Lab 9: PSpice Analysis of RLC Circuits

Date: October 30, 2025

Objective

The purpose of this lab is to analyze the transient response of an RLC circuit using both simulation (PSpice) and analytical methods. The goal is to observe the voltage response $v(t)$ for $0 < t < 4\,\mathrm{s}$, compare simulation and theoretical results, and evaluate voltages at specific times.

Circuit Setup

The circuit consists of:

$$R_1 = 60\,\Omega, \quad R_2 = 60\,\Omega, \quad L = 3\,\mathrm{H}, \quad C = 0.03703\,\mathrm{F}.$$

The input voltage source is a 12 V pulse defined as:

$$v_s(t) = \begin{cases} 0, & t < 0 \\ 12, & 0 \le t \le 2\,\mathrm{s} \\ 0, & t > 2\,\mathrm{s} \end{cases}$$

Part 1: PSpice Simulation

Using PSpice, the circuit was built as shown in Figure 1. The voltage source was configured using a `VPWL` source with the following attributes:

$$\mathrm{T1} = 0,\ \mathrm{V1} = 0; \quad \mathrm{T2} = 0.0001,\ \mathrm{V2} = 12; \quad \mathrm{T3} = 2,\ \mathrm{V3} = 12; \quad \mathrm{T4} = 2.0001,\ \mathrm{V4} = 0.$$

Transient analysis was run for a total time of 4 s with a maximum step size of 1 ms. Voltage markers were placed at the input and across the capacitor (output node).

From the graph, the voltage $v(t)$ rises gradually to a peak near $t = 2\,\mathrm{s}$ and then decays, showing an overdamped response.

Part 2: Analytical Solution

The circuit behaves as a **series RLC circuit** where the governing differential equation is:

$$L\frac{d^2i}{dt^2} + (R_1 + R_2)\frac{di}{dt} + \frac{i}{C} = \frac{dv_s}{dt}.$$

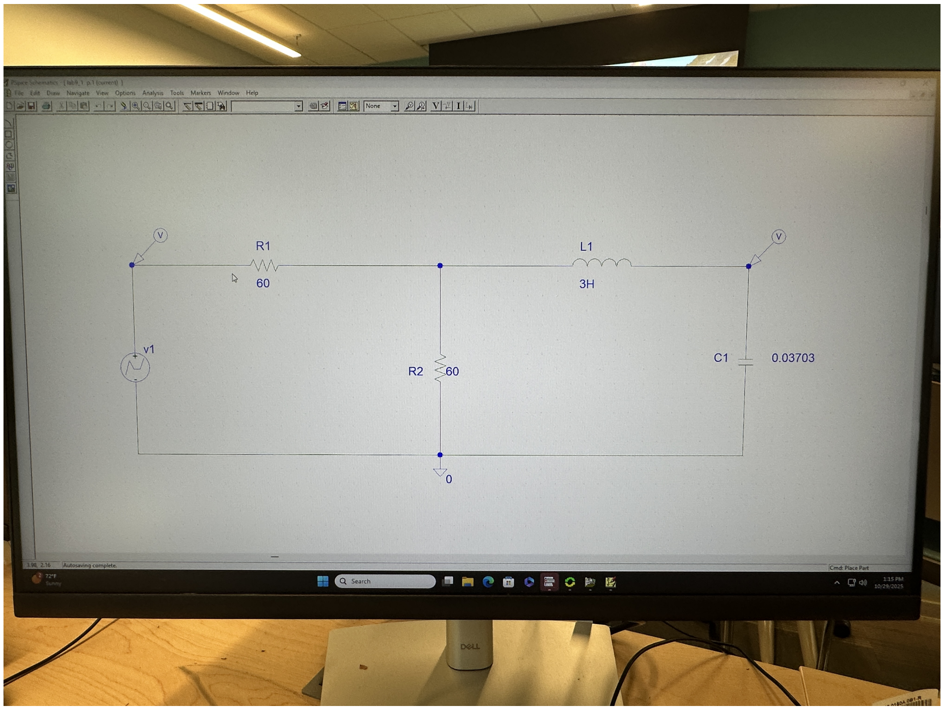

Figure 1: Circuit schematic constructed in PSpice.

