Faculty of Engineering & Technology
Electrical & Computer Engineering Department

CIRCUIT ANALYSIS ENEE2304

The Project

Prepared by:

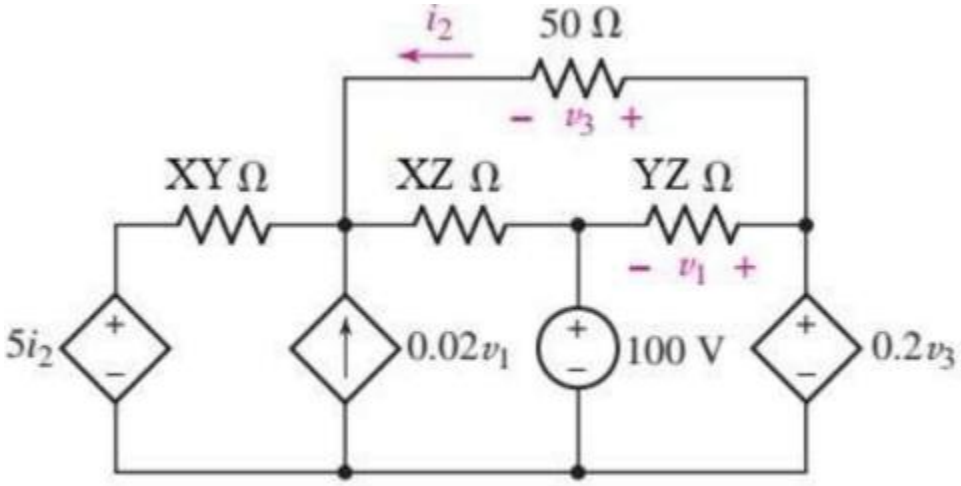
Mohammad Abu Shams 1200549

Instructor: Dr. Hakam Shehadeh

Section: 2

Date: 15-6-2022

Question 1: Construct a PSPICE schematic for the circuit shown in the figure below. Simulate the schematic and show voltages at each node and current in each branch.

