Laboratory 5: Common Source Amplifier Design Project

1. Introduction

In this lab you will design, build and demonstrate a common source amplifier using the BS170 NMOS transistor meeting the following set of specifications. This lab has three phases: hand design, SPICE simulation and lab verification. You must write a complete laboratory report describing your efforts.

2. Specifications

The design specifications are shown in Table 1.

Parameter	Specification
Mid-band gain ($|A_{mid}|$)	>50 V/V
High -3dB frequency (f_H)	>20 kHz
Low -3dB frequency (f_L)	<300 Hz
Output swing (undistorted)	>3 V_{ptp}
Supply voltage (V_{dd})	15 V
Output load (C_L)	1 nF

Table 1: Design specifications

The amplifier frequency response has three regions: low-band, mid-band and high-band. The gain is at its maximum value A_{mid} in the mid-band, and drops in the low-band and in the high-band. The f_H and f_L specs are defining the mid-band range, where the gain is 3 dB below A_{mid}. Also note that common source amplifiers are inverting.

The amplifier schematic is shown in Figure 1. It contains an attenuation network, a biasing network, the transistor itself and the output network. The required gain is the output voltage divided by the input voltage after the attenuation network:

$$|A| = \frac{v_{out}}{v_{i,div}}$$

The components with fixed values are shown on the schematic. Other components values are to be chosen by you to meet the specifications. Also, note that there is no output decoupling capacitor.

All resistors and capacitors must be standard 10% values (no parallel/series combination).
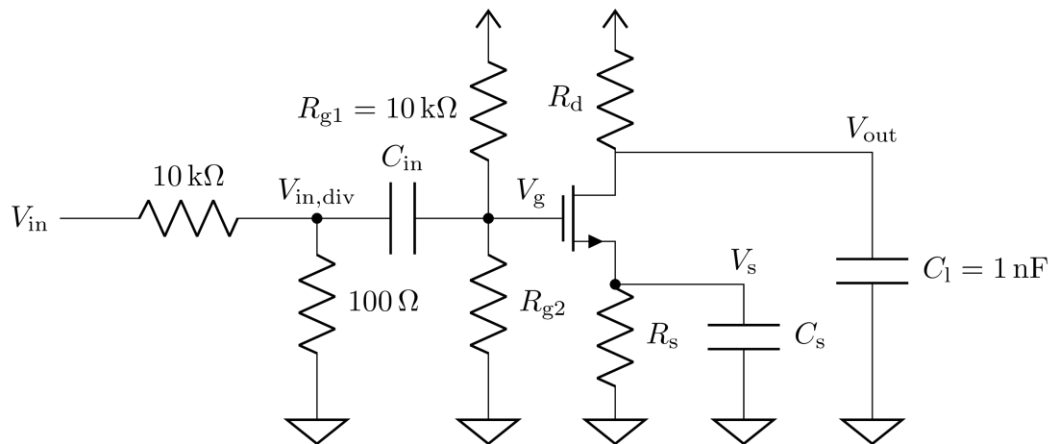
Figure 1: Common source amplifier schematic

3. Hand design

Biasing circuit

The biasing circuit includes the gate resistors, the source resistor and the drain resistor. These should be chosen to provide adequate gain as well as permitting sufficient output swing. A good starting point is to choose R_d and R_s such that each of them, as well as the transistor, drop $V_{dd}/3$ at DC. Note that the bypass capacitor is used to shunt the source resistor so the small signal gain can be expressed as:

$$A_v \approx -g_m R_d$$

You can tweak the gain at the expense of output swing (for a given bias current) by changing R_d.

Transfer function

The bypass and decoupling capacitors (respectively C_s and C_{in}) will determine the low frequency response of the amplifier since they form two high-pass RC filters. Because the attenuator network has low impedance (100Ω in parallel with 10kΩ), the decoupling high-pass filter frequency is mainly determined by C_{in}, R_{g1} and R_{g2}. Similarly, the bypass filter frequency is determined by C_s, R_s and the transistor equivalent source impedance ($1/g_m$).

The high frequency limit is determined by a gain-bandwidth tradeoff related to the output resistance (R_d), and by the output load capacitance C_l.

4. Simulation

For this project, you may use either HSPICE or LTspice (available at linear.com/ltspice). The BS170 trasistor can be described with the following model.

<u>Note</u>: Because the model is defined as a subcircuit, the transistor needs to use the X prefix instead of M. In HSPICE, you can add the model definition in your netlist file and define the transistor as X1. In LTspice, you can copy the model into a SPICE directive, and ctrl-right click on the transistor to change the prefix to X.

```
.SUBCKT BS170 1 2 3
* 1=drain  2=gate  3=source
Cgs  2 3 12.3E-12
Cgd1 2 4 27.4E-12
Cgd2 1 4 6E-12
M1 1 2 3 3 MOST1
M2 4 2 1 3 MOST2
D1 3 1 Dbody
.MODEL MOST1 NMOS(Level=3 Kp=0.100783 Rs=20m Vto=2 Rd=1.186 gamma=0.5276)
.MODEL MOST2 NMOS(VTO=-4.73 Kp=0.100783 Rs=20m)
.MODEL Dbody D(Is=125f N=1.023 Rs=1.281 Ikf=18.01 Cjo=46.3p M=.3423
+            Vj=.4519 Bv=60 Ibv=10u Tt=161.6n)
.ENDS
```

Simulate both the DC operating point, as well as the AC transfer function (both magnitude and phase). A transient analysis can be used to verify the output swing specification.

5. Lab

In lab you will verify your design (both hand calculations and SPICE simulations). After implementing your circuit, you should verify the design parameters and specifications using relevant measurements. In particular, make sure you measure the DC bias operating point (drain current and gate voltage), as well as the AC characteristics such as gain, output swing, low frequency cutoff and high frequency cutoff. Make sure you minimize any parasitic capacitances in your circuit, which will be especially important for the high frequency cutoff. During lab you may find that you need to tweak your design, for example if the gain is not large enough. In this case you should calculate or simulate the changes to describe their effect.

6. Report

Your report should be a concise yet comprehensive description and evaluation of your design, simulation and laboratory measurements. The report should contain no more than 10 pages of text and figures and should include proper caption and formatting.

Your report should contain the sections listed in Table 2.

Section	Weight
Introduction	10%
Hand design	30%
Simulation	20%
Experimental results	30%
Summary	10%

Table 2: Laboratory report sections

The introduction should include a description of how you approached the design problem and what trade-offs in the specifications you first encountered.

In the hand design section you should explain how you solved the design constraints, using equations where appropriate. Clearly explain how you designed both the operating bias point, and well and the frequency response. There is no need to derive the results shown in class, but you should show the equations in variable form before evaluating. Include a summary of the specifications achieved by the hand design.

The SPICE section should contain a description of the simulated circuit (either a netlist or a LTspice screenshot) as well as figures showing the simulation results. Include a summary of the simulated specifications.

In the experimental results, show the relevant measurements (multimeter, oscilloscope plots, etc.) proving that the circuit behaves as intended. If you changed any value from the hand design, you should justify in this section.

In the summary, include a table that compares the specified parameters from the hand design, the SPICE simulation and the lab measurements. Briefly explain any discrepancies.