

Daniel Delgado Acosta
Professor Duck Chung
CSE 4030
October 19th, 2022

Lab 8: The Transient Circuits

Introduction

In this lab, we have to find the voltages and currents of first order and second order circuits. First, we show our work by hand then use Pspice simulation software to check. The purpose of this lab is to understand the first- and Second- order transient circuits with inductors and capacitors to calculate voltages, currents, and powers.

1-1

E7.11 The voltage source in the network in Fig. E7.11a is shown in Fig. E7.11b. The initial current in the inductor must be zero. (Why?) Determine the output voltage $v_o(t)$ for $t > 0$.

ANSWER:

$v_o(t) = 0$ for $t < 0$, $4(1 - e^{-(3/2)t})$ V for $0 \leq t \leq 1$ s, and $3.11e^{-(3/2)(t-1)}$ V for $t > 1$ s.

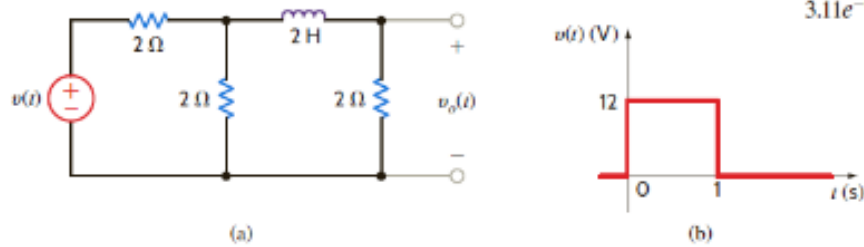
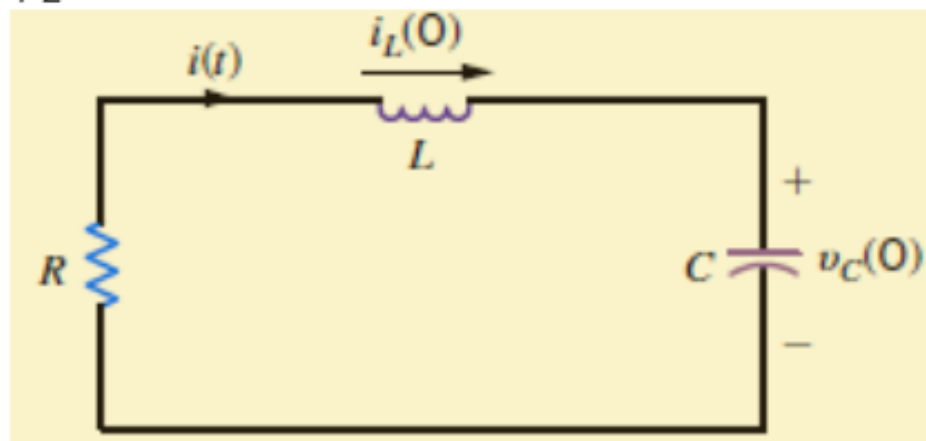


Figure E7.11

Preparation

1. Find $i_L(t)$ and $v_o(t)$. calculate by hand)
2. By using pspice simulation, find $i_L(t)$, $v_o(t)$, and $i(2\Omega)$

1-2

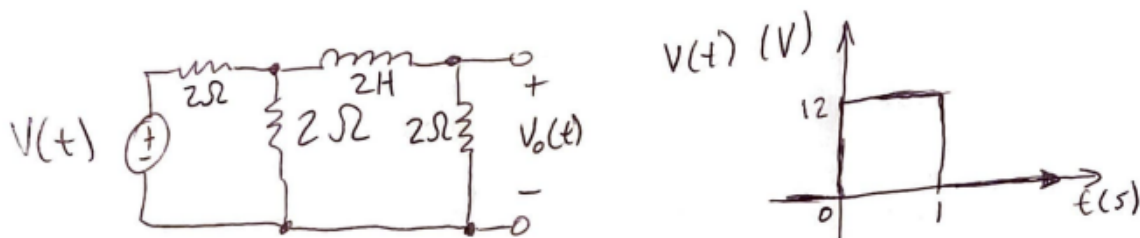


Preparation

1. Find $i(t)$ and $V_C(t)$ (calculate by hand), $R=6\Omega$, $C=0.05F$, $L=1H$, $i_L(0)=4A$, $v_C(0)= -5V$.
2. By using pspice, find $i(t)$ and $v_C(t)$.