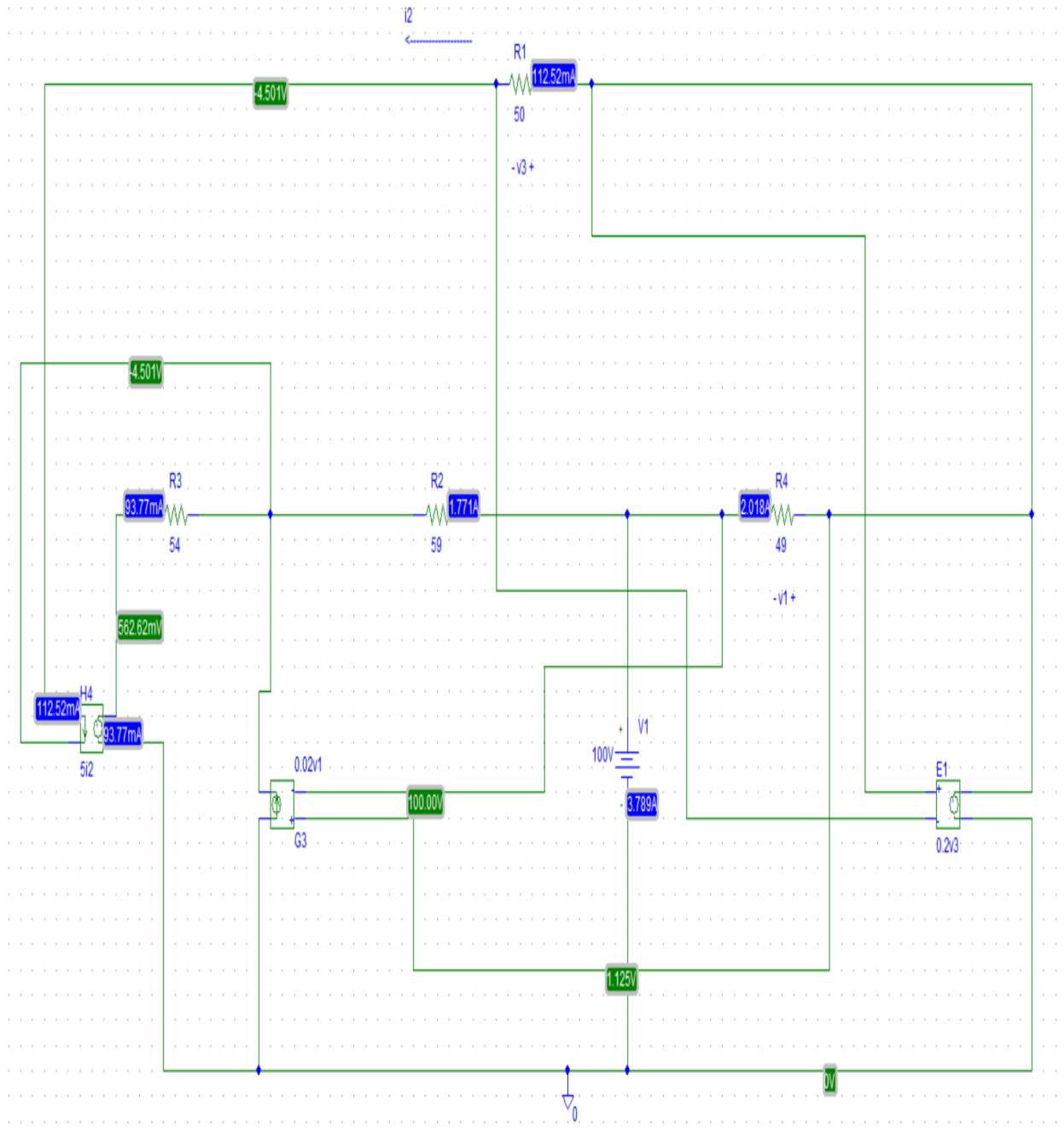


My ID : 1200549

X=5.

Y=4.

Z=9.



In this circuit, I use PSPICE to implement this circuit, I used a dependent voltage source (E1) in this circuit, with a value equal 0.2 v3. In addition, I used a current source with voltage controlled (G3) with a value 0.02 v1, also I used a voltage source with current controlled (H4) with a value 5 i2. When I finished build the circuit, I press on simulate then run this circuit, and the values for voltages and currents appeared in the circuit below.

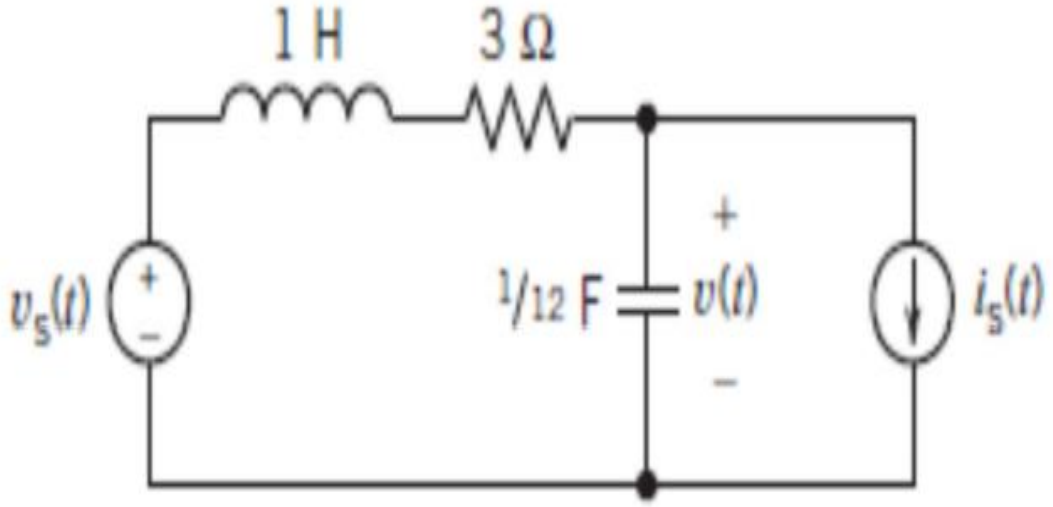
Question 2: The circuit shown in Figure below has two inputs, vs(t) and is(t), and one output, v(t). When inputs are given by Vs(t)= Vm sin 6t V and is(t)= Im A the output will be vo(t)=A sin(6t+θ) + B V 3Linearity requires that A be proportional to Vm and that B be proportional to Im. Consequently, we can write A= k1.Vm and B= k2.Im, where k1 and k2 are constants yet to be determined.

(a) Use PSpice to determine the value of k1 by simulating the circuit, using Vm =1 V and Im= 0.

(b) Use PSpice to determine the value of k2 by simulating the circuit, using Vm =0 V and Im= 1.

(c) Knowing k1 and k2, specify the values of Vm and Im that are required to cause vo(t)=5sin(6t+θ)+5 V. Simulate the circuit, using PSpice to verify the specified values of Vm and Im.

(d) Determine the average power delivered by vs(t) using Pspice.

