Analog Integrated Circuit

Lab: I

Task 1: Common Source Amplifier

Determining the operating point of a source circuit with resistive load

a Modify the circuit used in Project Part I as shown in Fig. 1. This is a *common source amplifier* circuit with resistive load. The input of this amplifier (not shown here!) is the small signal voltage at node "in" and the output is the voltage at node "out".

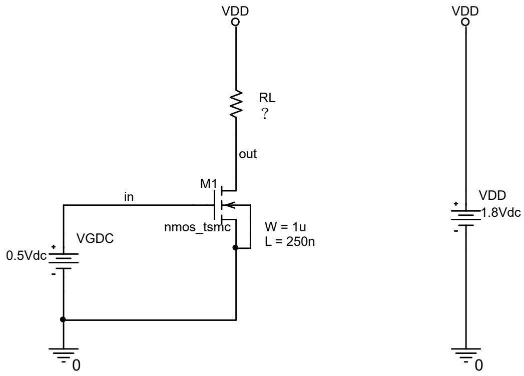

Figure 1: *Common source amplifier* circuit with resistive load RL.

b Based on the DC-Characteristic of the transistor, obtained in Task 1, determine the value of resistor *analytically* (without using the simulator!) such that the DC voltage at node "out" V_{out} is approximately equal to half of the power supply voltage: $V_{out} \approx \frac{V_{DD}}{2} = 0.9V$.
Hint: Use Kirchhoff's Voltage Law (KVL) in the mesh consisting of VDD, RL and Drain-Source voltage of the transistor and neglect for simplicity the channel length modulation!

c Perform a *Bias Point* simulation to determine the output voltage and the transistor current and compare your results with the expected values from part a.

d Based on the simulation result determine in which operation region (*saturation*, *triode*, or *cut-off*) the transistor is working. Justify your answer.

e Open the *Bias Point* simulation settings window (PSpice →Edit Simulation Profile) and modify the setting as shown below (see also Fig. 2): Check the "Include detailed bias point information ..." and "Calculate small-signal DC-Gain(.TF)" box and enter the following in the From input source name: "VGDC" and in the To output variable: "V(out)". Using this setup PSpice calculates the small signal parameters of all non-linear devices in the circuit (transistors, diodes, etc.) and will calculate the small-signal DC gain and write it to the "simulation output file". PSpice calculates the small signal (the variation) of node "out" for a "small" variation of the defined input source (in this case VGDC):

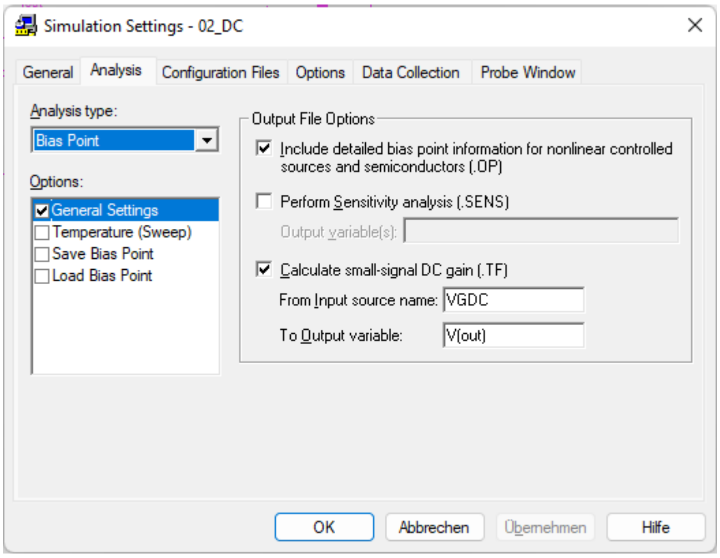

Figure 2: *Bias Point* simulation setup to determine the small signal parameters and calculate the small-signal DC gain (Transfer Function).

$$\text{DC-Gain} = \frac{dV_{out}}{dV_{in}}$$

.

f After running the simulation, the PSpice A/D Demo window opens. In the upper toolbar click in "View" → "Output File" to view output simulation file.

g Scroll to the bottom of the output file to find Determine g_m and r_{ds} and the calculated DC small signal gain.
Hint: The results of the small-signal analysis can be found in the output file below the line `**** SMALL-SIGNAL CHARACTERISTICS`.

In addition to the DC small signal amplification V(OUT)/V-VGDC, the input impedance INPUT RESISTANCE and the output impedance OUTPUT RESISTANCE at frequency 0 are calculated. [1]

h Draw the small signal equivalent circuit of this amplifier and calculate the small signal volatge gain $\frac{v_{out,ss}}{v_{in,ss}}$. Use the numerical values for the small signal parameters (g_m and r_{ds}) obtained by *Bias Point* simulation to calculate the small signal voltage gain and compare your result with the gain calculated by the *Bias Point* simulation simulation.
Hint: As shown in the lecture the small signal gain at very low frequencies is given by $\frac{v_{out,ss}}{v_{in,ss}} = -g_m(r_{ds} \parallel R_L)$.

[1]Note that the output impedance at the node "out" includes the effect of the load resistor RL.

