learning the crucial aspects of the design for these applications
- Practice the handling of the oscilloscope (Math Channel and Reference Channel)
- Finding ways to measure designated values

Lab Devices

You will find the following devices in the lab. Please make sure that you have the manuals ready. We also provide a breadboard with all required components and wires to setup your electrical circuit.

- **Digital Multimeter** GOSSEN METRAWATT METRA HIT 26S
- **Oscilloscope** Tektronix MDO3012
- **Differential Probe** EDITEST ELECTRONIC GE 8109
- **Frequency Generator** Agilent 33210A
- **DC Power Supply** Rohde & Schwarz NGT 20

1 MOSFET as Switch

Usually, a MOSFET gets used as switch when a high current needs to be switched. We use the n-channel MOSFET BS108 for that. In this lab task we have a microcontroller as a signal source, which gets emulated with a frequency generator. The output signal V_m of the microcontroller is a 3.3 V pulse width modulated (PWM) signal with a duty cycle of 50 % and a variable frequency f_{PWM}. The rise/fall time of the PWM-signal is 20 ns. We want to control a load R_L with our MOSFET which needs 50 mA. The supply voltage is given with $V_0 = 5\,\mathrm{V}$. The circuit diagram for this application is shown in figure 1. During both switch status of the MOSFET, the power loss of it is appromimately

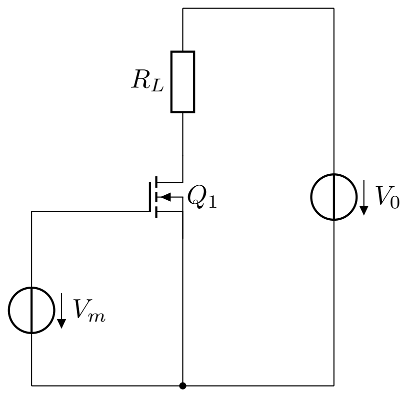

Figure 1: MOSFET as a switch

zero. The reason for that is either the voltage V_{DS} or the current I_D is zero. The electric power is defined as in equation 1 [1].

$$P = \frac{1}{T}\int_0^T v(t)i(t)\,\mathrm{d}t \tag{1}$$

- Calculate the equivalent load resistance for the given application.
- Simulate the circuit. Analyze the circuit to find the time where the highest power loss at the MOSFET occurs.

  Hint: To find this time you should plot the function $p(t) = v(t)i(t)$ over at least one period.
- Setup your circuit and find a way to measure the function $p(t) = v(t)i(t)$.
- Calculate the power per period with the help of equation 1. What does the result mean considering PWM frequency especially in battery powered systems?

2 MOSFET as Current Mirror

There is an easy way to use two MOSFETs BS108 as a constant current source. The name of this circuit is *n-channel current mirror* and it is shown in figure 2. The idea behind this circuit is to transfer a current value from the input I_{in} to the output I_{out} [2]. The combination between the source

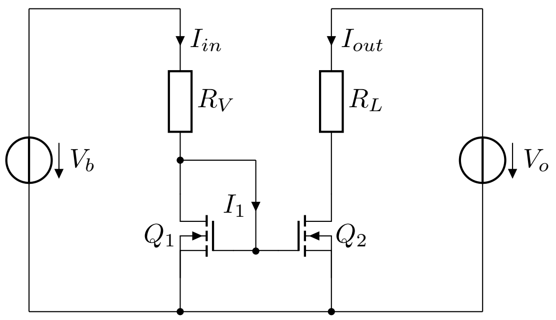

Figure 2: MOSFET as a current mirror

V_b, the voltage V_{DS} of Q_1 and R_V defines the current I_{in} which gets transferred to the load path. The load for the current mirror is emulated as resistor R_L. The ratio k_I between the two output I_{out} and input current I_{in} is defined in equation 2 [2].

$$k_I = \frac{I_o}{I_i} = \frac{K_2}{K_1}\left(1 + \frac{V_o}{V_A}\right) \approx \frac{K_2}{K_1} \qquad \text{for } V_o \ll V_A \tag{2}$$

There, V_A is the early voltage of the MOSFET and K_n the *transconductance coefficient* of MOSFET Q_n. Like any other power source, the circuit can be modeled as a real power source as shown in figure 3. In this circuit, I_0 represents the short circuit current of the source and R_i the inner resistance.

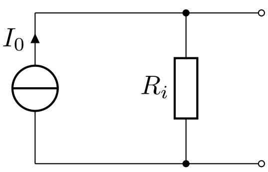

Figure 3: Real Current Source

- Find a dimensioning for the components to realize a load current $I_{out} = 10\,\text{mA}$.
- Simulate the circuit and vary the load resistance R_L. Find a way to determine the values I_0 and R_i of the corresponding real current source as shown in figure 3.
- Setup the circuit and verify your values for I_0 and R_i. Evaluate these values against the values of an ideal power source.