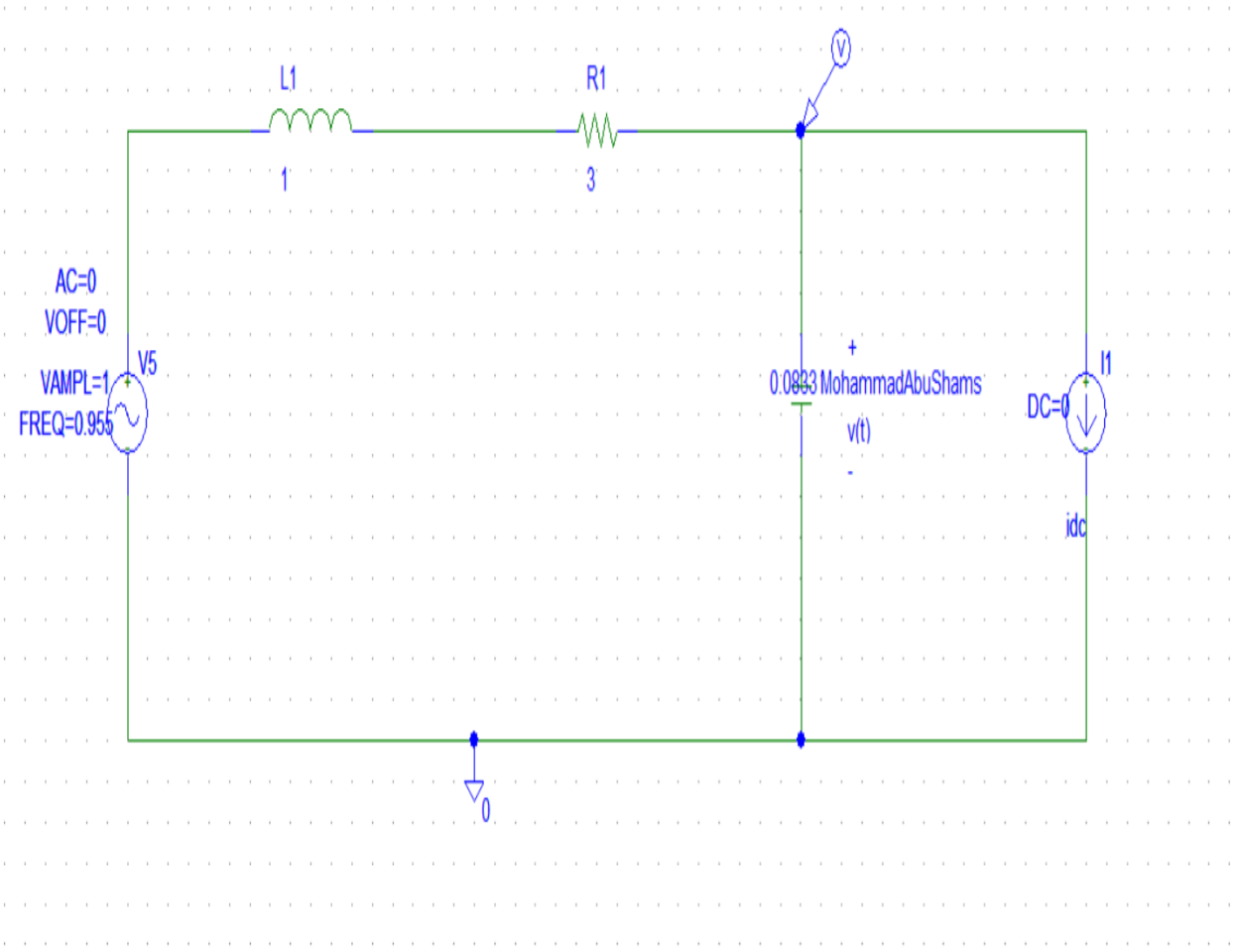


(a) Use PSpice to determine the value of k1 by simulating the circuit, using Vm =1 V and Im= 0.

