EXPERIMENT NUMBER 10

TRANSIENT ANALYSIS USING PLECS

Objectives:

By completing this experiment, the student is expected to:

- To be able to use PLECS to simulate transient repose of any passive circuit.
- To gain understanding on the effects of the damping factor ζ and natural frequency ω_o on the transient response of a passive circuit.
- To be able to deliver proper description of any transient response using a set of various metrics.

Scope:

This experiment focusses on the transient analysis of step response of a series RLC using PLECS; without losing in generality, all the learning objectives can apply to any other formats of learning higher order systems.

Experiment structure:

This experiment consists of a tutorial section and three parts:

1- The tutorial section is intended to teach the student how to use the PLECS software to simulate transient response of passive circuits.
2- Part-1 will focus on analyzing the effect of the damping factor ζ on the transient response of higher order systems.
3- Part-2 will focus on analyzing the effect of the natural frequency ω_o on the transient response of higher order systems.
4- Part-3 will focus the utilization of various metrics to qualitatively and quantitatively describe the transient response of higher order systems.

Background and theory:

This section is intended to review theoretical aspects related to the topic of the experiment only.

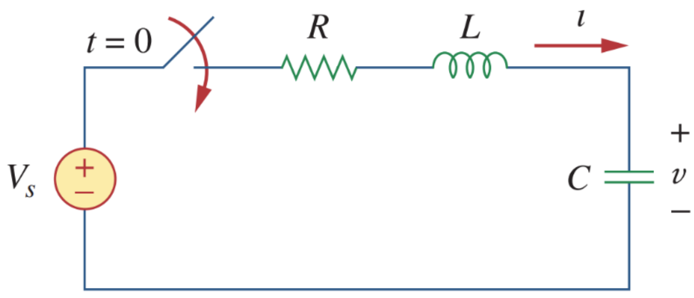

Figure 1: Step voltage applied to a series RLC circuit.

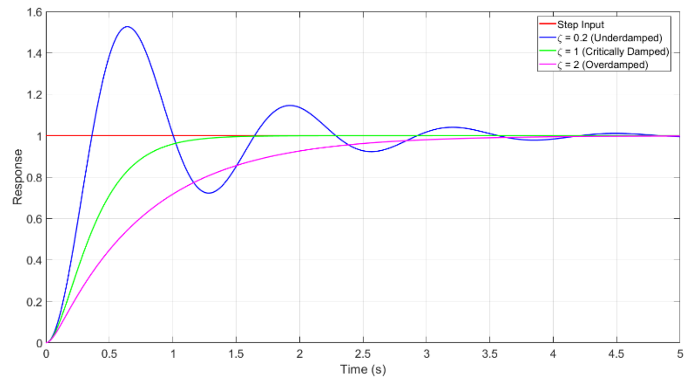

Figure 2: various formats of transient response.

Figure 1 showcase a step voltage applied to a series RLC circuit. A step voltage refers to a sudden change in applied voltage to the circuit. This concept is illustrated in Figure 1 using voltage source V_s in series with the switch such that the circuit is initially at rest and at $t = 0$, the switch closes, applying the step voltage V_s across the RLC components. This initiates a transient response characterized by oscillatory or exponential behavior. The transient response is the part of system's complete response that occurs immediately after a sudden change in input (e.g., step input) and *usually* settles out over time to a final value that is referred to as the steady-state value. Figure 2 illustrates how the sudden change in the step input (i.e., the red signal) could produce various possible transient responses (i.e., blue, green, and purple) that inhibit various oscillations, exponential decay, or overshoot before reaching the steady-state value. These various formats of the transient response occur due to the changes of the system dynamics that resulted due to changes in the RLC circuit.

Studying the transient response takes significant importance as it (i) characterizes the behavior of the system, (ii) determines the stability of the system, and (iii) is essential for controlling the system. Studying the transient response of RLC circuits is crucial in power systems (preventing surges and faults), signal processing (designing stable filters and amplifiers), and communication systems (ensuring reliable RF transmission). It's essential in control systems (servo motors and robotics), power electronics (DC-DC converters and inverters), and automotive electronics (ECUs and battery management). Additionally, it impacts medical devices like pacemakers and bioelectronic sensors, ensuring smooth operation.

The transient response can be characterized theoretically using the fundamentals of circuit analysis; for instance, Applying KVL around the loop in Figure 1 for $t > 0$ we get

$$L\frac{di}{dt} + Ri + v = V_s,$$

where L is the inductance, R is the resistance, i is the current in the loop, V_s is the source voltage and v is the voltage across the capacitor. Here, the current flowing through the capacitor (i.e., the current in the loop i) can be correlated to the voltage across the capacitor v as

$$i = C\frac{dv}{dt},$$

where C is the capacitance. We can now express the KVL as

$$\frac{d^2v}{dt^2} + \frac{R}{L}\frac{dv}{dt} + \frac{v}{LC} = \frac{V_s}{LC}.$$

