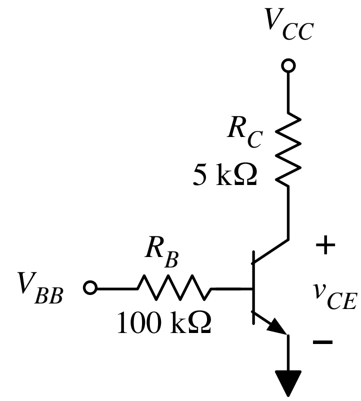
EE 230 — HW 5.3 Name____



For the simple inverter circuit (NOT gate), use SPICE to perform the following simulations. For the transistor, $\beta_F = 150$. (It will be necessary to change that for the *npn* model in SPICE. The default value in SPICE is 100.)

a. Find the DC values for i_C and v_{CE} for the following power supply combinations. Also, calculate the values by hand using the simplifying forward-active and saturation models for the BJT. You should use a *bias point* simulation for these DC calculations.

	i_C (manual)	i_C (SPICE)	v_{CE} (manual)	v_{CE} (SPICE)
$V_{BB} = 1.35$ V $V_{CC} = 10$ V				
$V_{BB} = 1.35$ V $V_{CC} = 5$ V				
$V_{BB} = 2.35$ V $V_{CC} = 10$ V				
$V_{BB} = 2.35$ V $V_{CC} = 20$ V				

b. With $V_{CC} = 10$ V, make a plot of the inverter characteristics (v_{CE} vs. V_{BB}). This will require a DC sweep simulation with V_{BB} sweeping from 0 to 10 V. Use at 10 points per volt when setting up the simulation.

c. Change V_{BB} to a sinusoidal source (VSIN) with amplitude of 0.1 V, frequency of 1 kHz, and DC offset of 1.35 V. Do a transient simulation, plotting the v_{CE} and V_{BB} waveforms together on a single set of axes. Plot at least two full periods of the sinusoid. From the voltage traces, note the gain of the amplifier.

Put your answers — and whatever work will fit — on this sheet. Staple any sheets with additional work behind.