RL, RC, AND RLC CIRCUITS

The primary goal of this assignment is to quickly review what you already know about capacitors, inductors, and AC circuits and to extend your new circuit analysis skills to cover sinusoidal signals. The assignment draws from Chapters 6-10 of your text. You will also learn how to use Spice to simulate time dependent effects and to analyze frequency dependent response in AC circuits.

We increasingly conceive of circuits in terms of terminal behavior. That is, we label the source signal v_{in} and the potential difference between some other pair of nodes v_{out}, and we think of the network of devices between the input and output terminals to be a filter that transforms v_{in} into v_{out}.

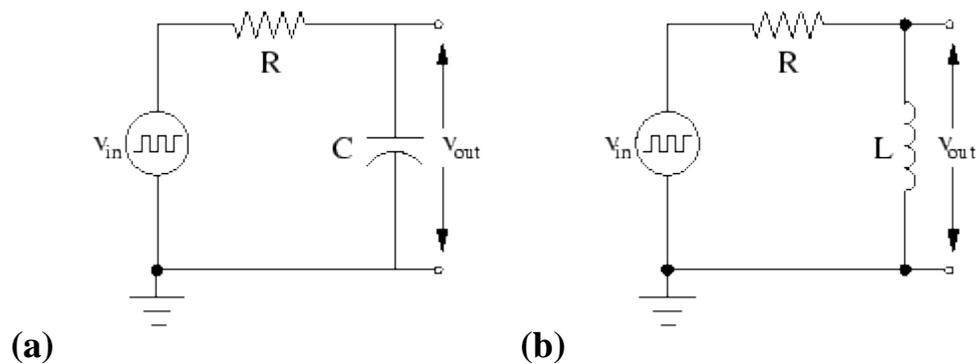# Measuring C and L: Natural/Step Response

1. **Experiment**

Figure 5: Series RC and RL circuits.

2. You have been provided with a breadboard, a resistor, a capacitor, an inductor, a function generator, and an oscilloscope. First, measure the resistance of the resistor by any means you like. Then construct the series LC and RC circuits shown in Figure 5. Drive the circuits with a square wave signal from the function generator.

3. For each circuit, pick a good frequency, and find the capacitance/inductance by measuring the time constant of the response with the oscilloscope. Explain what made your choice of frequency ``good.'' Use one channel of the oscilloscope to measure v_{in}, and trigger on that channel.

Measure v_{out} with the other channel to look at the natural and step response of the circuit. Measuring the half time and converting to the time constant may be easier than trying to measure the time constant directly. Use the $\times 10$ time base magnification feature of the oscilloscope to improve the precision of your time measurements. Give the uncertainties in your results.

4. **Simulation**

   Use a transient analysis (.TRAN analysis with a PULSE signal) to simulate one of your experiments of Section 1. You can measure the voltage levels and rise and fall times of the function generator with the oscilloscope to (possibly) produce a more realistic simulation. Compare and contrast the voltage plot from your simulation with your measured oscilloscope traces.

Source: http://webpages.ursinus.edu/lriley/ref/circuits/node3.html