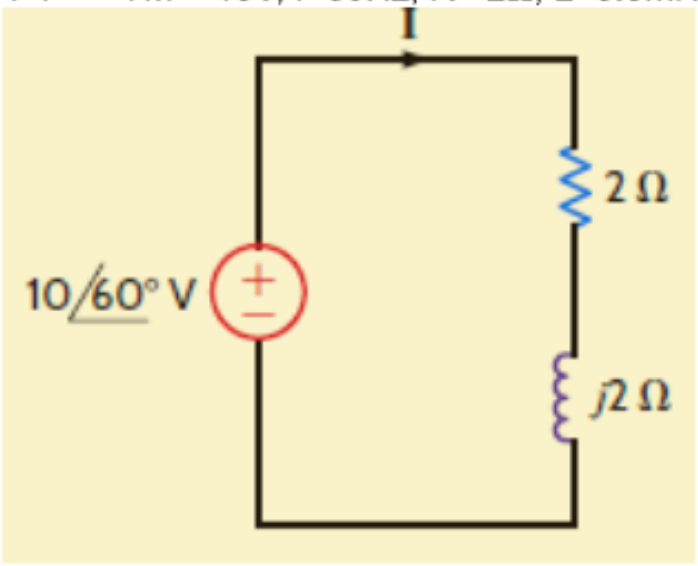
Daniel Delgado Acosta
Professor Duck Chung
CSE 4030
November 2nd, 2022

Lab 10: AC Steady State Power Analysis

Introduction

In this lab, we have to find the voltage, current, and power of two circuits that consist of resistors and inductors. First, we show our work by hand then use Pspice simulation software to check. The purpose of this lab is to understand the instantaneous and average power in ac circuits and compute the real power, reactive power, and power factor in an ac circuit.

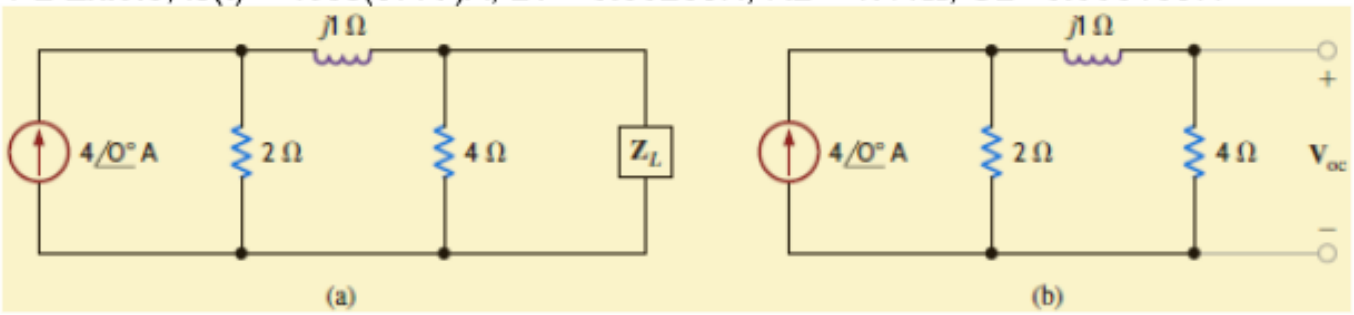
1-1 VM = 10V, f=60Hz, R= 2Ω, L=5.3mH.(Ex. 9.2)



Preparation

1. Find v(t), vL(t), i(t),and P_L. (.calculate by hand)
2. By using pspice simulation, find v(t),vL(t), i(t), and PL.

1-2 Ex.9.5, is(t) = 4cos(377t)A, L1 = 0.00265H, RL = 1.41Ω, CL =0.0061687F



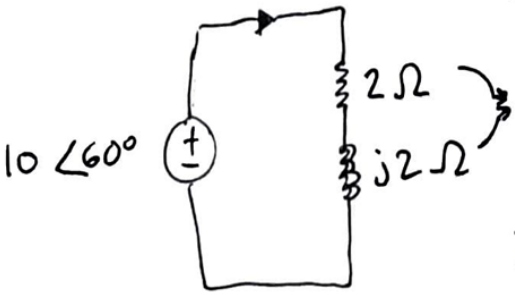
Preparation

1. Find vL(t), iL(t) and PL (calculate by hand),
2. By using pspice, find pL(t) when RL=1.41Ω and 5Ω.

Hand Written Work

Circuit 1:

$V_m = 10\,V,\ f = 60\,Hz,\ R = 2\,\Omega,\ L = 5.3\,mH$



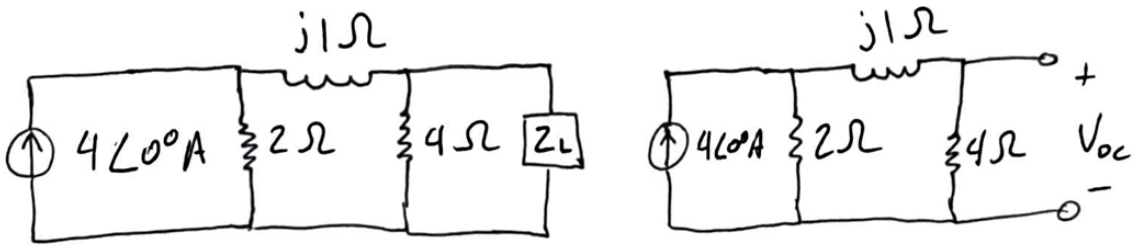
$Z = 2 + 2j = \sqrt{2^2+2^2}\ \angle \tan^{-1}\left(\frac{2}{2}\right)$

$= 2.83 \angle 45°$

$I_m = \frac{V_m}{Z} = \frac{10\angle 60°}{2.83\angle 45°} = \underline{3.53 \angle 15°\ A}$

