Brac University

Department of Electrical & Electronic Engineering

Semester Spring-25

Course Number: EEE203L

Course Title: Electrical Circuits II Laboratory

Section: 06

Lab Report

Experiment no. 03

Name of the experiment: Verification of KCL in AC Circuits (Software Simulation)

Prepared by:

Name: **Tanzeel Ahmed** ID: **24321367**

Group Number: 03

Other Group members:

Sl.	***ID***	***Name***
1.	24121083	Abontika Das
2.	24121225	Aditi Gupta
3.	24121219	Subha Tasfia Chowdhury
4.	24321022	Sumya Zaman

Experiment 3 (PSpice Simulation)

Set up the circuit:

- Construct the circuit for KCL in PSpice schematics as shown:

- For the sine wave source, set the amplitude to 2V, and the frequency to 2kHz.

Add Probes:

- Locate the probes for voltage and current on the toolbar at the top.
- Place the current probe on R_2 and R_3 as shown in the above figure.
- This will allow us to examine the waveshapes of the currents through the circuit.

Experiment 03 (Simulation):

Objective: This software experiment will be performed to learn the simulation steps for determining KCL in AC circuits and learn its properties by using the Pspice software

Equipments required:

1. Pspice software (Schematics)
2. Suitable PC or Laptop

Components required in software:

- Vsin voltage source
- Resistor (R)
- Capacitor (C)
- Ground (GND-Analog)
- Inductor (L)

Circuit diagram:

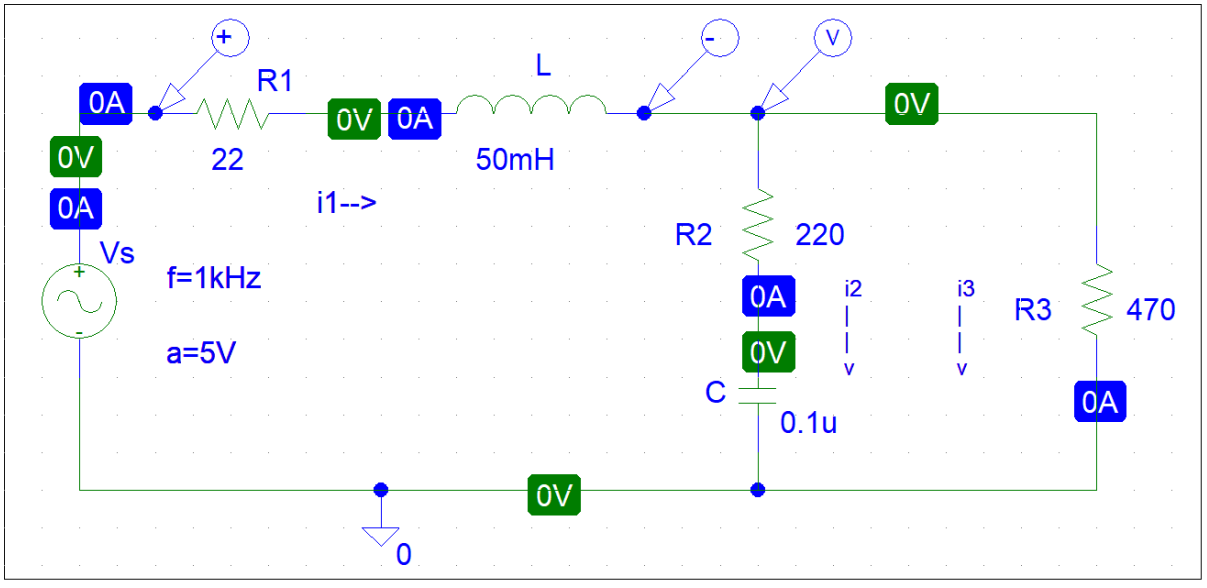
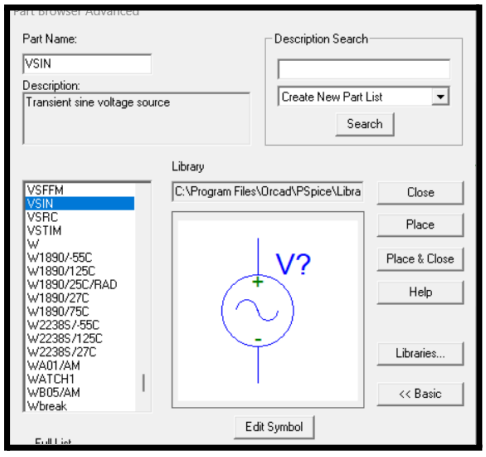
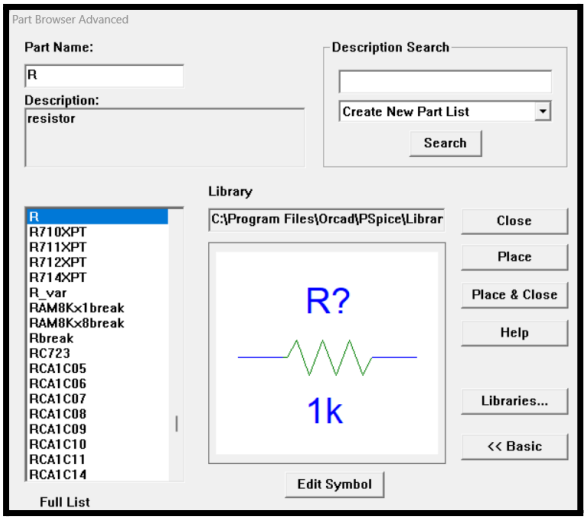