The previous equation is a second order differential equation. One way to solve it is express it a characteristic equation and solve the polynomial. More specifically, the corresponding characteristic equation is

$$s^2 + \frac{R}{L}s + \frac{1}{LC} = 0.$$

Solving for s

$$s = \frac{-R \pm \sqrt{R^2 - 4L/C}}{2L}.$$

The parameter s can also be described using the damping factor ζ and the natural frequency ω_o (measured in rad/s) as

$$s = -\zeta\omega_o \pm \omega_o\sqrt{\zeta^2 - 1},$$

where

$$\zeta = \frac{R}{2}\sqrt{\frac{C}{L}}, \qquad \text{and} \qquad \omega_o = \frac{1}{\sqrt{LC}}.$$

Here, based on the damping factor ζ, three formats are possible of the homogeneous solution $v_h(t)$ of the response:

- Case-1: Overdamped **(**$\zeta > 1$)**:** s has two real and distinct roots. The transient response in this case is

$$v_h(t) = Ae^{s_1 t} + Be^{s_2 t}.$$

- Case-2: Critically damped **($\zeta = 1$):** s has repeated real root. The transient response in this case is

$$v_h(t) = (A + Bt)e^{st}.$$

- Case-3: Underdamped **($\zeta < 1$):** s has complex conjugate roots. The transient response in this case is

$$v_h(t) = e^{-\alpha t}(A\cos(w_d t) + A\sin(w_d t)),$$

where

$$\alpha = \frac{R}{2L}, \qquad \text{and} \qquad \omega_d = \omega_o\sqrt{1-\zeta^2}.$$

The particular solution (i.e., the steady-state solution) describe the response under the steady-state (i.e., at $t = \infty$); for all the three cases the particular solution $v_p(t)$ is

$$v_p(t) = V_s$$

The general solution describes the overall transient response such as

$$v(t) = v_h(t) + v_p(t)$$

Thus, the general transient solution for the three cases is

- Case-1: Overdamped **($\zeta > 1$):** s has two real and distinct roots. The transient response in this case is

$$v(t) = Ae^{s_1 t} + Be^{s_2 t} + V_s.$$

- Case-2: Critically damped **($\zeta = 1$):** s has repeated real root. The transient response in this case is

$$v(t) = (A + Bt)e^{st} + V_s.$$

- Case-3: Underdamped **($\zeta < 1$):** s has complex conjugate roots. The transient response in this case is

$$v_h(t) = e^{-\alpha t}(A\cos(w_d t) + A\sin(w_d t)) + V_s.$$

Tutorial:

This section is intended to teach the student how to use the PLECS software to simulate transient response of passive circuits. This part should not be included in the student notebook.

PLECS (Piecewise Linear Electrical Circuit Simulation) is a fast and efficient simulation software for power electronics and electrical systems, available as a standalone tool or integrated with MATLAB/Simulink. It features a built-in component library for power electronics, supports electrical, thermal, and control modeling, enables real-time simulation, and allows co-simulation with Simulink. Additionally, it offers automatic code generation for embedded systems, making it a valuable tool for designing and testing power converters, motor drives, and grid-connected systems in both research and industry.

(A) Lunching PLECS

1- PLECS is not free or open-source software; however, Missouri S&T offers PLECS for all its students and staff using AppsAnyWhere. Start this tutorial by lunching AppsAnyWhere by visiting this link (https://appsanywhere.mst.edu).
2- Using AppsAnyWhere main window, type PLECS in the search bar and two versions of PLECS will appear. The PLECS Blockset is used if PLECS is intended to be utilized with MATLAB while PLECS Standalone is for the direct access of PLECS without MATLAB integration. For this experiment, we are using PLECS Standalone; click on Lunch to lunch PLECS Standalone. This step is shown in Figure 3.

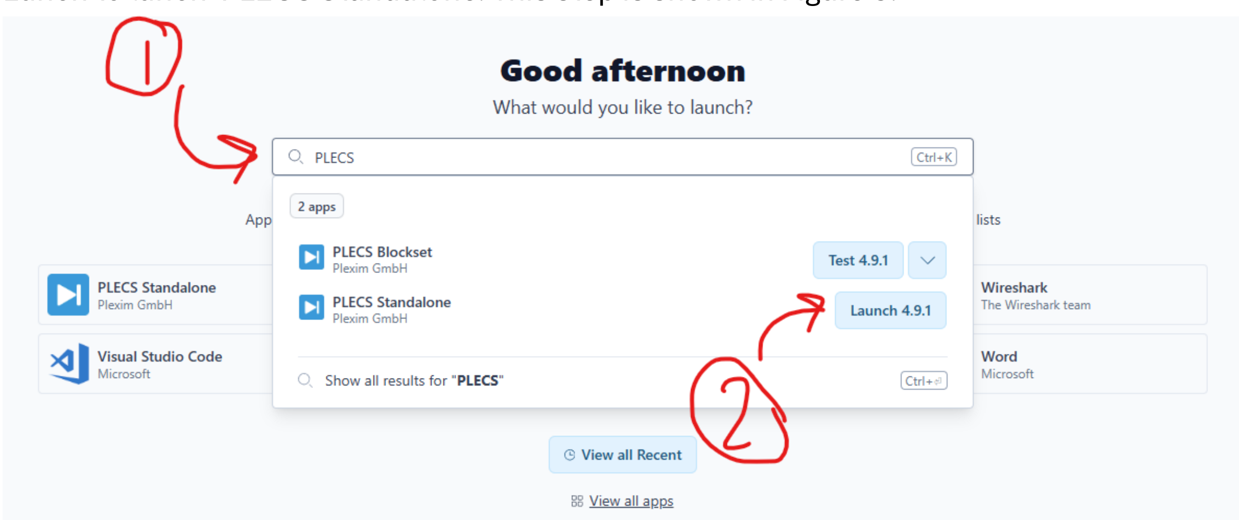

Figure 3: Lunching PLECS.

