

# 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.

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

*Prepared by:*

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

**Group Number: 03**

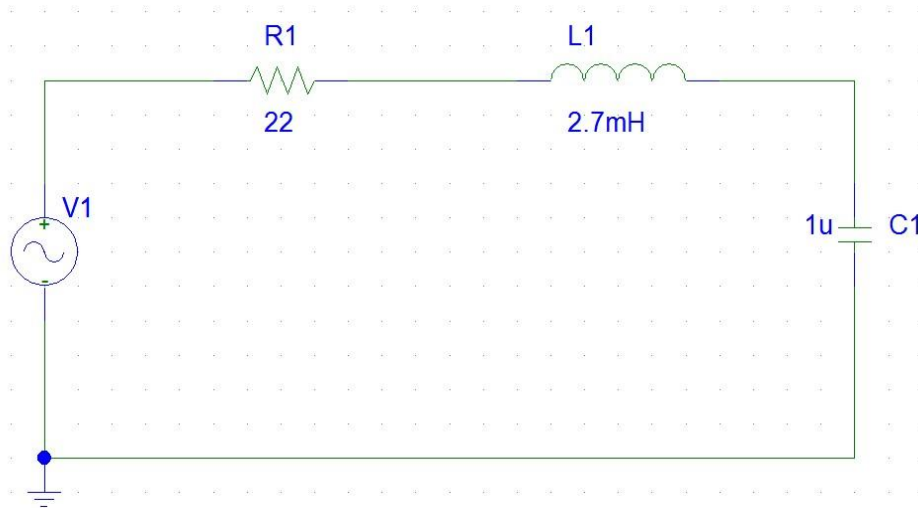
**Other Group members:**

<i><b>Sl.</b></i>	<i><b>ID</b></i>	<i><b>Name</b></i>
1.	24121083	Abontika Das
2.	24121225	Aditi Gupta
3.	24121219	Subha Tasfia Chowdhury
4.	24321022	Sumya Zaman

## Experiment 2 (PSpice Simulation)

### Set up the circuit:

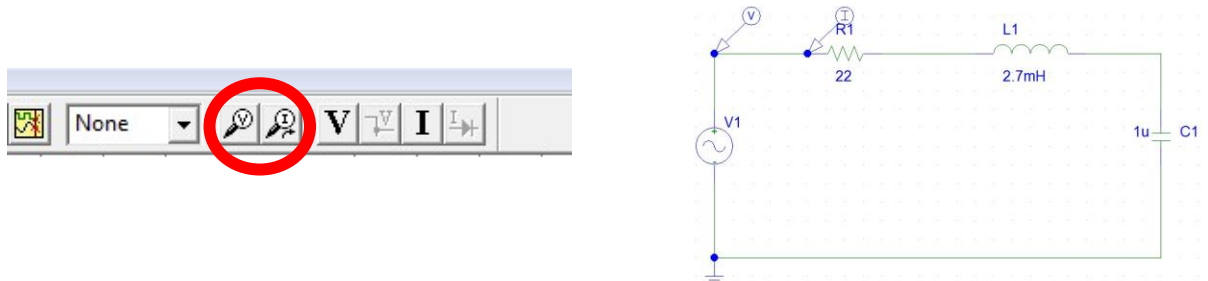
- Construct the circuit for KVL in PSpice schematics as shown:



- For the sine wave source, set the amplitude to 5V, and the frequency to 2kHz.

### Add Probes:

- Locate the probes for voltage and current on the toolbar at the top.
- Place the voltage probe across the source and the current probe on any one of the components.
- This will allow us to examine the waveshapes of the source voltage and the current through the circuit.