$V_L = 2\,I_m = \underline{7.07 \angle 15°\ V} \Rightarrow P_L = \frac{1}{2}(7.07)(3.53) = \boxed{12.5\,W}$

Circuit 2:



$I_s(t) = 4\cos(377t)\,A,\ L = 0.0026\,SH,\ R_L = 1.41\Omega,\ C_L = 0.0061687\,F$

~~$Z = 6 + j1 = \quad \angle \tan^{-1}\left(\frac{1}{6}\right) \Rightarrow \frac{6-j}{(6+j1)(6-j)} \quad \frac{6-j}{37}$~~

~~$\Rightarrow Z =$~~

$$V_{oc} = \frac{4\angle 0^\circ}{6+j1}(4) = \underline{5.26\angle -9.46^\circ V}$$

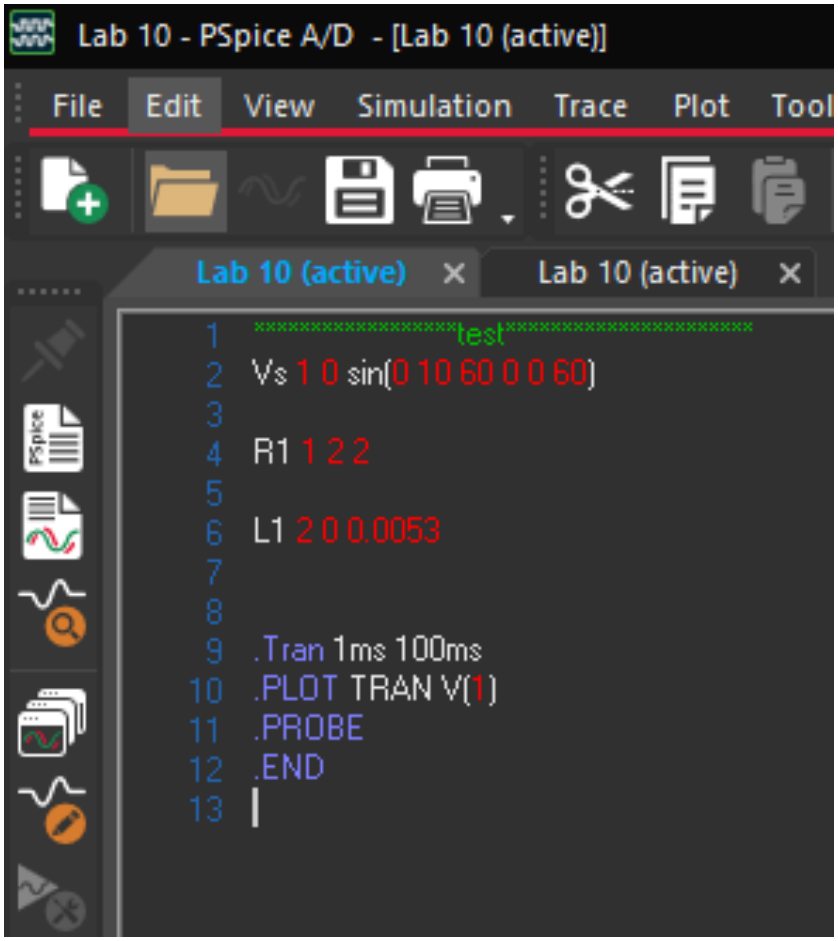
$$Z_{Th} = \frac{4(2+j1)}{6+j1} = 1.91 + j0.43\Omega \Rightarrow Z_L = 1.91 - j0.43\Omega$$

