ENGR 305 – Lab #9

2 weeks

NMOS Common-Source Amplifier

OBJECTIVES:

To study an NMOS-based common-source (CS) amplifier by:

- Completing the DC and small-signal analysis based on its theoretical behavior.
- Simulating it to compare the results with the paper analysis.
- Implementing it in an experimental setting, taking measurements, and comparing its performance with theoretical and simulated results.
- Measuring its output resistance.
- Qualitatively seeing the impact of transistor-to-transistor variations.

MATERIALS:

- Laboratory setup, including breadboard.
- 1 enhancement-type NMOS transistor (e.g., 2N7000)
- 3 large (e.g., 47-μF) capacitors
- Several resistors of varying sizes
- Wires

PART I: DESIGN AND SIMULATION

Design the amplifier to achieve a small-signal gain of at least A_v = -5 V/V. Use supplies of $V_+ = -V_- = 15$ V, $R_{sig} = 50\ \Omega$, $R_L = 10\ k\Omega$, $R_G = 10\ k\Omega$, and design the circuit to have $I_D = 1$ mA. Obtain the datasheet for the NMOS transistor that will be used. In your lab book, perform the following:

DC Operating Point Analysis (Assume for your calculations that $\lambda = 0.0146$/V, $k_n = 1.08$ mA/V^2, and $V_{tn} = 1.45$V.)

- Sketch a DC model of the circuit in your lab book, replacing the three "large-valued" coupling capacitors C_{C1}, C_{C2}, C_S by open circuits (for simplicity you may also omit v_{sig}, R_{sig}, and R_L). What is the DC current through R_G?
- Based on the information just given, you have enough information to calculate $V_{OV} = V_{GS} - V_{tn}$. What is its value? What is the value of g_m? What is V_{GS}? Remember: Your actual transistor will have a value of V_{tn} that will vary from its nominal value, which will alter your measurement results slightly!
- Calculate r_0.
- You now have enough information to calculate R_S. Show your calculation. Is the value you calculate for R_S available? Can you achieve the value by combining resistors?

- Note: At this stage we know neither V_{DS} nor R_D.

AC Analysis

- Sketch a small-signal model of the circuit in your lab book, replacing the transistor with its small-signal model, replacing the capacitors with short circuits (what happens to R_S?), and replacing V_+ with an AC ground. What happens to V_-? Label the gate of the transistor as v_i, i.e., the small-signal voltage at the input.
- What is the ratio of v_i/v_{sig}? How would you approximate it in further calculations?
- Derive an expression for $A_v = v_0/v_i$. What is the value of R_D that produces a small-signal voltage of *at least* A_v = -5 V/V? Is the value you calculated for R_D available? Can you achieve this value by combining resistors?
- What is the DC voltage at the drain? Does this satisfy the assumption that the transistor should be operating in the saturation region? Explain.
- What is the output resistance, R_o?

Simulation

- Simulate your circuit. Use capacitor values $C_{C1} = C_{C2} = C_S = 47$ µF, and the values of R_S and R_D based on your preceding calculations. Use a 10-$mV_{pk\text{-}pk}$, 1-kHz sinusoid with no DC component applied as v_{sig}. This voltage source is the same as used in Lab 3:

  Right-click on voltage source and click on *advanced.*
- Choose SINE:
  - Amplitude – 5 (V)
  - Frequency – 1000 (Hz)

Adding the Spice model directive for the 2N7000:

.model 2N7000 NMOS(kp=1.08m Vto=1.45 lambda=0.0146)

How to use it:

1. Add an NMOS component to your schematic
   a. Right-click the NMOS symbol in the component toolbar.
   b. Place the generic nmos symbol (NMOS) on your schematic.

2. Edit the diode's properties.
   a. Ctrl + Right-click on the NMOS symbol.
   b. In the "Value" field, change the name from NMOS to 2N7000.
   c. Click OK.

3. Add the Spice model directive.
   a. Go to Edit->Spice Directive.
   b. Enter the .model line from above into the dialog box.
   c. Place the directive anywhere on the schematic.

When you run the simulation, LTSpice will recognize the .model directive and apply the specified parameters to any NMOS component named 2N7000.

- From your simulation, report the DC values of V_{GS}, V_{DS}, and I_D. How closely do they match your calculations? (Remember: The simulator has its own more-complex model of the real transistor, so there should be some small variations.)
- From your simulation, report A_v. How closely does it match your calculations?

PART 2: PROTOTYPING

- Assemble the circuit onto your breadboard using the specified component values and those just calculated. Note that R_{sig} represents the output resistance of the function generator, and therefore you should ***not*** include it in your circuit.

PART 3: MEASUREMENTS

- *DC bias point measurement:* Using a digital multimeter, measure the DC voltages of your circuit at the gate (V_G), source (V_S), and drain (V_D) of your transistor.
- *AC measurement:* Using a function generator, apply to your circuit a 10-$mV_{pk\text{-}pk}$, 1-kHz sinusoid with no DC component. (*Note:* Some function generators allow only inputs as small as 50 $mV_{pk\text{-}pk}$. If this is the case, use that value instead.)
- Using an oscilloscope, generate plots of v_0 and v_i vs. t.
- *Output resistance R_0*: Replace R_L with a 1-MΩ resistor and repeat the AC measurement. What is the amplitude of the output waveform? Adjust R_L until you find a value such that the amplitude of the output waveform is approximately 50% of what it was for the 1-MΩ load. This new value of R_L is the output resistance R_0. How does it compare to the value you calculated earlier in Step 2? *Hint:* It cannot be greater than the value of R_D.
- *Further exploration*: What happens to the shape of the output signal as you increase the amplitude of the input signal, e.g., to 1 $V_{pk\text{-}pk}$? At what input amplitude do you begin to see significant distortion? Can you explain this?
- Using a digital multimeter, measure all resistors to three significant figures.

PART 4: POST-MEASUREMENT EXERCISE

- Calculate the values of V_{GS} and V_{DS} that you obtained in the lab. How do they compare to your pre-lab calculations? Explain any discrepancies.
- Based on the measured values of V_D and V_S and your measured resistor values, what is the real value of I_D based on your lab measurements?
- What is the measured value of A_v? How does it compare to your pre-lab calculations? Explain any discrepancies.

- *Hint:* The single biggest source of variations from your pre-lab simulation results will be due to variations in the transistor threshold voltage V_{tn}. Remember: Its value will be somewhere within the range indicated on the transistor datasheet.