I. OBJECTIVES

- To perform DC analysis of a single-stage BJT amplifier.
- To do small signal analysis and find the basic properties of a BJT amplifier.
- To simulate the design in PSPICE and verify operation.

II. PRE-LAB

1. For the amplifier Figure 1, assume Q1 has $\beta = 140$, $V_{BE} = 0.7V$, $V_A$ is very large, $V_T = 25 mV$.
   (a) Do DC analysis for the circuit and find its operating or Q point.
   (b) Record the value of $V_C$, $V_{BE}$, $V_E$, and $I_C$.
   (c) Calculate the necessary small signal parameters.
   (d) Calculate the total DC power consumed in this circuit.

2. Replace the transistor by its small signal equivalent model, and find
   (a) Overall Voltage Gain - $A_v$
   (b) Current Gain - $A_i$
   (c) Input resistance - $R_i$
   (d) Output Resistance - $R_o$

3. For a sinusoidal input signal $v_s$ of 10mV peak amplitude and 1KHz frequency draw
   the total instantaneous quantities $v_{BE}(t)$, $v_C(t)$ and $i_C(t)$ for one cycle of the input.
   Clearly mark all values, graphs and axes.

4. [DO this part for lab report only]
   What is the maximum value that the signal source $v_s$ can take in order to ensure
   small signal operation and that the transistor is always in active region?

---

**Figure 1**

- $R_s = 10K\Omega$
- $C = \infty$
- $v_s$
- $v_i$
- $i_i$
- $Q_1$
- $v_o$
- $R_e = 100K\Omega$
- $R_C = 10K\Omega$
- $R_L = 10K\Omega$
- $R = 10K\Omega$
- $- 10V$
III. LAB PROCEDURE

1. (a) Draw the amplifier circuit in Figure 1 using MICROSIM SCHEMATICS.
   (b) Choose the capacitors to be very large eg: 1Gf.
   (c) Choose the 2N2222 transistor model for Q1.

2. (a) Set the input voltage $v_s$ to be sinusoidal with 10mV peak amplitude and 1KHz frequency.
   Perform a transient analysis for two cycles of the input.
   (b) Observe the voltages $v_{BE}(t)$, $v_{C}(t)$, $v_{B}(t)$ and $v_{E}(t)$.
   (d) Observe the currents $i_C(t)$, $i_B(t)$ and $i_E(t)$.

3. Find
   (a) Voltage Gain - $A_v$ ($v_o / v_i$)
   (b) Overall Voltage Gain ($v_o / v_s$)
   (c) Current Gain - $A_i$ ($i_o / i_i$)
   (d) Input resistance - $R_i$ This is done by finding $v_i/i_i$ using PSPICE.
   (e) Output Resistance- $R_o$. To find the output resistance you need to find $v_o/i_x$ where $i_x$ is the current leaving the $v_x$ source at the output. For this analysis set input $v_s$ to 0V and give a sinusoidal input at $v_x$.

4. Find the total DC power consumed in this circuit through your simulation.

5. Verify that the calculated values from the pre-lab match the computer-generated values.
   Complete the below table. Explain why you have a difference between your hand calculated analysis and the PSPICE results.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Analysis</th>
<th>PSPICE</th>
<th>% Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>$R_i$</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$R_o$</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$A_v$</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$A_i$</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

6. Change the emitter circuit to make the resistor $R_E = 5K\Omega$ and add some emitter degeneration with the resistor $R_{EX}$ also of $= 5K\Omega$. Find
   (a) Voltage Gain - $A_v$ ($v_o / v_i$)
   (b) Overall Voltage Gain ($v_o / v_s$)
   (c) Current Gain - $A_i$ ($i_o / i_i$)
   (d) Input resistance - $R_i$
   (e) Output Resistance- $R_o$,
   Show how emitter degeneration changed each of these design parameters.

IV. REPORT

Prepare a detailed report that includes both preliminary design and PSPICE simulations, as well as measured data. A discussion should attest to design validity, address any discrepancies, and include any suggested improvements in design. The lab report should contain the following:

1 a) A clean paper and pencil analysis of the amplifier circuit.
   b) A schematic printout of the circuit.
   c) A printout of simulation plots and the circuit netlist.