$$I = \frac{5.26\angle -9.46}{2.8\Omega} = \underline{1.87\angle -9.46\,A}$$

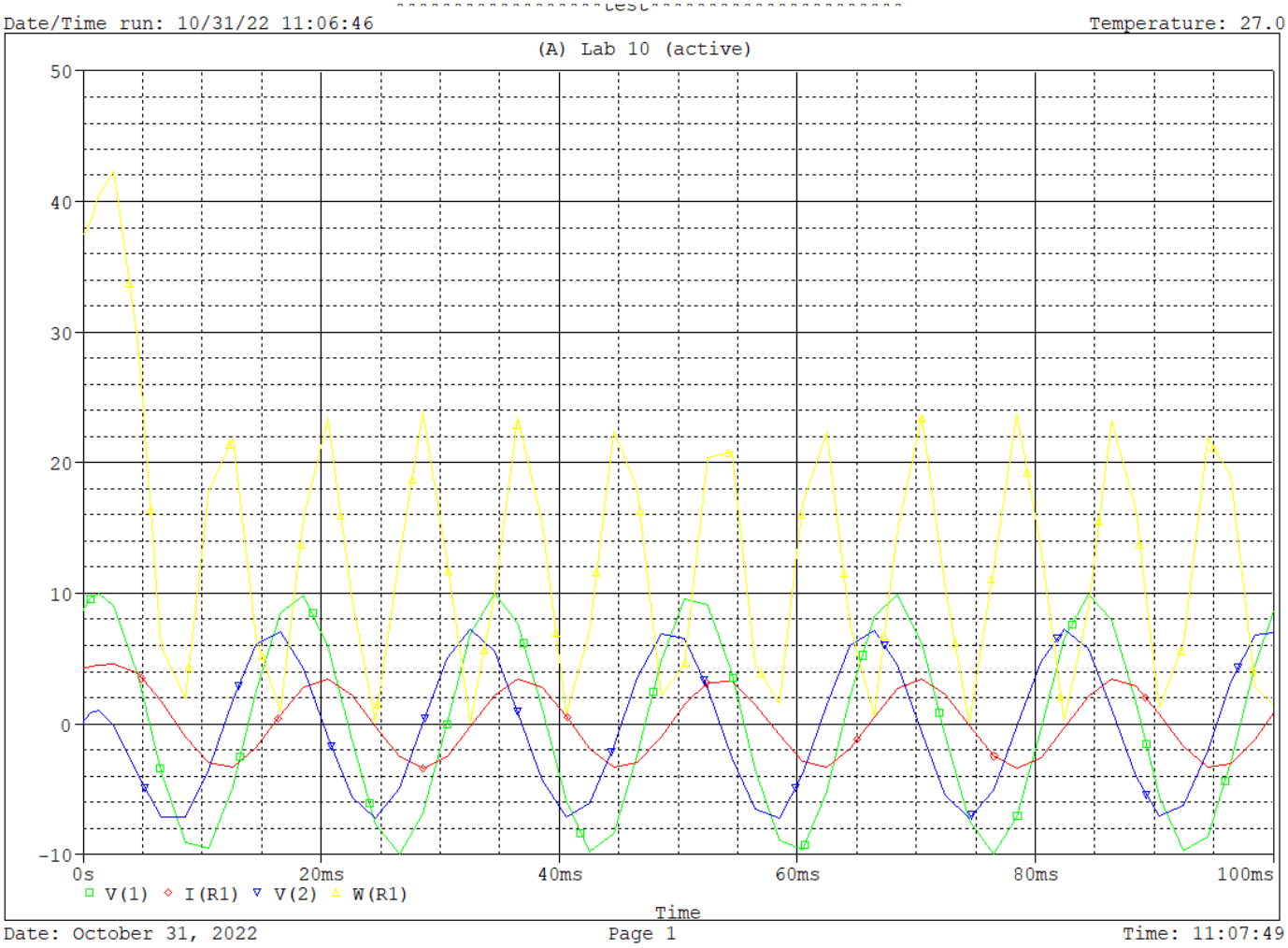
$$\Rightarrow P_L = \frac{1}{2} I_m^2 R_L = \frac{1}{2}(1.87)^2(1.41) = \boxed{2.47W}$$