(B) PLECS interface

When lunching PLECS for the first time, the Library Browser will show up. The Library Browser contains all the components that are to be utilized in the design. These components should be placed in the Model window. The Model window is the workspace that allow you to place and connect your model for simulation purposes. Open a new model window by clicking file → New Model from the Library Browser. The Model window will show up with yellow background. This step is illustrated in Figure 4. Note that you can access the Library Browser back from the Model window (i.e., in case if Library Browser has been closed) by clicking on Windows → Library Browser from the Model window.

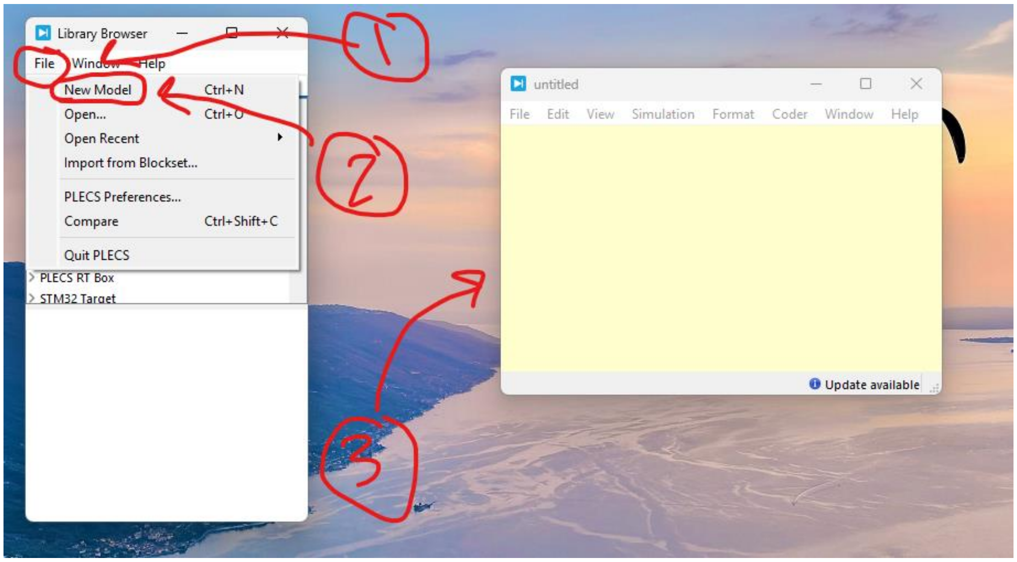

Figure 4: PLECS interface

(C) Libraries

The Library Browser consists of various libraries; each containing components that can be utilized to simulate various engineering systems. For this experiment, we are interested in the following two libraries: (i) Electrical and (ii) System. The Electrical library contains components related to various electrical systems. This library consists of various sub-libraries. Out of these sub-libraries, we will be using the (i) Sources, (ii) the Passive Components, and (iii) Meters. The Sources contains various electrical powering components, the Passive Components contain electrical circuit components that doesn't require to be power up externally to function, and Meters contain sensors that measure electrical voltage and current. The System library on the other hand include utility components that allows building and displaying the results.

For the tutorial part, we are interested in simulating the transient response of the circuit in Figure 1. As the figure shows, this circuit consists of a DC power source, a resistor, an inductor, and a capacitor all in series. Moreover, the voltage across the capacitor denoted v in the figure is the output voltage that we seek to simulate transient response for.

Start building the circuit in Figure 1 by placing the components in the figure. To do so, drag the following components from the library and drop it on the Model window. More specifically you need the following components:

- From Electrical → Sources → Voltage Source DC: this component generates constant voltage (DC power source).
- From Electrical → Passive Components → Resistor: this component represents a resistor.
- From Electrical → Passive Components → Inductor: this component represents an inductor.
- From Electrical → Passive Components → Capacitor: this component represents a capacitor.
- From Electrical → Meters → Voltmeter: this component represents a voltameter.
- From System → Scope: the scope is a utility component that allows you to plot a sensor measured value against time (i.e., similar to the oscilloscope).

Figure 5 shows the required components on the Model Window with the same order described above.

