American University of Beirut

DEPARTMENT OF ELECTRICAL AND COMPUTER ENGINEERING

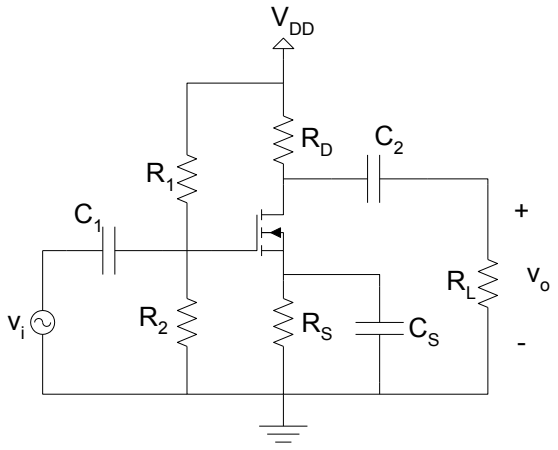
EECE 310 – Electronics Summer 2012

Due Tuesday July 31, 2012 at 11:00 am on Moodle

Design the amplifier shown in the figure below to achieve a small-signal voltage gain $A_v = v_o/v_i$ of -10 for $R_L = 47$ kΩ, and to have $R_i = 1$ MΩ.

Assume that the Q-point is chosen at $I_D = 0.1$ mA and $V_{DS} = 5$ V. Also, assume that $V_{DD} = 12$ V, and that all capacitors are *very large* (and therefore, the capacitors have zero impedance at the signal frequency, but infinite impedance at DC.)

For the MOSFET, $V_t = 0.7$ V, $k'(W/L) = 1$ mA/V^2 and $\lambda = 0.05$ V^{-1}.



Use PSpice to verify your results.

Start by placing the resistors with the values given or as designed.

Place the three capacitors. The capacitors should be "large". Specify their values to be `500uF`. At the signal frequency, which is 1 kHz, the capacitor impedance is $1/(2\pi \times 1\text{ kHz} \times 500\ \mu\text{F}) = 0.32\ \Omega$, which is quite negligible when compared with the other resistance values in the circuit.

Place a VDC part with a value of `12V`. Place the ground node and *make sure that your ground node is named* **0**.

Use a VSIN part for the sinusoidal input voltage source, and call it `Vi`. Set the value of VOFF to `0` (zero), the amplitude of the sine wave (VAMPL) to `1mV`, and the frequency (FREQ) to `1KHz`.

The MOSFET part to use is MbreakN from the BREAKOUT library. After placing the MOSFET, right-click on its symbol, then click on Edit Pspice Model. Replace the `.model` line in the Model Editor with the following line:

```
.model menhn nmos (kp=200u vto=0.7 lambda=0.05)
```

The .model line states that MOSFETs of type "menhn" are N-channel (`nmos`), with:

k' = 200 μA/V^2 (`kp`), V_t = 0.7 V (`vto`), λ = 0.05 V^{-1} (`lambda`)

In order to specify the values of W and L for the MOSFET, double click on the MOSFET symbol. The Property Editor window will open. Scroll horizontally until you see the column labeled "L". Click on "L", then click on the "Display…" button. Select "Name and Value" and click "OK". Scroll horizontally again until you see the column labeled "W". Click on "W", then click on the "Display…" button. Select "Name and Value" and click "OK".

You will now see "L=" and "W=" next to the MOSFET symbol on the schematic. Double click on "L=" to set the value of L to `10um`, then double click on "W=" to set the value of W to `50um`.

Connect all the parts using wires. Use a Net Alias for the input node, which is the non-grounded terminal of the sinusoidal source, and call it "in". To place a Net Alias, use Place → Net Alias from the menu. Place another Net Alias at the output node, which is the non-grounded terminal of resistor R_L, and call it "out".

Observe the node voltages v(in) and v(out) by clicking on the Voltage/Level Marker button on the Capture toolbar, and placing two markers at the input and output nodes.

Create a new simulation profile, and call it transient. Set the Analysis Type to "Time Domain (Transient)". Set the Run to time: value to `8ms`, the Start saving data after: value to `5ms`, and the Maximum step size: to `10us`.

a) Verify, using PSpice, the DC conditions in the circuit, specifically that I_D = 0.1 mA and V_{DS} = 5 V.
Use the V and I buttons on the Capture toolbar to display voltages and currents in the circuit.

List the values of the DC voltages and current in the MOSFET.

Show a printout of your circuit with DC voltages and currents.

b) Find, using Probe, the voltage gain, by measuring the peak-to-peak value of the voltage across the load resistor, and dividing by 2 mV, which is the peak-to-peak value at the input.

Show a printout of the Probe window showing the input and output voltage waveforms.