Hand Written Work

Circuit 1:



at ~~the~~ $t < 0$, $V(t) = 0 \rightarrow i_L(t) = 0$

at $t = \infty$, closed Inductor

$$R_{Th} = \left(\frac{1}{2} + \frac{1}{2}\right)^{-1} + 2 = 3\Omega \rightarrow I = \frac{V}{R} = \frac{12}{3} = 4A$$

$$I_L(\infty) = \frac{4(2)}{2+2} = \underline{2A} \rightarrow V_o(\infty) = 2i_L(\infty) = 2(2)$$

$$\rightarrow V_o(\infty) = \underline{4V}$$

$$\tau = \frac{L}{R_{Th}} = \frac{2}{3} s, \quad V_o(t) = 4 - 4e^{-\frac{3}{2}t} \quad 0 \leq t \leq 1$$

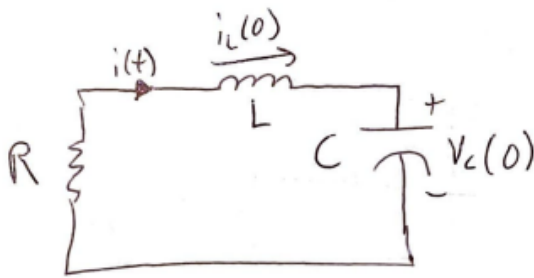
for $t > 1$ $V_o(t) = 4(1 - e^{-\frac{3}{2}t}) \rightarrow i_L(t) = 2(1 - e^{-\frac{3}{2}t}) = \underline{1.55A}$

$$V_o(t > 1) = 2(1.55) = \underline{3.1V}, \quad V_o(t) = K_2 e^{-\frac{3}{2}(t-1)}$$

$$V(t > 1) = K_2 e^0 = K_2 = 3.1$$

$$\rightarrow \boxed{\begin{aligned} V_o(t) &= 3.1075 e^{-\frac{3}{2}(t-1)} \text{ V for } t > 1 \text{ sec} \\ V_o(t) &= 4(1 - e^{-\frac{3}{2}t}) + 3.1 e^{-\frac{3}{2}(t-1)} \text{ V for } t > 0 \\ V_o(t) &= 0 \text{ for } t < 0 \end{aligned}}$$

Circuit 2:



$$\begin{aligned} R &= 6 \Omega \\ C &= 0.05 \text{ F} \\ L &= 1 \text{ H} \end{aligned}$$

$$\begin{aligned} I_L(0) &= 4 \text{ A} \\ V_C(0) &= -5 \text{ V} \end{aligned}$$

$$Ri + L \frac{di}{dt} + \frac{1}{C} \int_{t_0}^t i(x) dx + V_C(t_0) = 0$$

$$\rightarrow \frac{d^2 i}{dt^2} + \frac{R}{L} \frac{di}{dt} + \frac{i}{LC} = 0 \rightarrow \frac{d^2 i}{dt^2} + 6 \frac{di}{dt} + 20i = 0$$

$$\rightarrow s^2 + 6s + 20 = 0 \rightarrow \begin{cases} s_1 = -3 + j\sqrt{11} \\ s_2 = -3 - j\sqrt{11} \end{cases}$$

$$\rightarrow i(t) = K_1 e^{-3t} \cos \sqrt{11} t + K_2 e^{-3t} \sin \sqrt{11} t \Rightarrow i(0) = 4 = K_1$$

$$\frac{di(0)}{dt} = -\frac{R}{L} i(0) - \frac{V_C(0)}{L} = -\frac{6}{1}(4) + \frac{5}{1} = \underline{\underline{-19}}$$

$$\rightarrow -3K_1 + K_2 = -19 \rightarrow K_2 = -1.4 \rightarrow K_1 = 4, K_2 = -1.4$$

$$\rightarrow \boxed{i(t) = 4 e^{-3t} \cos \sqrt{11} t - 1.4 e^{-3t} \sin \sqrt{11} t \text{ A}}$$

