Studying the Response of RLC Circuits to Sinusoidal Inputs Using Simulink

Farrokh Mansouri, Raymond Kwong

Abstract
In this experiment you will use Fourier analysis to explore the response of second order systems to sinusoidal inputs, specifically the response of an RLC circuit.

IMPORTANT

1. **First, calculate the *hash value* for your group. Look up you lab group number in Quercus which we denote as N. Then calculate in Matlab:** $round(((mod(N*113,128))+1)/128+1,2)$

   This will be your hash value and henceforth, we will denote by Θ.

2. **To submit your assignment for this lab, you must upload a SINGLE PDF file which includes the answer sheet, as well as the associated screenshots and the answers you have been asked to provide in the instructions.**

Introduction

Signal processing is extremely useful for studying the behaviour of various systems. Fourier analysis, one of the most powerful techniques in signal processing, enables you to explore the oscillating nature of a signal. In this lab, we will use Fourier analysis to explore the response of second order systems to sinusoidal inputs.

In this lab, you will learn to model and analyze the response of an RLC circuit to various input signals. An RLC circuit consists of a resistor, an inductor and a capacitor. Its dynamics can be described by a second-order ordinary differential equation (ODE), and is characterized by its natural frequency and damping ratio. ODEs are easily modelled using Simulink. You will develop a Simulink model for an RLC circuit and investigate its properties through its responses to various waveforms.

You have already studied the properties of resistors, capacitors and inductors in your other courses. The behaviour of resistors, capacitors and inductors can be modelled using the following equations.

$$\begin{aligned} Resistor\ Equation:\ V_R &= Ri \\ Capacitor\ Equation:\ V_C &= \frac{1}{C}\int idt \\ Inductor\ Equation:\ V_L &= L\frac{di}{dt} \end{aligned}$$

If we connect the components in series, we obtain the following RLC circuit. Note that same current passes through each of these components.

Kirchhoff's Voltage Law (KVL) states that the sum of voltages around a loop is equal to zero. Therefore, we can sum over all voltages across all the components in this circuit and write the following equation.

$$L\frac{di}{dt}+Ri+\frac{1}{C}\int idt=0$$

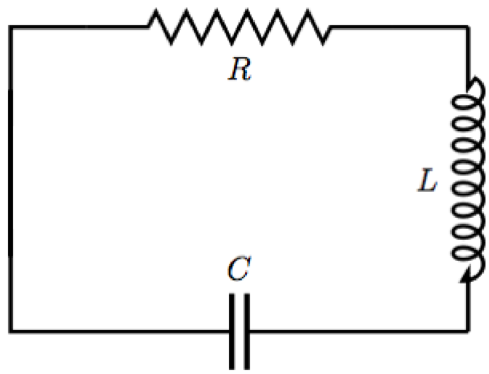

Figure 1. Schematics of a series RLC circuit

This equation can be solved using Simulink. It is generally recommended not to use derivatives in Simulink and use integrators instead. Therefore, we will rearrange the equations to replace differentiation with integration. The following Simulink model demonstrates how to use Simulink to provide a numerical solution of a differential equation.