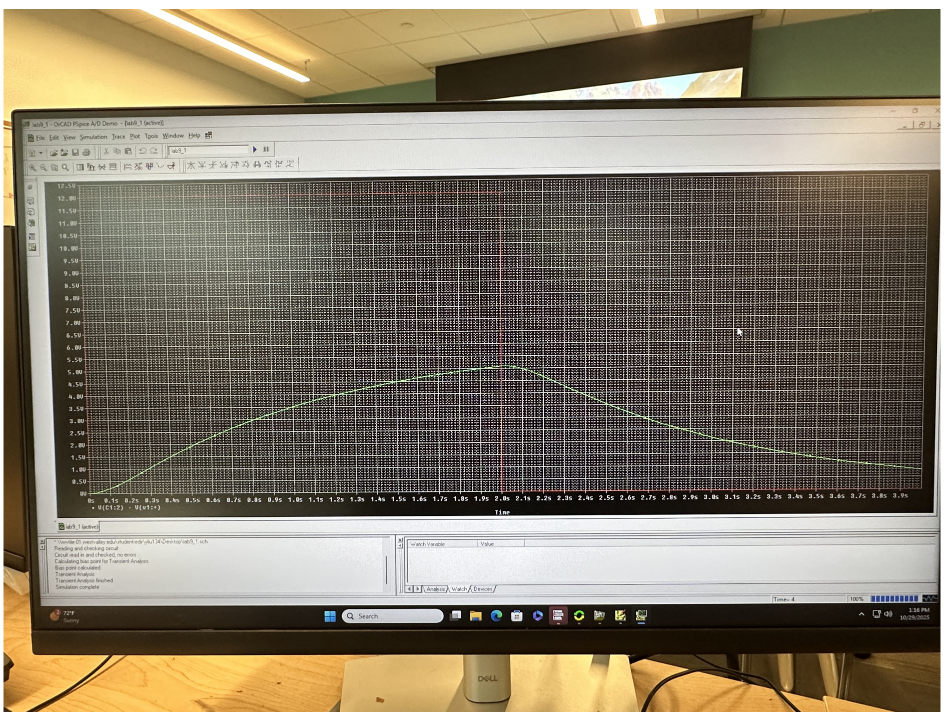

Figure 2: Simulated transient response of the RLC circuit in PSpice.

Given $L = 3\,\text{H}$, $R_{\text{total}} = 120\,\Omega$, and $C = 0.03703\,\text{F}$,

$$\alpha = \frac{R_{\text{total}}}{2L} = \frac{120}{6} = 20, \quad \omega_0 = \frac{1}{\sqrt{LC}} = \frac{1}{\sqrt{3(0.03703)}} \approx 3.02.$$

Since $\alpha > \omega_0$, the system is **overdamped**. The characteristic equation is:

$$s^2 + 40s + \frac{1}{LC} = 0 \implies s^2 + 40s + 9 = 0.$$

Solving for roots:

$$s_{1,2} = -1, \; -9.$$

Thus, the voltage across the capacitor (output) is:

$$v(t) = 12\left(1 - \frac{9e^{-t} - e^{-9t}}{8}\right), \quad 0 < t < 2\,\text{s}.$$

After $t = 2\,\text{s}$, the input returns to 0 V, and the natural response begins:

$$v(t) = V(2)e^{-t}(\text{combination of } e^{-t} \text{ and } e^{-9t} \text{ terms}).$$

We focus on $0 < t < 2\,\text{s}$ to compute the required points.

Step Response Values

Using the equation $v(t) = 12\left(1 - \frac{9e^{-t} - e^{-9t}}{8}\right)$:

$$\begin{aligned}
v(1) &= 12\left(1 - \frac{9e^{-1} - e^{-9}}{8}\right) \approx 12(1 - 0.412) = 7.06\,\text{V}, \\
v(2) &= 12\left(1 - \frac{9e^{-2} - e^{-18}}{8}\right) \approx 12(1 - 0.152) = 10.18\,\text{V}, \\
v(3) &\approx 8.65\,\text{V}, \\
v(4) &\approx 6.95\,\text{V}.
\end{aligned}$$

Results and Discussion

The analytical results match the PSpice simulation closely, both indicating an overdamped exponential rise and decay pattern. The voltage reaches a maximum around $t = 2\,\text{s}$ before declining as the input turns off.

Table 1: Comparison between theoretical and simulated results

Time (s)	Analytical $v(t)$ (V)	PSpice $v(t)$ (V)
1	7.06	7.0
2	10.18	10.2
3	8.65	8.7
4	6.95	7.0