Figure: Circuit diagram for determining KCL simulated in Pspice Schematics

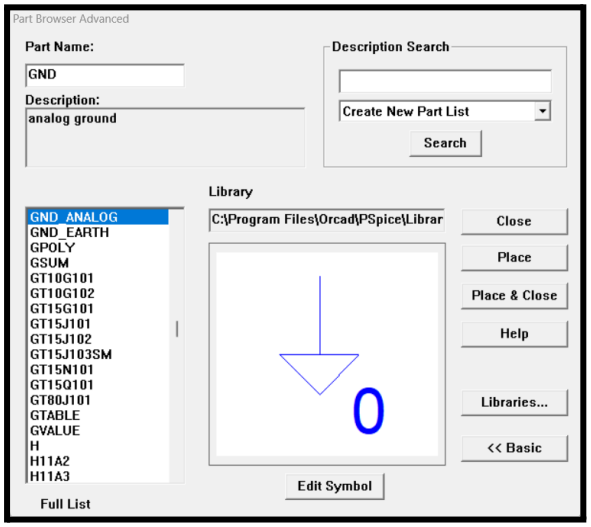
Tools, values and parameter setup menu:



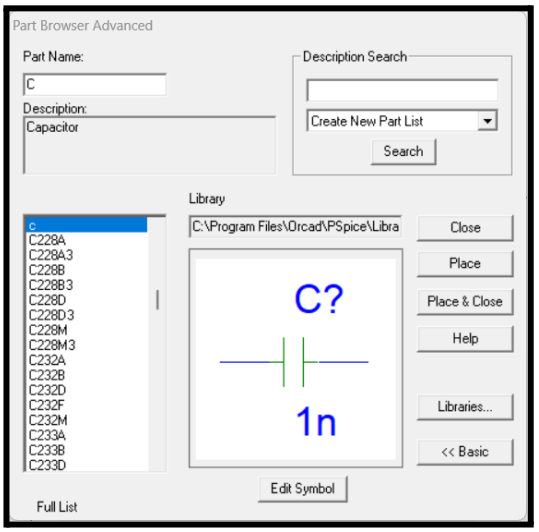
Selection of AC Source (Sin wave)



