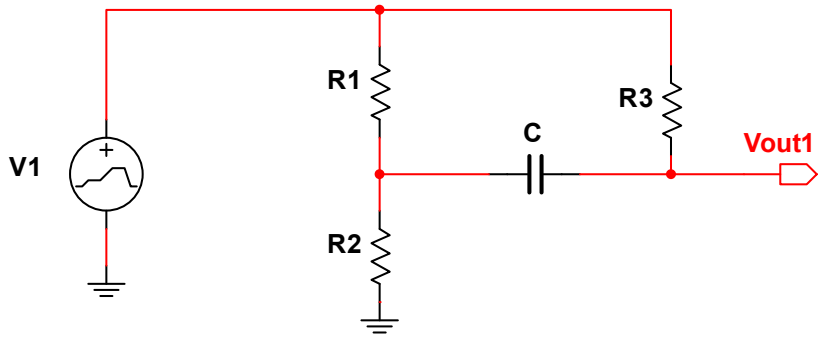
ECE 3043 Spring 2020
Homework Problem Set No 3 for Experiment No. 3

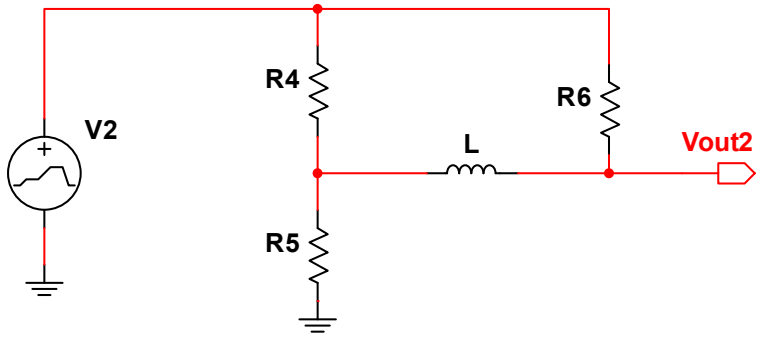
Due Week of February 3

1. The excitation for both circuits shown below is $v_i(t) = 10\,\mathrm{V}u(t)$. For Circuit 1 plot the capacitor voltage and capacitor current for 3 time constants using either Mathcad or Matlab. Assume the terminal on the right is positive. Repeat for Circuit 2 for the inductor voltage and current. The component values are: $R_1 = R_4 = 100\,\mathrm{k}\Omega, R_2 = R_5 = 100\,\Omega, R_3 = R_6 = 3.9\,\mathrm{k}\Omega, C = 1\,\mathrm{nF}$, and $L = 1\,\mathrm{mH}$.

2. Make the same plot as in Problem 1 using National Instruments SPICE (Multisim). Hint:. Multisim will not list either the capacitor current or voltage as a variable to be plotted. So for an output variable specify an expression as the voltage across R_3 divided by R_3. (Ohms's Law). The current will be numerically smaller than the voltages. So for it to display plot the current divided by 0.001 which will yield the current in mA.Otherwise the current plot would be a horizontal line.



Circuit 1. Capacitor



Circuit 2. Inductor