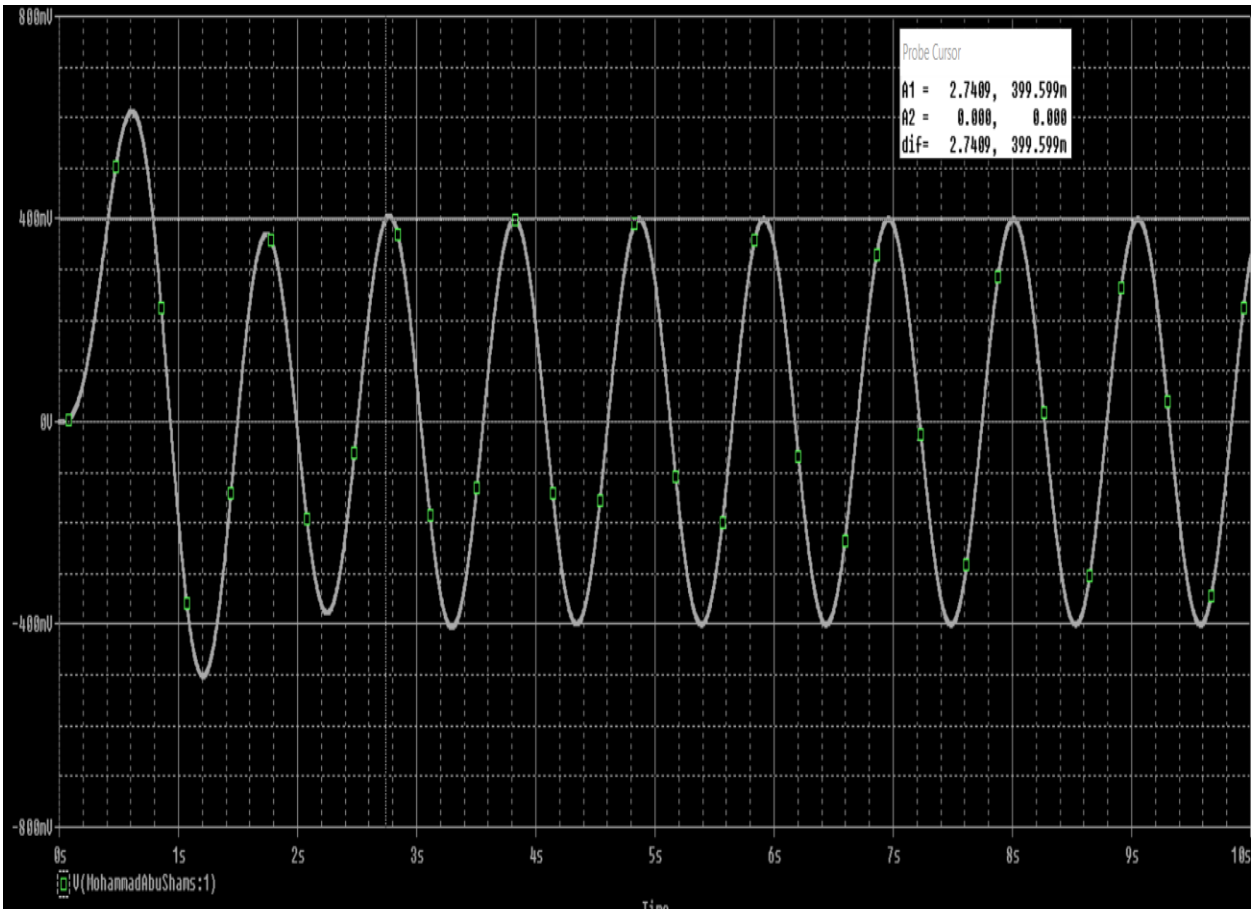
When Vm=1V, and Im =0 A

Is(t)= 0 A ,, and Vs(t)= sin (6t) V , so w=6 rad/s .. and the frequency equal w/(2*pi) =6/(2*3.14)= 0.955 Hz

I built the circuit below



Waves:



According to this diagram

the value of K1 = 399.599m /1

= 0.4

k1=0.4

(b) Use PSpice to determine the value of k2 by simulating the circuit, using Vm =0 V and Im= 1.

