Lab 4 - RLC Circuit

Overview

In this lab we will be modeling RLC circuits in Simulink using the Simscape Electrical toolbox and observing their behavior.

Resources Required:
MATLAB
Simulink
Simscape
Simscape Electrical

Description of the System

The below RLC circuits will be the circuits that will be modeled in this lab. Each of the systems are AC circuits with one voltage source. Each of the circuits will share constant parameters whose values are detailed in table1.

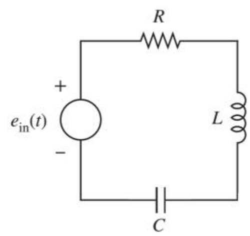

Figure 1: RLC Circuit 1

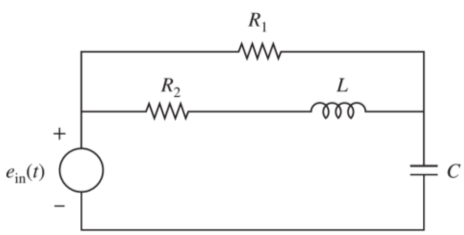

Figure 2: RLC Circuit 2

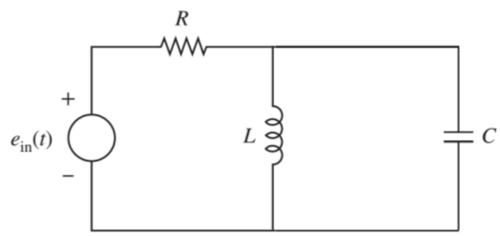

Figure 3: RLC Circuit 3

Table 1: System Parameters of RLC Circuit

Property	Value	Units
AC Voltage Peak Amplitude	10	V
AC Voltage Frequency	1	kHz
Resistor (R, R_1, R_2)	1	kOhm
Inductor (L)	100	mH
Capacitor (C)	100	nF

Task 1: RLC Circuit 1

For this task, we will create the first RLC circuit. The block diagram for circuits is created as you would expect, in the same manner in which the circuit is drawn. For our model, we also will add a current sensor to view the circuit's current as well as voltage sensors across each of the components to view the voltages across them.

NOTE: Be sure to save your models for each circuit so that you can go back and run the simulation. You will need to be able to do so to answer the post lab questions correctly.

