Note: All transistor experiments in this Lab are done with 2N3904 NPN transistor. For your analytical analysis use the following values: $\beta = 200$, $h_{ie} = 3 \text{k}\Omega$, and $h_{oe} = 10 \mu\text{S}$.

**Experiment 1: Common Emitter with no Emitter Resistance**

Circuit Analysis: Compute (analytically) the Q point parameters, amplifier gain, cut-off frequency, and input and output resistances of the circuit.

PSpice Simulation: a) Simulate this circuit with PSpice and find the bias points and compare with your analysis of part a and the design values.

b) Simulate the frequency response (10 Hz to 10 MHz) of the circuit to a sinusoidal source with an amplitude of 0.02 V. Find the lower cut-off frequency and the amplifier gain and compare with analytical results of part a.

c) Calculate the input resistance of the amplifier circuit by plotting $v_i/i_i$ using PSpice simulation of part b. Compare with your analytical calculations.

Lab Exercise:

a) Set up the circuit and measure bias voltages and currents and compare with analytical/simulation results. Explain your observations.

b) Set the function generator to produce a sinusoidal wave with a frequency of 2 kHz and amplitude of 0.02 V. (Use 20 dB attenuation button on the function generator). Print the input and output signals (is there a phase shift?), calculate the amplifier gain, and compare with design/simulation results.

c) Measure the lower cut-off frequency of the amplifier circuit and compare with design/simulation results.

d) Set the input frequency back to 2 kHz. Increase the input signal amplitude and observe the output signal shape. At some value, the output signal departs from a sinusoidal wave. Report the output voltage that this happen. Explain the results.
Experiment 2: Common Emitter with Emitter Resistor

Circuit Design and Simulations:

a) Design a common emitter amplifier circuit with a gain of 4 and a cut-off frequency of 200 Hz using a 2N3904 NPN transistor. Set the Q point to have $V_{CE} = 7.5$ V and $I_C = 2.5$ mA.

b) Simulate the circuit with PSpice and find the bias point details and compare with your analysis of part a and the design values.

c) Simulate the frequency response (10 Hz to 10 MHz) of the circuit to a sinusoidal source with an amplitude of 0.5 V. Find the cut-off frequency and compare with analytical results of part a. Compare the amplifier gain with design value. Note that PSpice simulations shows both a lower and a higher cut-off frequencies. Explain why.

Lab Exercise:

a) Build the circuit that you have designed in the Lab. Measure bias voltages and currents and compare with simulation results. Explain your observations.

b) Attach the function generator to the input. Set it to produce a sinusoidal wave with a frequency of 2 kHz and amplitude of 0.5 V. Sketch the input and output signals (is there a phase shift?) , calculate the amplifier gain and compare with design/simulation results.

c) Measure the input resistance of the amplifier circuit by adding a 5 kΩ resistor in series with the function generator.

d) Measure the lower cut-off frequency of the amplifier circuit and compare with design/simulation results. To do so, find the frequency range in which the output signal is the largest (maximum gain). Measure and note the output voltage (you can adjust the input amplitude until the output voltage is a nice round number). Change the frequency of the function generator (without changing the input signal amplitude) until the output voltage is dropped to 0.71 times its maximum. That would be the cut-off frequency.

e) Set the input frequency back to 2 kHz. Increase the input signal amplitude and observe the output signal shape. At some point, the output signal departs from a sinusoidal wave. Report the output voltage that this happen. Explain the results.

f) Compare this circuit with circuit of experiment 1. What are your observations?
Experiment 3: Common Emitter with Emitter Resistor and Bypass Capacitor

Circuit Analysis: Compute (analytically) the Q point parameters, amplifier gain, cut-off frequency, and input and output resistances of the circuit. Note the similarities and differences of this circuit and that of Experiment 2. Explain the function of the 240 Ω resistor.

PSpice Simulation: a) Simulate the circuit with PSpice and find the bias point details and compare with your analytical results.

b) Simulate the frequency response (10 Hz to 10 MHz) of the circuit to a sinusoidal source with an amplitude of 0.2 V. Find the cut-off frequency and the amplifier gain and compare with analytical results.

Lab Exercise:

a) Set up the circuit and measure bias voltages and currents and compare with analytical/simulation results. Explain your observations.

b) Attach the function generator to the input. Set it to produce a sinusoidal wave with a frequency of 2 kHz and amplitude of 0.2 V. Sketch the input and output signals (is there a phase shift?), calculate the amplifier gain and compare with design/simulation results.

c) Measure the input resistance of the amplifier circuit by adding a 5 kΩ resistor in series with the function generator.

d) Measure the cut-off frequency of the amplifier circuit and compare with design/simulation results.

f) Set the input frequency back to 2 kHz. Increase the input signal amplitude and observe the output signal shape. At some value, the output signal departs from a sinusoidal wave. Report the output voltage that this happen. Explain the results.