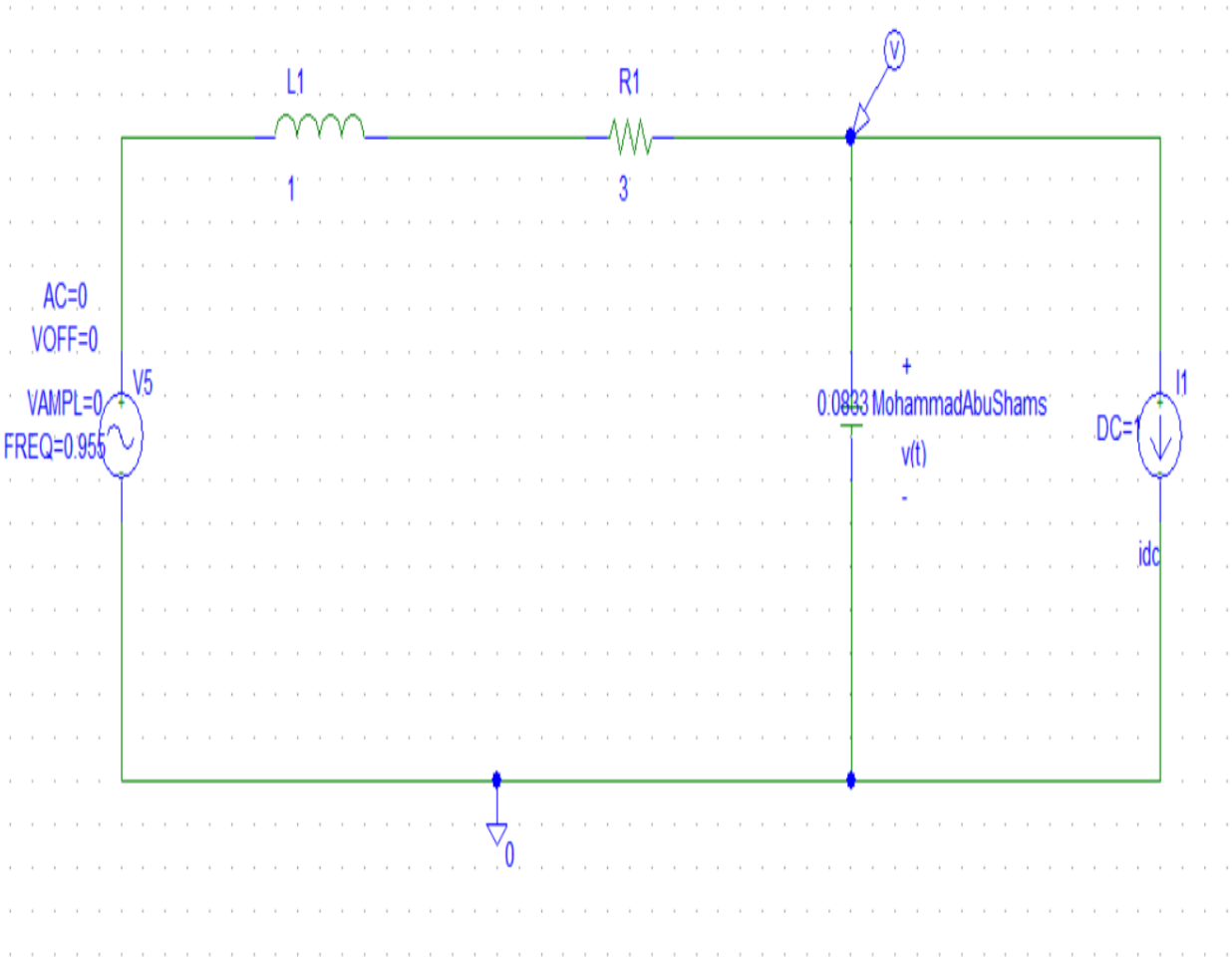
When Vm=0V, and Im =1 A

Is(t)= 1 A ,, and Vs(t)=0 V …. w=6 rad/s .. and the frequency equal w/(2*pi)

