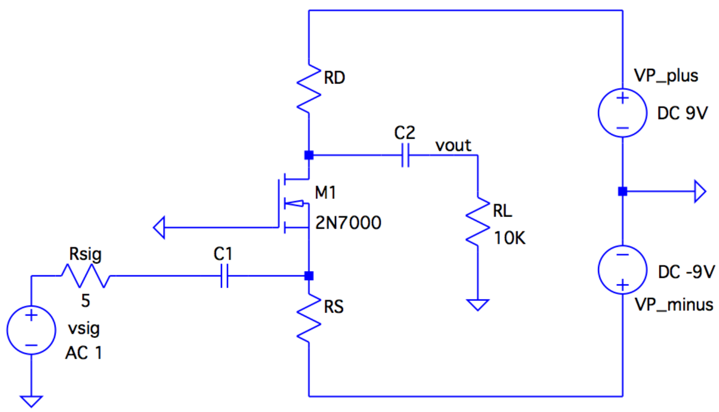
Objectives:

- Performing DC and small signal AC analysis of the circuit
- Simulating the circuit and comparing the simulation results with hand analysis
- Implementing the circuit in an experimental setting, taking measurements and comparing the circuit behavior with theoretical and simulated results

Pre-Lab

Consider the following amplifying circuit using a 2N7000 nMOST. The SPICE model of the 2N7000 is as follows:

```
.SUBCKT 2N7000 3 4 5
* Nodes         D G S
M1 3 2 5 5 MOD1
RG 4 2 343
RL 3 5 6E6
D1 5 3 DIODE1
.MODEL MOD1 NMOS VTO=2.474 RS=1.68 RD=0.0 IS=1E-15 KP=0.296
+CGSO=23.5P CGDO=4.5P CBD=53.5P PB=1 LAMBDA=267E-6
.MODEL DIODE1 D IS=1.254E-13 N=1.0207 RS=0.222
.ENDS 2N7000
```



Design the circuit to achieve a small signal gain of at least Av = vout/vsig = 5 V/V. Assume the following specifications: RL=10KΩ, Rsig=50Ω, VP_plus=9V, VP_minus=–9V. Use a matlab script to carry on all the required computations.

1. DC operating point analysis

    a. Sketch the DC model of the circuit

b. Make an adequate design choice for the value of the DC drain current

ID = ______________________________________

c. Calculate Vov (show the symbolic equation you used)

VOV = ______________________________________

d. Calculate gm (show the symbolic equation you used)

gm = ______________________________________

e. Calculate VGS (show the symbolic equation you used)

VGS = ______________________________________

f. Calculate ro (show the symbolic equations you used)

ro = ______________________________________

g. Calculate RS (show the symbolic equations you used)

RS = ______________________________________

2. AC analysis

a. Sketch the small-signal model of the circuit:

b. Calculate the ratio vs/sig (show the symbolic equation you used)

vs/vsig = ____

c. What is the value of RD that produces vout/vsig = 5 V/V?

RD = ____

d. How much is the DC voltage at the drain?

VD = ____

3. Attach the matlab script you used to carry on all required computations

Lab

1. Simulate the circuit with SPICE:

Parameter	***SPICE value***	***Theoretical value***	***% Error***
VGS =			
VDS =			
ID =			
VOV =			
VTH =			
gm =			
ro=1/go =			
Av = vout/vsig =			
BW =			

% error = 100 x (spice – theoretical) / theoretical

2. What SPICE analysis did you run to obtain the data in the table above (write the exact SPICE command you used):

____

____

Attach the corresponding plots/tables.

a.

b.

3. Build the given amplifier and measure the DC operating point.

Parameter	***SPICE value***	***Measured value***	***% Error***
VGS =			
VDS =			
ID =			

4. Use the AWG (Arbitrary Waveform Generator) to generate the signal vsig to be applied at the input of the amplifier. Make vsig a sinusoidal waveform.

   What is the amplitude of vsig you selected?

   amplitude = ____

   Explain shortly how did you select the amplitude?

   ____

   ____

   What is the frequency you selected for vsig?

   freq = ____

   Explain shortly how did you select the amplitude?

   ____

   ____

   Use the oscilloscope to measure the amplitude of vout (that is the small signal voltage taken across RL).

   What is the amplitude of vout?

   amplitude = ____

   vout amplitude / vsig amplitude = ____

5. Use the network analyzer to measure the gain and bandwidth of the amplifier.

   a. Attach the Bode plot.

   b. Compare the values measured with the values you obtained using SPICE:

Parameter	***SPICE value***	***Measured value***	***% Error***
Gain = 20 Log (Av) =			
BW =			