1. Start by creating the block diagram in Simulink for circuit 1 as you see it in figure 4.

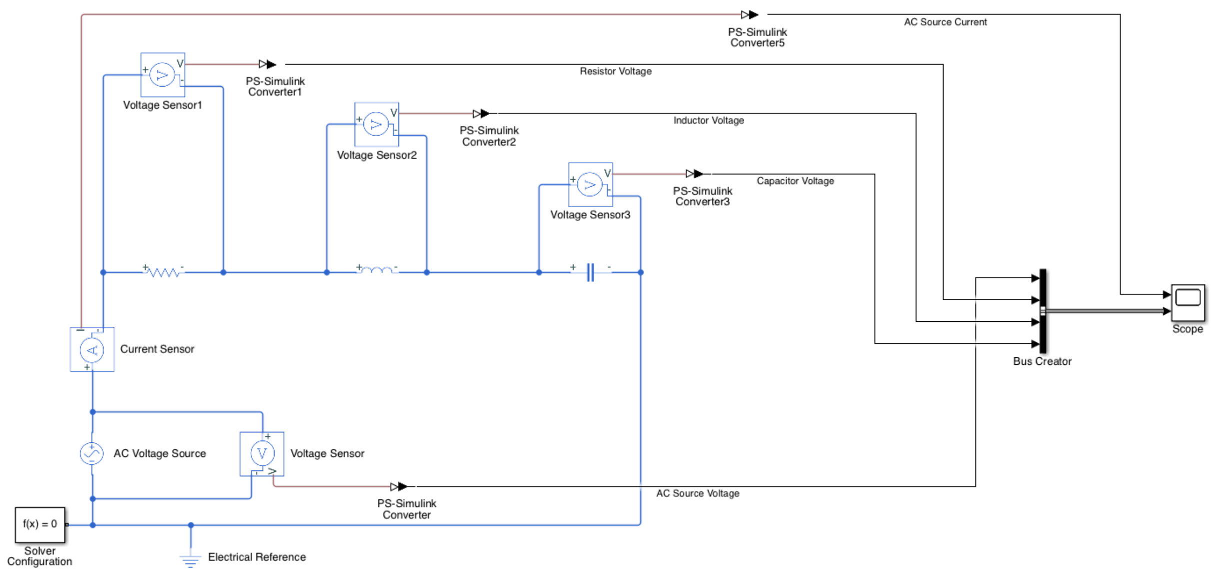

Figure 4: RLC Circuit 1 Block Diagram

NOTE: Be careful when selecting the components as there are two different types. Select the components from the "Foundation Library/Electrical/Electrical Source" library as shown in figure 5.

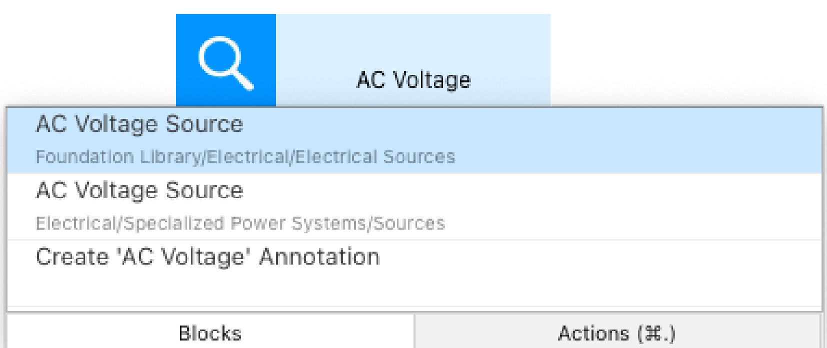

Figure 5: Correct Block Library Location

2. After placing all of the blocks in place, define the parameters for the system as described from table 1.
   To recap:
   a. AC Voltage Source “Peak amplitude” = 10 V
   b. AC Voltage Source “Frequency” = 1kHz
   c. Resistor (R, R_1, R_2) “Resistance” = 1 kOhm
   d. Inductor (L) “Inductance” = 100 mH
   e. Capacitor (C) “Capacitance” = 100 nF
3. It will now be important to specify the stop time for the simulation. We need the simulation to last only a few periods so that we can see the plot correctly going from its transient state to steady state. If the simulation time is too long, the plot that is generated will not look correct. Go into the “Configuration Parameters” and change the stop time to 3E-3 (seconds) and the change the solver to “ode23t (mod.stiff/Trapezoidal)” Figure 6 displays these changes.

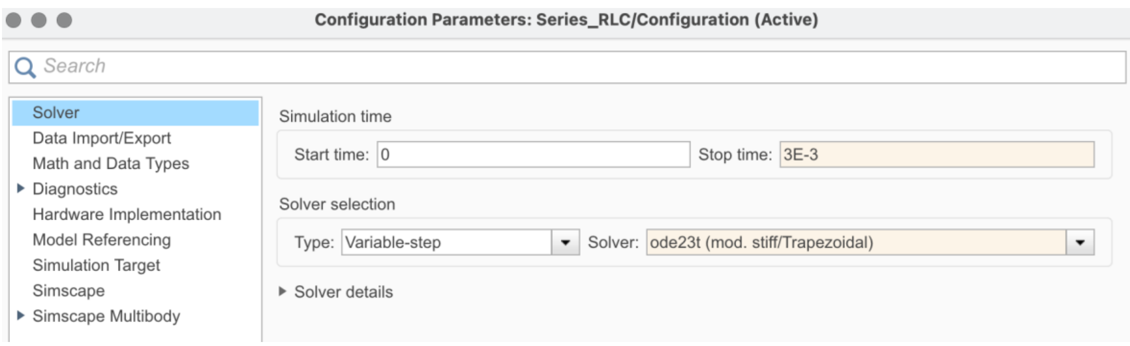

Figure 6: Simulation Configuration Parameters

4. The Scope block is capable of receiving multiple inputs and then display them in a specified layout. To increase the number of ports on the Scope to two as shown in figure 4, open the Scope block and hover over “View”, then Configuration Properties and change the “Number of Input Ports” to 2. This is shown in figure 7.