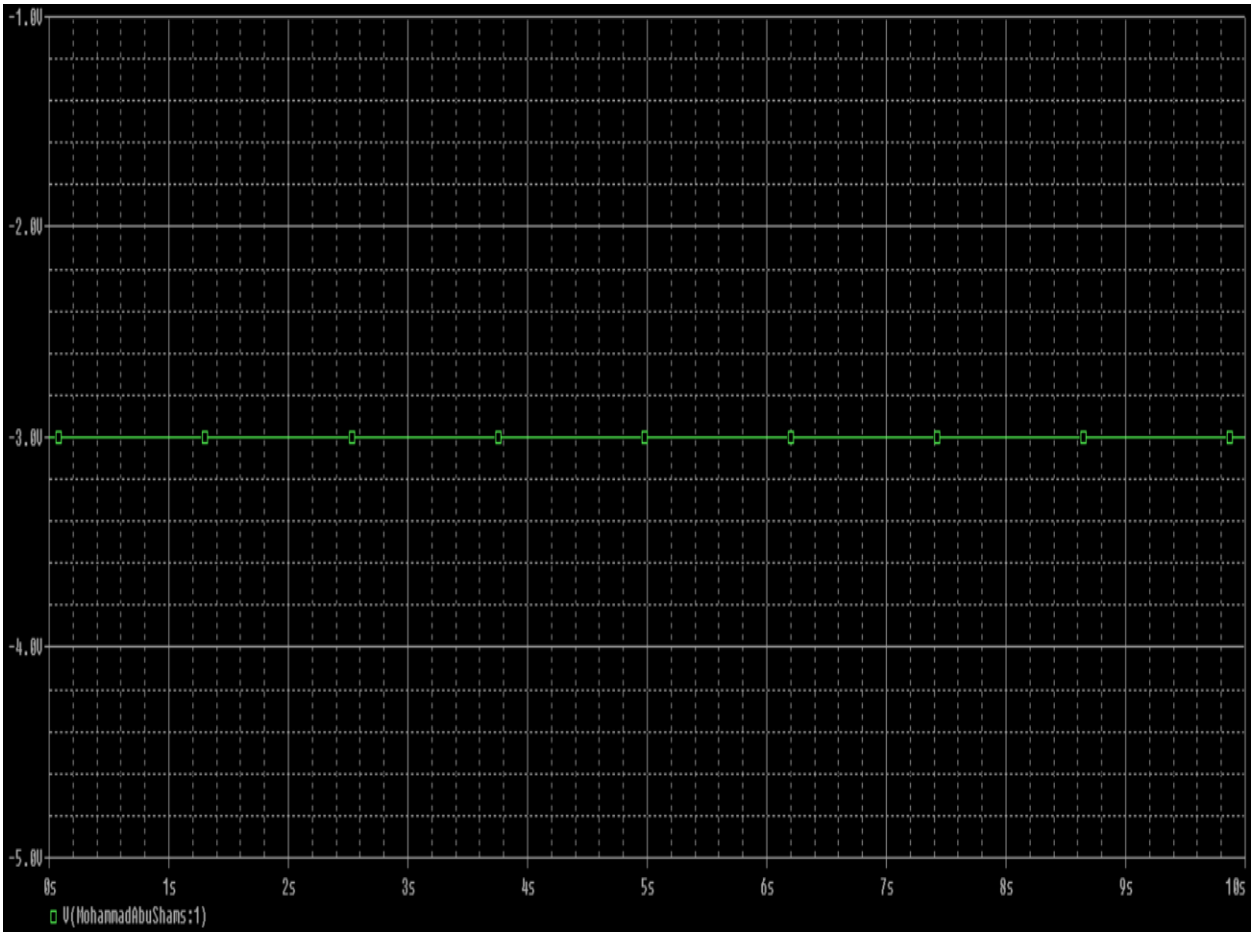
=6/(2*3.14)= 0.955 Hz

And A=k1 ,, B=0.

I built the circuit below



Waves:



According to this diagram, the value of K2 = -3

k2= -3

(c) Knowing k1 and k2, specify the values of Vm and Im that are required to cause vo(t)=5sin(6t+θ)+5 V. Simulate the circuit, using PSpice to verify the specified values of Vm and Im.

vo(t)=5sin(6t+θ)+5 V

For the voltage

1 input volt → 0.4 output volt (k1)