## **Experiment 02 (Simulation):**

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

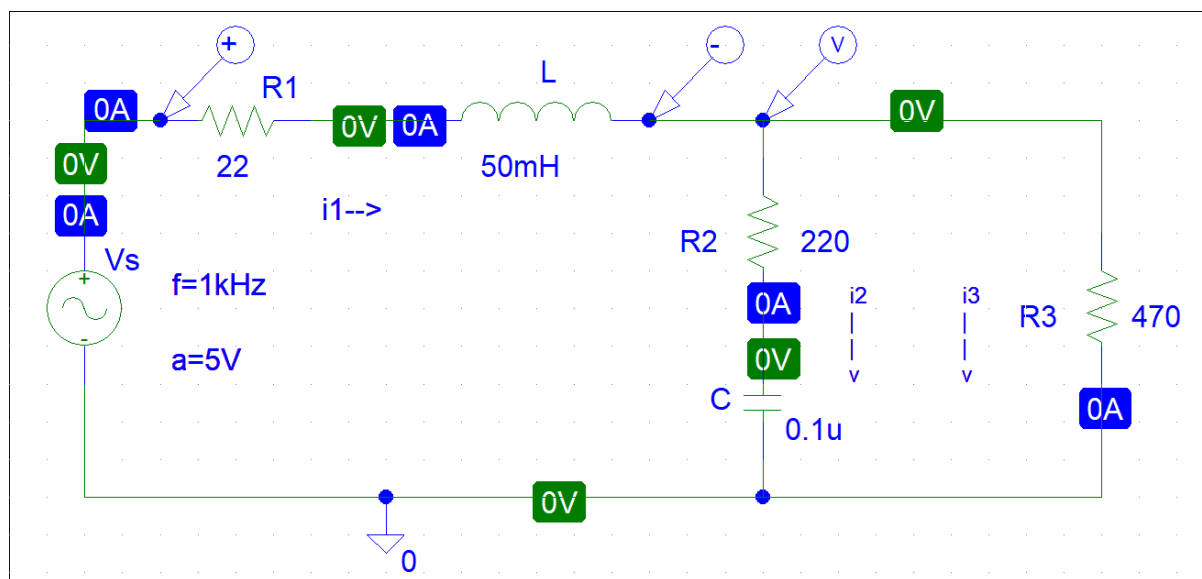
### **Equipments required:**

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

### **Components required in software:**

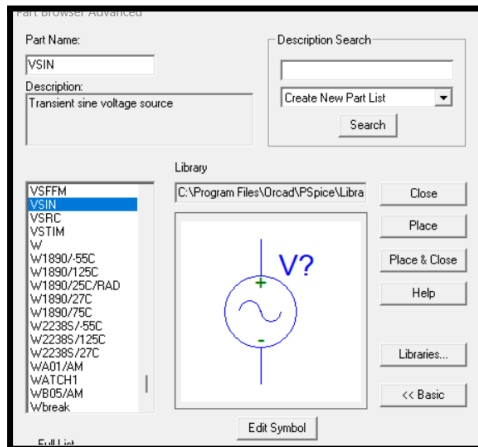
- Vsin voltage source
- Resistor (R)
- Capacitor (C)
- Ground (GND-Analog)
- Inductor (L)

