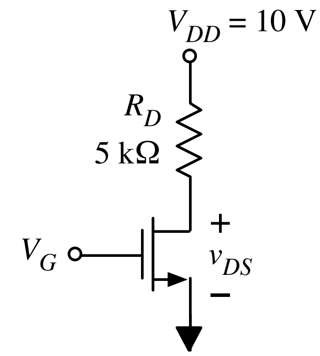
EE 230 — HW 6.7 Name____



For the simple inverter circuit, use SPICE to perform the following simulations.

For the transistor, V_T = 1 V and K_n = 0.25 mA/V². (It will be necessary to change the K value for the NMOS model in SPICE.)

a. Find the DC values for i_D and v_{DS} for the following power supply combinations. You should use a *bias point* simulation for these DC calculations.

	i_D (SPICE)	v_{DS} (SPICE)
V_G = 3 V V_{DD} = 10 V		
V_G =3 V V_{DD} = 5 V		
V_G = 5 V V_{DD} = 10 V		
V_G = 10 V V_{DD} = 10 V		

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

c. Change V_G to a sinusoidal source (VSIN) with amplitude of 0.1 V, frequency of 1 kHz, and DC offset of 3 V. Do a transient simulation, plotting the v_{DS} and V_G 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.