X input volt → 5 output volt

So, X= (5*1)/0.4 = 5/0.4= 12.5 volt

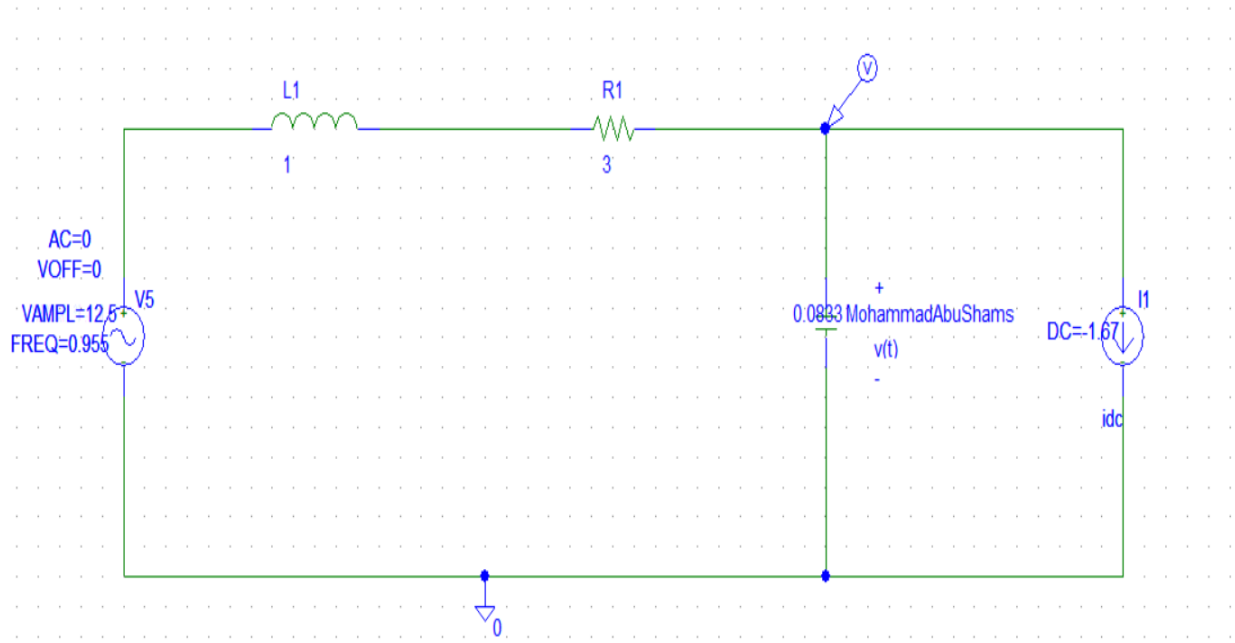
For the current

1 input Ampere → -3 output Ampere

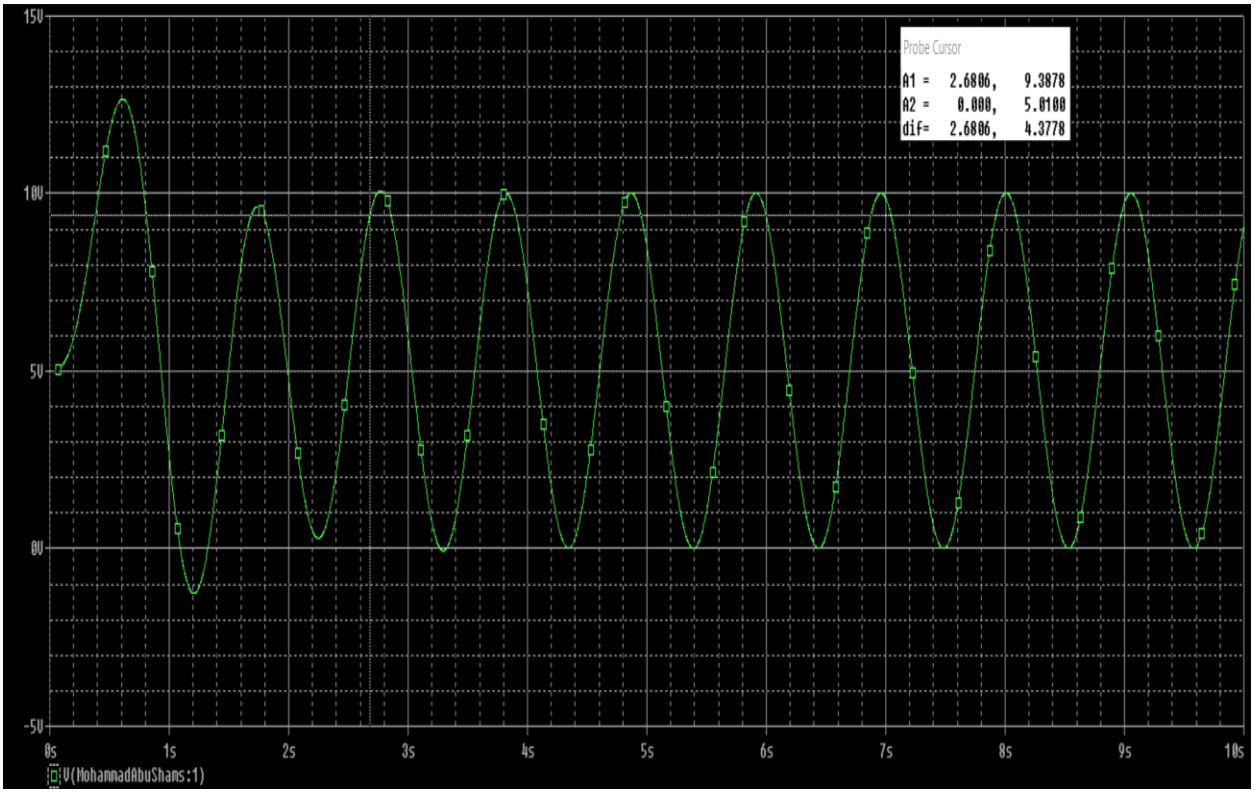
X input Ampere → 5 output Ampere

So, X= (5*1)/-3=5/-3=-1.67 Ampere

Then I built the circuit below ….