### **Circuit diagram:**

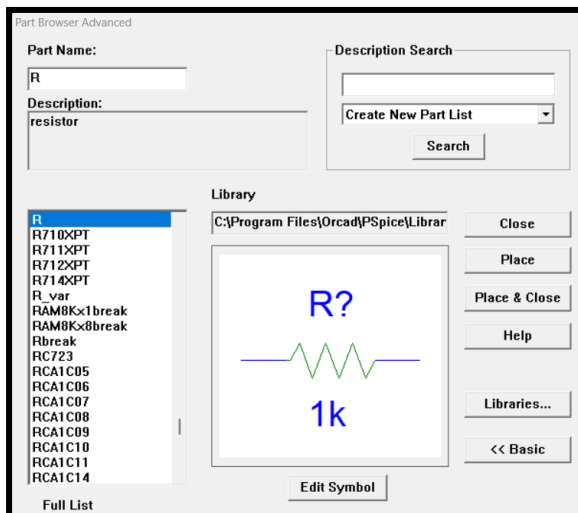


*Figure: Circuit diagram for determining KVL simulated in Pspice Schematics*

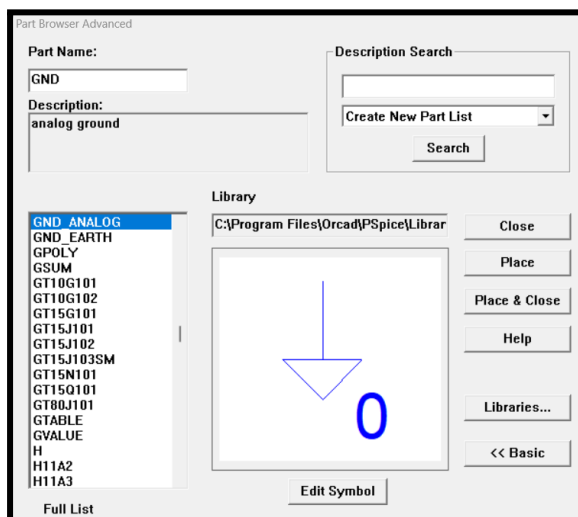
## Tools, values and parameter setup menu:



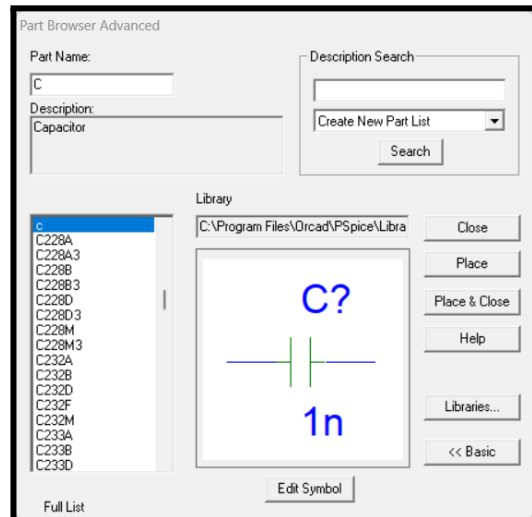
Selection of AC Source (Sin wave)