Pspice simulation

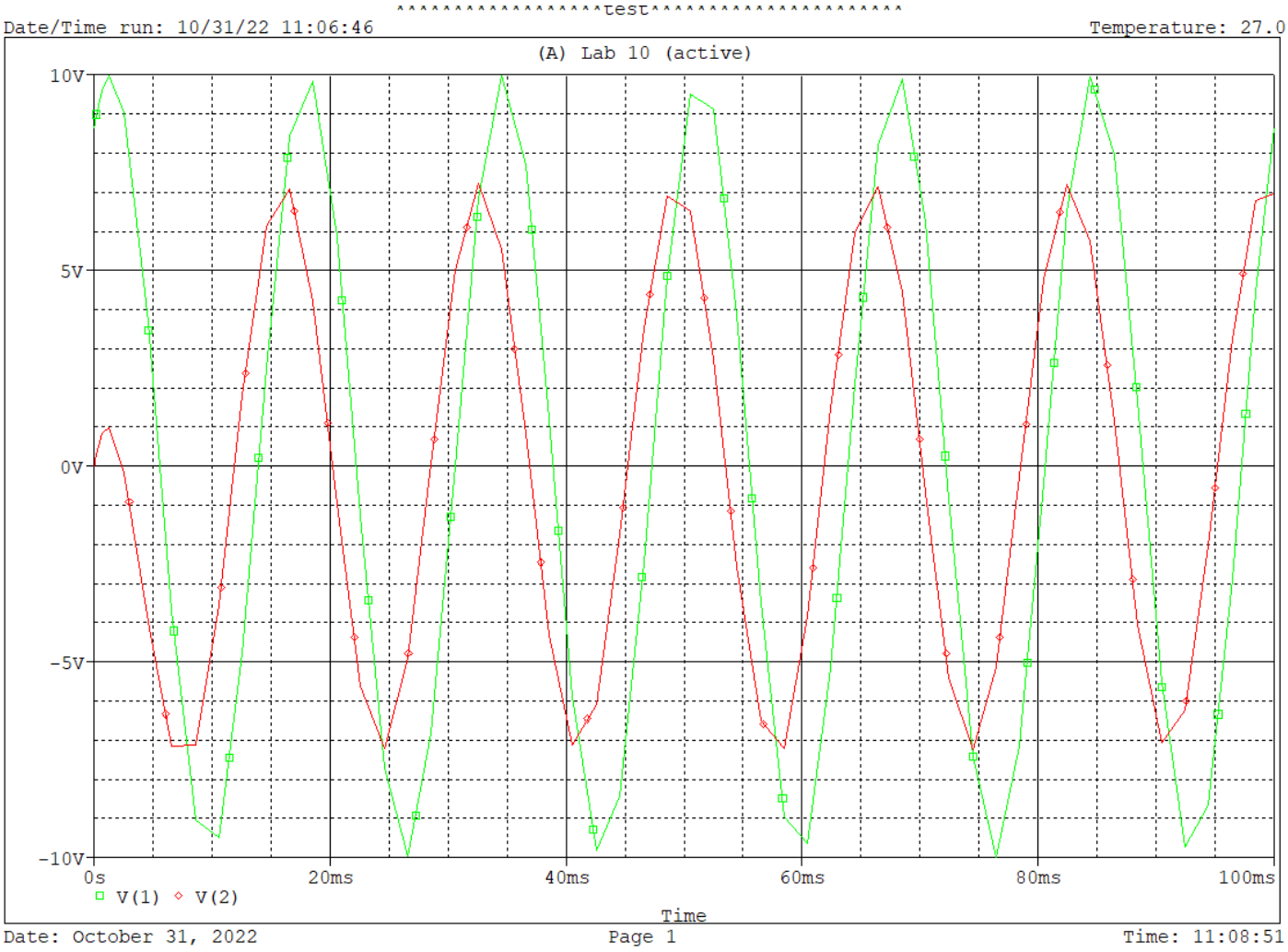
Circuit 1:
Code used



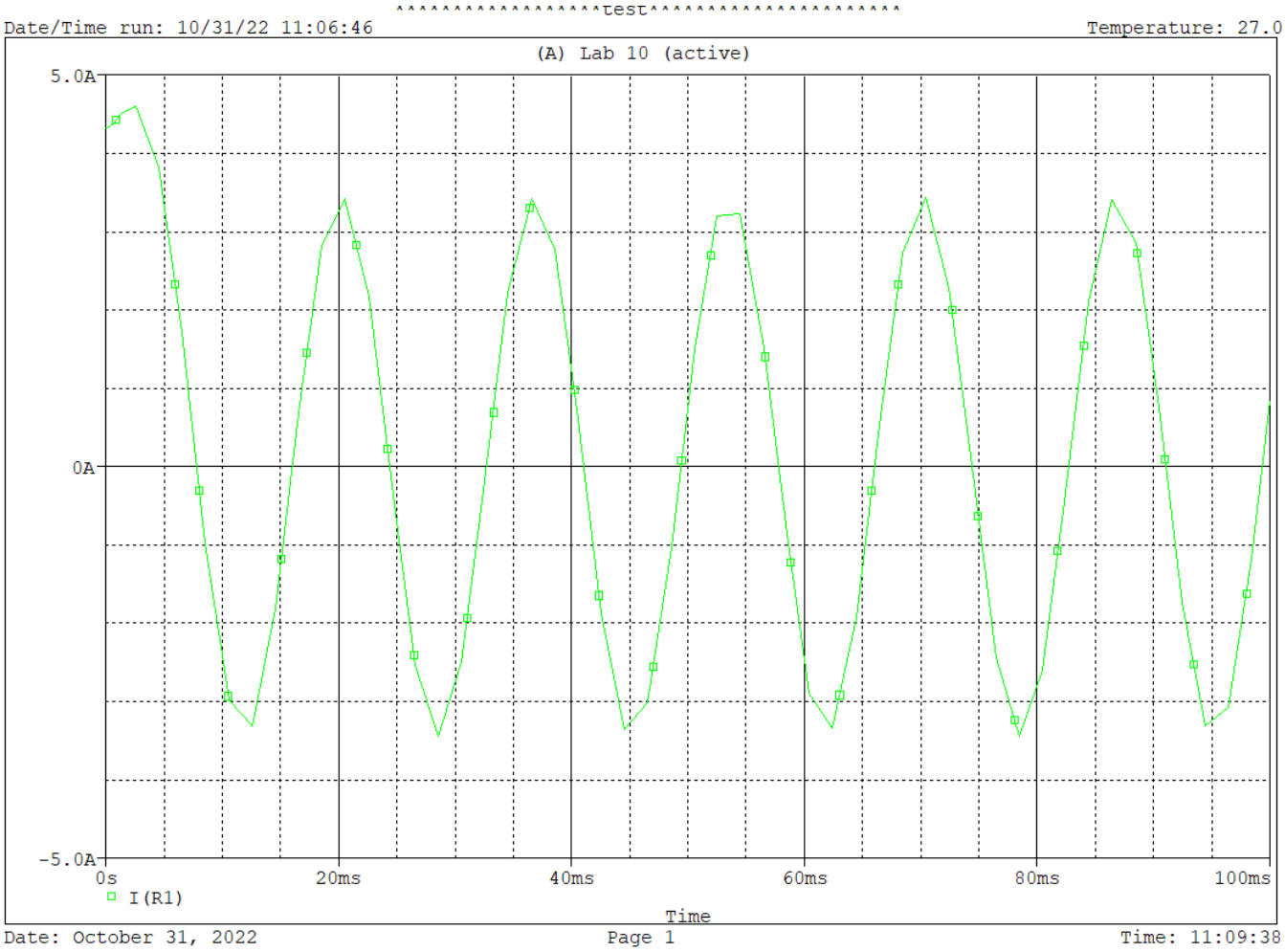
All traces


Date/Time run: 10/31/22 11:06:46
Temperature: 27.0
(A) Lab 10 (active)
50
40
30
20
10
0
-10
0s
20ms
40ms
60ms
80ms
100ms
V(1)
I(R1)
V(2)
W(R1)
Time
Date: October 31, 2022
Page 1
Time: 11:07:49