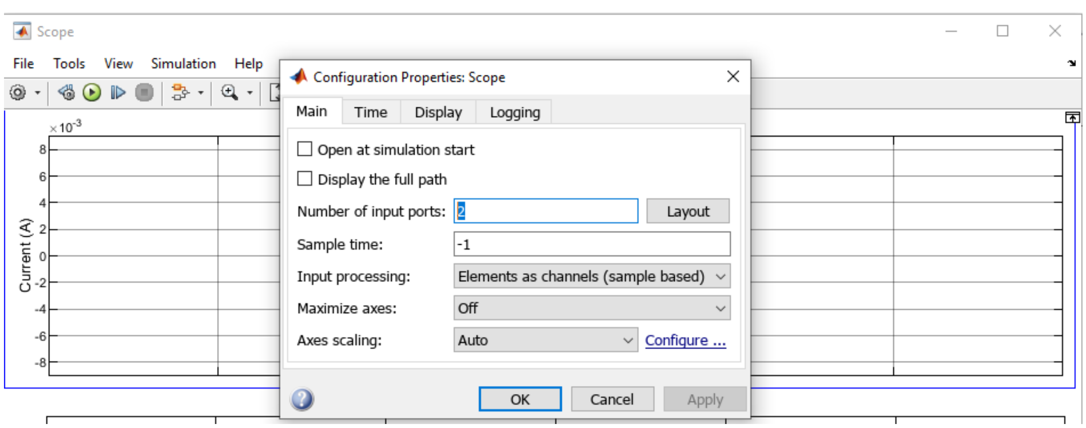

Figure 7: Changing Number of Ports on Scope

5. Within the scope, click on "View" and then "Layout…" a new window will appear. Use your mouse to select the layout option as it does in figure 8. This will take the two incoming ports and display them in a 2x1 view.

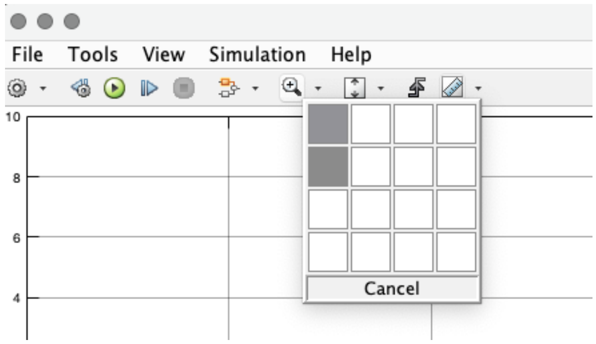

Figure 8: Changing the Scope's Layout View

6. Run the simulation, and view the scope. You should see that the AC Source Current is on the top plot while all of the voltage reading throughout the circuit is viewed on the bottom plot.
7. To individually format the plots, go into the "Configuration Properties" and under the "Display" tab, select the display you want to format. For instance, in figure 9, the 2nd display (the Voltage Vs. Time plot) is selected. Once selected, you can change some of the formatting for the selected plot.
8. Format the plots appropriately to include a title, labels, and legend for each plot.

