EE223 Laboratory
Second-Order Circuits

Objectives
1) Gain an intuitive understanding of the step response for second-order circuits that are overdamped, critically damped, and underdamped.
2) Maintain, develop oscilloscope skills
3) Maintain Spice skills

Prelaboratory
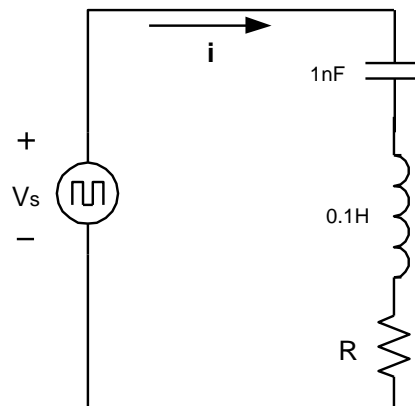
Consider the following circuit.

Figure 1: Prelab RLC circuit

Overdamped Response

1.   Let R=100kΩ in the above circuit. Let $v_s(t) = 5\,u(-t)$, which will create some initial conditions. Analytically (i.e. using methods covered in class) find i(t), t≥0. Not sure what u(-t) is? Graph it first by plugging in different values for t and evaluate by hand u(-t) at those points. Once you have an analytic solution, graph the first 0.5 ms of it using Matlab. To do this, make a time vector using `linspace`, and plot the result using `plot`. (Remember to use `help linspace`, for example, to get online help). Print out your Matlab solution.

2.   Complete a Spice simulation of the above circuit, but this time using a squarewave input for $v_s(t)$ that varies between 0 and 5V with a frequency of 1kHz. It should be 0V for $0 \leq t < 0.5$ms, and then 5V for 0.5ms $\leq t < 1$ms. Simulate this circuit to find i(t) for $0 \leq t \leq 1$ms. Spice should show a classic decaying exponential that matches the first 0.5ms of your Matlab simulation. Print out the Matlab plot. Use the Spice walkthrough to help you, especially with defining the squarewave.

Critically-Damped Response

3.   Determine the resistor's value that will produce a critically-damped circuit. Derive a new analytic solution using that value for i(t), $0 \leq t < 0.5$ms, using the same $v_s(t) = 5\ u(-t)$V input voltage you analyzed earlier, and print it out as you did in Problem 1 using Matlab.

4.   Generate a new Spice simulation for this circuit with the new R using the same squarewave you used in Problem 2. Print out i(t) over the same 1ms period as you did in the previous Spice simulation.

Underdamped Response

5.   Do the same hand analysis of i(t) as you did for Problems 1 and 3, but this time using $R = 2$kΩ. Print out i(t) for $0 \leq t \leq 0.5$ms using Matlab as you did previously.

6.   Do the same Spice simulation with this new value of R, and print out the resulting i(t) waveform over the same 1ms period as you did above in Problems 2 and 4.

Note that you can check your work – your Matlab plot for Problems 1, 3, and 5 should match the Spice results for Problems 2, 4, and 6 respectively for the first 0.5ms of time. (After 0.5ms, the Spice input will change its input voltage from 0V to 5V since it is a squarewave).

Laboratory

Procedures:

1. Overdamped response

Connect the circuit of Figure 2 with R=100kΩ. Set the function generator to produce a 1 kHz squarewave that varies between 0 and 5V. Traditionally, channel 1 is used to display an input and channel 2 to display the response.
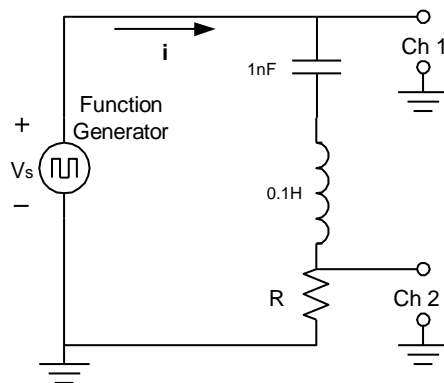

Figure 2: Laboratory series RLC circuit

Since an oscilloscope cannot directly measure current, instead use channel 2 to measure the voltage dropped by the resistor and recognize this is proportional to the current through the circuit. Sketch this waveform in your notebook, and fully label your diagram with x and y axis labels and scales. For your laboratory report, include labeled graphs of the input voltage waveform and the output current waveform. Since the input is zero, this represents the natural response of the circuit. Record both the maximum absolute current in the circuit and the time at which it occurs. You will find that adjusting the horizontal time base to a shorter sweep will permit you to get a much more accurate reading of time of maximum current flow.

2. Critically damped response

Set the value of R in the circuit of Figure 2 to the value calculated in the prelaboratory exercise. Again, sketch the voltage waveform across the resistor. Make note of the maximum absolute current in the circuit, and the time at which this occurs.

3. Underdamped response.

Use the value of R in Figure 2 to 2kΩ, and observe the resulting current waveform making note of the frequency of the decaying sinusoid. Record a sketch of this waveform.

Discussion Questions:

Compare analytic solutions, Spice solutions, and oscilloscope results of the natural current response for each case. How can you compare them (e.g. time to rise, time to fall, maximum value, time constant, frequency,…)? Do all appear to generate substantially equivalent results? At a minimum, compare both general shape and maximum absolute current values. What do you think was primarily responsible for any differences? For the underdamped case compare the calculated frequency and the envelope time constant with the observed values. Do not be a qualitative humanities major ("they all look similar"); be a quantitative engineer ("the risetime of the Matlab model is within 1% of the Spice simulation, but is 12% lower than the observed in the lab. This is probably caused by…")

Do you need to be concerned about the non-ideal characteristics of the oscilloscope (i.e. input resistance of 5MΩ is not infinite) and function generator (i.e. output resistance of 600Ω is not zero)? If so, correct your simulations to take this into account. If not, give an example of a value of R in the circuit for which you would need to include the generator's non-ideal behavior, and an example of an R that would make the oscilloscope appear to load the circuit.