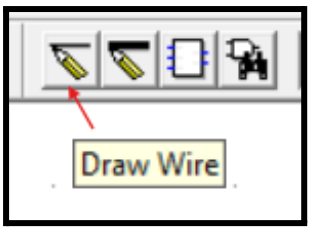
Selection of Resistor



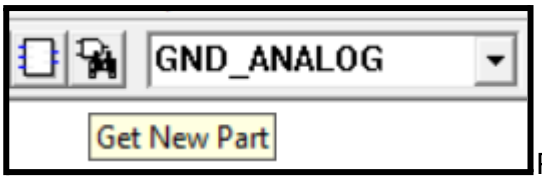
Selection of Ground



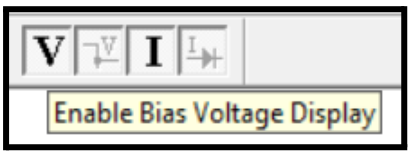
Selection of Capacitor



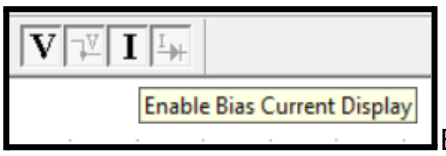
Wire tool



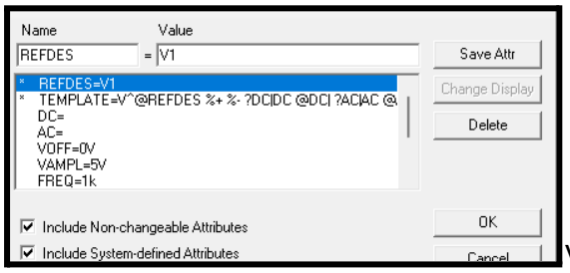
Parts menu



Bias Voltage



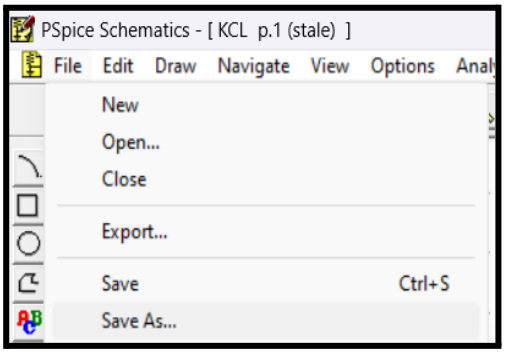
Bias Current



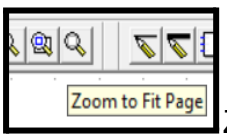
Values set in VSIN

Resistor set (R value set)