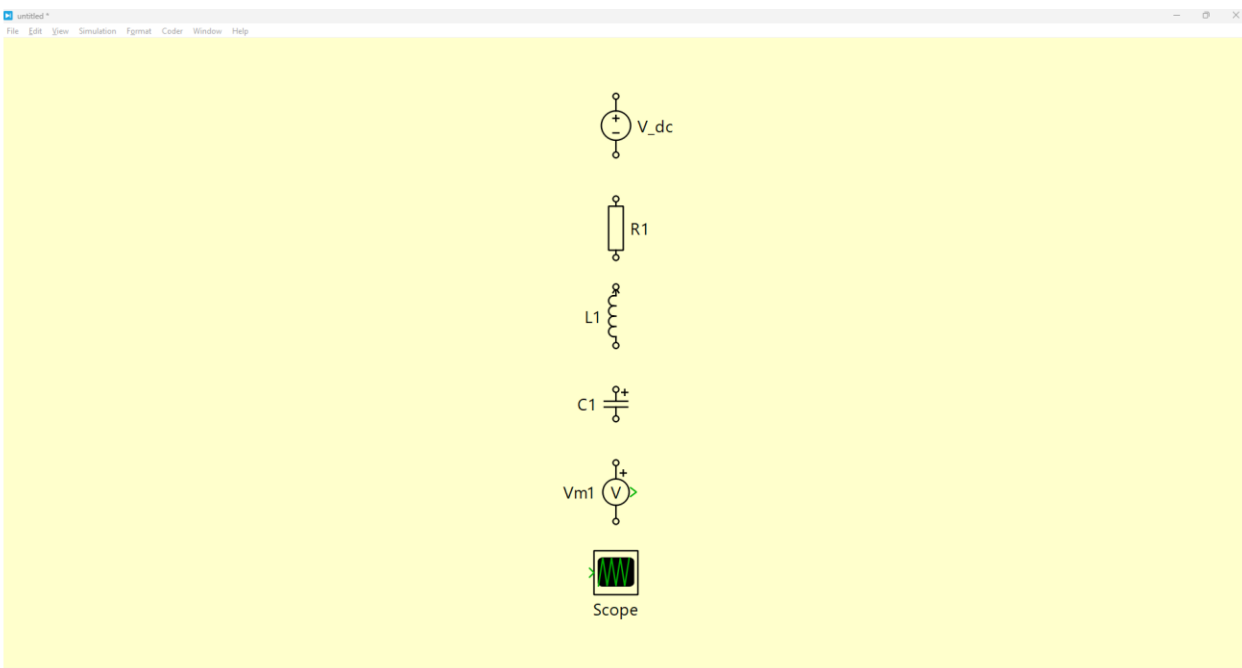

Figure 5: The required components

(D) Building the circuit

Build the circuit by connecting the components as it is connected in Figure 1. For connecting the components, note that all the inductor, the capacitor, and the voltameter have polarities and should be connected with respect the expected direction of the current. The voltameter should be connected in parallel with the capacitor as the output voltage to be illustrated is the capacitor voltage $v(t)$. The voltameter should also be connected to the scope to show $v(t)$ against time similar to an oscilloscope. The correct connected circuit should look similar to what is illustrated in Figure 6.

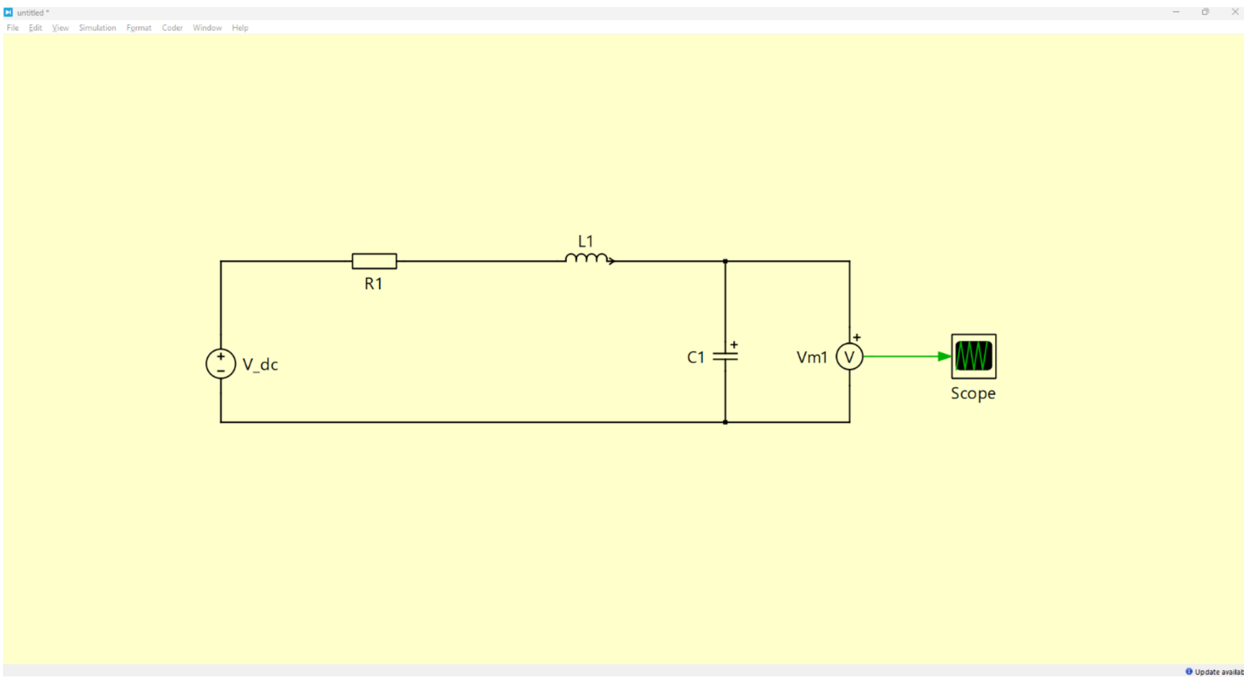

Figure 6: The connected circuit.

