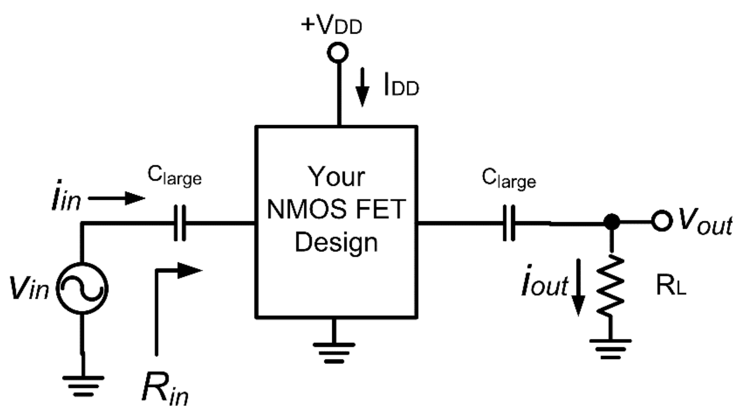
Design a **single stage FET amplifier** to the following specifications. *This is an individual effort; similar designs will not get the extra credit.*

DESIGN SPECIFICATIONS:

- Transistor: NMOS FET type V2N2222, $Kn = 0.06$ A/V^2, $V_{TN} = +2.6$V, r_0 = large.
- Resistor-only DC biasing (no DC current sources allowed)
- Single DC voltage supply, $\mathbf{V_{DD} = +30.0\ V}$
- All resistors <= 500kΩ
- All coupling and bypass capacitors = 1 (large = 1 Farad)
- Load Resistor, R_L = last 4 digits of your student N#, if $R_L < 500$, then $R_L = R_L + 500$
- Target small signal voltage gain (magnitude) is based on your load resistance and is equal to the value in following table $|A_v| = \left| v_{out} / v_{in} \right|$

R_L **(ohms)**	**Gain,** $|A_V|$
500-2,000	2
2,001-8,000	5
8001-10,000	10

- Maximum DC power supply current: I_{DD} **< 100mA**
- Maximum DC power dissipation for transistor: P_D **< 500mWatt**
- Assume that the transistor small signal r_o **= infinite**
- **Extra Points** for a design with low power dissipation, P_D.
- **Extra Points** if your design is stable with changes in temperature.



1. **HAND CALCULATIONS:** Design a single stage NMOS FET amplifier to achieve the above specifications. As this is an amplifier circuit, you should design for a FET in saturation mode.
    a. This is a design project; show all steps for selection of resistors and bias point. Submit all hand calculations for voltage gain, input resistance and output resistance. Show that your transistor is saturated. Extra points for neatness.
    b. What is your load resistance, R_L?
    c. Submit your complete schematic and label all resistor values.

d. On a table, list the following calculated parameters
   i. Voltage Gain, $\boldsymbol{A_v}$?
   ii. Input Resistance, $\boldsymbol{R_{in}}$?
   iii. Output Resistance $\boldsymbol{R_{out}}$? (not including R_L)

e. Calculate the amplitude of the input source, v_{in}, to have the transistor just operating at the condition for small signal. What is this **amplitude for v_{in}**?

f. **Extra Points**: Discuss how your design was optimized for low power dissipation?

2. **PSPICE:** Simulate your design using an input sinusoid, v_{in}, operating at 1kHz with a peak amplitude equal to 1mV. If using Orcad, use the PSpice model for the VN2222 found in the "EVALAA" library.

   a. Simulate the DC performance of your design. Submit your PSpice **schematic** showing values for DC voltage and DC current on the schematic. Your schematic should clearly show the resistor values and DC voltages/currents.

   b. What are the simulated DC bias parameters? **Submit a table showing the simulated values for $\boldsymbol{I_{DD}}$, $\boldsymbol{I_D}$, $\boldsymbol{V_{GS}}$, $\boldsymbol{V_{DS}}$, and $\boldsymbol{P_D=(V_{DS})(I_D)}$**? Is your power supply current, $\boldsymbol{I_{DD} < 100mA}$**?** Is your transistor **power rating < *500mW***?

   c. **Simulated Gain:** Simulate a Transient analysis, **submit** a plot showing v_{in} and v_{out} versus time under small signal conditions. The time scale is 0 to 3 msec, adjust the "max. time step" for a smooth sinusoidal curve (0.02msec). Ideally, plot both curves on the same graph but separate graphs are also acceptable. Are your waveforms *in-phase* or *out-of-phase*? Is there any distortion in the output sinusoidal waveform, v_{out}? What is the simulated voltage gain, $A_v = v_{out}/v_{in}$? Use cursors to measure the peak amplitudes of each voltage waveform and calculate the small signal gain by taking the ratio. Compare the simulated value to your calculated value, what is the percent difference between the two$\left(\frac{simulated-calculated}{calculated}x100\%\right)$?

   d. **Simulated Input Resistance**: Place voltage and current probes on the left side of the input capacitor. Simulate a Transient analysis, **submit** a plot of v_{in} and i_{in} versus time under small signal conditions, these waveforms should be on two separate plots. What is the simulated input resistance, $R_{in} = v_{in}/i_{in}$? Use cursors to measure the peak amplitudes of each waveform and calculate the ratio. Compare the simulated value to your calculated value.

   e. **Simulated Output Resistance**: Remove the input voltage source. Place a short across the input. Remove the load resistor, R_L. Please a voltage source at the output. Simulate a Transient analysis. **Submit** a plot of v_{out} and i_{out} versus time under small signal conditions. What is the simulated input resistance, $R_{out} = {}^{v_{out}}\!/_{i_{out}}$? Use cursors to measure the peak amplitudes of each waveform and calculate the ratio. Compare the simulated value to your calculated value.

f. **Extra Points: Simulated Gain Over Temperature:** Simulate the transient output voltage over temperature. You will need to return the input voltage source and the load resistance back into the circuit and remove the output voltage source. Using Orcad, in the *simulation analysis profile*, there is a temperature sweep setting. Add this checkbox to the transient simulation. Enter the following three temperatures: -40C, +25C and +85C. Run the transient simulation. The three temperature plots will be superimposed onto the same graph. **Submit** the plot showing the output voltage with changing temperature. Calculate the gain at each temperature knowing the input voltage is 1mV. Is your gain stable over temperature? **Submit** a table of the percentage change in gain over temperature and use the gain at +25C as the reference.