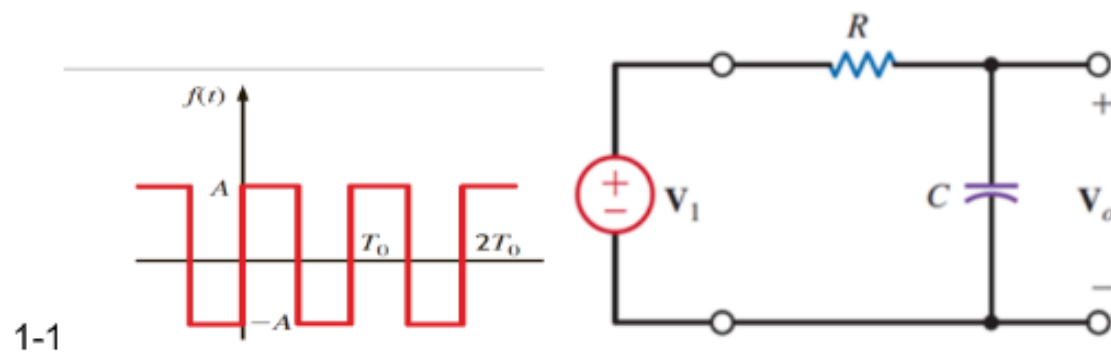


Daniel Delgado Acosta  
Professor Duck Chung  
CSE 4030  
November 30th, 2022

## **Lab 14: Fourier Transform**

### **Introduction**

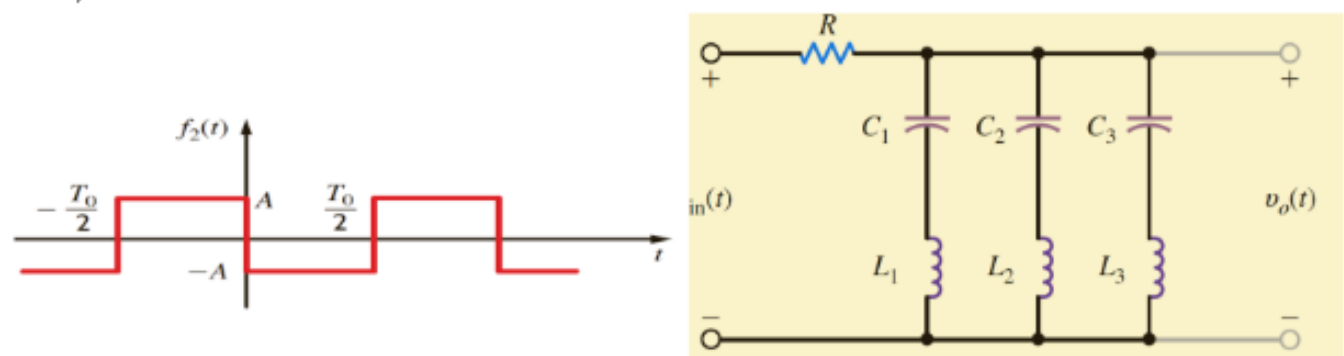
In this lab, we have to find the missing value of each capacitor by using the Fourier transformation method on two different circuits. First, we show our work by hand then use Pspice simulation software to check. The purpose of this lab is to understand how to use Fourier Transform.



### Preparation

- Find  $f(t)$   $\frac{a_0}{2} + \sum_{n=1}^{\infty} \left[ a_n \cos 2n\pi \frac{t}{T} + b_n \sin 2n\pi \frac{t}{T} \right]$   
by using Fourier Transform.( calculate by hand).  
1-1 A = 10V,  $T_0 = 10\text{ms}$ ,  $R = 1\text{K}\Omega$ .
- Find the value of C to keep on fundamental term and remove kth harmonic term
- By using pspice simulation, find  $v_o(t)$ .  
Vi N1 N2 PULSE(10 -10 0 2n 2n 5m 10m) (V1 V2 TD TR TF PW PER)