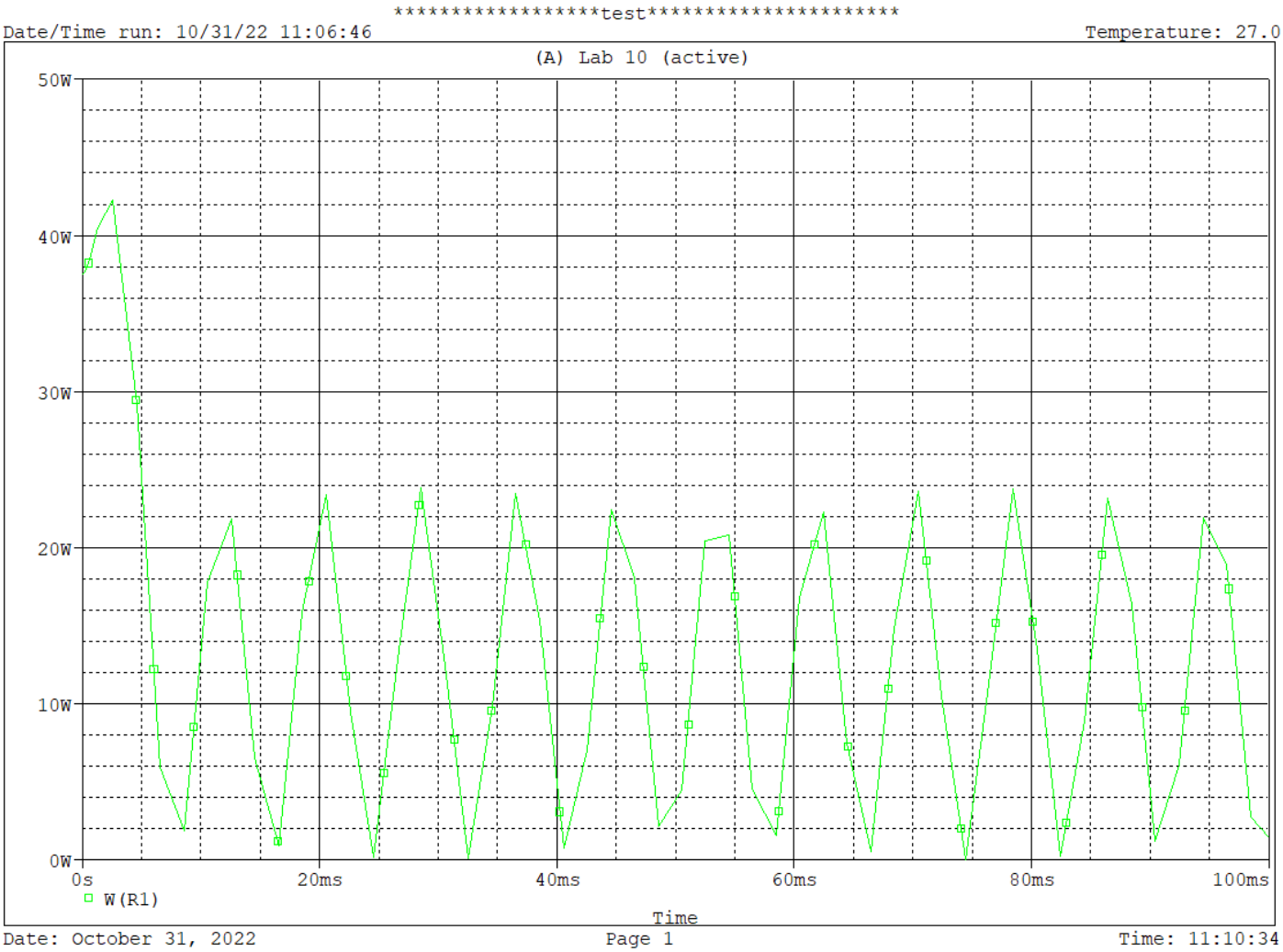
Voltage traces



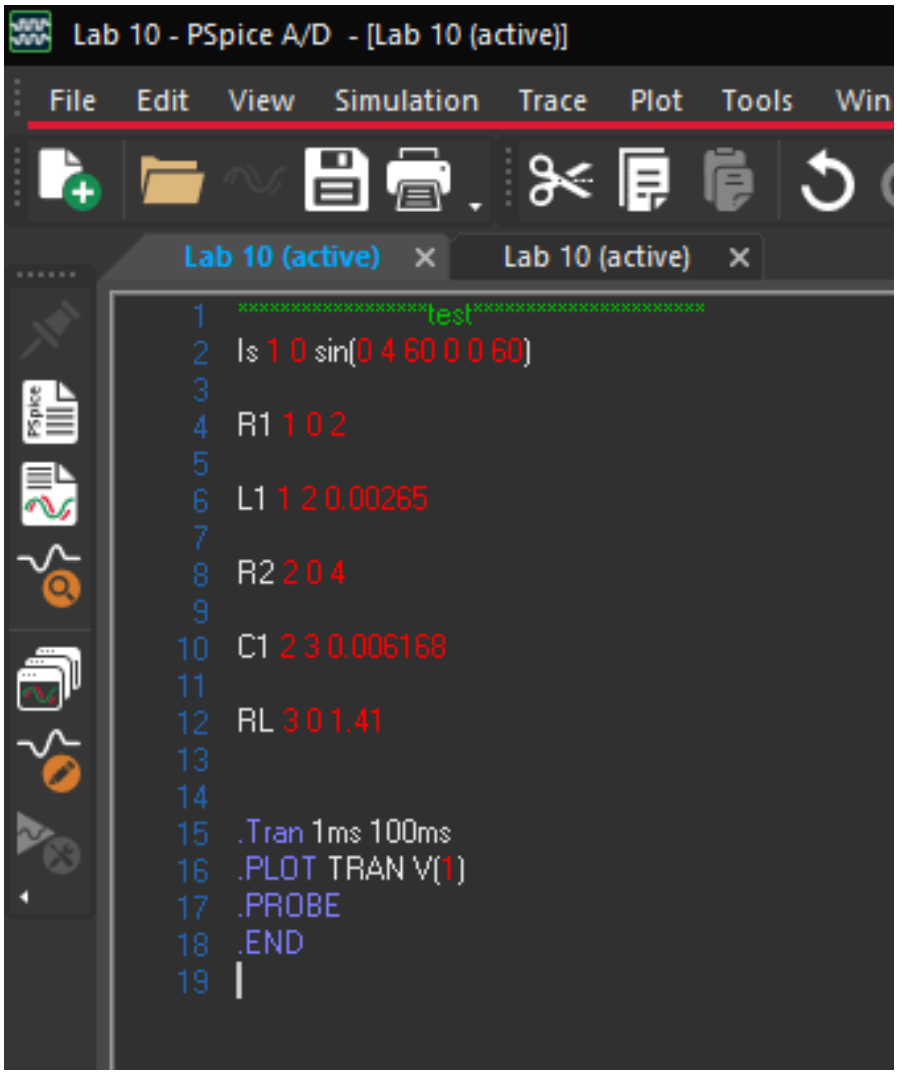
Current trace:



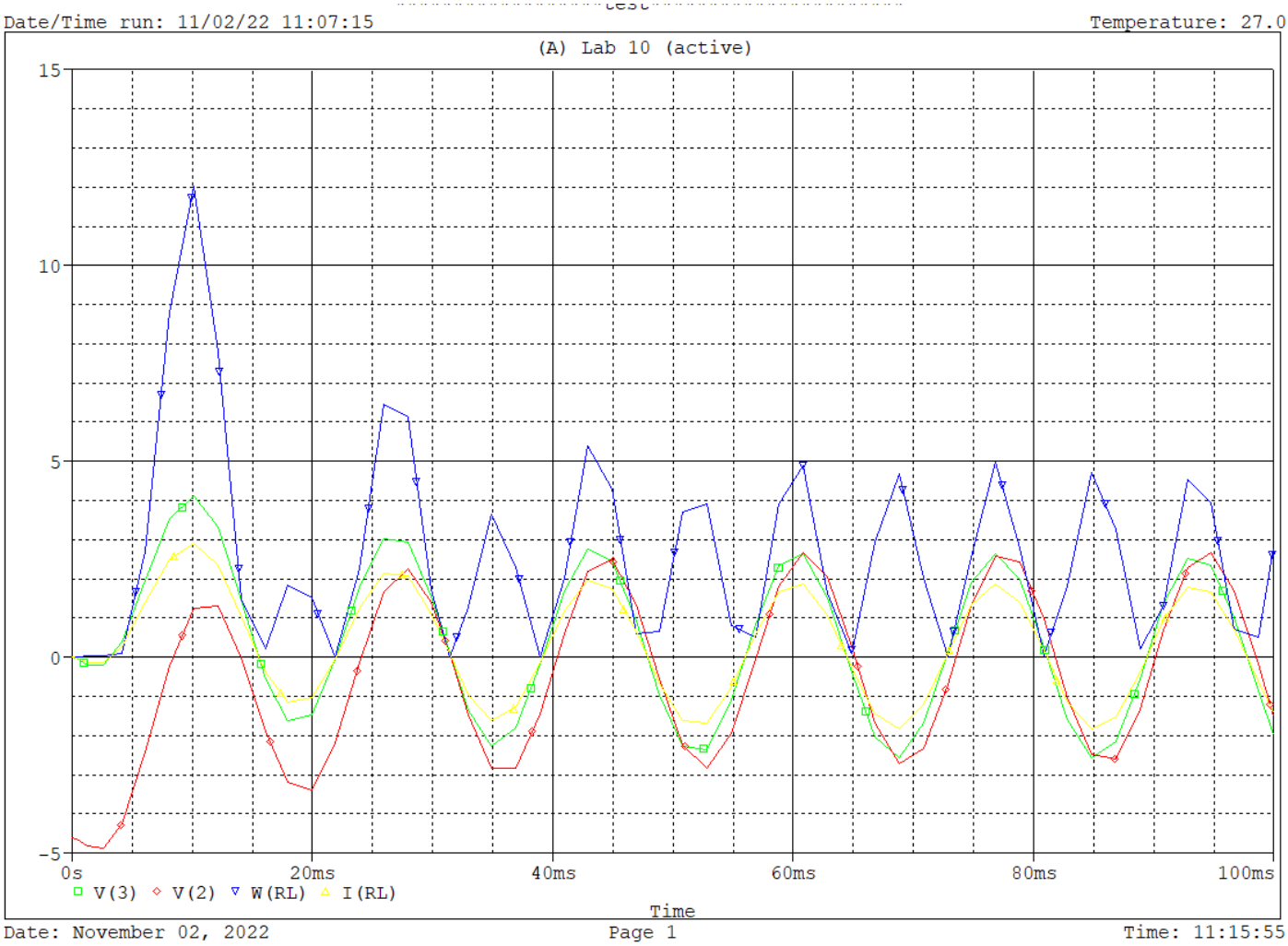
Power trace:


**********test**********
Date/Time run: 10/31/22 11:06:46
Temperature: 27.0
(A) Lab 10 (active)
50W
40W
30W
20W
10W
0W
0s
20ms
40ms
60ms
80ms
100ms
W(R1)
Time
Date: October 31, 2022
Page 1
Time: 11:10:34