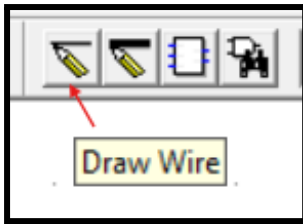
Selection of Resistor



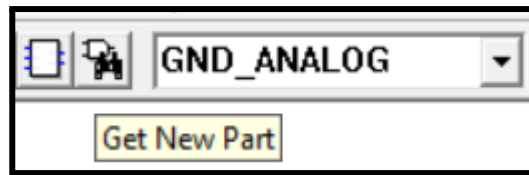
Selection of Ground



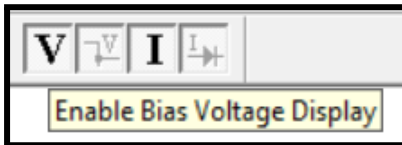
Selection of Capacitor



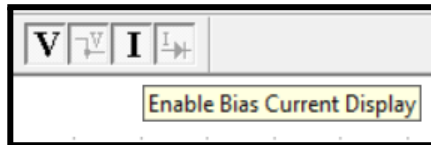
Wire tool



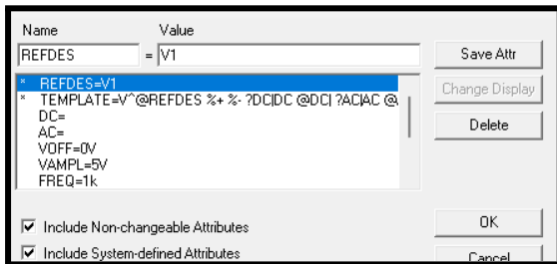
Parts menu



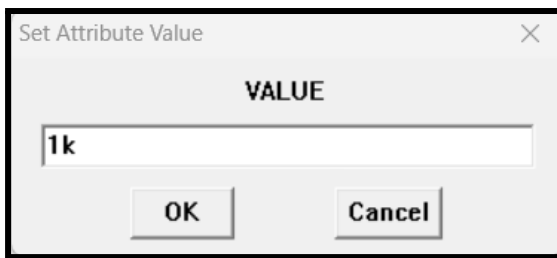
Bias Voltage



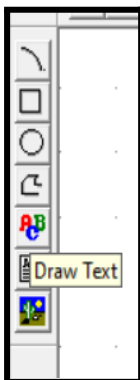
Bias Current



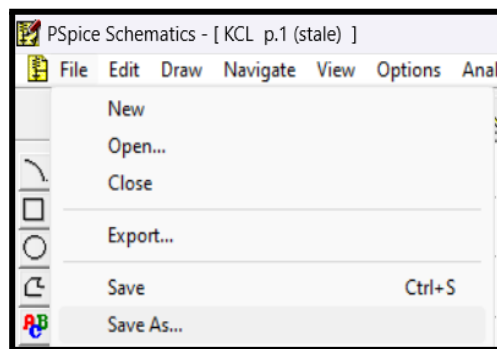
Values set in VSIN



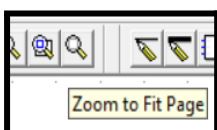
Resistor set (R value set)