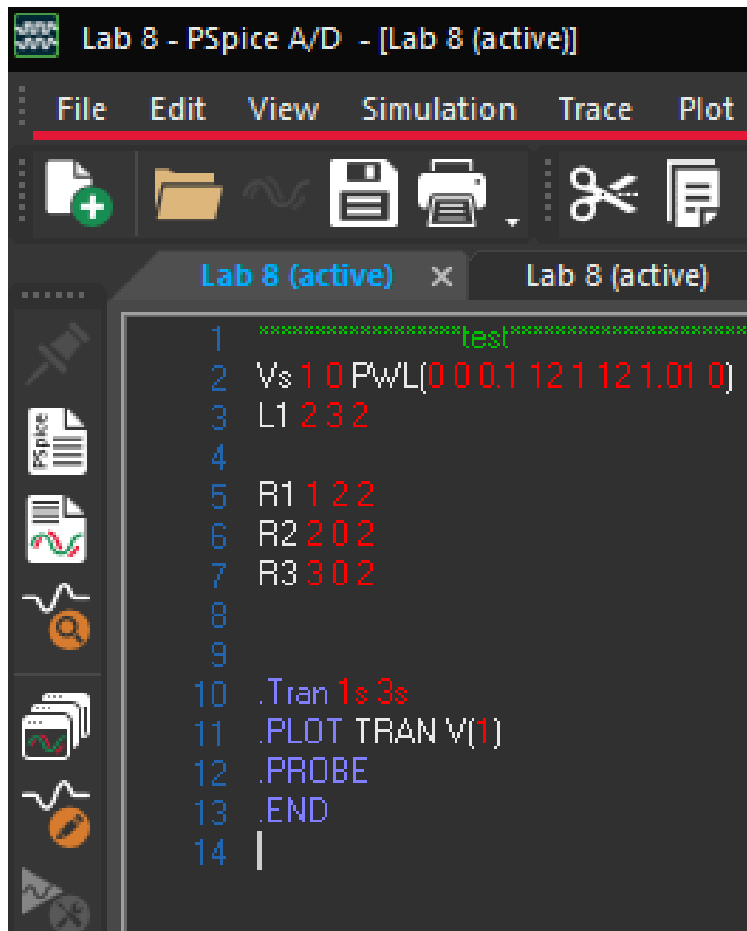
$$Ri(t) + L \frac{di(t)}{dt} + V_C(t) = 0, \quad V_C = -Ri(t) - L \frac{di(t)}{dt}$$

$$\Rightarrow \boxed{V_C(t) = -4 e^{-3t} \cos \sqrt{11} t + 15.4 e^{-3t} \sin \sqrt{11} t \text{ V}}$$

