1. Perform SPICE simulations for each of the circuits that will be built in the procedure for the Discrete Op Amp. Use the SPICE parameters for the NPN and PNP transistors given in the Preliminary SPICE Simulations sections. Assume that the current source for the differential amplifier is an ideal current source that has an $I_T$ equal to the value used for the procedure section. Use the actual values measured for the resistors and the compensation capacitor used in lab. Obtain the dc operating point, an ac analysis, and a transient analysis sufficient to show the clipping behavior and the slew rate. For design calculations you may invoke the infinite $\beta$ assumption for the BJTs which means the collector and emitter currents are the same and the base currents are zero.