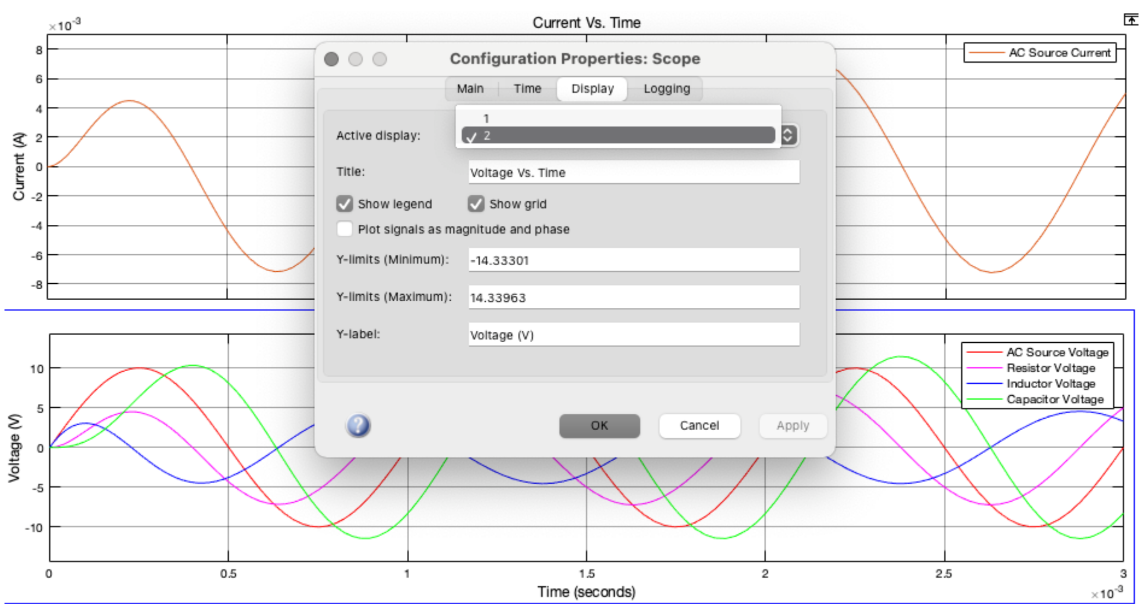

Figure 9: Selecting Individual Plots for Formatting

9. Scale the axes for each of the plots by selecting the plot from the "Display" tab, then after selecting the individual plot, go to the "Main" tab and select "Auto" under "Axes Scaling". This should give a better fit of the plot within its scale range.

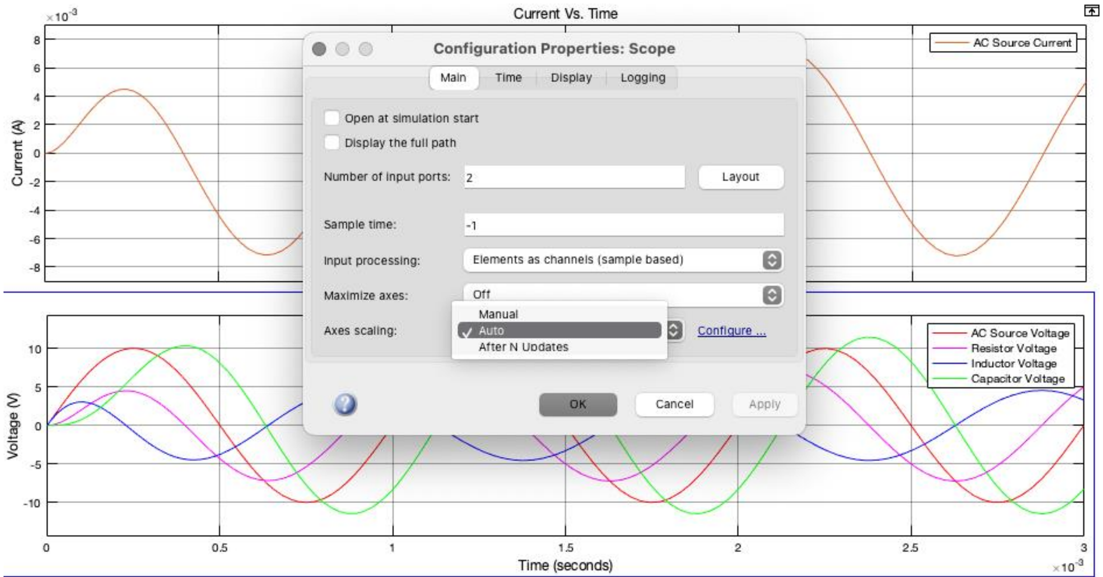

Figure 10: Scaling the Plots

10. You also will need to select the individual displays when changing the color layout under the "Style" tab. Once you have completed formatting the plots. You should have a plot similar to the one shown in figure 11.
11. Format and save the final block diagram and your plot appropriately and include them in your report.