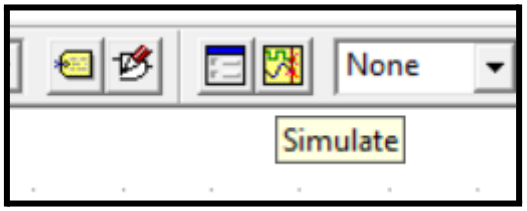
Text tools



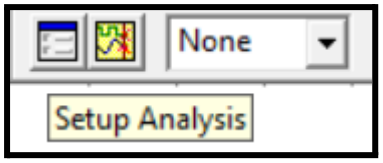
File saving



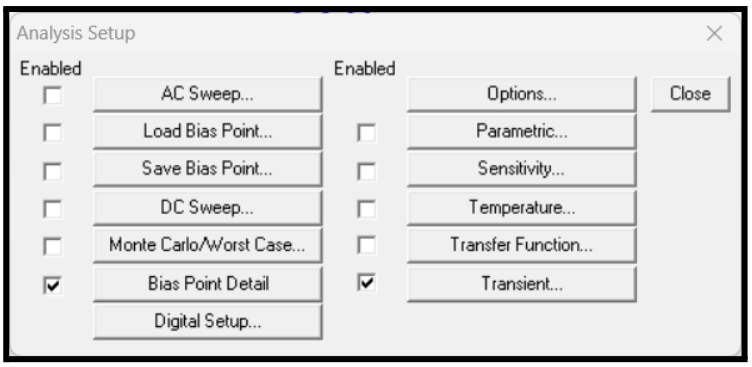
Zoom



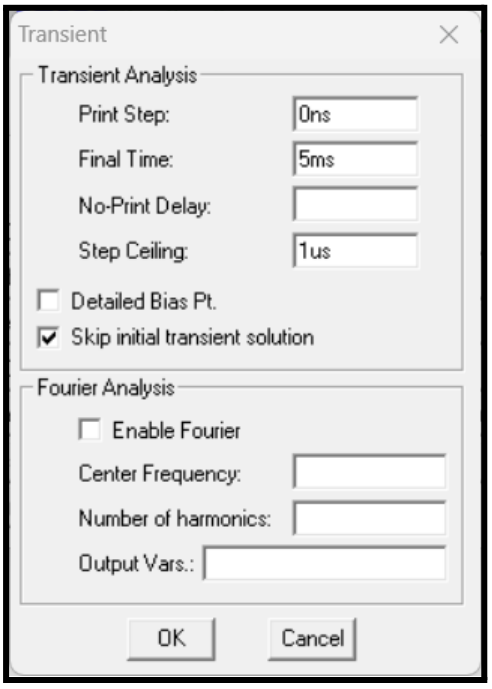
Begin Simulation



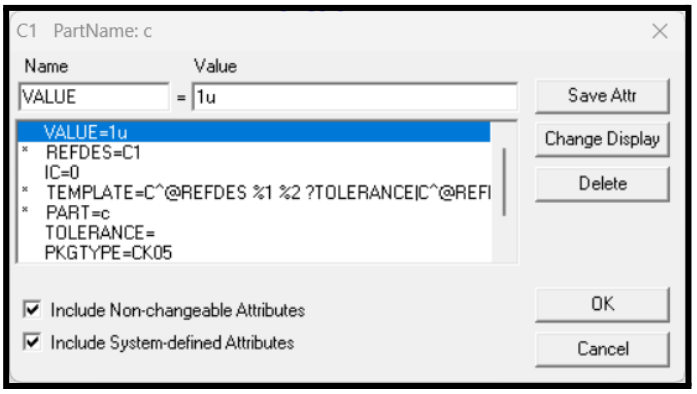
Setup Analysis Icon



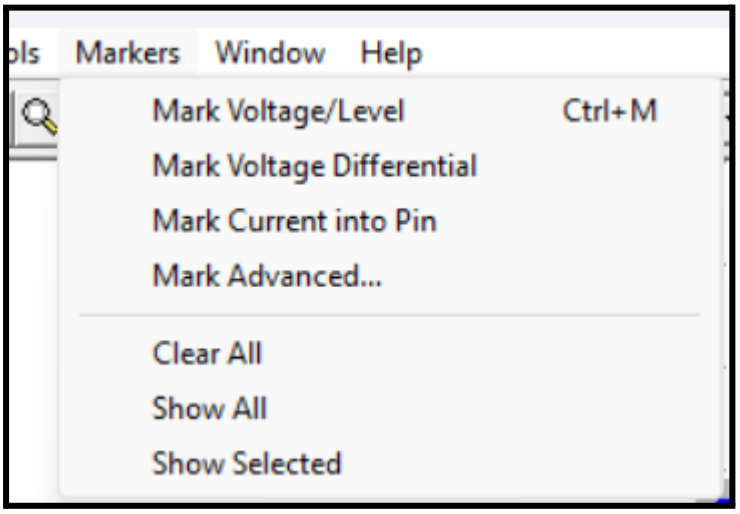
Analysis Setup Menu (Transient On)