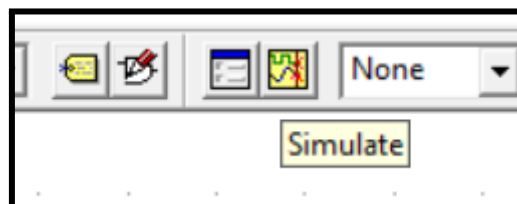
Text tools



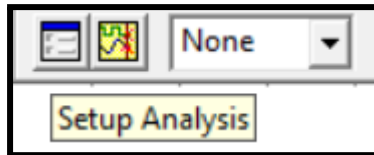
File saving



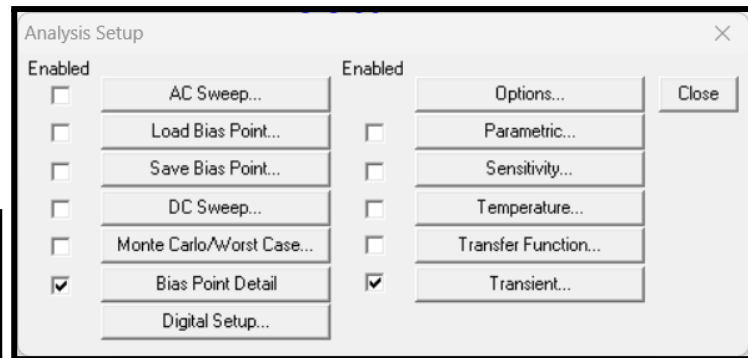
Zoom



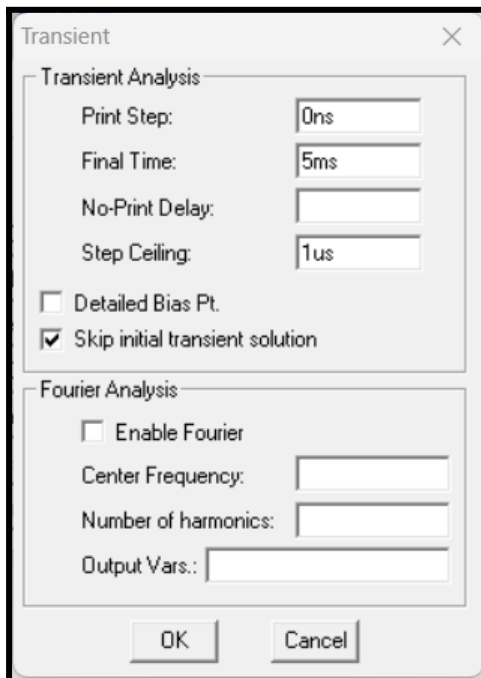
Begin Simulation



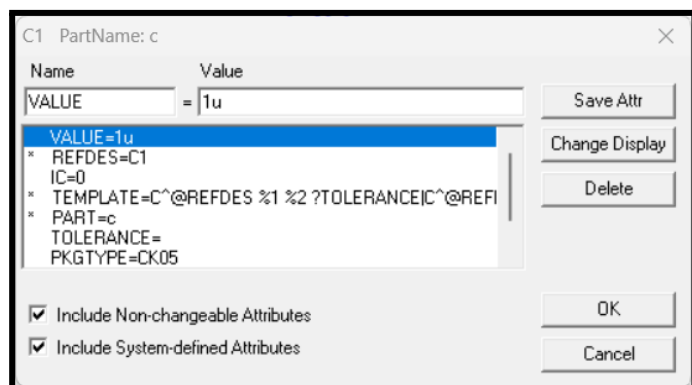
Setup Analysis Icon



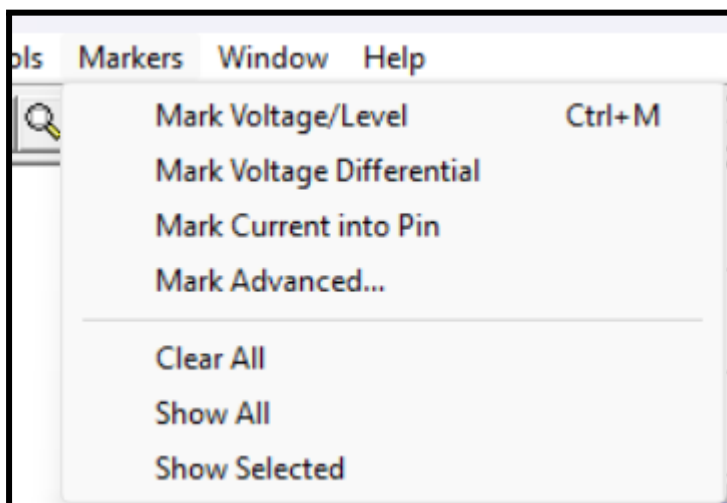
Analysis Setup Menu (Transient On)



