University of California, San Diego  
Department of Electrical and Computer Engineering

ECE60L, Spring 2004

Lab 6, BJT Amplifiers: Biasing and Emitter Follower

Note: All transistor experiments in this Lab are done with 2N3904 NPN transistor. For your analytical analysis use the following values: $\beta = 200$, $r_i = h_{ie} = 3 \text{k}\Omega$, $\beta = h_{fe} = 200$, and $h_{oe} = 10 \mu\text{S}$. Use formulas for emitter follower given in Page 113 of lecture notes and see Example in page 114.

Experiment 1: Poor biasing

Set up the circuit with $R_B = 1.5 \text{M}\Omega$, $R_C = 5 \text{k}\Omega$, and $V_{CC} = 15 \text{V}$.

Circuit Analysis: Compute $I_C$, $I_B$, and $V_{CE}$.

PSpice Simulation:

a) Simulate the circuit with PSpice (bias point details only) and compare values of $I_C$, $I_B$, $V_{CE}$, and $V_{BE}$ from PSpice simulations with your analytical calculations.

b) Rerun your PSpice simulations for temperatures of 0 and 60°C. Make a table of $I_C$, $I_B$, $V_{CE}$, and $V_{BE}$ for the above temperatures and that of part a (which is for default temperature). Explain your observations.

Lab Exercise:

a) Set up the circuit and measure $I_C$, $I_B$, $I_E$, $V_{CE}$, $V_{BE}$, and $V_{BC}$. Note that these measurements are not straightforward. You have to consider the sensitivity of your meter as well as its internal resistance. In some cases, you have to measure the quantities “indirectly.” Check if the transistor obeys KVL and KCL.

b) Compare your measurements of $I_C$, $I_B$, $V_{CE}$, and $V_{BE}$ with your PSpice simulations. Explain why they are different.

c) Set the voltmeter to measure $V_{CE}$. Hold the transistor between two fingers without touching the rest of the circuit. Transistor will warm up slightly by your body temperature. What happens to $V_{CE}$ as transistor warms up? Explain your observations.
Experiment 2: Good biasing

Set up the circuit with \( R_2 = 12 \, \text{k}\Omega \), \( R_1 = 39 \, \text{k}\Omega \), \( R_C = 2 \, \text{k}\Omega \), \( R_E = 1 \, \text{k}\Omega \), and \( V_{CC} = 15 \, \text{V} \).

*Circuit Analysis:* Compute \( I_C \), \( I_B \), and \( V_{CE} \).

*PSpice Simulation:*

a) Simulate the circuit with PSpice (bias point details only) and compare values of \( I_C \), \( I_B \), \( V_{CE} \), and \( V_{BE} \) from PSpice simulations with your analytical calculations.

b) Rerun your PSpice simulations for temperatures of 0 and 60°C. Make a table of \( I_C \), \( I_B \), \( V_{CE} \), and \( V_{BE} \) for the above temperatures and that of part a (which is for default temperature). Explain your observations. Compare your results with Experiment 1.

*Lab Exercise:*

a) Set up the circuit and measure \( I_C \), \( I_B \), \( I_E \), \( V_{CE} \), \( V_{BE} \), and \( V_{BC} \). Compare your measurements with your PSpice simulations. Explain why they may be different.

b) Set the voltmeter to measure \( V_{CE} \). Hold the transistor between two fingers without touching the rest of the circuit. Transistor will warm up slightly by your body temperature. What happens to \( V_{CE} \) as transistor warms up? Explain your observations.

c) Compare your results from this experiment with those of Experiment 1. Explain why this is a better biasing circuit.

Experiment 3: Emitter Follower

Set up the circuit below with \( R_1 = 47 \, \text{k}\Omega \), \( R_2 = 39 \, \text{k}\Omega \), \( R_E = 1.5 \, \text{k}\Omega \), \( C_1 = C_2 = 100 \, \text{nF} \), and \( V_{CC} = 15 \, \text{V} \). Use a 100 kΩ resistor as \( R_L \). The input signal, \( v_i \), is sinusoidal with an amplitude of 1 V and frequency of 5 kHz.

*Circuit Analysis:*

a) Compute the bias point.

b) Compute voltage gain, input and output resistances, and the lower cut-off frequency of this amplifier.

*PSpice Simulation:*

a) Simulate the circuit with PSpice (ask for bias point details) and compare bias point values with analytical results.

b) Simulate the frequency response of this circuit in the frequency range of 10 Hz to 1 MHz. Plot the gain (in dB) and phase shift of the signal at the transistor base (point B), at the
emitter (point E) and at the output (point O). Compare your simulation results with the analytical calculations.

c) Use your simulations of part b, plot the input resistance ($R_i v_i / i_i$) of this amplifier as a function of frequency. Compare with your analytical calculations.

*Lab Exercise:*

a) Set up the circuit (do not attach $v_i$) and measure $I_C, I_B, I_E, V_{CE}, V_{BE}$, and $V_{BC}$. Compare your measurements with your circuit analysis and PSpice simulations.

b) Attach the input signal to the circuit. Sketch and compare signals at the input with those at points B, E, and O. Explain your observations.

c) Note your calculations and simulations of the cut-off frequency of the circuit. Scan the input frequency from $0.1 f_c$ to $10 f_c$. Measure and plot the gain (in dB) and phase shift of signals at the output. Compute the cut-off frequency and compare with your analytical/simulation results.

d) Measure the input resistance of this amplifier circuit by attaching a resistance in series with the input signal (similar to Lab 3). Use your analytical/simulation results to guide you in choosing the proper value for this resistor.