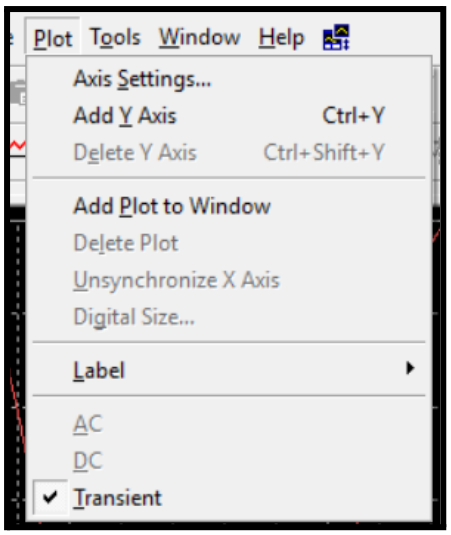
Values set in Transient



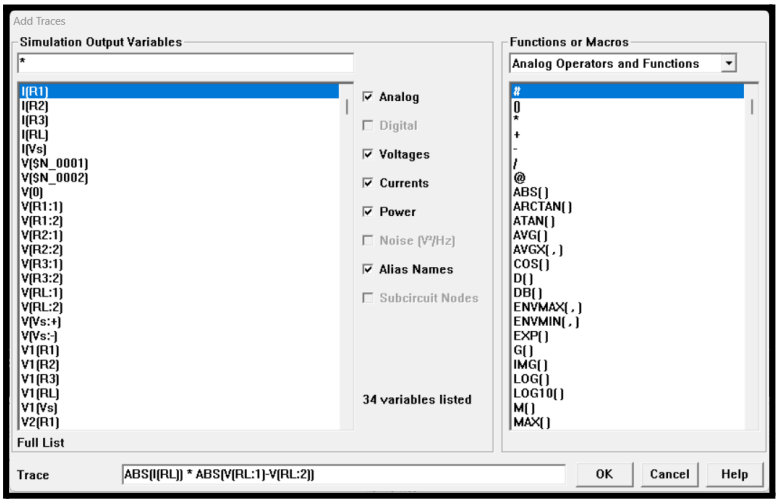
Values set in Capacitor



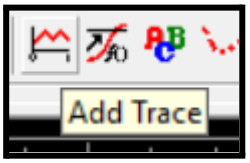
Marker Menu



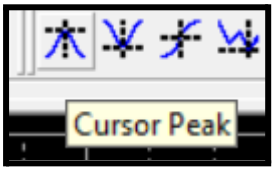
Adding new plot



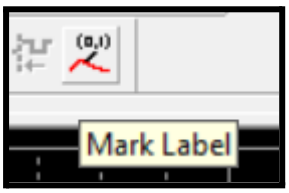
Add trace value



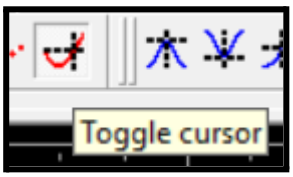
Add trace tool



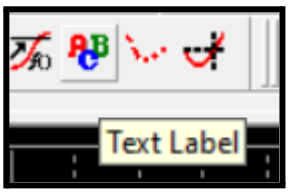
Cursor Peak tool



Mark Label Tool

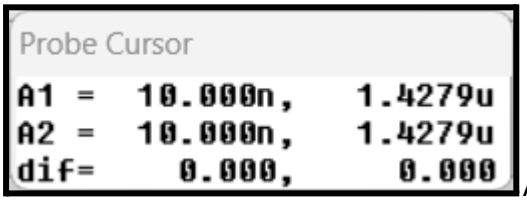


Toggle Cursor Tool



Text Label tool

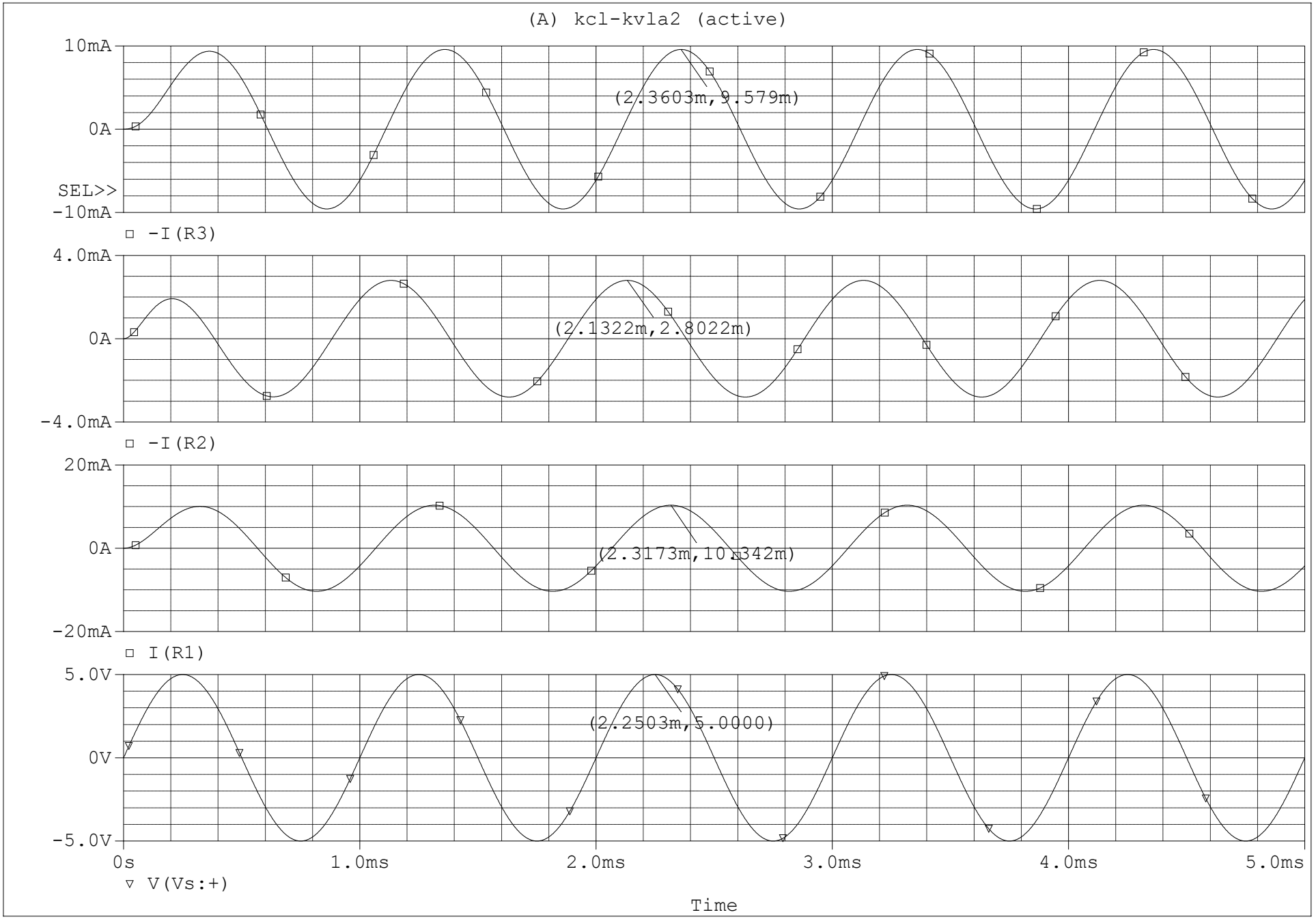
Selected Plot



After enabling cursor peak (Probe Cursor Window)

Experiment Procedure:

1. Open Pspice Schematics software
2. Open the parts menu
3. Search the necessary parts and place them according to the diagram
4. Using the wire tool connect all the parts in the circuit
5. Rename all the parts for easier identification
6. Use the draw text and text box tool to mark necessary information
7. Set the value of resistor and capacitor.
8. Double click on VSIN and set VOFF=0V, VAMPL=5V and FREQ=1k
9. Open setup analysis and select the transient menu.
10. In the transient menu, set Print Step=0ns, Final Time=5ms, Step Ceiling=1us and tick the Skip initial transient solution option.
11. Double click on the capacitor and set IC=0V.
12. Add Mark Voltage/Level and Current into Pin on the circuit.
13. Begin circuit simulation.
14. Shift to the graph interface menu
15. Add the necessary trace values following the graphs below.
16. Use toggle cursor to mark peak of all graphs
17. Calculate the necessary information and fill the data tables.



Graph for verifying KCL where V, I1, I2 and I3 have been plotted

Setup Analysis:

- We wish to obtain the current waveshapes as function of time. To do this, go to "Analysis Setup", and to "Transient".