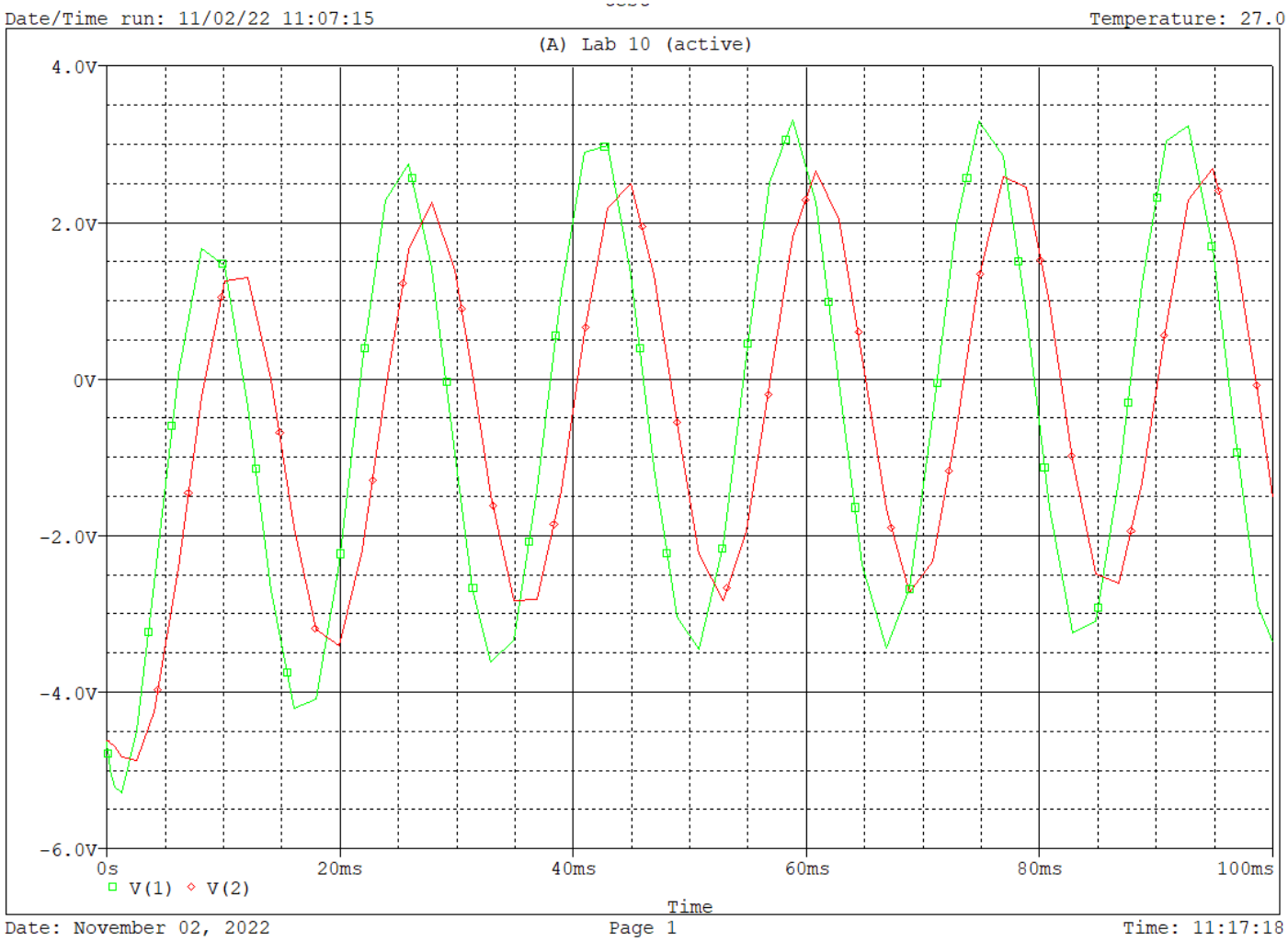
Circuit 2:
Code used



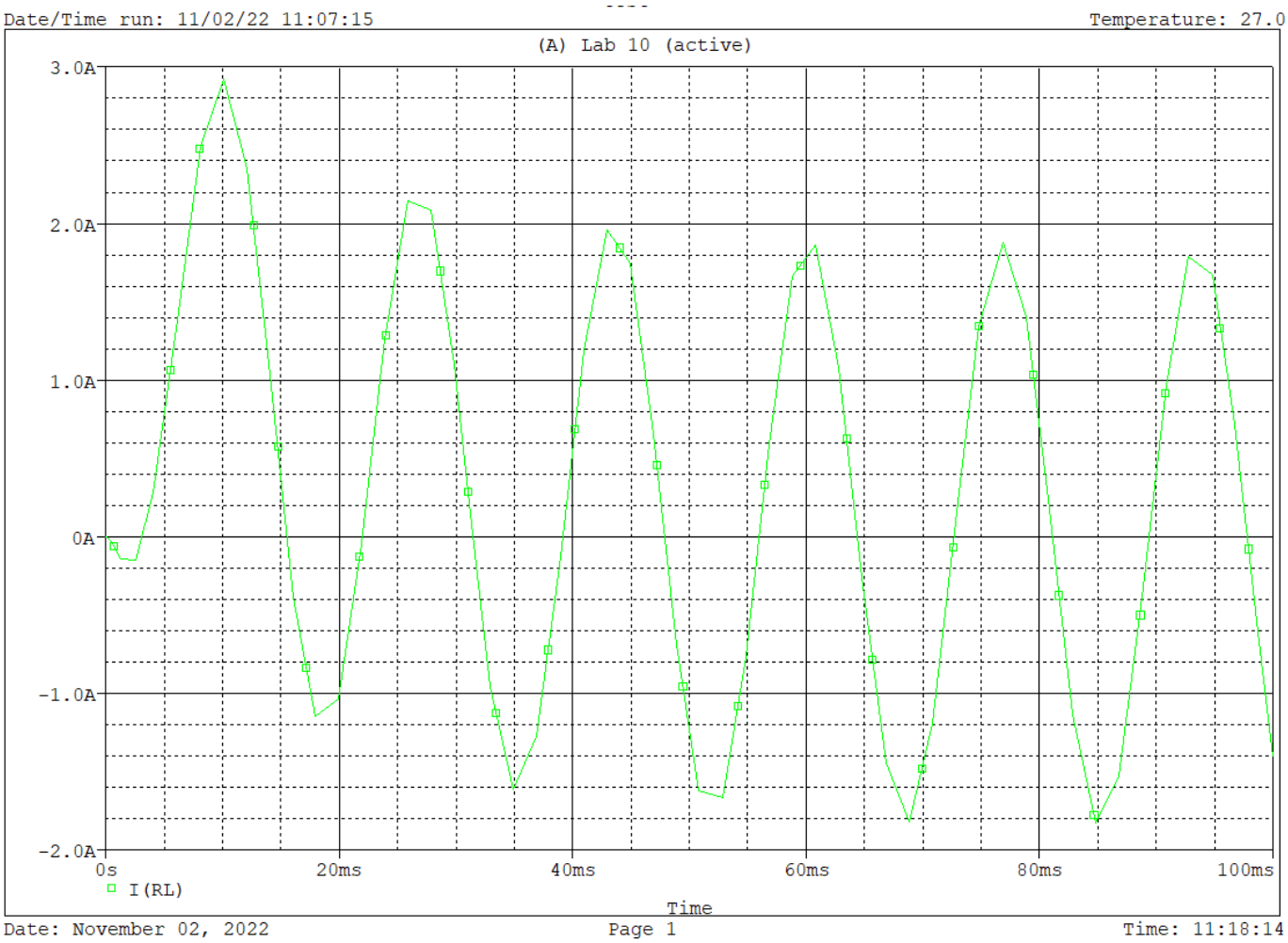
All traces:


Date/Time run: 11/02/22 11:07:15
Temperature: 27.0
(A) Lab 10 (active)
15
10
5
0
-5
0s
20ms
40ms
60ms
80ms
100ms
V(3) V(2) W(RL) I(RL)
Time
Date: November 02, 2022
Page 1
Time: 11:15:55

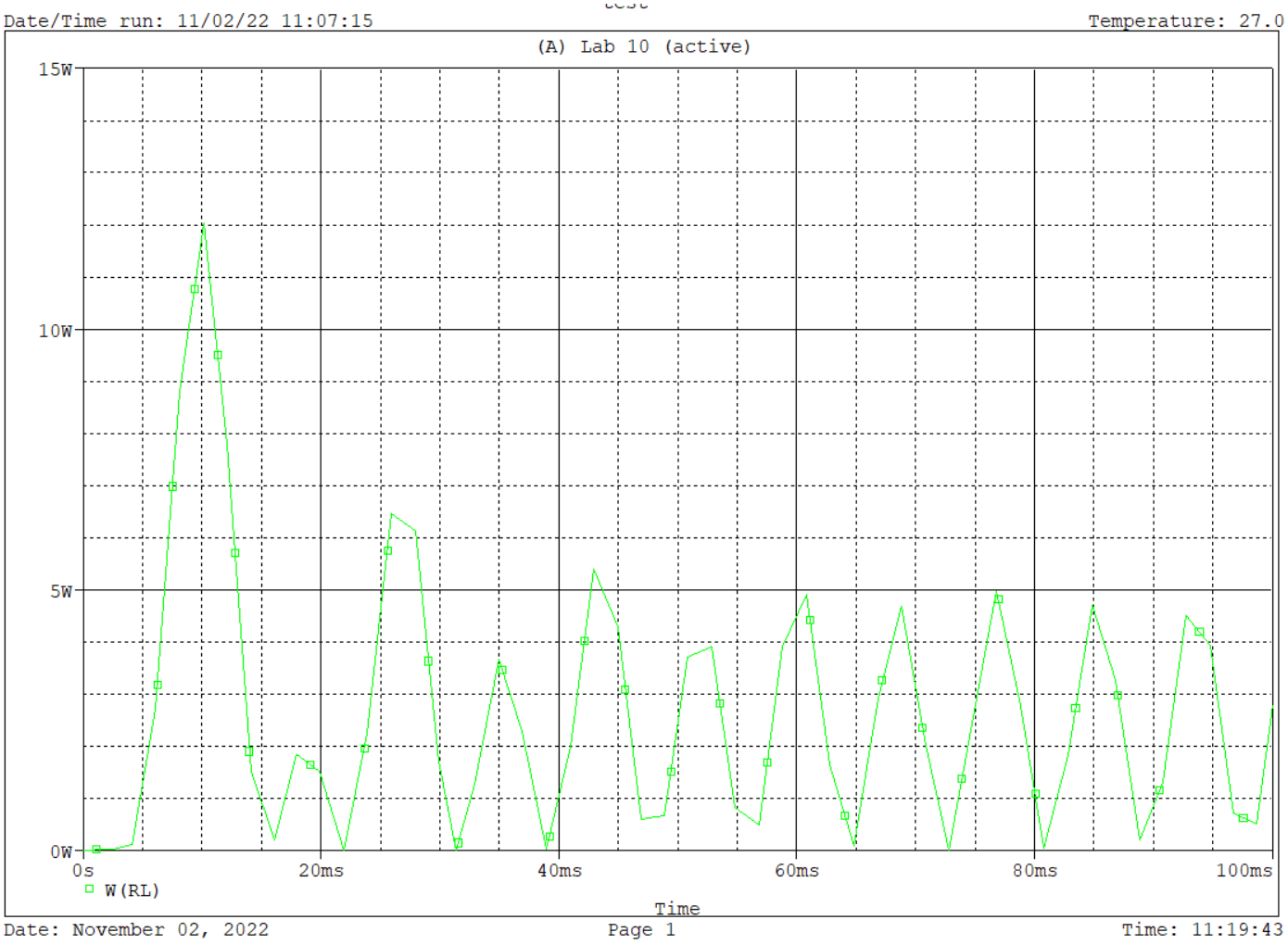
Voltage trace:



Current traces:



Power trace:


Date/Time run: 11/02/22 11:07:15
Temperature: 27.0
(A) Lab 10 (active)
15W
10W
5W
0W
0s
20ms
40ms
60ms
80ms
100ms
W(RL)
Time
Date: November 02, 2022
Page 1
Time: 11:19:43

Conclusion

In this lab, I learned how to find the voltage, current, and power over two different circuits. After reviewing the answers obtained from handwritten work and pspice, I can conclude the answers concur and are correct.