Training Unit 3: RLC simulation

1 INTRODUCTION TO TRAINING UNIT

After successfully completing Training Unit 2, which focused on data analysis, manipulation, storage types, and visualization, this unit will shift towards modeling an electronic circuit. For this, in addition to MATLAB, you will need Simulink, LTSpice, and Simscape.
If any issues arise during the training, be sure to consult the Matlab documentation online or use the “F1” key in MATLAB, where you'll find numerous examples and copy-paste code. The same applies to any challenges with the Simulink add-on "Simscape."
By the end of this unit, you will be able to model differential equations using Simulink and verify the simulation results through different modeling approaches and tools.

2 PREPARATION

- Access to Matlab / Simulink / Simscape / LTSpice
- Mathematical background of differential equations

3 TECHNICAL BACKGROUND – RLC CIRCUIT [1]

In this training unit, you will model a series RLC circuit. This circuit functions as a harmonic oscillator for the current (iii) and resonates similarly to an LC circuit. The addition of a resistor increases the decay of these oscillations—known as damping—and reduces the peak resonant frequency. Some resistance is inevitable, even if a resistor is not explicitly included (e.g., the series resistance of the inductor).
RLC circuits have numerous applications, especially in oscillator circuits. They are commonly used in radio receivers and television sets for tuning, allowing them to select a narrow frequency range from surrounding radio waves. In this context, the circuit is often called a tuned circuit. An RLC circuit can serve as a band-pass filter, band-stop filter, low-pass filter, or high-pass filter. Tuning applications are an example of band-pass filtering.
The RLC filter is a second-order circuit, meaning that any voltage or current in the circuit can be described by a second-order differential equation in circuit analysis.
Resonance frequency is defined by the impedance presented to the driving source. Even after the driving source is removed or subjected to a voltage step (including stepping down to zero), the circuit may continue oscillating for a period of time.

$$Natural\ frequency = \omega_0 = \frac{1}{\sqrt{LC}}$$

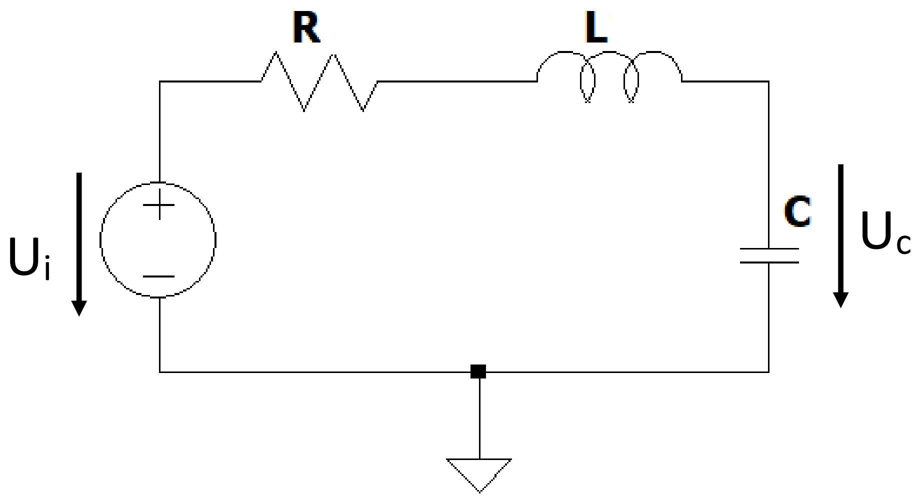

Figure 1: RLC series circuit

The RLC series circuit has following values for the single component parameters:

parameter	value
Ui (t < 2 s)	0 V
Ui (t >= 2 s)	3 V
R	56 kΩ
L	100 µH
C	10 µF

4 LABORATORY WORK

4.1 Derive differential equation for the RLC circuit

4.1.1 Subtask description

- To be able to model the RLC circuit presented in chapter 3 (see Figure 1) in Simulink, you need to derive the ordinary differential equation first, that describes this circuit.
    1. Write down the characteristic component equations for every component in the given circuit diagram
    2. The input for this circuit is the voltage source Ui, the outputs of this circuit is the voltage drop over the capacitor C
    3. Use Kirchhoff's current and voltage law to create the system equation that describes the relationship between Ui and Uc as differential equation

4.1.2 Presenting your results

- After finishing this subtask, you shall have the system equation derived for the RLC series circuit
- Show in your report how you derived the system ODE (a photo of the handwritten process is sufficient as long as the writing is readable (photo quality plus handwriting itself), or use your Matlab script in combination with a Matlab Equation or Latex Equation (see "Insert" tab))
- Characterise the derived ODE according to the ODE classification possibilities that were discussed in LU3 and add it to your report

4.2 Model the differential equation for the RLC circuit using Simulink

4.2.1 Subtask description

- After successfully deriving the system equation, you can model the differential equation using Simulink. For this task, consider following steps:
    1. Start modelling with inserting the "Integrator" block (remember: the order of the ODE is equivalent with the amount of integrator blocks necessary)
    2. Name the signal lines according to the variable it represents (e.g. *R*C*, or *dUc/dt*, U_dot, etc.)
    3. Use the model properties of the Simulink model by *right click, model properties, Callbacks, PreloadFCN* to implement the variable declaration and initialisation for all parameters (R, L, C, Ui) or use a data dictionary to add the variables as constants to your Simulink worspace
        - With this option you can store the parameters within the Simulink model file and as soon as the model is opened, the commands written herein are called and the parameters will appear in the workspace.
        - Now you are able to use the parameters throughout the model