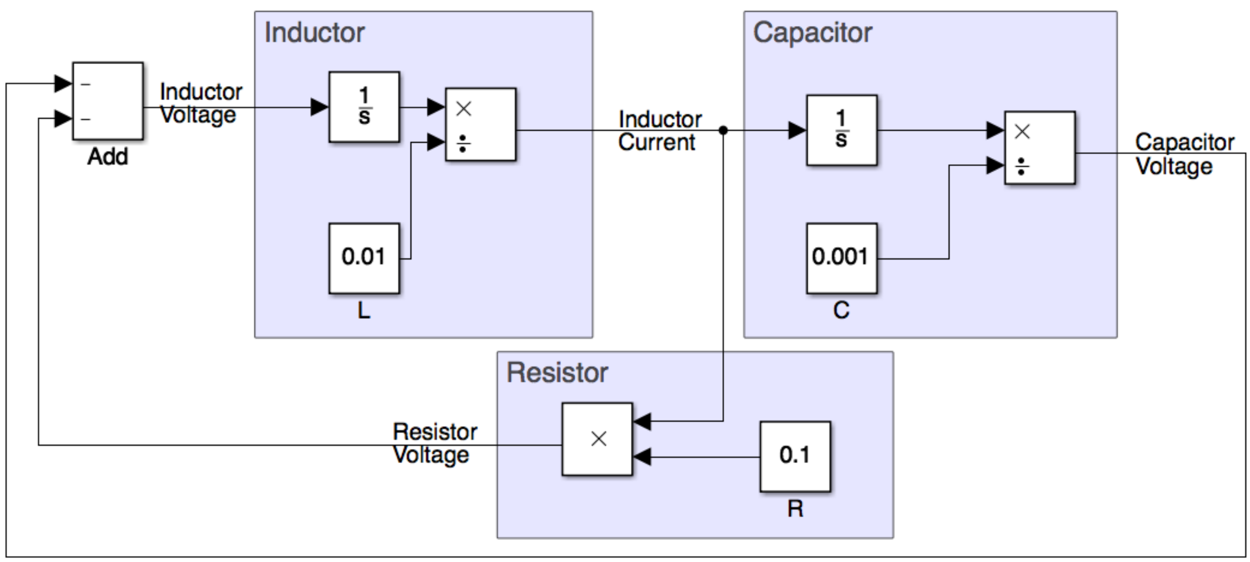

Figure 2. Simulink model of RLC circuit

The second order nature of the RLC circuit can be readily seen from the fact that since $i = C\frac{V_C}{dt}$, V_C satisfies the differential equation

$$LC\frac{d^2V_C}{dt^2} + RC\frac{dV_C}{dt} + V_C = 0$$

Second order differential equations with positive coefficients can be rewritten in the form

$$\frac{d^2y}{dt^2} + 2\zeta\omega_n\frac{dy}{dt} + \omega_n^2 y = 0$$

where ω_n is called the natural frequency, and ζ is called the damping ratio.

1. Natural frequency of oscillation of the RLC circuit

In this part, we will investigate the natural frequency of the RLC circuit. To do that, you will need to first create the model as presented in the figure below. Of course, the labels are not a part of the model and you do not have to use them, but make sure you understand which blocks are controlling the parameters of each component in the model. Note that the Simulink model consists of the RLC circuit previously described, with the capacitor voltage connected to some output blocks to view the response.

Once you are done making the model, there are several parameters that you can adjust to observe behaviour of the system. You can adjust the inductance, resistance and capacitance in the model by changing the constant block values. Further, each integrator has an initial condition value; that is, the value of the output of the integral at time zero. You can adjust this value by double clicking on the integral block. The default value for the integrator block is zero and you should change it to see how the initial value affects the behaviour of the model.

Another component in the model that you will use for your analysis is spectrum analyzer module (from DSP toolbox), which applies Fourier analysis and displays the spectrum of the signal; that is, the amplitude vs. frequency plot. This block should be used with a sampling device, which in Simulink is the zero-order hold, because the integrator blocks work in continuous time domain, while Fast Fourier Transform (FFT) used in the spectrum analyzer can only be applied to a discrete signal. The zero-order hold block converts continuous time to discrete time. You can set the sample time by double clicking on the zero-order hold block.

Note that the scope displays the plot of the signal in time domain, while the spectrum analyzer displays the spectrum of the signal, in frequency domain.

