1. The excitation for both circuits shown below is $e(t) = 10\, V u(t)$. Plot the voltage $v_o(t)$, the reactive voltage (assume the left terminal is positive) and current (flowing through the reactive element from left to right) for the two circuits using Mathcad. Plot the circuit variables as $t$ varies from 0 to two time constants for the circuits. The values of the circuit components are $R_1 = 4.3\, \text{k}\Omega$, $R_2 = 200\, \Omega$, $R_3 = 5.1\, \text{k}\Omega$, $R_4 = 2.4\, \text{k}\Omega$, $L = 3\, \text{mH}$, and $C = 0.022\, \mu\text{F}$.

2. Make the same plot as in Problem 2 using Matlab.

3. Make the same plot as in Problem 2 using National Instruments SPICE (Multisim).

4. Make the same plot as in Problem 2 using LTSpice (text editor input mode).