The next step involves specifying the parameters for each component. To do so, double click on each component and insert the value of the parameter.

- The Voltage Source DC has only one parameter that is the amount of the DC voltage.
- The resistor has only one parameter that is the amount of resistance.
- The inductor has two parameters that is the amount of inductance, and the initial current stored within the inductance.
- The capacitor has two parameters that is the amount of capacitance, and the initial voltage stored within the capacitance.
- The voltmeter has no parameters to adjust.

As for the tutorial part, specify the parameters as you desire. Note that you can use the exponential notation to specify your values; for example, $300\ \mu F$ is $300e-6$.

(E) Simulation Parameters

Before you start your simulation, you need to adjust the Simulation Parameters. To do so, click on Simulation → Simulation Parameters from the Model window. A window will show up with various settings related to simulation configuration. We will be changing only three parameters of interest; these are:

- Start time (s): this represents the starting time of the simulation. Keep this value on $0\ s$ unless you want to make special simulation that doesn't start at $t = 0\ s$.
- Time span (s): this represents the stop time of the simulation (i.e., when the simulation stop). Setting this parameter extra small will result an incomplete simulation (i.e., simulation ends before showing the full response), while setting this parameter extra large will compress the response signal in a way that it is not longer readable. For this purpose, the stop time should be specified appropriately. One way to do so is to rely on some evaluation metrics. More specifically, the settling time will be used for this purpose. The Settling time t_s is the time required for the response of a system to remain within a specified percentage (commonly 2% or 5%) of its final steady-state value after a disturbance or input change, such as a step input. The settling time can be determined as follows:

$$t_s \approx \frac{4}{\zeta \omega_o}$$

  Set your span time based on the settling time of your system.
- Max step size (s): this represents the resolution of your simulation. It is recommended to use a very small value for this purpose. A $10^{-6}\ s$ should be sufficient.

Figure 7 illustrates how to specify your simulation parameters.

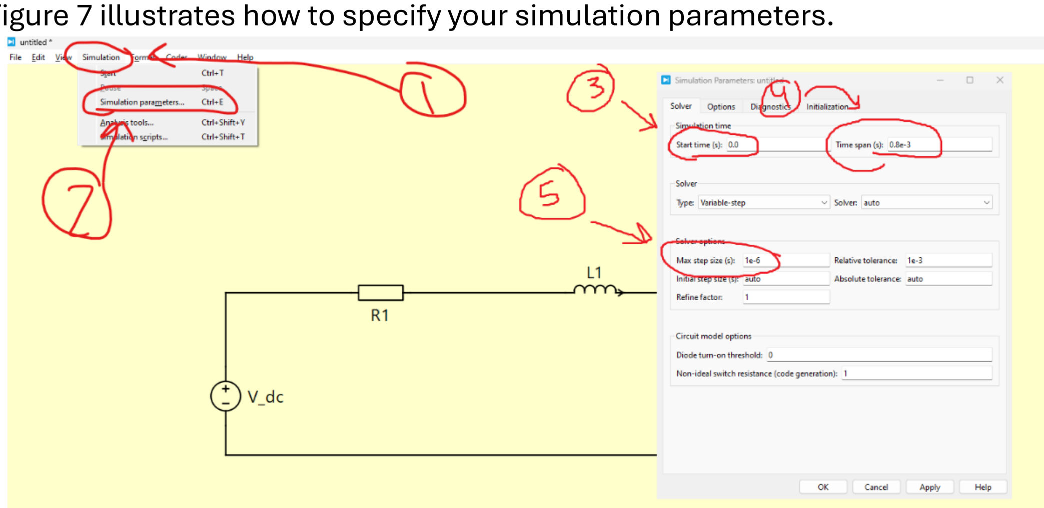

Figure 7: Setting the simulation parameters.

(F) Simulation Start

Now everything is ready for your simulation. To start your simulation click on Simulation → Start from the Model window. To view your simulation results double click on the Scope. These steps are shown in Figure 8.