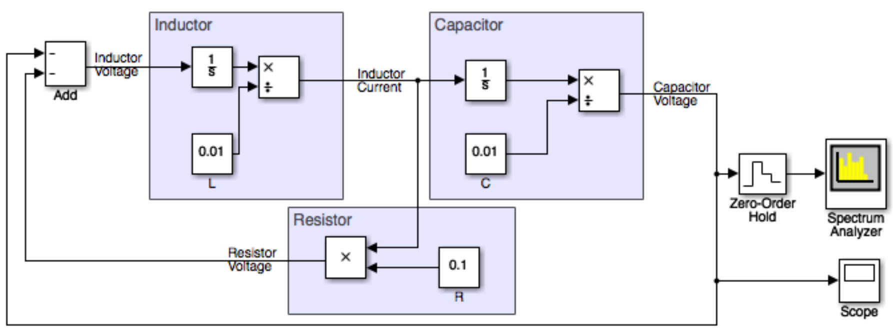

Figure 3. Simulink model of RLC circuit with scope and spectrum analyzer

Now, your job is to explore the behavior of this system. You should be able to determine the natural frequency of the system and show how the inductance and capacitance of the system affect its oscillations. Keep in mind that the resistance value contributes also to how fast the response decays and to how much it is damped.

Use these parameters to set up your model initially:

Inductor integrator initial condition (inductor current)	0.1
Inductor constant (Inductance)	0.01
Capacitor integrator initial condition (capacitor voltage)	0.1
Capacitor constant (capacitance)	0.01
Resistor Constant (Resistance)	0.1
Zero-order hold sample time	2^{-10}
Simulation time	2 sec

Exercise 1.1:

Plot the capacitor voltage over time and show that the system starts its oscillations at the natural frequency. The natural frequency, corresponding to the frequency where the peak amplitude occurs, can be read from the spectrum plot. Your plot should look like the example below. Make sure to use the “Peak finder” tool to display the frequencies.

Do the above experiment for the following pairs of values for C and L and observe how the natural frequency changes (recall that Θ is the hash value calculated for your group):