1-2 ,

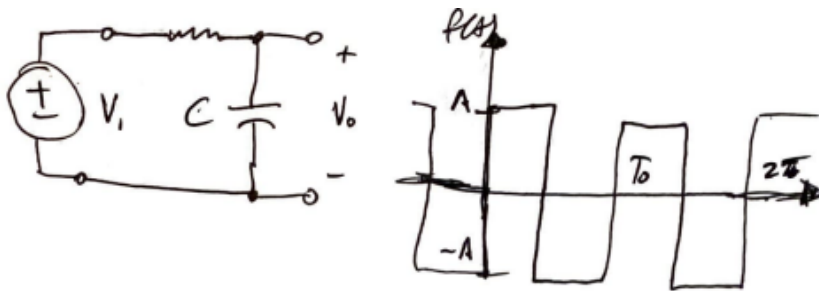


### Preparation

- Find  $f(t)$   $\frac{a_0}{2} + \sum_{n=1}^{\infty} \left[ a_n \cos 2n\pi \frac{t}{T} + b_n \sin 2n\pi \frac{t}{T} \right]$   
by using Fourier Transform.( calculate by hand).  
1-2 A = 10V,  $T_0 = 10\text{ms}$ ,  $R = 1\text{K}\Omega$ ,  $L = 10\mu\text{H}$
- Find the value of and C to keep on fundamental term and remove 3<sup>rd</sup>, 5<sup>th</sup>, 7<sup>th</sup> harmonic term

## Hand Written Work

Circuit 1:



$$\frac{a_0}{2} + \sum_{n=1}^{\infty} \left[ a_n \cos 2\pi n \frac{t}{T} + b_n \sin 2\pi n \frac{t}{T} \right]$$

$$A = 10V, T_0 = 10ms, R = 1k\Omega$$

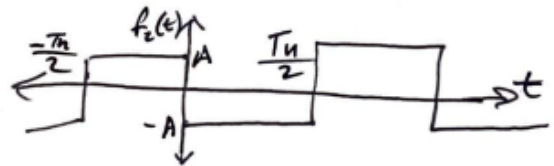
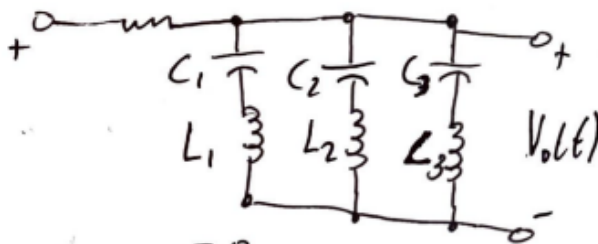
$$\rightarrow f(t) = \sum_{n=-\infty}^{\infty} C_n e^{jn\omega_0 t} \Rightarrow C_n = \frac{2}{T_0} \int_{t_1}^{t_1+T_0} f(t) e^{-jn\omega_0 t} dt \quad \begin{cases} a_n = \frac{2}{T_0} \int_{t_1}^{t_1+T_0} f(t) \cos n\omega_0 t dt \\ b_n = \frac{2}{T_0} \int_{t_1}^{t_1+T_0} f(t) \sin n\omega_0 t dt \end{cases}$$

$$\rightarrow a_n = 0, b_n = \frac{20}{n\pi} [1 - (-1)^n] \rightarrow f(t) = \sum \quad f = \frac{1}{T_0} = \frac{1}{10ms} = 100 Hz \times 2 = 200 Hz$$

$$X_C = \frac{1}{\omega C} = \frac{1}{2\pi f C} \Rightarrow C = \frac{1}{2\pi f X_C} = \frac{1}{2\pi (200)(1000)} \quad \text{~~200000~~$$

$$\Rightarrow \boxed{C \approx 0.8 \mu F}$$

Circuit 2:



$$A = 10V, T = 10ms, R = 0.1\Omega, L = 10\mu H$$

$$\frac{V_o}{V_x} = \frac{R_2}{R_1 + R} \rightarrow \text{if } f = 300\text{ Hz}, 500\text{ Hz}, 700\text{ Hz} \text{ then } V_o = 0$$

$$C_1: \omega L = \frac{1}{\omega C} \Rightarrow \omega^2 LC = 1 \Rightarrow C_1 = \frac{1}{(2\pi(300))^2(10 \times 10^{-6})} = \boxed{0.0281\text{ F}}$$