4.2.2 Presenting your results

- After finishing this subtask, you modelled the characteristic equation in Simulink. The input voltage jump Ui(t) results in a step response from RLC circuit model
- Make sure, that your model is aligned properly (this is necessary for following subtasks as well)
    1. Avoid signal line crossings as much as possible for readability reason
    2. Make sure the signal lines are named correctly
    3. Follow basic signal flow rules (signal flow from left (input) to right (output))
    4. Use "Annotation" or "Area" blocks to help express your thoughts directly in the model
    5. Use scopes to plot your modelling results
        - Use the scope properties to enable **legends** in your diagram
        - Use the scope properties to activate **axis description** and make sure to implement the correct **units** for the scope axis
    6. As you will need to upload your model to Moodle, make sure it is readable and understandable

4.3 Model RLC circuit using LTSpice

4.3.1 Subtask description

- After successfully modelling the differential equation in Simulink, use LTSpice to check Simulink's result. For this task, consider following steps:
    1. Redraw the circuit diagram given in chapter 3 figure 1 using the circuit simulation program "LTSpice"
    2. Simulate the RLC circuit and examine the results optically. Consider using measurements options in LTSpice to get some discrete data points and compare them with Simulink data points. Do the results match?

4.3.2 Presenting your results

- After finishing this subtask, you modelled the RLC circuit using LTSpice and reviewed the simulation results
- Upload your *.asc file with your other simulation results

4.4 Compare your LTSpice results with Simulink simulation results within your Simulink model

4.4.1 Subtask description

- After successfully modelling the differential equation in Simulink and LTSpice, you need to import the LTSpice data to be able to compare them with the Simulink results
    1. Export your LTSpice simulation results into a *.txt file
    2. Import the data in the Matlab workspace, using a separate Matlab function for this task (similar to training unit 2)
    3. Once imported to your workspace, you need to import the data into Simulink, to be able to compare the LTSpice data within a Simulink scope
    4. Add a scope, that plots the Simulink results in combination with the LTSpice results

4.4.2 Presenting your results

- After finishing this subtask, you imported the LTSpice simulation results using a Matlab function and plotted both results using a Simulink scope.
- Hint: Make sure, that both programs use the same simulation stop time, otherwise Simulink will not be able to compare the results properly, especially if LTSpice data is shorter than Simulink data
- Make sure you name the signals properly, to improve readability of the model and the scope results

4.5 Derive the transfer function of the RLC circuit

4.5.1 Subtask description

- After successfully comparing the simulation results, we add a third possible modelling option by using a transfer function for the circuit. Before a transfer function can be implemented, you need to derive the transfer function equation for the RLC circuit first:
    1. Start by setting-up the transfer function equation by using the simple voltage divider rule
    2. After restructuring the complex fractional expression, use Laplace transform to receive the transfer function for this circuit

4.5.2 Presenting your results

- After finishing this subtask, you successfully derived the transfer function for this circuit
- Be sure to include the process of your transfer function derivation in your folder for the final upload (a photo of your handwritten work is sufficient, as long as it is readable (photo quality and handwriting))

4.6 Model the transfer function with Simulink

4.6.1 Subtask description

- After successfully deriving the transfer function of the RLC circuit, it is time to model it with Simulink:
    1. Use the "Transfer function" block for this subtask
    2. Compare the simulation results with your previous work

4.6.2 Presenting your results

- After finishing this subtask, you successfully implemented the transfer function and compared it with your previous work
- Be sure to name the signals properly for better understanding

4.7 Model the circuit diagram using Simscape

4.7.1 Subtask description

- Another possible way to simulation the RLC circuit is the use of Simulinks Add-on Simscape
    1. Redraw the circuit diagram using Simscape
    2. Make sure to add a solver configuration block to the circuit diagram
    3. Use the parameters stored in the workspace as values for the components in Simscape

4.7.2 Presenting your results

- After finishing this subtask, you successfully simulated the circuit in Simscape and compared it with your previous work
- Be sure to name the signals properly for better understanding
- Compare the all previous results with Simscape's simulation results using a scope block
- What is the advantage of storing Simulink parameters in the workspace (R, L, C) and reference them in all blocks (constant blocks, gains, etc.) instead of using discrete values for R, L, C and Ui? Explain your answer in the Simulink model by using an Annotation block.

4.8 Model the current in the RLC circuit

4.8.1 Subtask description

- Can the current **i** in the RLC circuit be plotted in the ODE Simulink model (model that was created in subtask 2) as well?
    1. If yes, plot it in a scope
    2. If no, explain why not. Which modifications are necessary to be able to plat the current?
- Can the current **i** of the RLC circuit be plotted from the transfer function output?
    1. If yes, plot it in a scope
    2. If no, explain why not. Which modifications are necessary to receive the current?
- Evaluate the current from the Simscape model and compare it to the previous current results, if possible.

4.8.2 Presenting your results

- Add a new scope to the Simulink model, that shows the current of the RLC circuit model. Depending if it is possible or not to compare all three Simulink / Simscape results the scope must have more or less inputs

5 REPORT

- The report for this training unit are all your modelling files (Simulink, LTSpice). All the files that are necessary to run your model shall be uploaded via Moodle.
- Make sure to use explain your findings properly in your model. Use annotation blocks, add pictures, add formulars to your model, to improve your documentation.
- Be sure to check the correct function of your model before uploading it to Moodle. If your model **does not work**, you **will not receive any points** for this training unit.
- Make sure, that the script shows **your own work** on this training unit task. Using solutions from your fellow students will result in zero points for you and your fellow students.

- **Caution (!)** be sure to check the **due date** that is set for uploading the files on Moodle!

6 SOURCES

[1] – Wikipedia, Online: https://en.wikipedia.org/wiki/RLC_circuit