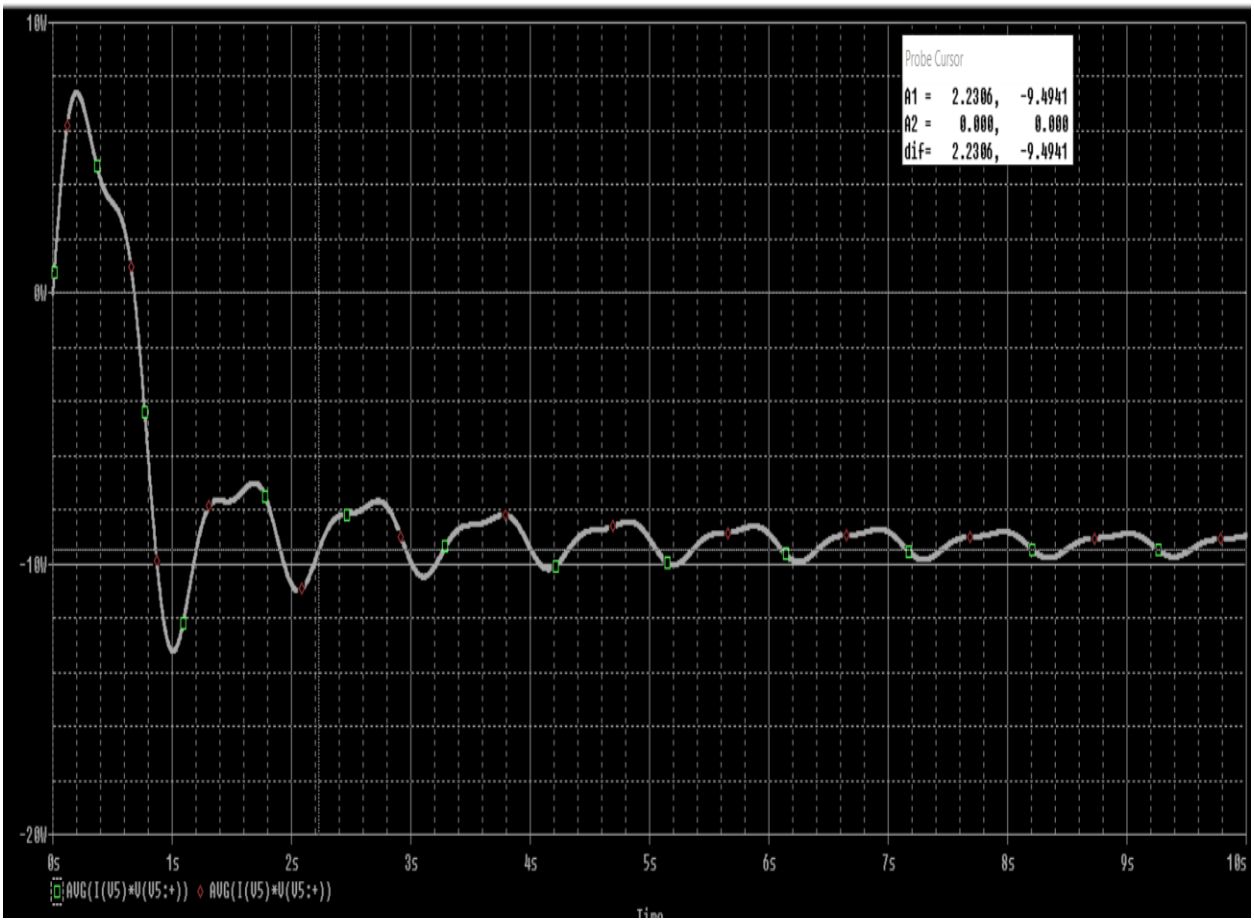

L1
1
R1
3
V
AC=0
VOFF=0
VAMPL=12.5
FREQ=0.955
V5
0.0833 MohammadAbuShams
v(t)
I1
DC=-1.67
idc
0

Waves:


Probe Cursor
A1 = 2.6806, 9.3878
A2 = 0.000, 5.0100
dif= 2.6806, 4.3778
15V
10V
5V
0V
-5V
0s
1s
2s
3s
4s
5s
6s
7s
8s
9s
10s
V(MohammadAbuShams:1)

(d) Determine the average power delivered by vs(t) using Pspice.

For the circuit before, After simulation, I pressed to the (trace) and then pressed to (add trace) then I added the voltage and the current



from this red disgram, I noticed that the average powe = 9.5 watt delivered = -9.5 watt.

Approximately 10 watt.

Thank you Dr. Hakam Shehadeh