Pspice simulation

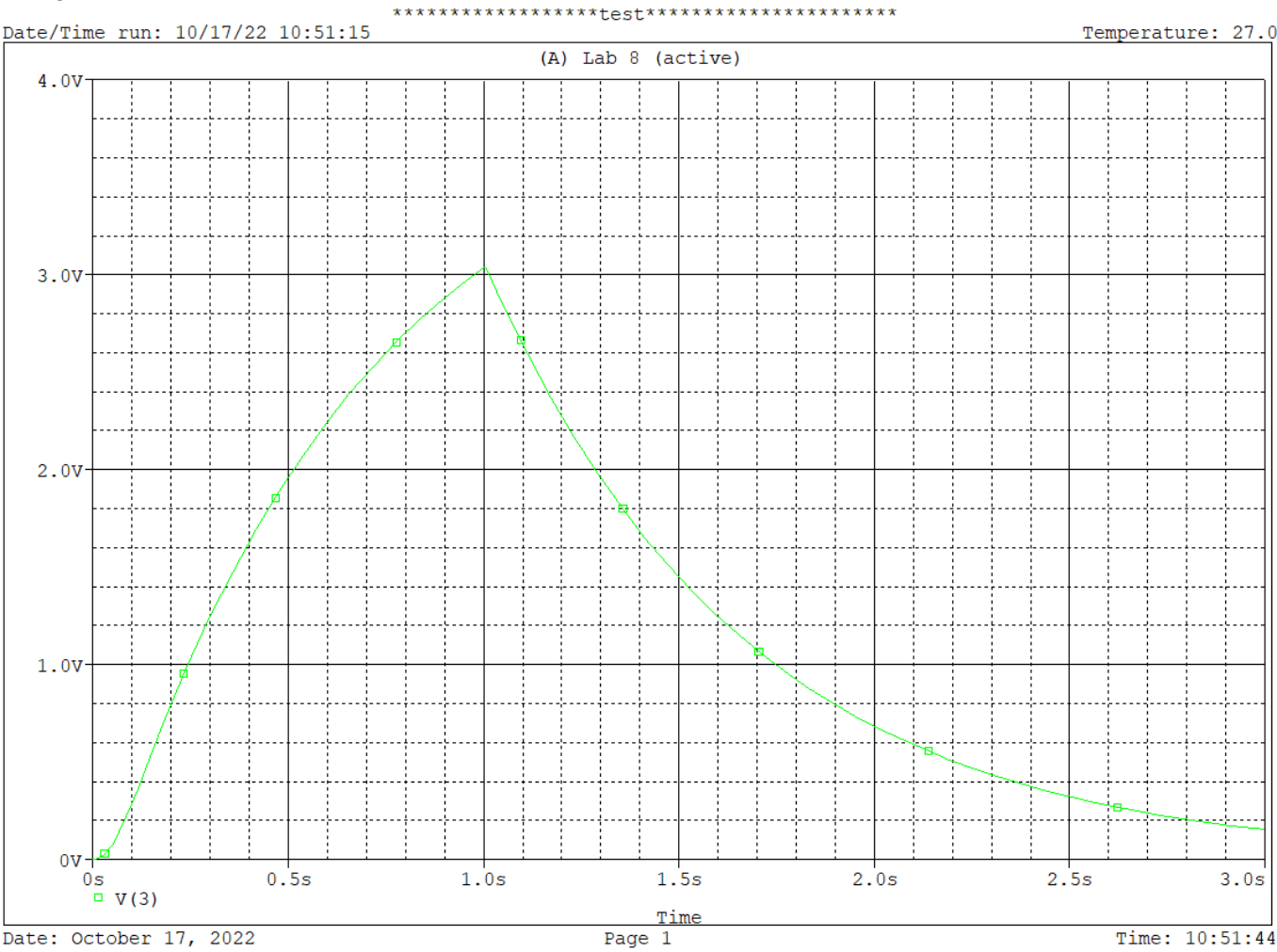
Circuit 1:

Code used

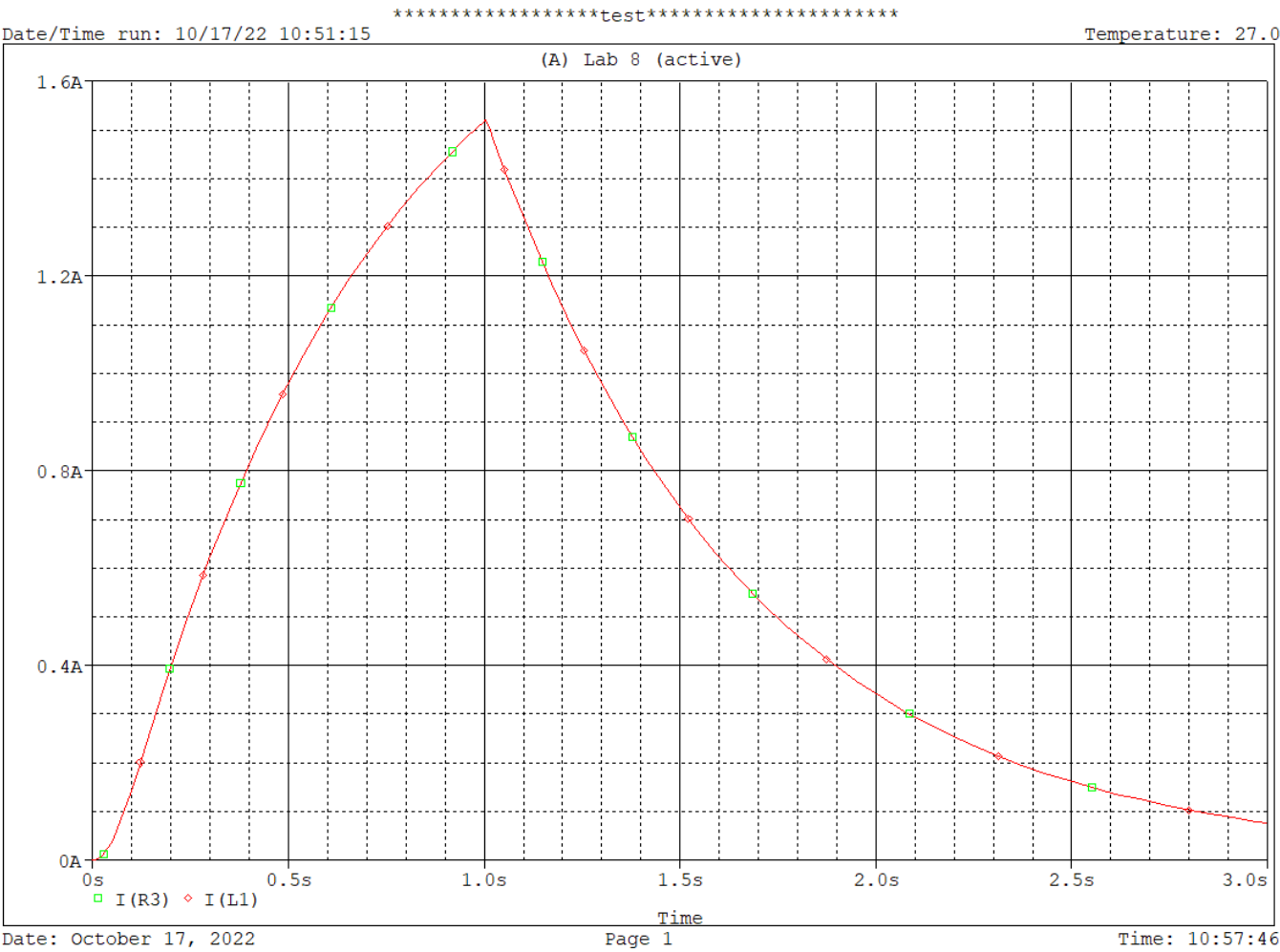
The image shows the PSpice A/D software interface. The title bar reads "Lab 8 - PSpice A/D - [Lab 8 (active)]". The menu bar includes "File", "Edit", "View", "Simulation", "Trace", and "Plot". The toolbar contains icons for creating a new file, opening a file, saving, printing, cutting, and pasting. Below the toolbar, there are two tabs labeled "Lab 8 (active)". The main workspace displays a circuit simulation code in a text editor. The code is as follows:

```
1  test
2  Vs 1 0 PwL(0 0 0.1 12 1 12 1.01 0)
3  L1 2 3 2
4
5  R1 1 2 2
6  R2 2 0 2
7  R3 3 0 2
8
9
10 .Tran 1s 3s
11 .PLOT TRAN V(1)
12 .PROBE
13 .END
14 |
```

