ECE 3043 Spring 2020

Homework Problem Set No. 4 for Experiment No. 4

Due Week of February 10

1. For the circuit shown below, use National Instruments SPICE (Multisim) to plot the Bode plot of the complex transfer function V_o/V_i, as the frequency of the voltage source, $v_i(t)$, varies from $0.1f_o$ to $10f_o$.where f_o is the resonant frequency of the circuit. Compare the simulation results with the theoretical results for f_o, Q, and Δf . Assume that the voltage source $v_i(t)$ is a sine wave with an rms value of 1 V. The component values are $R_1 = 20\,\text{k}\Omega$, $R_2 = 20\,\text{k}\Omega$, $L = 3\,\text{mH}$, and $C = 1\,\text{nF}$. Use either Mathcad and Matlab to plot the magnitude and phase of the voltage $v_o(t)$; assume that the phase of the source $v_i(t)$ is zero with the positive side up. (Note that since the input is unity this is equivalent to finding the Bode plot.)

2. For the circuit shown below plot $v_o(t)$ versus t using either MathCad or Matlab. The range of t for the plot is from 0 to $4T_d$ where $T_d = 1/f_d$, $f_d = f_o\sqrt{1-\zeta^2}$, $\zeta = (G/2)\sqrt{L/C}$ and $G = 1/R$ where R is the Norton resistance seen by the parallel combination of L and C.. The input is $v_i(t) = E_o u(t)$ where $E_o = 10\,\text{V}$ and $u(t)$ is the unit step function. Compare the theoretical and simulation results for the driven frequency, f_d, and the attenuation factor of the envelope, α, (use the $i_c(t)$ plot for this with Multisim). The component values are the same as for Problem 1.