

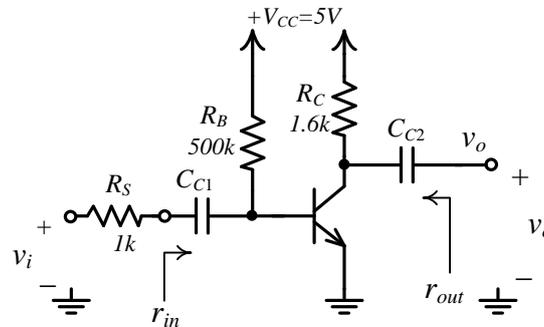
Student Name:

Student ID:

EHB262E Electronics II

Homework 1

Deadline: 01/11/2012 (before the lecture)



Common-emitter amplifier

Construct the above circuit using SPICE. Coupling capacitors C_{C1} and C_{C2} should take high values; you can select 1mF for both. Use Philips 2N2222 model for the transistor.

- Find V_B and V_C of the transistor by performing bias point (DC operating point) analysis in SPICE.
- Find small signal resistances r_{in} and r_{out} as well as the small signal voltage gain v_o/v_i by performing transient analysis in SPICE. Use a sine signal with 1mV peak-to-peak amplitude and 1kHz frequency as a small signal voltage source.
 - Hint: calculate r_{in} , r_{out} , and v_o/v_i as a ratio of the corresponding sine signals' amplitudes.
- Construct a linear model for the amplifier by using the values obtained in b). The model should consist of R_S , r_{in} , r_o , and a **controlled voltage source**.
- Determine the minimum value of R_L (load resistance connected to the output) such that v_o/v_i does not drop below **90%** of the gain achieved without R_L .

Grading: a)15 %, b) 45%, c)20%, d)20%

Note: Do not forget to attach SPICE **output file** prints to your homework!