(i) $C = 0.01 \times \Theta,\ L = 0.01 \times \Theta$

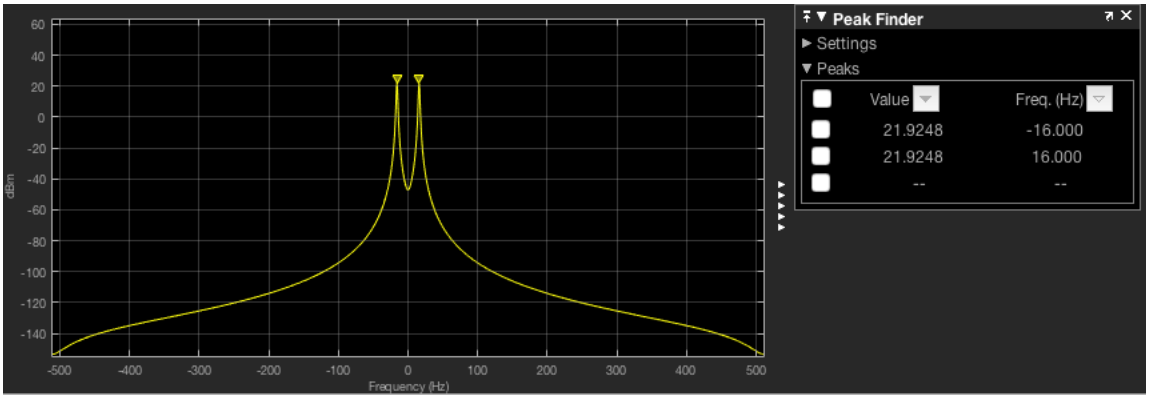

Figure 4. Spectrum of the output with the peaks identified using peak finder

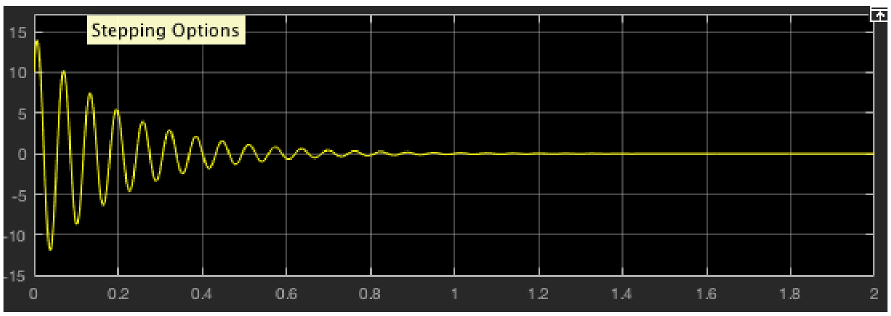

Figure 5. Capacitor voltage over time in series RLC circuit

(ii) $C = 0.02 \times \Theta$, $L = 0.01 \times \Theta$

(iii) $C = 0.04 \times \Theta$, $L = 0.01 \times \Theta$

(iv) $C = 0.01 \times \Theta$, $L = 0.02 \times \Theta$

(v) $C = 0.01 \times \Theta$, $L = 0.04 \times \Theta$

Calculate the natural frequency analytically and record the values on the answer sheet. Compare them with the experimental results. Obtain a screenshot of the spectrum plot to show your results and include with your submission. Note that if you set the capacitance C to be a variable, you can dynamically change its value in your Matlab command window and the new value will be reflected in Simulink.

Exercise 1.2:

Since there is no input to this model and there is a resistor that will damp the oscillation, you should expect that the voltages and the current drop to zero over time. Keep the values of $C = L = 0.01 \times \Theta$. Demonstrate how the value of the resistor will affect the damping and the rate at which oscillation decays as the value of R goes through $0.01, 0.1, 0.2, 0.5, 1, 1.5, 2, 2.5$. Find out the value of R for which the response is critically damped, i.e., it goes from underdamped (oscillating) to overdamped (never crosses the x-axis). Obtain a screenshot of the voltage over time and include it with your submission. Do the same for the case when the values of C and L are changed to $C = L = 0.02 \times \Theta$.

2. RLC circuit response to an external voltage source

Now that we know how to model RLC circuits, we will apply external voltage input to the model and will examine how this external input drives the RLC circuit, by examining its output. As in part 1 we will connect resistor, capacitor and inductor in series. Now we add a voltage source in series to the other components. The following is the schematic of the circuit that we are going to model.

The equation for the RLC circuit derived in part 1 can be used for this step. We just need to add the presence of the voltage source. This is how you can use the schematic form part 1 to model the new circuit.

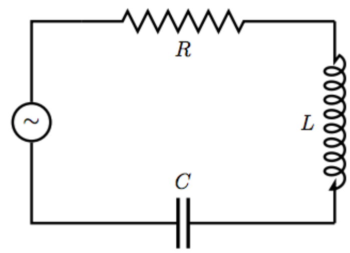

Figure 6. Schematics of series RLC circuit with voltage source

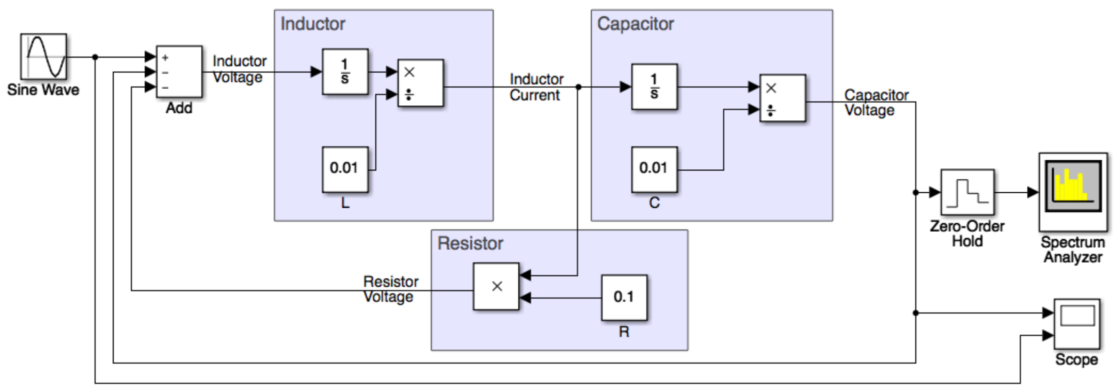

Figure 7. Simulink model of RLC with voltage source

Now you can explore the effects of various voltage sources on the behaviour of your RLC system. We start by applying alternating sine wave voltage source. You can adjust the frequency of your source to see what the response looks like. Try applying sine wave with frequencies close to natural frequency of the system and then try to apply sine waves with frequencies larger or lower than the natural frequency of the system. Make sure to keep the sample time at zero. That will insure the simulation is in continuous-time mode.