**** MOSFETS

NAME	M M1
MODEL	nmos_tsmc
ID	$I_{D,DC}$
VGS	VGS
VDS	VDS
VBS	VBS
VTH	V_{TH}
VDSAT	VDSAT
Lin0/Sat1	Lin0/Sat1
if	if
ir	ir
TAU	TAU
GM	GM
GDS	GDS
GMB	GMB
CBD	CBD
CBS	CBS
CGSOV	CGSOV
CGDOV	CGDOV
CGBOV	CGBOV
CGS	CGS
CGD	CGD
CGB	CGB

**** SMALL-SIGNAL CHARACTERISTICS

V(OUT)/V_VGDC = Dc_gain

INPUT RESISTANCE AT V_VGDC = Rin

OUTPUT RESISTANCE AT V(OUT) Rout

Figure 3: Simulation output file.

Transient Analysis

a Modify the circuit as shown in Fig. 4. Vsig is a sinusoidal voltage source ("VSIN" in PSpice library "SOURCE") with amplitude 10mV, frequency of 1kHz and a DC-offset of 0V. This voltage is superimposed on the dc voltage VGDC and is modeling a "small-signal" input of the amplifier (for example a signal coming from a microphone).

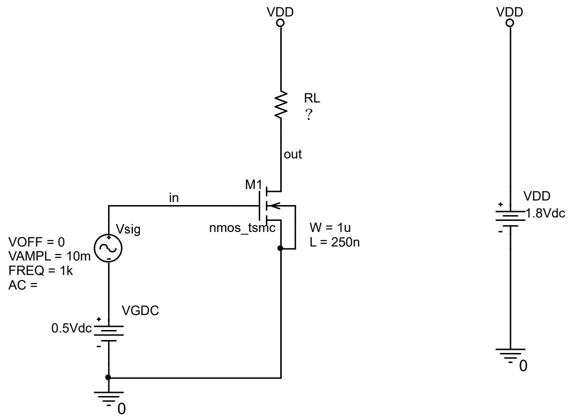

Figure 4: *Common source amplifier* circuit with resistive load RL. The input signal is a sinusoidal voltage source Vsig with Amplitude =10mV and Frequency=1kHz.

b Perform a *Transient* simulation and plot the voltage at the input node "in" v_{in} and the output node "out" v_{out}.

c Using the *Transient* simulation results, determine the amplitude of the small signal (ac) and DC values of the voltages v_{in} and v_{out} and calculate the small signal gain of the amplifier.

d Compare the results with the results of *Bias Point* simulation simulation.

e Calculate the maximum amplitude of the input voltage which results in a "clipping" of the output voltage. Modify the amplitude to this maximum value and repeat the simulation and plot the input and output voltages.
Discuss why the output voltage is "distorted" and why the small signal model cannot be used anymore.

Frequency (AC) Analysis

An AC analysis allows you to find the behavior of a circuit for small signals. The simulation will first determine the operating point, after which small sinusoidal excitations are applied on top of the calculated biasing conditions. Each voltage, current or other simulated quantity will then vary along with the excited signal and will show an amplitude and phase shift that is represented by a single complex number. This is useful when characterizing the frequency response of amplifier, characterizing a filter, or when analyzing stability of systems that have feedback in them.

By assuming the perturbations are small and sinusoidal, the circuit can be linearized which makes this type of simulation relatively fast. The result is a Complex number, which contains the amplitude and phase information.

For each AC analysis, we need (at least) an AC source that will excite the circuit. The amplitude and phase of the excitation can be set by using the acmag and acphase parameters for a VoltageSource object.

Please not that the resulted circuit is linearized. The output signals will always be proportional to the input voltage, regardless of the amplitude of the input signals! Therefore, no non-linear effects (e.g. clipping) can be simulated using an AC analysis.

a Modify the circuit shown in Fig. 4 to define an AC input signal with magnitude 1V (see Fig. 5).

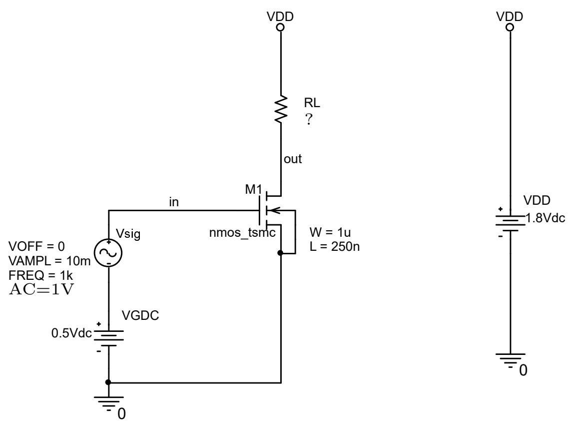

Figure 5: *Common source amplifier* circuit with resistive load RL. The input signal is a sinusoidal voltage source Vsig with Amplitude =10mV and Frequency=1kHz.

b Perform a *AC* simulation in a frequency range from 1Hz to 10MHz and plot the voltage at the input node "in" v_{in} and the output node "out" v_{out}.

c Using the *AC* simulation results, determine the magnitude (amplitude) and phase of the at output v_{out} and calculate the small signal gain of the amplifier.

d Compare the results with the results of *Bias Point* simulation and *Transient* simulation simulation.

e Now increase the amplitude of the input signal to a large value (e.g. 1V) and determine the gain again. Compare this result with the result in part x and the *Transient* simulation simulation result.
Explain the discrepancies (if any) between the *Transient* simulation simulation result and the AC simulation .