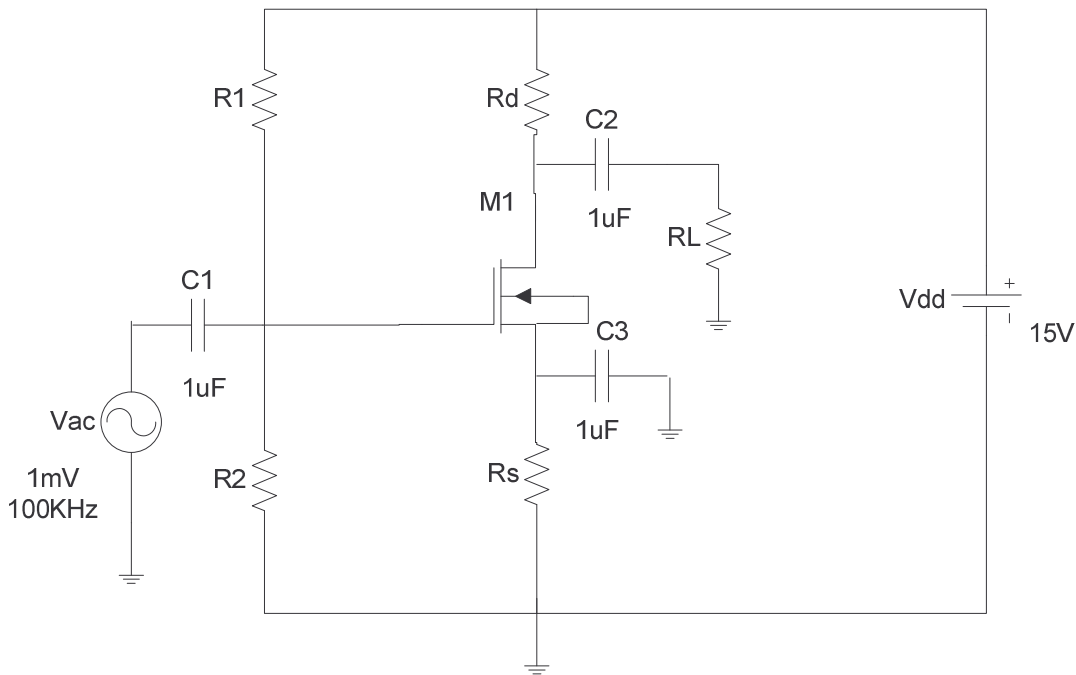


# SPICE Analysis of the MOSFET Common Source Amplifier



Given:  $V_T = 1.45\text{V}$ ,  $\mu_o C_{ox}(W/L) = 500\mu\text{A}/\text{V}^2$ ,  $R_L = 10\text{ k}\Omega$

## 2. Required Specifications

- $V_s = 5\text{V}$ ,  $V_D = 10\text{ V}$
- $V_{GS} = 2\text{V}$
- $I_{DS} = 75\mu\text{A}$

## 3. Derivation of Gain

Derive the gain of the circuit in terms of the resistors and other given parameters.

Gain =  $V_{RL}/V_{ac}$  : \_\_\_\_\_

## 4. Calculation of Resistor Values

From the project specifications in, calculate the values required for resistors  $R_1$ ,  $R_2$ ,  $R_d$  and  $R_s$ .

Show your work, attaching worksheets.

Be sure to label your calculations.

## 5. Building the circuit on Pspice

### 5.1 Getting started

To run Capture:

- Select Start → All Programs → Mosaic XP→Engineering→Electrical→ Cadence PSPICE → Capture CIS.
- Select “PCB Design Studio with Capture CIS”, then click “OK”
- File → New → Project.
- Choose “Analog or Mixed-signal Circuit Wizard”
- Type in the name of the new project and specify the location where it will be saved. Be sure to save it in a subdirectory (for instance, U:\ECGR3131\Project 1), because PSPICE cannot re-open files that have been stored in the root directory and your project will be lost.
- Then you will be prompted to create a PSPICE project. Choose “Create a Blank Project” and click ‘OK’.

### 5.2 Drawing the circuit

The first step in drawing the circuit is acquiring the parts libraries. You do this by opening the Place Part browser and then choosing the Add Library option. You can invoke the Place Part browser in one of the following ways.



- Clicking on the Place Part icon located on the right-hand toolbar.
- Pressing shift + p.
- Choosing *Place > Part* from the Capture CIS menu

To add the libraries click the Add Library button. This action opens a Browse File. This is the default directory for the PSpice libraries. The libraries you will need are analog.olb, source.olb, and fairchild.olb. All the three libraries can be found in the pspice.

To add a library, double click on it. The library will appear in the Libraries window in the Place Part browser. Add all three libraries and then click *OK* then The X, close button, to close the Place Part browser.

After adding all the necessary libraries you will be able to select and add parts to the schematic.


a) Reopen the Place Part browser. To place a part, first highlight the library where the part resides, and then select the part in the Parts List and click *OK*. For instance to add a resistor, highlight ANALOG in the Libraries window then in the Part List window click on R for resistor and click *OK*. The part attaches itself to the cursor and is ready to be placed. The part can be placed by left clicking on the schematic, try it. To add a second resistor simply moves the cursor to another area of the schematic and left click again. The place part mode can be terminated by right clicking and choosing the End Mode option, then you are free to choose another part to place or move on to a new task.

b) Using the same procedures outlined above add the following parts to your schematic:

- Capacitor C (from ANALOG library).
- 2N7000/FAI (from the Fairchild library). Select the transistor, right click and edit its pspice model. In the netlist specified change all VTO to 1.45V and all Kp to 500u (u=micro)
- Power sources VAC and VDC (from source library).
- Add a Ground by clicking the Place Ground icon on the right hand toolbar or pressing shift + g and choose 0 from source library.



c) Create a new simulation profile by selecting the menu from **pspice** tab. Give some name for the simulation profile.

### 5.3 DC Analysis

From the Pspice tab, edit simulation profile, choose DC sweep and under options check save bias points. Now run the circuit by clicking on the  icon. The simulation annotates the bias points on the circuit. Compare the simulated DC currents and voltages to the values obtained using your hand calculation. Make sure that they are within a few percentage of each other.

Attach a print out of the schematic annotated with the DC voltages and currents.

## 5.4 Transient Analysis

Replace VAC source with VSIN source, edit the frequency and voltages(AC) and set them to 100kHz and 1mV respectively. From the Pspice tab, select edit simulation profile, set the simulation profile to transient and set the run time to 100 $\mu$ s. Place voltage level markers  at the input and the output terminals of the circuit. Now run the circuit by clicking on the  icon. Calculate the gain from the simulation.

Attach the plot to the report.

## 5.5 AC Analysis

Switch back the VSIN with VAC and set the AC voltage to 1V, select AC Sweep from the edit simulation tab, set start and end frequencies to 10Hz and 10MHz respectively. Delete both the markers. Select dB magnitude of voltage marker from pspice  $\rightarrow$  markers  $\rightarrow$  advanced analysis. Run the circuit. Find the peak voltage minimum and maximum 3dB frequency points, and the 3dB frequency from the plot.

Attach the plot to the report.

## 6. Results

Gain of the circuit from the hand calculations : \_\_\_\_\_ V/V

Gain calculated from the transient analysis : \_\_\_\_\_ V/V

Gain calculated from the AC Analysis : \_\_\_\_\_ dB

Range of frequency of operation : \_\_\_\_\_ Hz to \_\_\_\_\_ Hz

3dB Bandwidth of the circuit : \_\_\_\_\_ Hz