LAB 5: Computer Simulation of RLC Circuit Response using PSpice

PURPOSE

To use a computer simulation program (PSpice) to investigate the response of an RLC series circuit to:
(a) a sinusoidal excitation.
(b) a step excitation under conditions of underdamping, critical damping and overdamping.

EQUIPMENT

(i) Student computer network station.
(ii) OrCad Capture PSpice circuit simulation software.

This lab is to be done individually unless equipment shortages require otherwise.

BACKGROUND

Part 1. Frequency Domain Response of RLC Circuit to Sinusoidal Excitation

Figure 1 shows a series RLC circuit driven by an ideal voltage source, V_S.

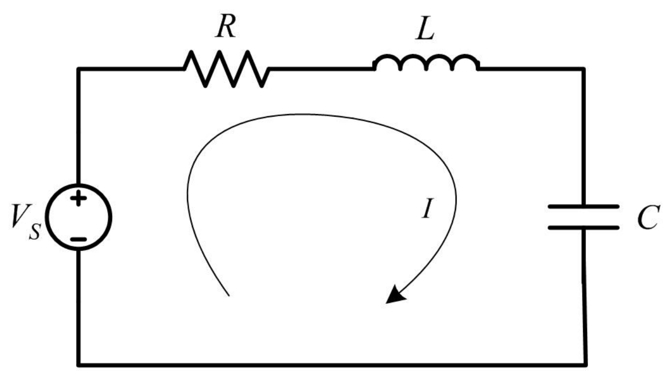

Figure 1. Series RLC circuit.

The loop current I is given by:

$$I = \frac{V_S}{R + j\omega L + \dfrac{1}{j\omega C}} = \frac{V_S}{R + jX} \tag{1}$$

where:

$$X = \omega L - \frac{1}{\omega C} \tag{2}$$

The magnitude of the current phasor is:

$$|I| = \frac{V_S}{\sqrt{R^2 + X^2}} \tag{3}$$

The current has a maximum value when X = 0, which occurs when:

$$\omega L = \frac{1}{\omega C} \quad \text{or} \quad \omega = \omega_0 = \frac{1}{\sqrt{LC}} \tag{4}$$

This phenomenon is called resonance. At the resonant frequency ω_0, the magnitude of current takes the value:

$$|I| = |I|_{MAX} = \frac{|V_S|}{R} \tag{5}$$

and the voltage across the reactance X, i.e. the voltage across the combined impedance of L and C in series, is zero. For frequencies higher or lower than ω_0, the current is reduced below its peak value. At the resonant frequency, the voltage across the capacitor is:

$$V_C = |I|_{MAX} \frac{1}{j\omega_0 C} \tag{6}$$

From (4) and (5):

$$|V_C| = |V_S| \frac{1}{\omega_0 CR} = |V_S| \frac{\omega_0 L}{R} \tag{7}$$

We define the ratio:

$$\frac{|V_C|}{|V_S|} = \frac{|V_L|}{|V_S|} = Q \tag{8}$$

where Q is the *quality factor* of a series RLC circuit, and is given by:

$$Q = \frac{1}{\omega_0 CR} = \frac{\omega_0 L}{R} \tag{9}$$

If R is small, Q can be greater than unity, i.e. voltages appearing across C and L are larger than the applied voltage, V_S. This represents a voltage magnification equal to Q.

Part 2. Time Domain Response of RLC Circuit to Step Excitation

The theory of the RLC circuit behavior when excited by a step voltage is beyond the scope of this course. However, we present here a brief description required to complete the laboratory assignment.

The application of KVL around the loop of the RLC circuit shown in Figure 1 gives:

$$iR + \frac{1}{C}\int_0^t i\,dt + L\frac{di}{dt} = v_S(t) \tag{10}$$

for zero initial conditions.

