1. Shown below are two common collector aka emitter follower circuits. Simulate the second circuit shown below with National Instruments SPICE, Multisim. Use the SPICE parameters for the NPN transistor given in the Preliminary SPICE Simulations sections of the lab manual. Obtain the dc operating point, an ac analysis, and a transient analysis sufficient to show the clipping behavior. Perform the analysis for $I_E = 1$ mA, 5 mA, and 10 mA. (For the second circuit compute the values of $R_1$, $R_{E2}$, and $R_{E3}$ to produce the desired $I_E$.) For the calculations assume $\beta = \infty$. Pick a value for $R_{E3}$ and $I_{C3}$ then compute $R_1$ and $R_{E2}$.) Use the component values: $C_1 = C_2 = 22 \mu F$, $C_E = 330 \mu F$, $R_B = 51 k\Omega$, $R_E = 1 k\Omega$, and $R_L = 10 k\Omega$. The dc power supply voltages are: $V^+ = -V^- = 15 V$. 