Use these parameters to set up your model:

Inductor integrator initial condition (inductor current)	0
Inductor constant (Inductance)	$0.01 \times \Theta$
Capacitor integrator initial condition (capacitor voltage)	0
Capacitor constant (capacitance)	$0.01 \times \Theta$
Resistor Constant (Resistance)	0.1
Zero-order hold sample time	2^{-10}
Simulation time	2 sec

Exercise 2.1:

For this part, make sure the initial values for the integrators are set to zero. Drive the system using a sine wave input having exactly the natural frequency of the system and record the amplitude of the system output. Repeat using a sine wave input with the following frequencies:

(i) Natural frequency/5

(ii) Natural frequency/2

(iii) Natural frequency

(iv) Natural frequency*2

(v) Natural frequency*5

Record the amplitude of the output in steady-state (when the transient has died out) as well as the maximum and minimum output values in each of the cases above, i.e., when the input sine wave has exactly the natural frequency and has the corresponding frequencies above, respectively. Obtain a screenshot of the output signal over time and of its spectrum in each case and include with your submission. Also include an explanation of how the amplitude of the output changes as you change the frequency of the sine wave input.

Exercise 2.2:

Now instead of the sine wave, use a different periodic signal, the square wave. To generate square waves, you can use "Repeating Sequence Stair" module. You should set the output values to 1 and -1; that will produce a square wave between 1 and -1. Also, you can adjust the frequency of the square wave by changing the sample time. Set the period to be $\frac{1}{32}$ seconds. Your sample time should equal to period/2. This results in a square wave where it takes the value 1 in half of the period and -1 in the other half. Create a new model where you directly connect square wave voltage source to a spectrum analyzer. Make sure to set the output values equal to [1 -1] and the sample time to Period/2. Obtain a screenshot of the square wave signal (its time-domain plot and its frequency-domain spectrum) and include with your submission. Include also an explanation of why you see so many peaks in the spectrum analyzer plot.