When differentiated with respect to time, we get:

$$L\frac{d^2 i}{dt^2} + R\frac{di}{dt} + \frac{1}{C}i = \frac{dv_S}{dt} \qquad (11)$$

This is a second-order linear differential equation. The solution of this equation describes the response of the circuit. Recall that the response of the RC and RL series circuits to a voltage step yielded a first-order linear differential equation, whose solution was of the form:

$$v(t) = K_1 + K_2 e^{-\frac{t}{\tau}} \qquad (12)$$

where K_1 and K_2 are constants. Using intuition, the response of the RLC circuit must be of the form:

$$v(t) = K_0 + K_1 e^{s_1 t} + K_2 e^{s_2 t} \qquad (13)$$

where K_0, K_1 and K_2 are constants, and s_1 and s_2 are called the ***natural frequencies*** of the circuit.

The actual response of the circuit depends on the relation between and the nature of s_1 and s_2. There are three possible cases that must be considered:

Case 1: s_1 and s_2 are both real and unequal.

In this case, the solution is given by:

$$v(t) = K_0 + K_1 e^{-\frac{t}{\tau_1}} + K_2 e^{-\frac{t}{\tau_2}} \qquad (14)$$

This indicates that the response of the RLC circuit is the sum of two decaying exponentials. The circuit is said to be ***overdamped***.

Case 2: s_1 and s_2 are complex conjugates of each other.

In this case, the solution is given by:

$$v(t) = K_0 + K_1 e^{-(\sigma - j\omega)t} + K_2 e^{-(\sigma + j\omega)t} \qquad (15)$$

$$= K_0 + e^{-\sigma t}(K_1 e^{j\omega t} + K_2 e^{-j\omega t}) \qquad (16)$$

The response is an exponentially damped sinusoid. The circuit is said to be ***underdamped***, and its behavior is described as ***ringing***.

Case 3: s_1 and s_2 are both real and equal.

In this case, the solution is given by:

$$v(t) = K_0 + K_3 e^{-\frac{t}{\tau_3}}(A_1 + A_2 t) \qquad (17)$$

The response is similar to Case 1, but it reaches its peak earlier and also decays faster. The circuit is said to be ***critically damped***.

The three possible responses are shown in Figure 2. The RLC circuit is excited by a step input voltage, and the output voltage is measured across the capacitor.

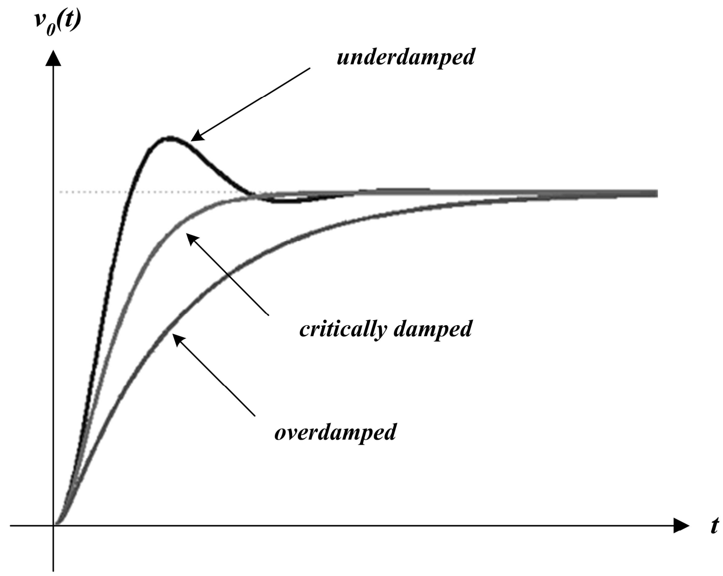

Figure 2. Comparison of overdamped, critically damped and underdamped responses.

PRELAB

1. (_/0.5) Calculate the values of **L** (from Part 1, step 1.3), $\mathbf{R_1}$ (from Part 2, step 2.2), and $\mathbf{R_3}$ (from Part 2, step 2.2) to **five significant digits**. Recall xxx represents the last three digits of your student number.

2. (_/0.5) Calculate the resonant frequency ω_0 for the circuit given in Figure 1 using the component values as given in Part 1, step 1.3. Also convert the resonant frequency into Hertz for f_0.

3. (_/0.5) Calculate the value of the quality factor Q for the circuit given in Figure 1 using the component values as given in Part 1, step 1.3.

4. (_/0.5) Draw a phasor diagram of the circuit in Figure 1 *at resonance* using the component values as given in Part 1, step 1.3.

