1. For the circuit shown below, use both National Instruments and LT SPICE to plot the magnitude and phase of the voltage $v_o(t)$ as a function of frequency (frequency log, mag lin, phase lin) as the frequency of the voltage source, $v_i(t)$, varies from 10 kHz to 1 MHz. Compare the simulation results with the theoretical results. Assume that the voltage source $v_i(t)$ is a sine wave with an rms value of 1 V. The component values are $R_1 = 510 \, \Omega$, $R_2 = 11 \, \Omega$, $L = 3 \, \text{mH}$, and $C = 1 \, \text{nF}$. Use Mathcad and Matlab to plot the magnitude and phase of the voltage $v(t)$; assume that the phase of the source $e(t)$ is zero with the orientation shown. (Note that since the input is unity this is equivalent to finding the Bode plot.) For Multisim, Mathcad, and LTSpice use 2,000 logarithmically spaced points. Compare the simulation with the theoretical value for the quality factor and resonant frequency.

2. For the circuit shown below, use Mathcad, Matlab, National Instruments and LT SPICE to plot the voltage $v_o(t)$ versus $t$ as $t$ varies from 0 to $3/f_o$. Also plot the current flowing from left to right through the resistor $R_1$. Compare the simulation results with the theoretical value for the attenuation factor for the envelope and the driven frequency using the current plot. The voltage source is $v_i(t) = 10u(t) \, \text{V}$.