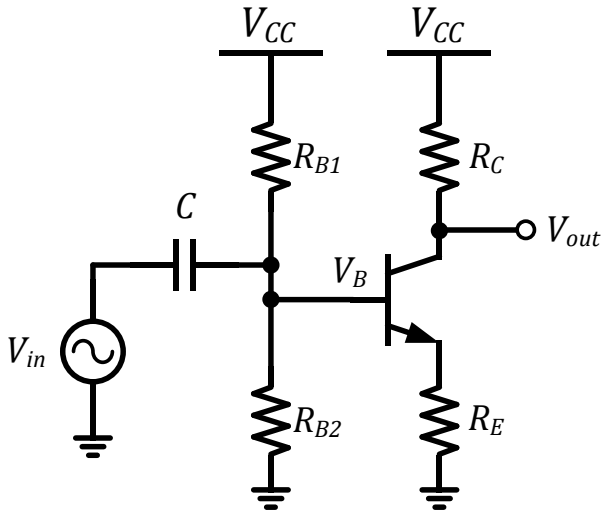
EE 538 Spring 2020
Analog Circuits for Sensor Systems
University of Washington Electrical & Computer Engineering

Instructor: Jason Silver
Homework #2 (40 points)
Due Saturday, April 18, Submit on Canvas

Please show your work.

Problem 1: Common-emitter amplifier



For the following, use the figure and the simplified NPN model from Section 2.1.1 in AoE. V_{CC} = 5V, C = 1μF, and β = 100. Use the *npn* Ltspice component for your simulations.

a) Select values for R_{B1} and R_{B2} to achieve V_B = 1V and a high-pass corner frequency (f_{3dB}) of 10Hz. You may ignore base current for this step.
b) Choose R_C and R_E for a gain of −10V/V (at 10kHz) and a collector current of 1mA. What is the DC value of V_{CE}?
c) Derive the transfer function for V_{out}/V_{in} and plot the frequency response (magnitude and phase) in MATLAB/Python.
d) Perform an AC simulation of the circuit in Ltspice. Export the frequency response data for comparison to the ideal response from Part c (plot them together). Provide reasons for any discrepancies between the two.

Problem 2: Emitter follower

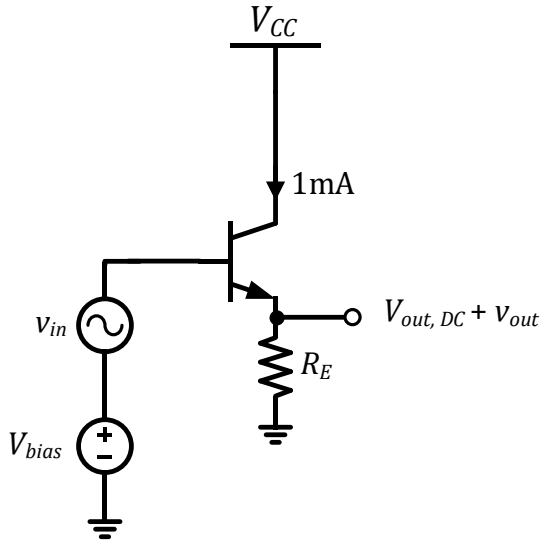


Figure 2a. Emitter follower

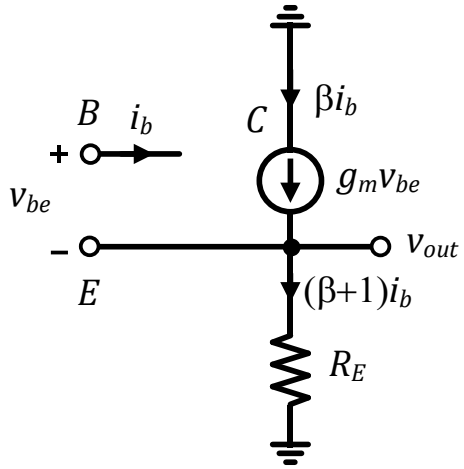


Figure 2b. Small-signal equivalent circuit

Use the Ebers-Moll model of the BJT and the figures to answer the following questions. V_{CC} = 5V, I_S = 10^{-16}, and β = 100. When determining input/output resistances, connect a test voltage to the small-signal circuit and determine the resistance as $r = v_{test}/i_{test}$. Use the *npn* Ltspice component for your simulations.

a) Design the biasing of the emitter follower (i.e. determine V_B and R_E) such that the collector current is 1mA and the DC level of V_{out} is 1V. You can do this by hand or use a MATLAB/Python script.
b) Use the small-signal model (Fig. 2b) to determine the input resistance of the circuit.
c) Use the small-signal model (Fig. 2b) to determine the output resistance of the circuit.
d) Verify your design in Ltspice and include all relevant SPICE schematics and results in your submission.