PROCEDURE

This lab is meant to be an introduction to circuit simulation using PSpice. The PSpice graphical user interface is intuitive, and follows the standard menus that are common to many Windows applications. The lab manual does *not* give you every single step required to complete the exercises. However, you are given enough time to explore, learn and experiment with the different options and settings of the PSpice simulator.

Use the following resources to help you with PSpice since the version available for your use changes from time to time:

- http://www.doe.carleton.ca/~nagui/Web2501/PSpicePrimer.html.
- PSpice help menu and web tutorials.

Part 1. Frequency Domain Response of RLC Circuit to Sinusoidal Excitation

1.1 For your first time on a particular workstation: Open *Cadence → Release 16.5 → Configure.* Click Next, select accept terms, click Next, click Install, click Finish.

Open *Cadence → Release 16.5 → OrCAD Capture CIS Demo*. The first time it is opened select Allegro PCB Librarian XL and click OK.

Create a new project with the following parameters:

- Include your name in the project name (this will enable you to distinguish your printouts from others').
- Choose 'Analog or Mixed A/D' option in the project dialogue box.
- Select the H: drive to store your design
- Choose 'Create a blank project' in the 'Create PSpice Project' dialogue box.

1.2 In the Schematic window that opens up, note the icons located on the right:

- Place part – add a circuit component (C – capacitor, R – resistor, L – inductor, etc.). When choosing this icon for the first time, press 'Add Library' icon (or alt-A) and add the 'ANALOG' and 'SOURCE' libraries. To change the value of a component, double click on the displayed value. The values of the elements can be specified using scaling factors (n – 10^{-9}, u – 10^{-6}, m – 10^{-3}, k – 10^{3}, meg – 10^{6} etc.).
- Add wire – add an interconnecting wire between circuit elements.
- Place net alias – assign names to nets or nodes.
- GND – add a ground to the circuit.

1.3 Draw the circuit shown in Figure 1 using the following values:

R = 40 Ω $L = 80e^{\left(\frac{xxx-800}{800}\right)} mH$ C = 0.01 μF V_S = 1 V

For the AC voltage source use the VAC component from the Source library.

1.4 Place the ground on an appropriate node in your circuit.

1.5 Label each node in your circuit with an appropriate label, e.g. NODE1, NODE2, etc. using the 'Place net alias' icon. This will help you in plotting in the following steps.

1.6 In this part of the lab, we investigate the *frequency* response of the circuit to sinusoidal excitation. Thus we must specify the desired frequency range, which can be done by going to *PSpice / New simulation profile*. Choose logarithmic 'AC Sweep/Noise' for the analysis type.

First, we are interested in looking at a very wide range of frequencies to see the rough shape of the response. Therefore, set the frequency range to 20 Hz – 20 kHz, with **intervals of 20 frequency points** per decade.

1.7 Simulate your circuit (*PSpice / Run*) and make sure there are no errors.

1.8 (_/0.5) Once the circuit is simulated with no errors, the PSpice A/D Demo window will come up. Graph the phase difference between V_C and V_S by going to *Trace / Add trace* and choosing the appropriate variables and functions. Make sure the horizontal axis is set to log scale.

1.9 (_/0.5) Add a second plot to the existing one (*Plot / Add plot to Window*). Graph $|V_C / V_S|$ in decibels on the top plot. Print the resultant plots and include them in your lab report.

1.10 From your plot in step 1.9, you should be able to identify a narrower frequency range of interest (approximately f_0-0.250kHz to f_0+0.250kHz). Set the frequency sweep to this narrower range with about 1500 points per decade. This will allow you to see a more precise shape of the magnitude and phase plots around the resonance point of the circuit.

1.11 Simulate the circuit and obtain the two graphs as in steps 1.8-1.9. Set the horizontal axis range to f_0-0.250kHz to f_0+0.250kHz by double-clicking on the axis, if needed. Do not print your plots at this time.

1.12 (_/0.5) If Q is large, then the values of ω_0 and *3 dB bandwidth* can be obtained from the magnitude plot as shown in Figure 3.