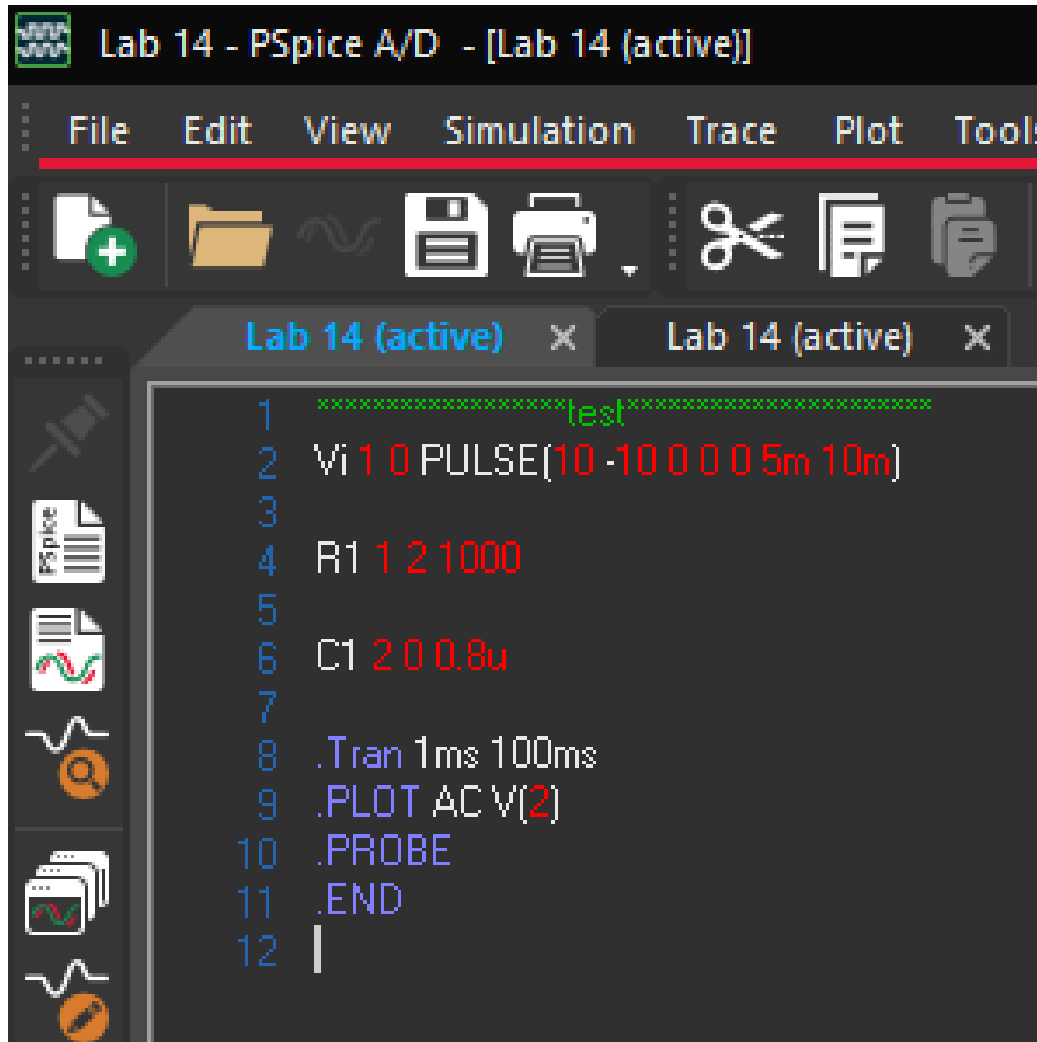
$$C_2: C_2 = \frac{1}{(2\pi(500))^2(10 \times 10^{-6})} = \boxed{0.0101\text{ F}}$$

$$C_3: C_3 = \frac{1}{(2\pi(700))^2(10 \times 10^{-6})} = \boxed{0.00517\text{ F}}$$

## Pspice simulation

Circuit 1:

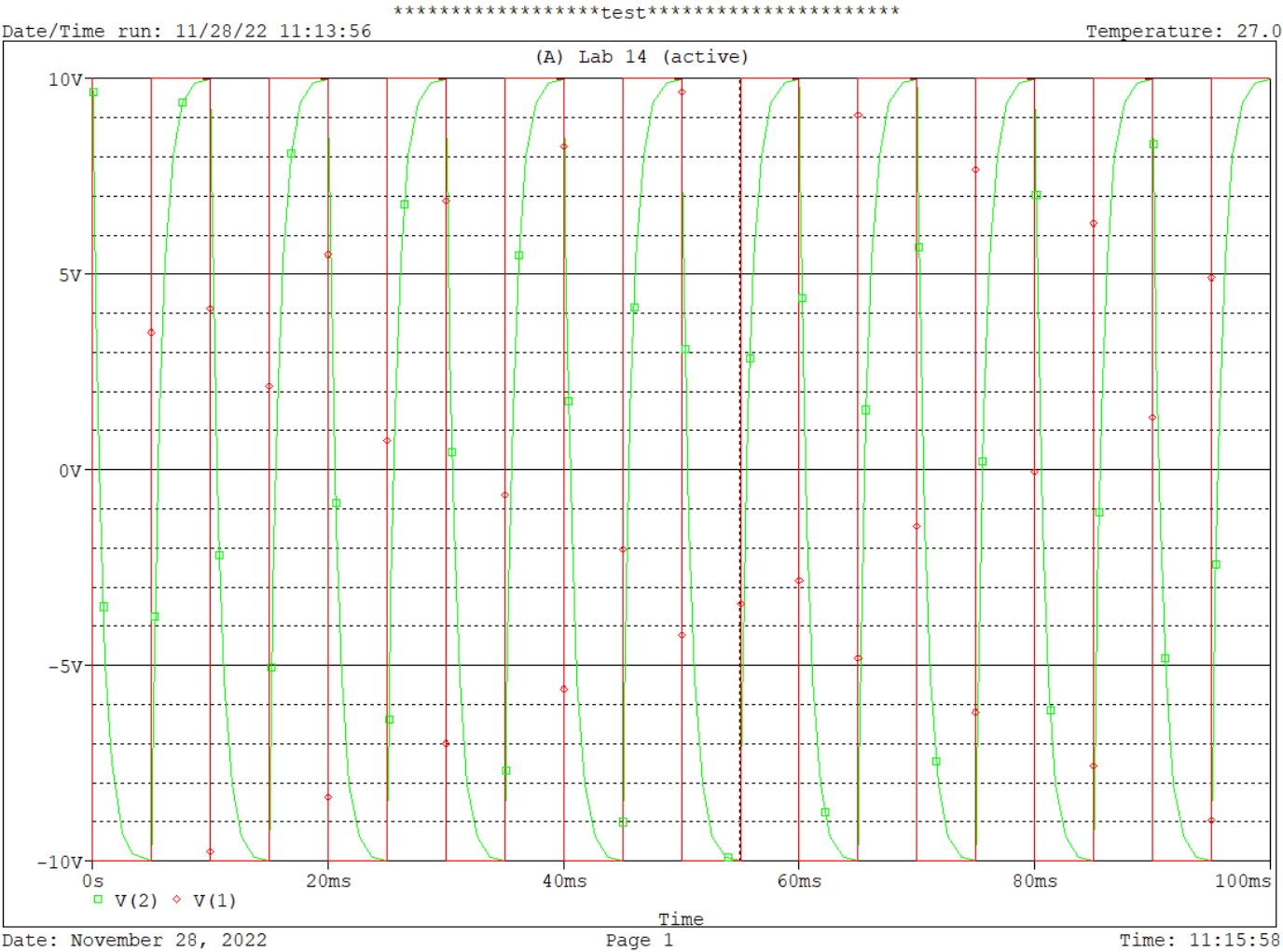
Code used



The screenshot shows the PSpice A/D software interface. The title bar reads "Lab 14 - PSpice A/D - [Lab 14 (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 a file, printing, cutting, copying, and pasting. The workspace area shows two tabs, both labeled "Lab 14 (active)". The active tab displays a circuit simulation code in a text editor. The code is as follows:

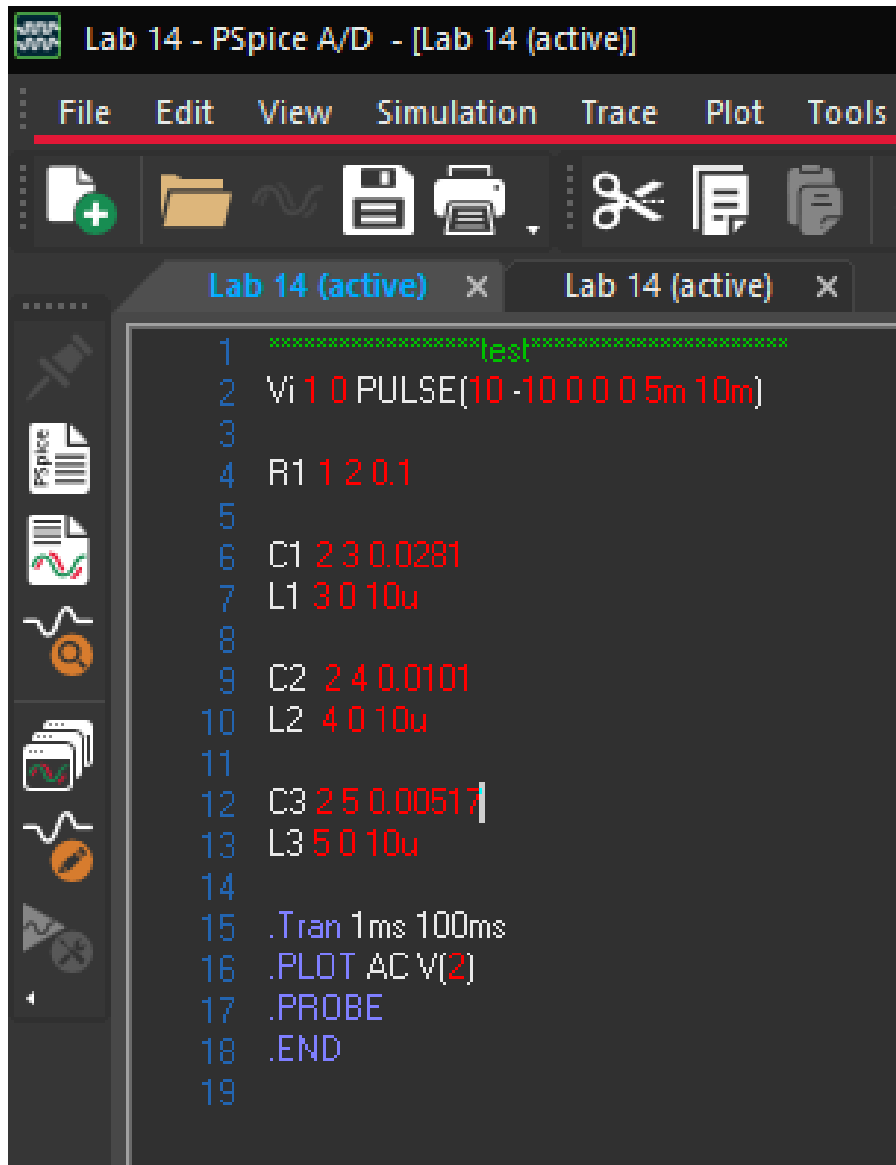
```
1  test
2  Vi 1 0 PULSE(10 -10 0 0 5m 10m)
3
4  R1 1 2 1000
5
6  C1 2 0 0.8u
7
8  .Tran 1ms 100ms
9  .PLOT AC V(2)
10 .PROBE
11 .END
12 |
```

