

ECE 4043 Spring 2015

Homework Problem Set No 4 for Experiment No 4

Due Week of February 8

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 $I_E = 1\text{ mA}$, 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: $C_1 = C_2 = 22\ \mu\text{F}$, $C_E = 330\ \mu\text{F}$, $R_B = 51\ \text{k}\Omega$, $R_E = 1\ \text{k}\Omega$, and $R_L = 10\ \text{k}\Omega$. The dc power supply voltages are: $V^+ = -V^- = 15\ \text{V}$.