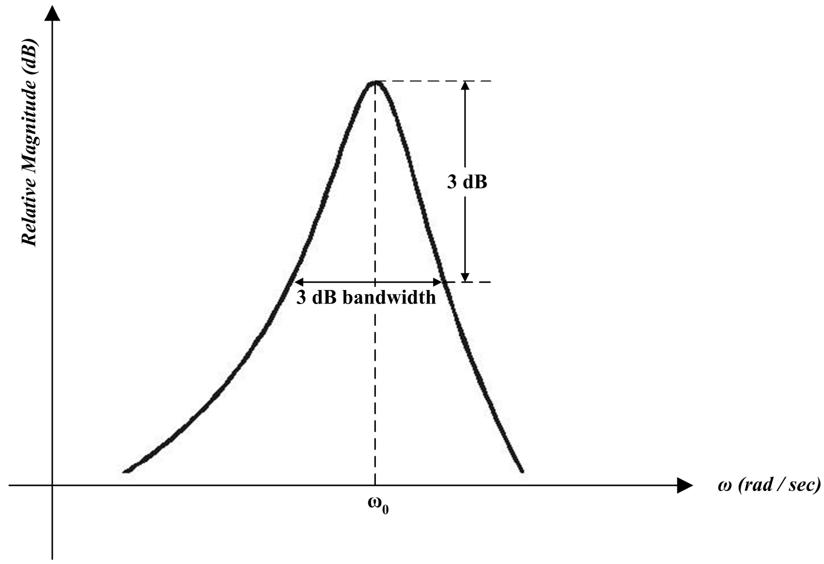

Figure 3. Definition of 3 dB bandwidth.

Explore the options in the graphing utility program (PSpice A/D Demo), and figure out how to use cursors (*Trace / Cursors*) to obtain measurements of the plotted data.

Using cursors, find the value of ω_0, the resonant frequency, and mark the relevant point on the graph. Compare this value to the one calculated in question 2 of the prelab.

1.13 (_/2) Using cursors, find the 3 dB bandwidth, as defined in Figure 3. Mark the relevant points and their values on the graphs. Print the resultant plots and include them with your lab report.

1.14 (_/0.5) **Calculate** Q based on the linear relationship:

$$Q = \frac{\omega_0}{3\ dB\ bandwidth} \tag{10}$$

Compare this value to the one calculated in question 3 of the prelab.

Part 2. Time Domain Response of RLC Circuit to Step Excitation

2.1 Create a new project as in Part 1, and include your name in the project name.

2.2 Figure 4 shows three independent circuits, connected to a common input signal. This configuration will enable you to observe the under damped, critically damped and over damped cases simultaneously. In a new file, draw the circuit shown in Figure 4 using the following values:

$L_1 = L_2 = L_3 = 100$ mH $\qquad R_1 = \dfrac{xxx + 400}{2}\ \Omega \qquad R_3 = R_1 + 4000\ \Omega$

$C_1 = C_2 = C_3$ = 54 nF R_2 = 2721 Ω V_{IN} = 1 V step

For the step voltage source use the VPULSE component from the Source library. Specify appropriate values for the $V1$, $V2$ and TD parameters of VPULSE ($V1$ and $V2$ correspond to the minimum and maximum voltages of the pulse, respectively, and TD corresponds to the delay from time zero to the first rising edge). Label the nodes of the circuit as shown in Figure 4.

2.3 To specify the time range for the simulation, go to *PSpice / New Simulation Profile* and choose 'Time domain (transient)' analysis type. Make sure you specify an appropriate time range to display the outputs clearly. You may need to iterate through steps 2.3 and 2.4 a number of times to obtain the required output plot (in that case, go to *PSpice / Edit simulation profile* to edit the settings).

2.4 Simulate the circuit and, on the same plot, graph the input signal v_{IN}, as well as the three output signals, v_3, v_5 and v_7, as specified in Figure 4.

2.5 (_/1) Identify which one of the circuits is **under** damped, which one is **over** damped, and which one is **critically** damped.

2.6 (_/1) In the case of the under damped circuit:

(a) What is the frequency of the oscillatory response?

(b) Does the frequency approximately agree with the resonance condition in equation (4)?

