

ECE 3043

Summer 2018 Homework Problem Set No. 10 for Experiment No. 11

Due Second Meeting Week of July 9

1. Use both Multisim and LT SPICE to plot the voltage transfer characteristic V_o versus V_i for the circuit shown below for $-5\text{ V} \leq V_i \leq 5\text{ V}$. Assume that the op amp is ideal and that each of the four diodes has as its dominant SPICE parameters: Reverse Breakdown Voltage 100 V, Emission Coefficient 1.8, and Reverse Saturation Current 12.61 fA . The component values are: $V^+ = 15\text{ V}$, $V^- = -15\text{ V}$, $R_1 = 1\text{ k}\Omega$, $R_2 = 3\text{ k}\Omega$, $R_3 = 5.1\text{ k}\Omega$, and $R_4 = 5.1\text{ k}\Omega$. Compare the simulation results with the theoretically expected values with regard to break points and slopes. Also plot the currents in the four diodes as functions of V_i .

2. Use both Multisim and LT SPICE to plot the output voltage $v_o(t)$ as a function of time for the circuit shown below if the input is

$$v_i(t) = A \sin(\omega t)$$

where $A = 5\text{ V}$ and $f = 1\text{ kHz}$ for two cycles of $v_i(t)$. Compare the peak value of the output from the simulation with the theoretically expected value. Use both Multisim and LT SPICE to plot the spectra output voltage as a function of frequency from 0 Hz to 20 kHz.