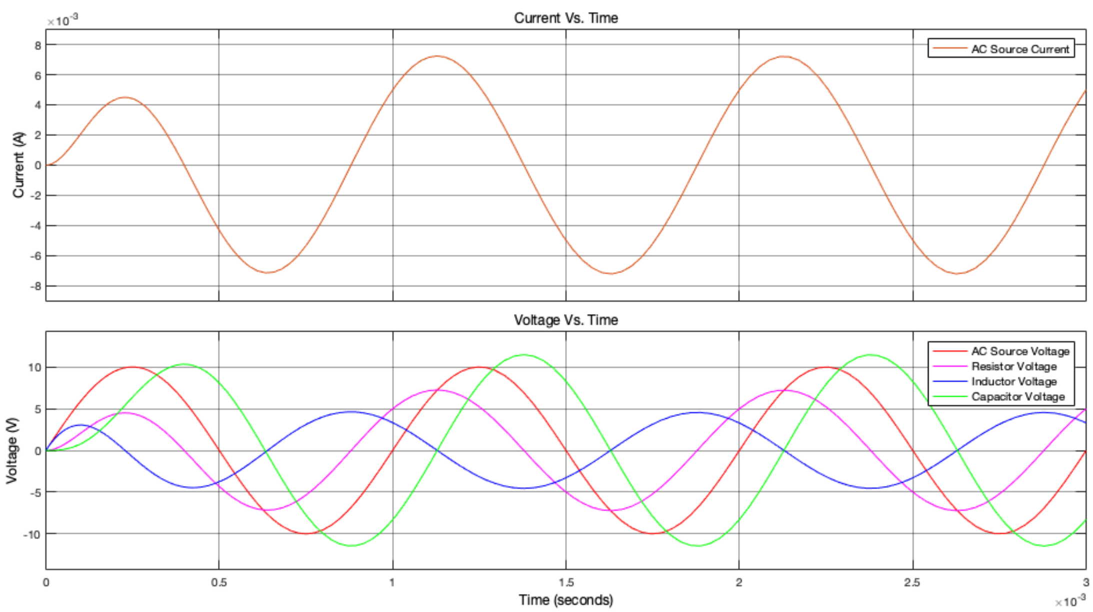

Figure 11: Complete Plot for RLC Circuit 1

To show completion of this task, save your formatted scope plot and show the TA.

Task 2: RLC Circuit 2

For task 2, we will create and simulate the second RLC circuit. Follow the same procedure as described in Task 1 to create the block diagram.

1. Be sure to use the parameter values as specified in the Description of the System section. To recap:
    a. AC Voltage Source "Peak amplitude" = 10 V
    b. AC Voltage Source "Frequency" = 1kHz
    c. Resistor (R, R_1, R_2) "Resistance" = 1 kOhm
    d. Inductor (L) "Inductance" = 100 mH
    e. Capacitor (C) "Capacitance" = 100 nF
2. Also be sure to change the solver to "ode23t (mod.stiff/Trapezoidal)" and the stop time to 3E-3 (seconds) as described in Task 1 step 3.