2.7 (_/1) In the case of the **over** damped and the **critically** damped circuits, measure the *rise-time*, that is, the time each output takes to reach 90% of its maximum voltage. Mark the measured points and their values on the graph. Print the plot and include it with your lab report.

2.8 (_/1) Comment on your results.

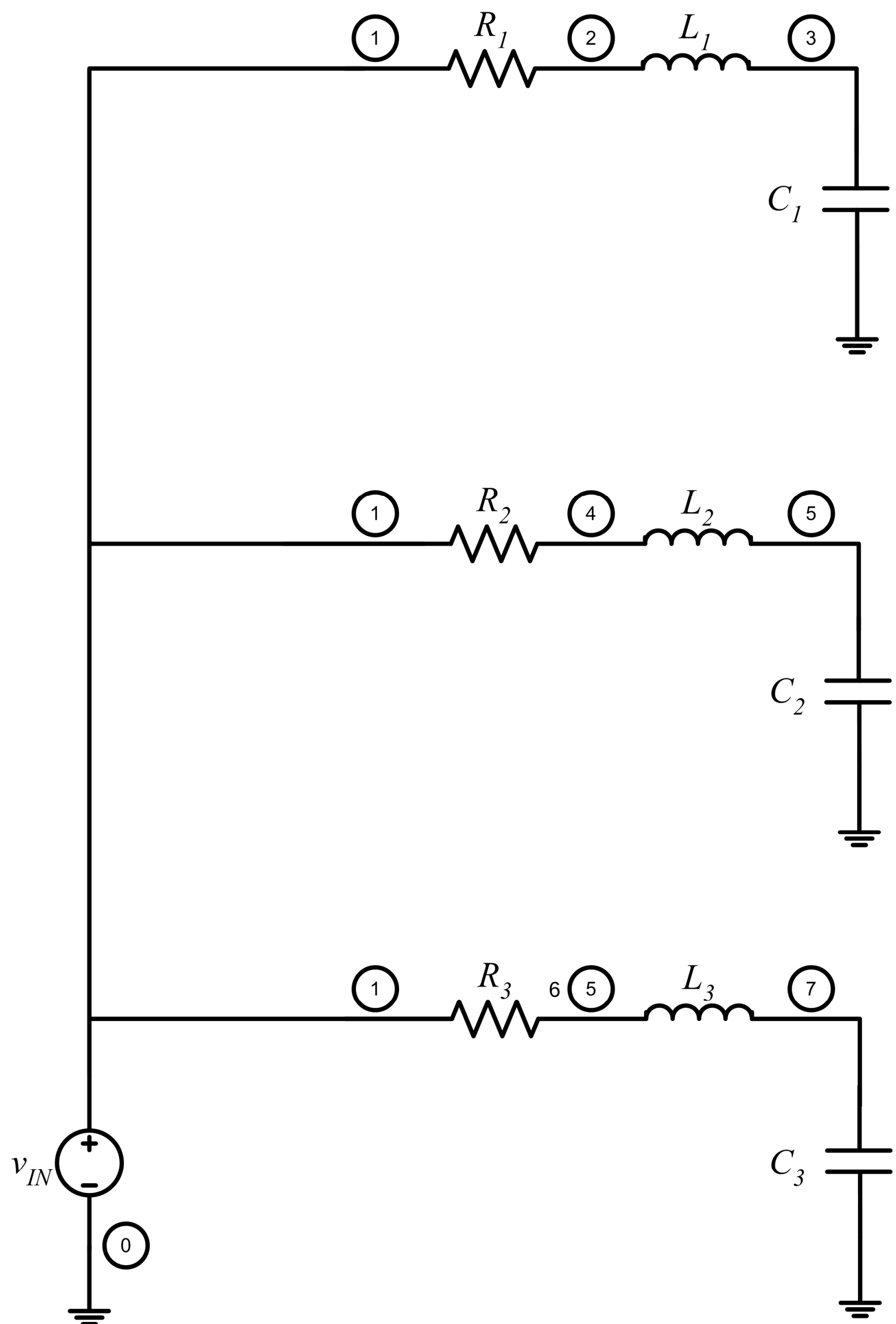

Figure 4. Circuit configuration for Part 2.