3 MOSFET as Single-Ended Amplifier

One of the three basic amplifier circuits is the common-source circuit. It is shown in figure 4. The objective of this task is the analysis of this circuit. The first step to use this circuit is the setting of

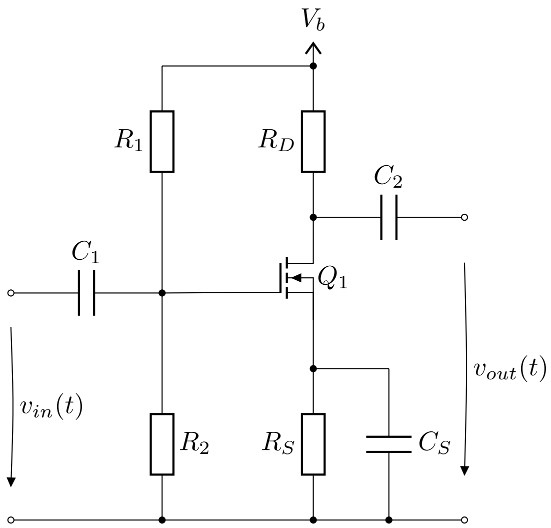

Figure 4: Common-Source Circuit

the operating point. For this step is it necessary to define the desired characteristics of the amplifier. These parameters are:

$$v_{in}(t) = 0.1\,\mathrm{V} \cdot \sin(2\pi t \cdot 1\,\mathrm{kHz}) + 0.1\,\mathrm{V} \tag{3}$$

$$v_u = \frac{v_{out,pp}(t)}{v_{in,pp}(t)} = -20 \tag{4}$$

The amplification v_u only describes the amplification of the peak-to-peak voltage of the respective voltages. One way to find the correct dimensions for the components is a graphical approach. For this way is it necessary to have the transfer function and the output characteristic field ready. You have measured the transfer function in the first lab task. The output characteristic field can be exported from a simulation. The way to define the operation point and the values of the resistors for the desired amplification can be seen in figure 5 [1]. Additionally, the following equations need to be used:

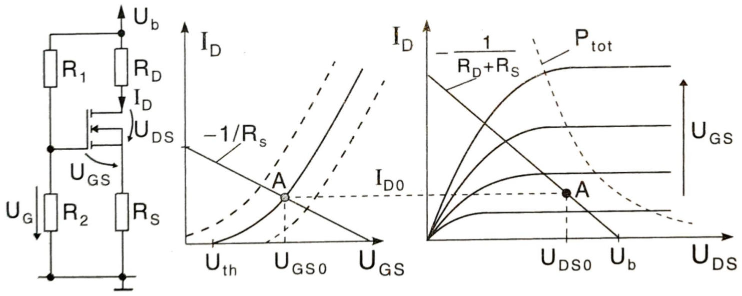

Figure 5: Operation Point of the Common-Source Circuit [1]

$$R_D = \frac{V_b - V_{DS0}}{I_{D0}} - R_S \tag{5}$$

$$R_S = \frac{V_G - V_{GS0}}{I_{D0}} \tag{6}$$

$$\frac{R_1}{R_2} = \frac{V_b - V_G}{V_G} \tag{7}$$

$$v_u = -S \cdot R_D \tag{8}$$

There, S is the *forward transconductance* of the MOSFET.

$$S = \frac{\mathrm{d}I_D}{\mathrm{d}V_{GS}} \tag{9}$$

The values with the indices 0 describe the parameters of the operation point. Usually V_{DS0} is set between $0.3...0.5V_b$.

- Find the dimensioning of all components for the given requirements. Explain the purpose of each component.
- Simulate the circuit to verify it and to optimize your value selection.
- Setup your circuit. Measure the DC voltages and currents. Verify your operation point values.
- Measure the input signal $v_{in}(t)$, the output signal $v_{out}(t)$ and the amplification v_u at the same time. Analyze the amplification over time and explain the shape.
- Try to overlap the input signal $v_{in}(t)$ and output signal $v_{out}(t)$ as good as possible. Analyze differences (distortions) between the two signal.
- **Optional** What is the input and output resistance of the circuit? How can you measure the output resistance?

References

[1] R. Kories and H. Schmidt-Walter, *Taschenbuch der Elektrotechnik : Grundlagen und Elektronik* (Edition Harri Deutsch). Europa Lehrmittel : H. Deutsch, 2013, ISBN: 9783808556696.

[2] U. Tietze, C. Schenk, and E. Gamm, *Electronic Circuits: Handbook for Design and Application* (Electronic Circuits). Springer Berlin Heidelberg, 2015, ISBN: 9783540786559.