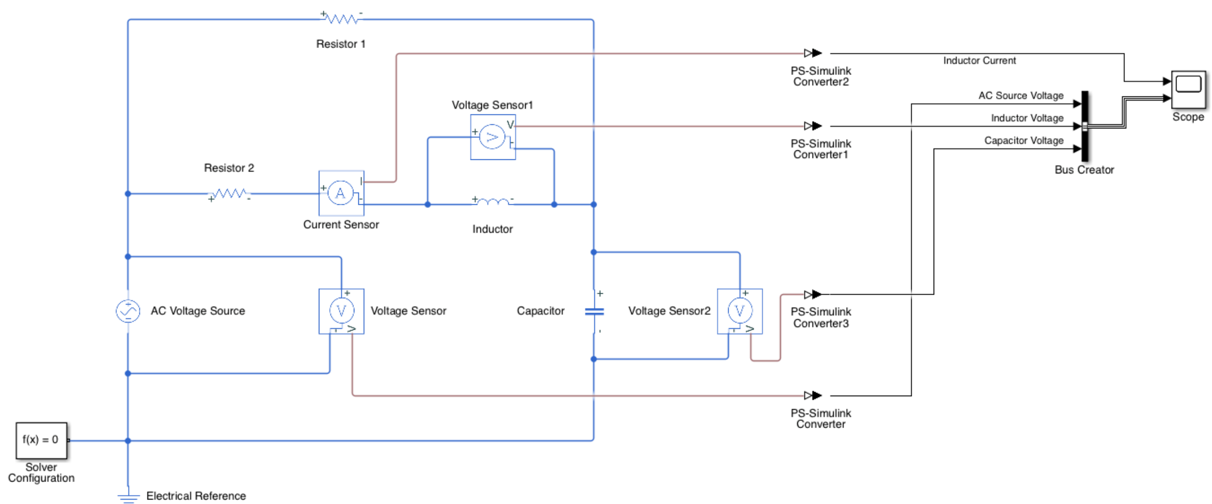

Figure 12: RLC Circuit 2 Block Diagram

3. Once you have completed the block diagram, run the simulation and view the scope.
4. Be sure to format the scope plots appropriately as described in Task 1.
5. Format and save the final block diagram and your scope's plot appropriately and include them in your report.

To show completion of this task, save your formatted scope plot and show the TA.

Task 3: RLC Circuit 3

For task 3, create and simulate the third RLC circuit as shown in the Description of the System section. Follow the same procedure as you did in tasks 1 & 2 to create the model. You should now be able to create and simulate the model by simply looking at the circuit diagram and knowing the parameter values.

Some things to keep in mind while creating the model:

1. Include the following sensors
    a. Voltage Sensor across the AC Voltage Source
    b. Voltage Sensor across the Inductor
    c. Voltage Sensor across the Capacitor
    d. Current Sensor down the branch of the Capacitor
2. Be sure to use the parameter values as specified in the Description of the System section. These will be the same values used in tasks 1 and 2.
3. Be sure to change the solver to and the stop time as done previously.
4. Be sure to name/ label the lines coming from the “PS-Simulink Converter” blocks to the bus creator and scope ports appropriately as you did in the previous steps. The scope will use these line labels to create the legend for your plots.
5. Once you have completed the block diagram, run the simulation and view the scope.
6. Be sure to format the scope plots appropriately as did in the previous tasks.
7. Format and save the final block diagram and your scope’s plot appropriately and include them in your report.

To show completion of this task, save your formatted scope plot and show the TA.

NOTE: You should now have a total of 3 block diagrams and 3 plot sets (6 individual plots) to include in your report. Be sure they look appropriate.

Post-Lab Questions

To answer the questions below pertaining to the signal's values. You will need to view the models' scope output and view the "curser measurements" and or the "signal statistics" for the particular trace. While in the scope, you can open the two tools by clicking on "Tools" then "Measurements". You can also access them by clicking on the small ruler icon. It is the far-right icon at the top menu bar within the scope. Play around with these tools to get a sense on how they work.

From task 1:

1. What is the peak-to-peak voltage across the resistor to 3 decimal places?
2. What is the relationship (leading or lagging) between the current and the capacitor's voltage? Is this behavior expected?

From task 2:

3. What is the max voltage across the inductor to 3 decimal places?
4. What is the relationship (leading or lagging) between the current and the inductor's voltage? Is this behavior expected?

From task 3:

5. What is the RMS voltage across the capacitor to 3 decimal places?
6. How does the plot for the inductor's voltage and capacitor's voltage differ? Why?