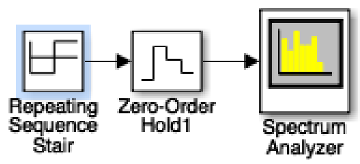

Figure 8. Simulink model to analyze the spectrum of a square wave

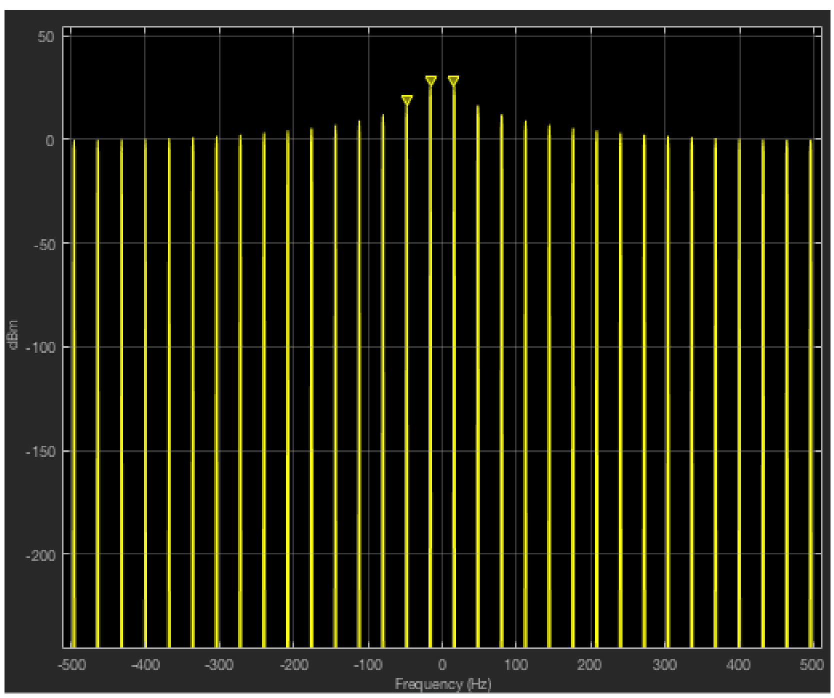

Figure 9. Frequency spectrum of a square wave

3. Applying Fourier Series in circuit analysis

It can be shown that a square wave with period T can be expanded in a Fourier series given by

$$Square(t) = \frac{4}{\pi} \sum_{n=1,3,5,\ldots}^{\infty} \frac{1}{n} \sin \frac{2\pi nt}{T}$$

We will now explore using the Fourier series to approximate the square wave as well as compare the output response.

Exercise 3.1:

With the square wave connected to the spectrum analyzer, record the frequency of the first 4 peaks on the amplitude spectrum plot and compare them to the values determined from the first 4 terms of the Fourier series expansion. Display the square wave and the Fourier series approximation using 4 terms (time-domain plot) and the frequency-domain spectrum of the Fourier series approximation. Include a screenshot showing how closely the Fourier series approximation matches the square wave.

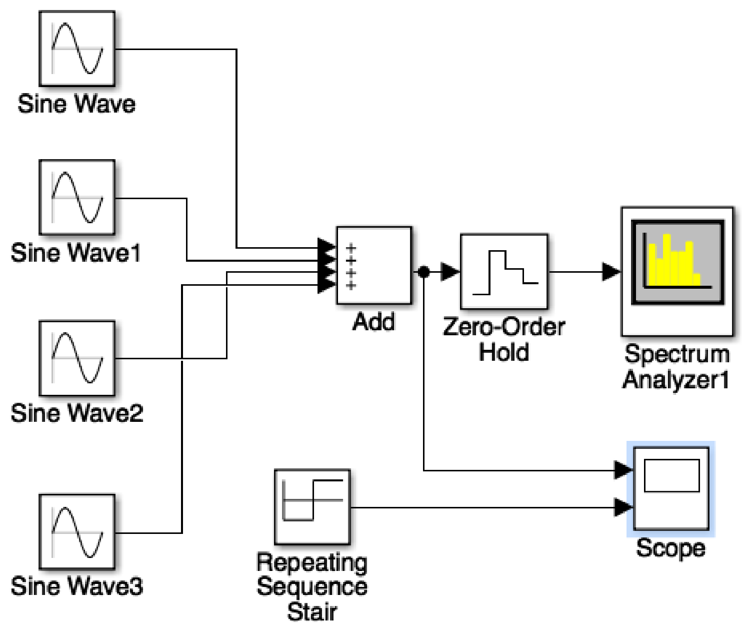

Figure 10. Simulink model to approximate a square wave using sine waves

Exercise 3.2:

Now connect the square wave as the voltage source (input) to the RLC circuit. Set the period of the square wave to be $\frac{1}{32}$ seconds and examine the RLC circuit output response for the square-wave input. Take now the first 4 terms in the Fourier series, add them, and use the sum as the voltage source input, replacing the square-wave input. Show a screenshot of the output response of the RLC ircuit for the square-wave input and for the 4-term Fourier series approximation input, respectively.

Exercise 3.3:

Repeat Exercises 3.1 and 3.2 with the first 8 terms of the Fourier series expansion. Show a screenshot of the approximation of the square wave. Examine also the improvement in approximating the output response of the RLC circuit. Answer the following question based on your screenshots: Are there any differences in the degree of improvement in the two cases?