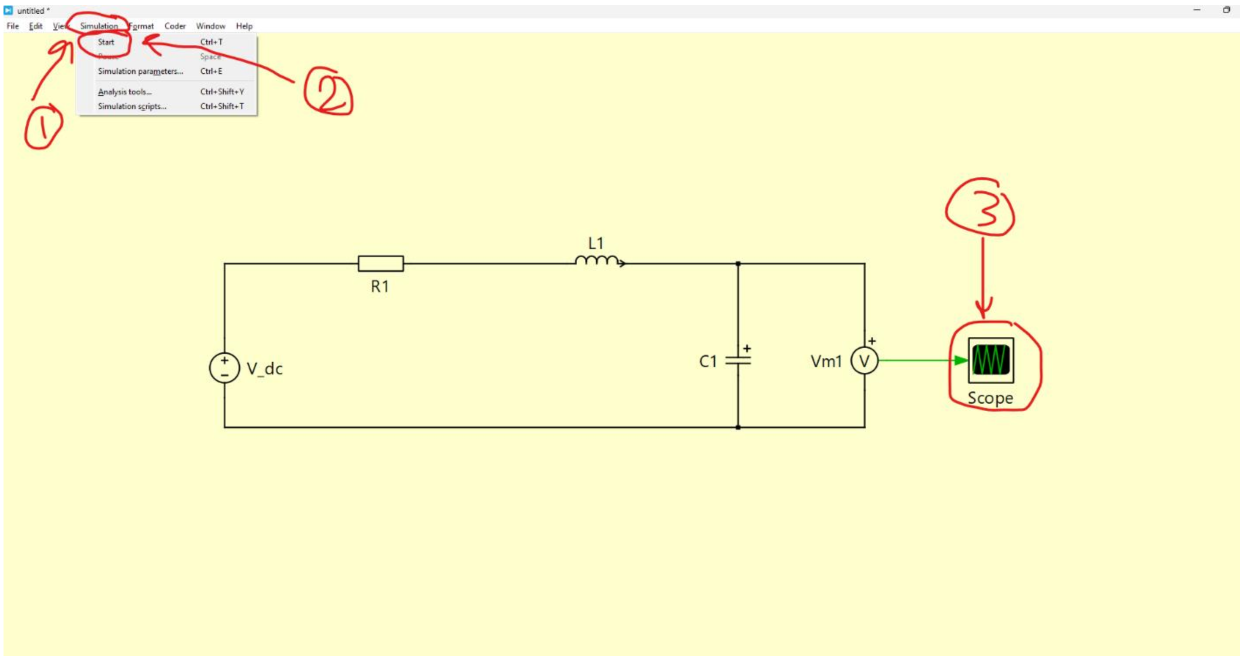

Figure 8: Start the simulation.

(G) Scope

Scope will be utilized to display results of your simulation. When double clicking on the Scope a window of your response against time will show up. The scope has the following features:

- You can zoom to specific time horizon by clicking the cursor on the start time and dragging the cursor horizontally to the end time.
- You can zoom to specific y-axis horizon (e.g., voltage in this case) by clicking the cursor on the start voltage and dragging the cursor vertically to the end voltage.
- You can undo any changes by clicking on the left arrow icon (←) tabs menu.
- You can export the figure in CSV format by clicking File → Export → as CVS → All...
- You can add another sub-figure to the scope to illustrate two figure on the same screen by clicking File → Scope parameters... and then a new window will show

up; set the Number of plots for how many plots you wish to see at the same scope. This step is illustrated in Figure 9.

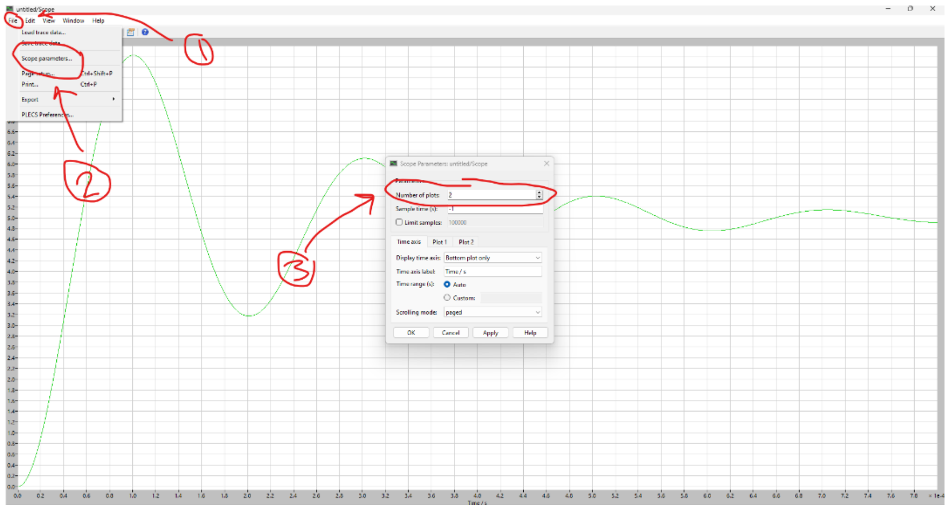

Figure 9: changing the Number of plots.

Test your understanding by creating a new parallel RLC circuit. Use the same exact values for the resistor, capacitor and inductor. Replace the Voltage Source DC with a Current Source DC. You can find this component in the following library path Electrical → Sources → Current Source DC. This component represents a DC current source. Change the value of the DC current source to any appropriate value. Connect the results of your simulation with to the same exact scope of the series RLC circuit. Add caption to each circuit by clicking on empty yellow space above each of the two circuits. You might change the font size and color of your caption if needed. Your configuration should look similar to Figure 10. Start the simulation and observe the results.