Voltage trace

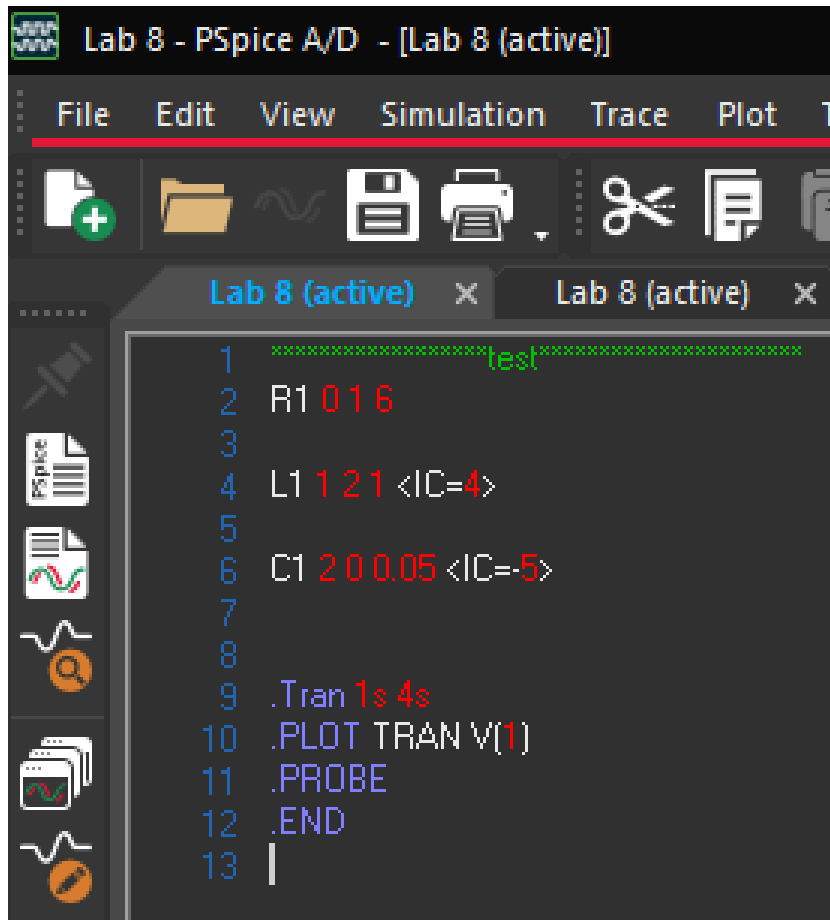


Current traces:



Circuit 2:

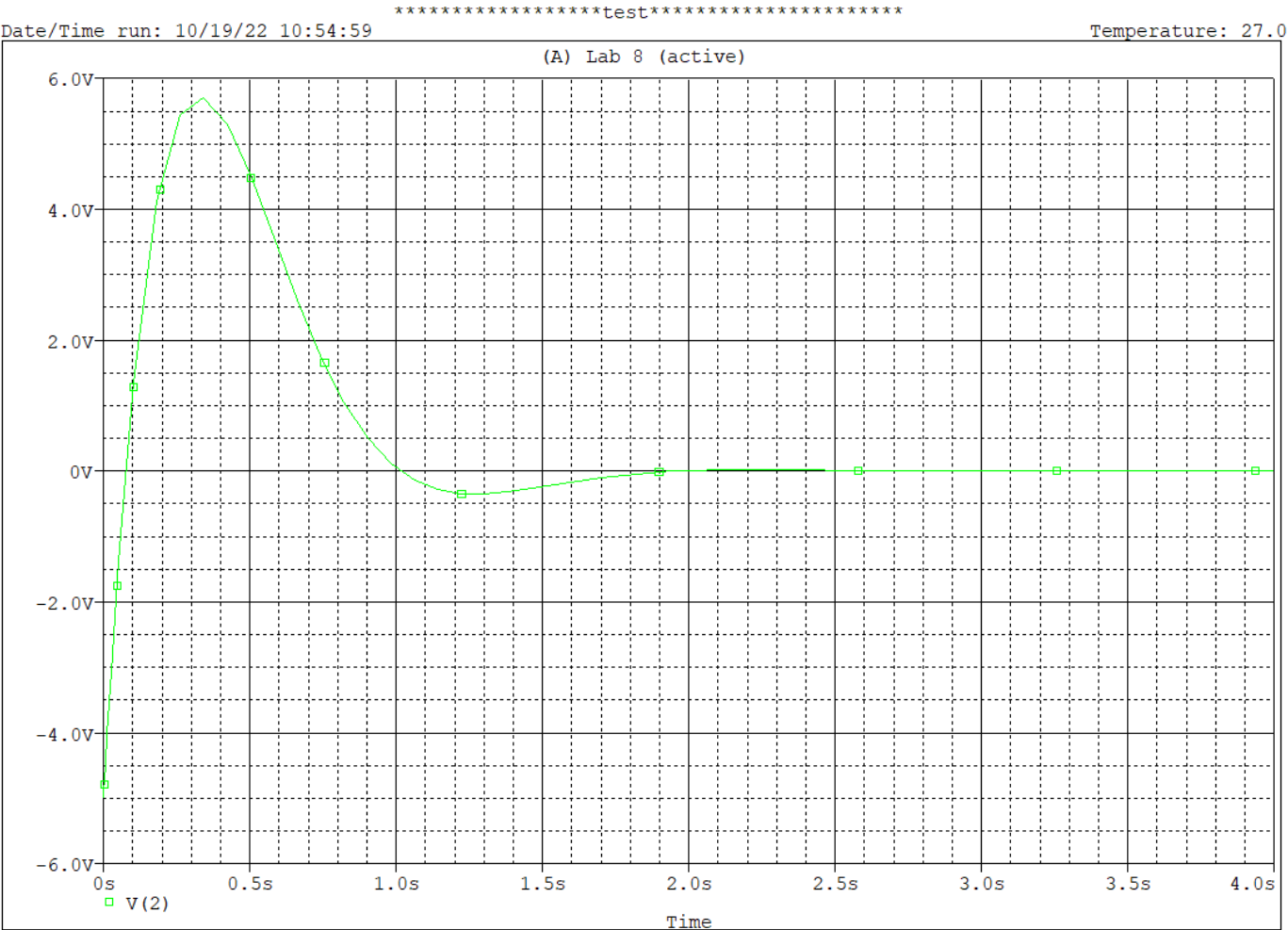
Code used



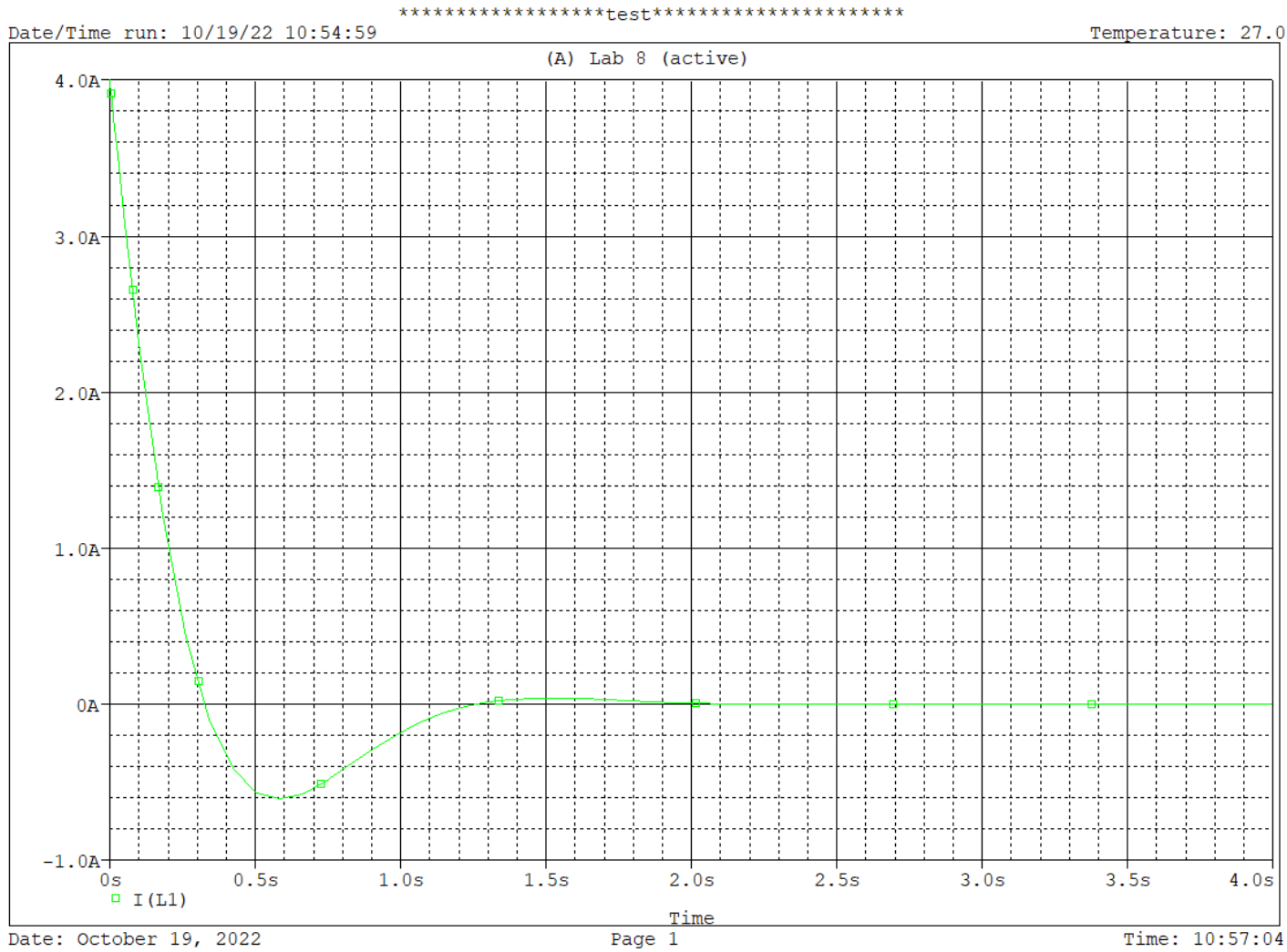
The screenshot shows the PSpice A/D software interface. The title bar reads "Lab 8 - PSpice A/D - [Lab 8 (active)]". The menu bar includes "File", "Edit", "View", "Simulation", "Trace", "Plot", and "Tools". The toolbar contains icons for creating a new file, opening a file, saving, printing, cutting, copying, and pasting. The workspace shows two tabs, both labeled "Lab 8 (active)". The active tab displays a circuit simulation code file named "test". The code is as follows:

```
1 *****test*****
2 R1 0 1 6
3
4 L1 1 2 1 <IC=4>
5
6 C1 2 0 0.05 <IC=-5>
7
8
9 .Tran 1s 4s
10 .PLOT TRAN V(1)
11 .PROBE
12 .END
13 |
```


Voltage trace:



Current trace:



Conclusion

In this lab, I learned how to find the voltage and current of a first order circuit and a second order circuit. After reviewing the answers obtained from handwritten work and pspice, I can conclude the answers concur and are correct.