- Enter a stop time so that you are able to see 6 complete cycles of the input waveform.
- Finally, save and simulate the circuit.
- Use cursors to find the phase difference between the current and voltage. See whether this matches with that obtained from your phasor diagram.

Data:

Table for I_{R1}					
$\lvert I_{R1} \rvert$ (mA)	Sign (+/-)	Δt (ms)	f (kHz)	360fΔt (o)	$\angle I_{R1}$ (o)
10.342	-	0.067	1	24.12	-24.12

Table for I_{R2}					
$\lvert I_{R2} \rvert$ (mA)	Sign (+/-)	Δt (ms)	f (kHz)	360fΔt (o)	$\angle I_{R2}$ (o)
2.5022	+	0.1181	1	42.516	-42.516

Table for I_{R3}					
$\lvert I_{R3} \rvert$ (mA)	Sign (+/-)	Δt (ms)	f (kHz)	360fΔt (o)	$\angle I_{R3}$ (o)
9.579	-	0.11	1	39.6	-39.6

Table for $I_{R2} + I_{R3}$	
$\lvert I_{R2} + I_{R3} \rvert$ (mA)	$\angle I_{R2} + I_{R3}$ (o)
7.3129 angle -24.32	7.313 angle -24.03

Calculation:

Table for IR1 = |IR1|= 10.342 and t=0.067, f=1 so 360f∇lt= 24.12 and ∠IR1= -24.12
Table for IR2 = |IR2|= 2.5022 and t=0.1181, f=1 so 360f∇lt= 42.516 and ∠IR2= -42.516
Table for IR3 = |IR3|= 9.579 and t=0.11, f=1 so 360f∇lt= 39.6 and ∠IR3=-39.6

Table for IR2+IR3 = |IR2+IR3|= 7.312∠-24.32 so ∠IR2+IR3= 7.313∠-24.03

Discussion:

We were able to build and observe a RLC parallel circuit which had an alternating current (AC) source using the Pspice Schematics software. Sinusoidal waveform was generated after the simulation was completed and after calculations, we correlated with practically measurable values such as rms, phase angle and time period. The usage of inductors in a simulated circuit was also established. From the simulation graph we successfully determined that KCL in phasor form can be verified. All steps necessary to complete the circuit and experiment were mentioned in the instructions. In conclusion, we were able to successfully verify KCL of the given circuit.

Schematics Drive Link: EEE203L-EXP3