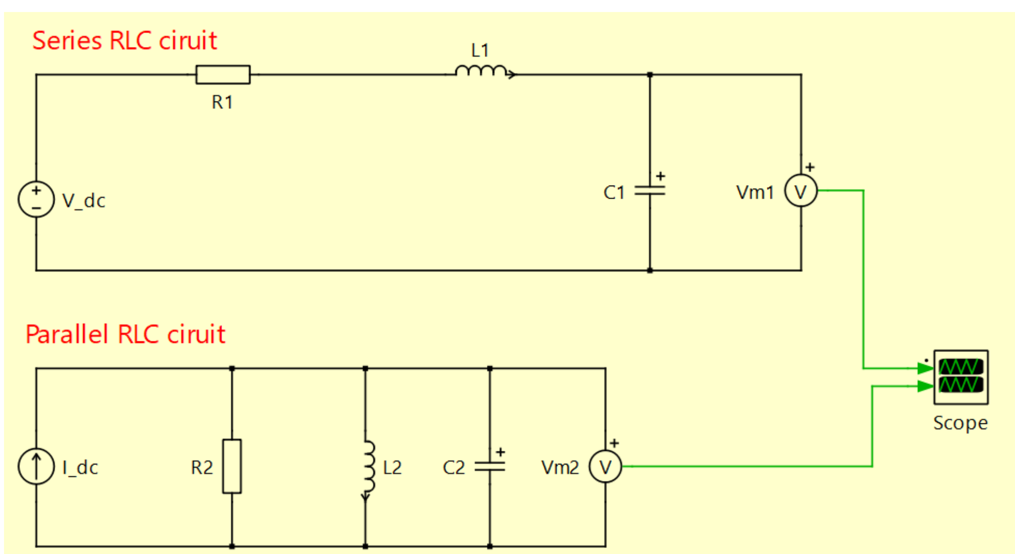

Figure 10: simulating two circuits simultaneously.

Part-1: Analyzing the effect of the dampening factor ζ.

In this section, you will simulate various circuits with different damping factors ζ and observe their effects on the transient response. To ensure a fair comparison, we will keep the natural frequency ω_0 constant while varying only the damping factor ζ.

Start by constructing three series RLC circuits powered by a DC power voltage source. Use the following parameters in Table 1 for each of the three circuits

Table 1: part-1 simulation parameters.

V_s in $[V]$	R in $[\Omega]$	L in $[mH]$	C in $[\mu F]$	ζ	ω_o in $[rad/s]$	t_s in $[s]$
5	1.0	10.0	100.0	???	???	???
5	6.32	10.0	100.0	???	???	???
5	12.0	10.0	100.0	???	???	???

For each of the three circuits, calculate damping factor ζ, the natural frequency ω_0 and the settling time t_s. Add caption of these missing values on the top of each corresponding circuit. Connect the output voltage (i.e., voltage across the capacitor) of the three circuits for a single scope. Your circuit should look similar to Figure 11. Set the Time span of your simulation to be the maximum settling time of the three circuits (i.e., $t_{stop} = \max(t_{s,1}, t_{s,2}, t_{s,3})$. Run the simulation and observe the results.

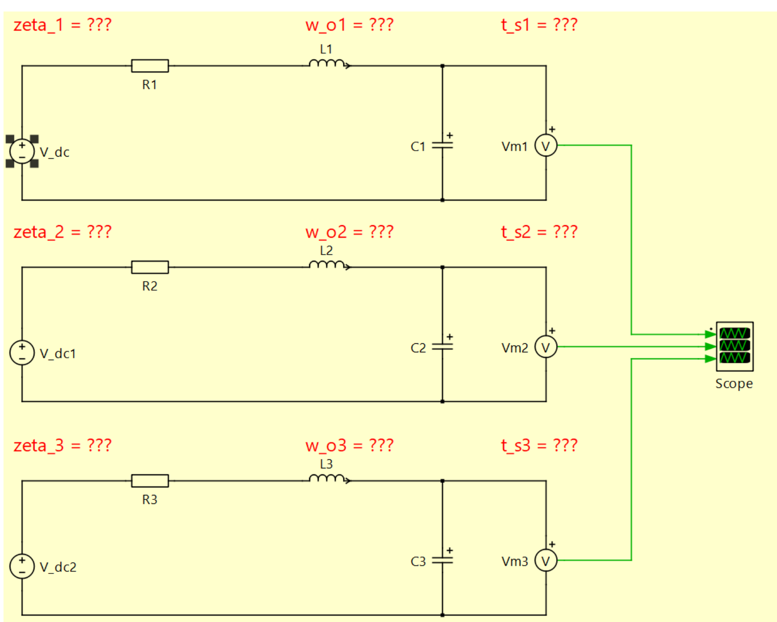

Figure 11: Part-1 simulation circuits.

In your notebook report include the following:

1- Include a screenshot of the constructed three circuits. Make sure that the correct damping factor, natural frequency and settling time are captioned appropriately and clearly on the top of each constructed circuit in your screenshot.
2- Include the transient responses of the three circuits obtained from the scope. Make sure that your figure(s) is high-quality with readable axes and features. You ***might*** need to export the data as CSV and re-plot the reposes for higher quality if needed.
3- What type of response does each of the transient responses represent and why? Correlate your answer with your theoretical understanding.
4- Qualitatively describe how each of the three transient responses look like.
5- For the series RLC circuits that you have constructed, what circuit parameter would you change to change the response type and why? Correlate your answer with your theoretical understanding.
6- Give your final conclusion; what is the effect of changing the damping factor ζ on your transient response?

Part-2: Analyzing the effect of the natural frequency ω_o

In this section, you will simulate various circuits with different natural frequencies ω_o and observe their effects on the transient response. To ensure a fair comparison, we will keep the damping factor ζ constant while varying only the natural frequency ω_0.