Voltage plot:  $C = 0.8\mu F$



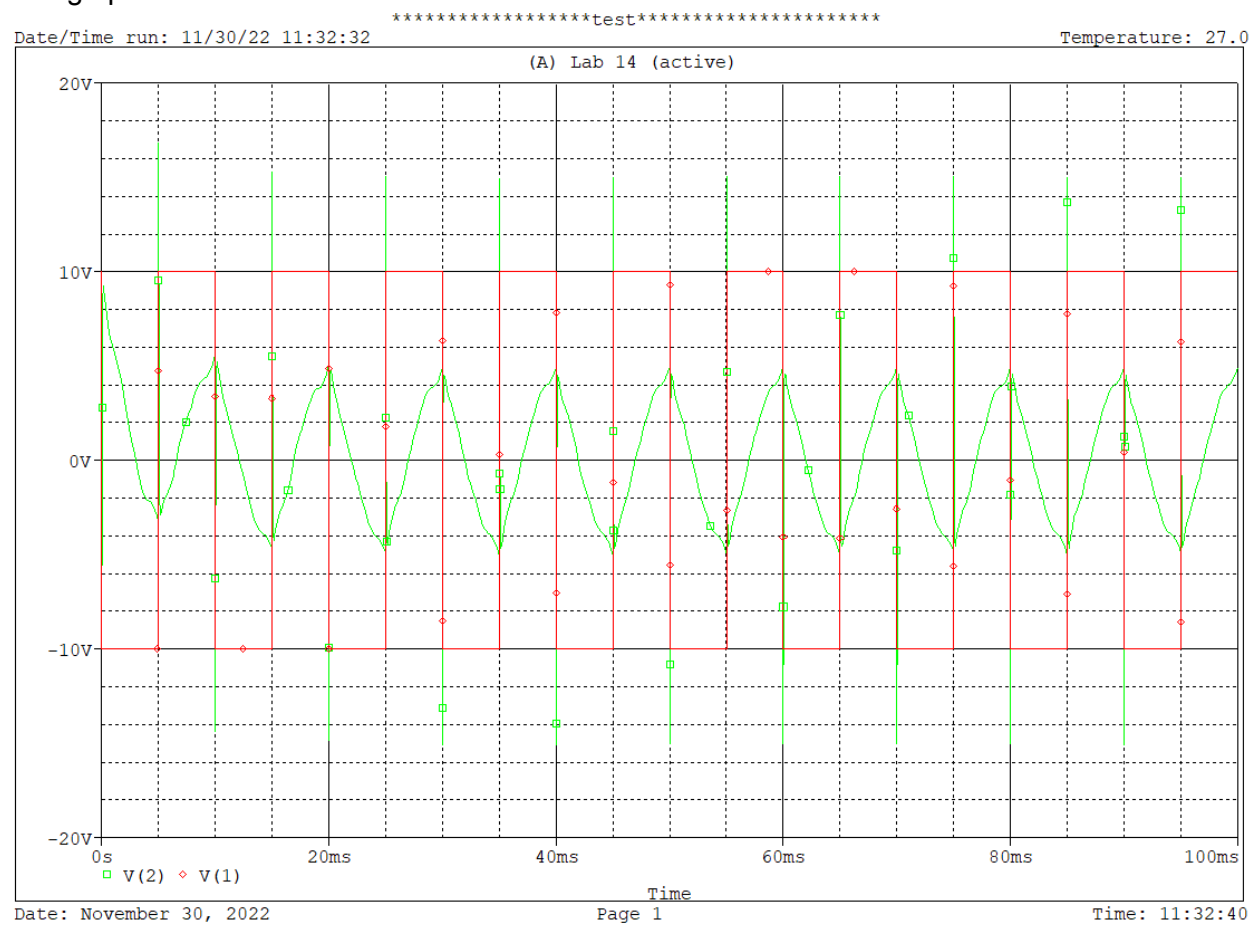
Circuit 2:

Code used

The image shows the LTSPICE software interface. At the top is a title bar that reads "Lab 14 - PSpice A/D - [Lab 14 (active)]". Below this is a menu bar with the following options: File, Edit, View, Simulation, Trace, Plot, and Tools. Underneath the menu bar is a toolbar containing icons for opening a file, saving, printing, cutting, copying, and pasting. Below the toolbar are two tabs, both labeled "Lab 14 (active)". On the left side of the main workspace is a vertical toolbar with icons for various simulation and analysis tools. The main workspace contains a list of circuit components and simulation commands, numbered 1 through 19. The components include a voltage source (Vi), a resistor (R1), and three capacitors (C1, C2, C3), along with three inductors (L1, L2, L3). The simulation commands include a transient analysis (.Tran), a plot command (.PLOT), a probe command (.PROBE), and an end command (.END).

```
1  *****test*****
2  Vi 1 0 PULSE(10 -10 0 0 0.5m 10m)
3
4  R1 1 2 0.1
5
6  C1 2 3 0.0281
7  L1 3 0 10u
8
9  C2 2 4 0.0101
10 L2 4 0 10u
11
12 C3 2 5 0.00517
13 L3 5 0 10u
14
15 .Tran 1ms 100ms
16 .PLOT AC V(2)
17 .PROBE
18 .END
19
```

Voltage plot:





## **Conclusion**

In this lab, I learned how to find the missing capacitor values of two different circuits by applying fourier transformations. After reviewing the answers obtained from handwritten work and pspice, I can conclude the answers concur and are correct.