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

<u>Pre-Lab</u>

Consider the following amplifying circuit using a 2N7000 N-channel MOST. For the 2N7000 build your own "simple" SPICE model. Your spice model should look "somehow" similar to the following line:

.model 2N7000 VDMOS KP=0.236 VT0= 1.61 Cgdmax=80p Cgdmin=12p Cgs=50p Cjo=50p



Design the circuit to achieve a small signal gain of at least Av = vout/vsig = 5 V/V. Assume the following specifications: RL=10K Ω , Rsig=50 Ω , 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 =_____

EE 303 - Laboratory

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 =		N/A	N/A

% 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.

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 =			