Start by constructing three series RLC circuits powered by a DC power voltage source. Use the following parameters in Table 2 for each of the three circuits

Table 2: part-2 simulation parameters.

V_s in $[V]$	R in $[\Omega]$	L in $[mH]$	C in $[\mu\ F]$	ζ	ω_o in $[rad/s]$	t_s in $[s]$
5	1.0	5.0	50.0	???	???	???
5	1.0	7.0	70.0	???	???	???
5	1.0	10.0	100.0	???	???	???

For each of the three circuits, calculate damping factor ζ, the natural frequency ω_0 and the settling time t_s. Add caption of these missing values on the top of each corresponding circuit. Connect the output voltage (i.e., voltage across the capacitor) of the three circuits for a single scope. Your circuit should look similar to Figure 11. Set the Time span of your simulation to be the maximum settling time of the three circuits (i.e., $t_{stop} = \max(t_{s,1}, t_{s,2}, t_{s,3})$. Run the simulation and observe the results.

In your notebook report include the following:

1- Include a screenshot of the constructed three circuits. Make sure that the correct damping factor, natural frequency and settling time are captioned appropriately and clearly on the top of each constructed circuit in your screenshot.

2- Include the transient responses of the three circuits obtained from the scope. Make sure that your figure(s) is high-quality with readable axes and features. You <u>*might*</u> need to export the data as CSV and re-plot the reposes for higher quality if needed.

3- What type of response does each of the transient responses represent? Did you get similar types of responses or different and why? Correlate your answer with your theoretical understanding.

4- Qualitatively describe how each of the three transient responses look like.

5- For the series RLC circuits that you have constructed, what circuit parameter would you change to change the response type and why? Correlate your answer with your theoretical understanding.

6- Give your final conclusion; what is the effect of changing the natural frequency ω_o on your transient response?

Part-3: Transient Response Evaluation

You can qualitatively and quantitatively describe the transient response of any system using various of metrics. The following are definitions of some of these metrics:

- Settling Time t_s: The time required for the response to stay within a certain percentage (commonly 2% or 5%) of its final value. The settling time is defined as

$$t_s \approx \frac{4}{\zeta \omega_o}$$

- Peak Time t_p: The time at which the system response reaches its maximum (peak) value.

$$t_p = \frac{\pi}{\omega_d}$$

- Rise Time t_r: The time taken for the response to rise from 10% to 90% of its final value.

$$t_r \approx \frac{1.8}{w_n}$$

- Maximum overshoot M_p (in percentage %): The amount by which the response exceeds the final steady-state value, expressed as a percentage.

$$M_p = e^{\left(\frac{\zeta \pi}{\sqrt{1-\zeta^2}}\right)} \times 100\%$$

- Steady-State Error e_{ss}: The difference between the final output and the desired value.

Now construct the circuit that is shown in Figure 12. Use the parameters in Table 3.

Table 3: parameters for part-3 simulation circuits.

	R_1 in $[\Omega]$	R_2 in $[\Omega]$	L in $[\mathrm{mH}]$	C in $[\mu\mathrm{F}]$	V_{DC} in $[V]$	Time span in $[ms]$
Value	30	100	30	10	10	20

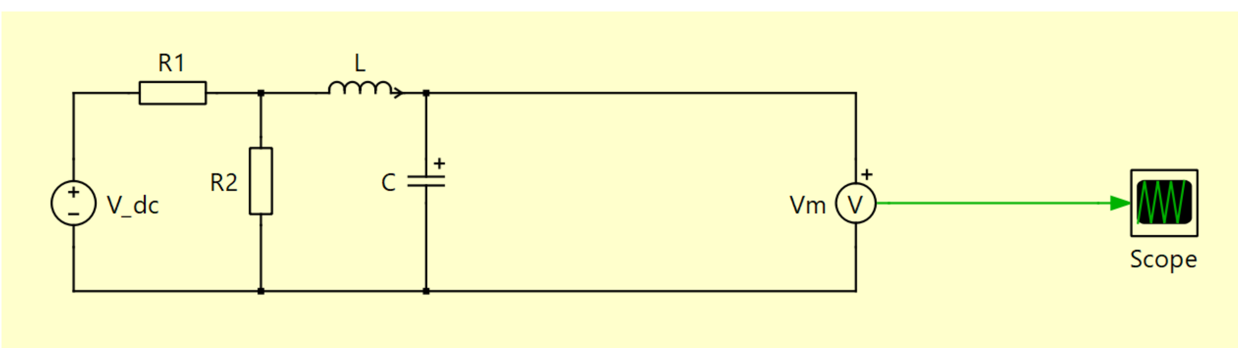

Figure 12: Part-3 simulation circuits.

In your notebook report include the following:

1- Include the transient responses of the three circuits obtained from the scope. Make sure that your figure(s) is high-quality with readable axes and features. You *might* need to export the data as CSV and re-plot the reposes for higher quality if needed.
2- What type of response did you get and why?
3- Use the resulted response plot to find the settling time t_s, peak time t_p, rise time t_r, maximum overshoot M_p, and steady-state error e_{ss}.
4- Estimate the damping factor and natural frequency using the parameters your collected from the previous part. Does this match the response type you have predicated?
5- Critique the obtained response based on the parameters you found. What is wrong in this response?