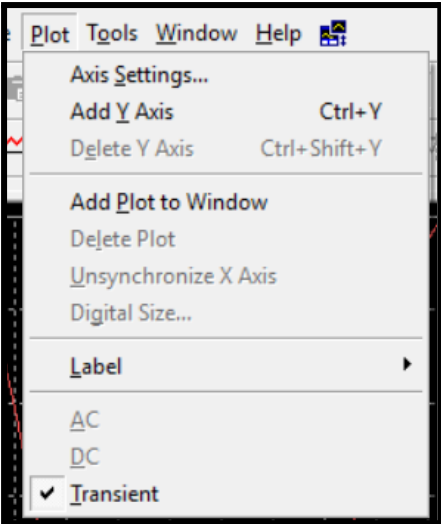
Values set in Transient



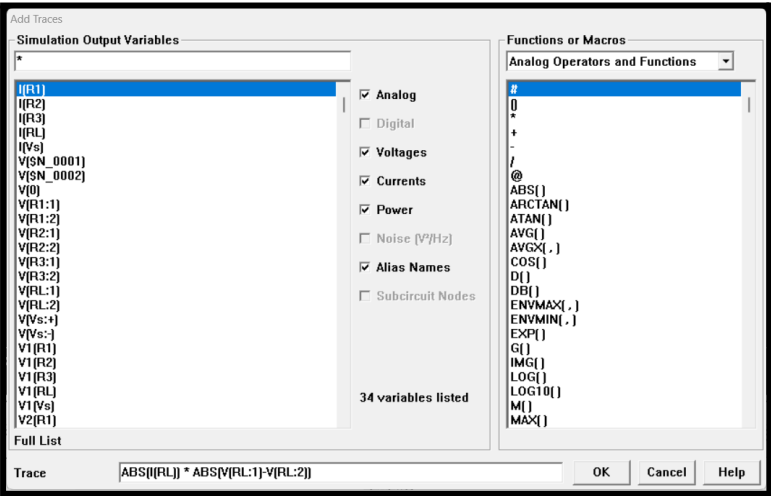
Values set in Capacitor



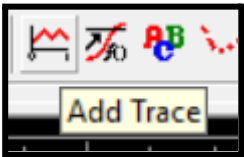
Marker Menu



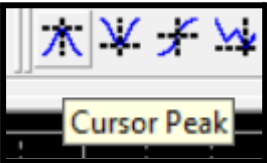
Adding new plot



Add trace value



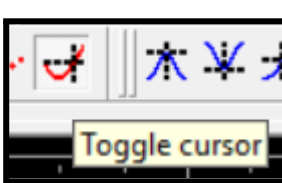
Add trace tool



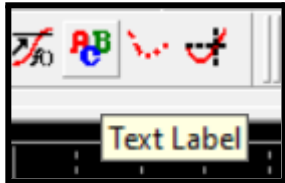
Cursor Peak tool



Mark Label Tool



Toggle Cursor Tool



Text Label tool



Selected Plot

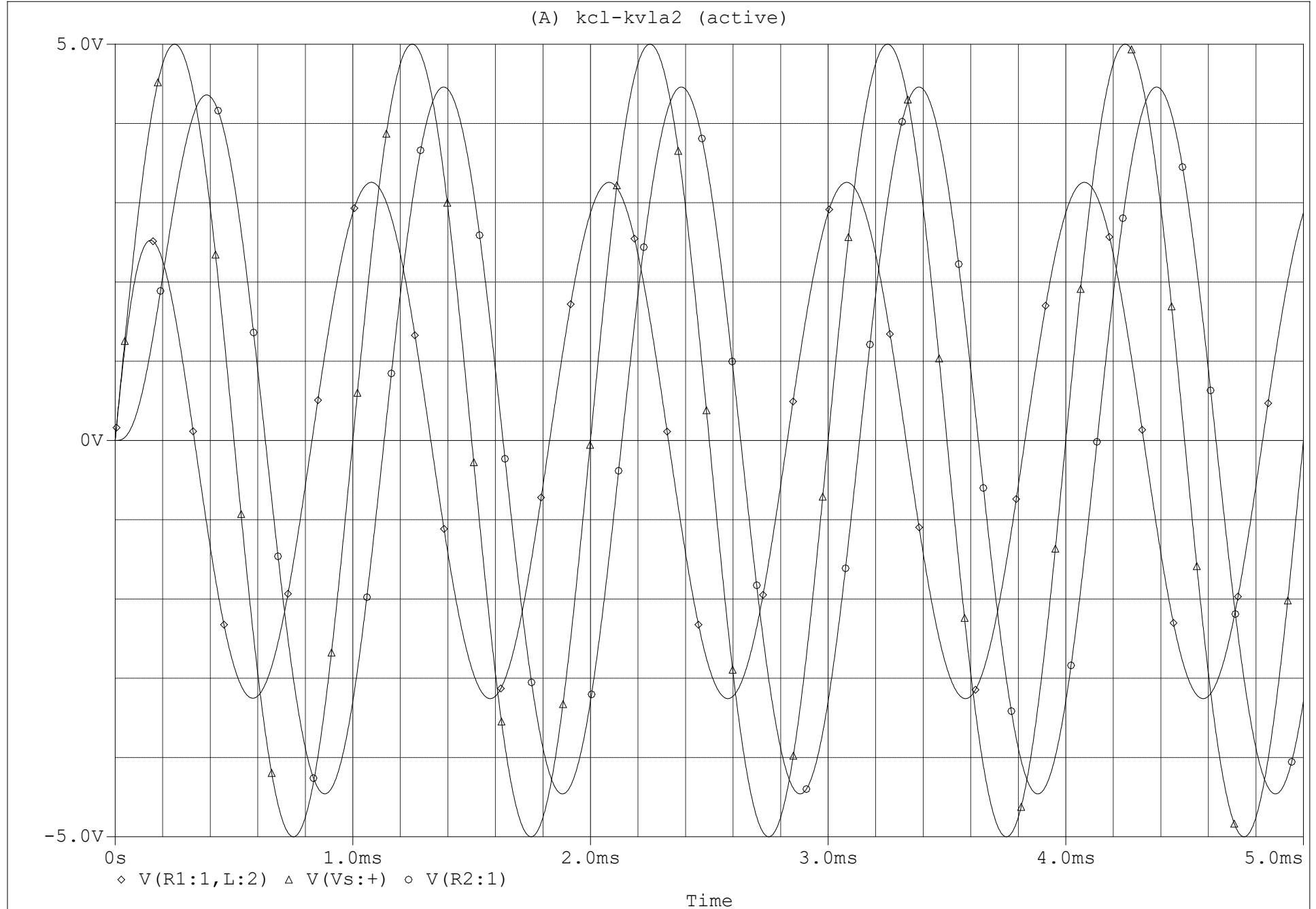


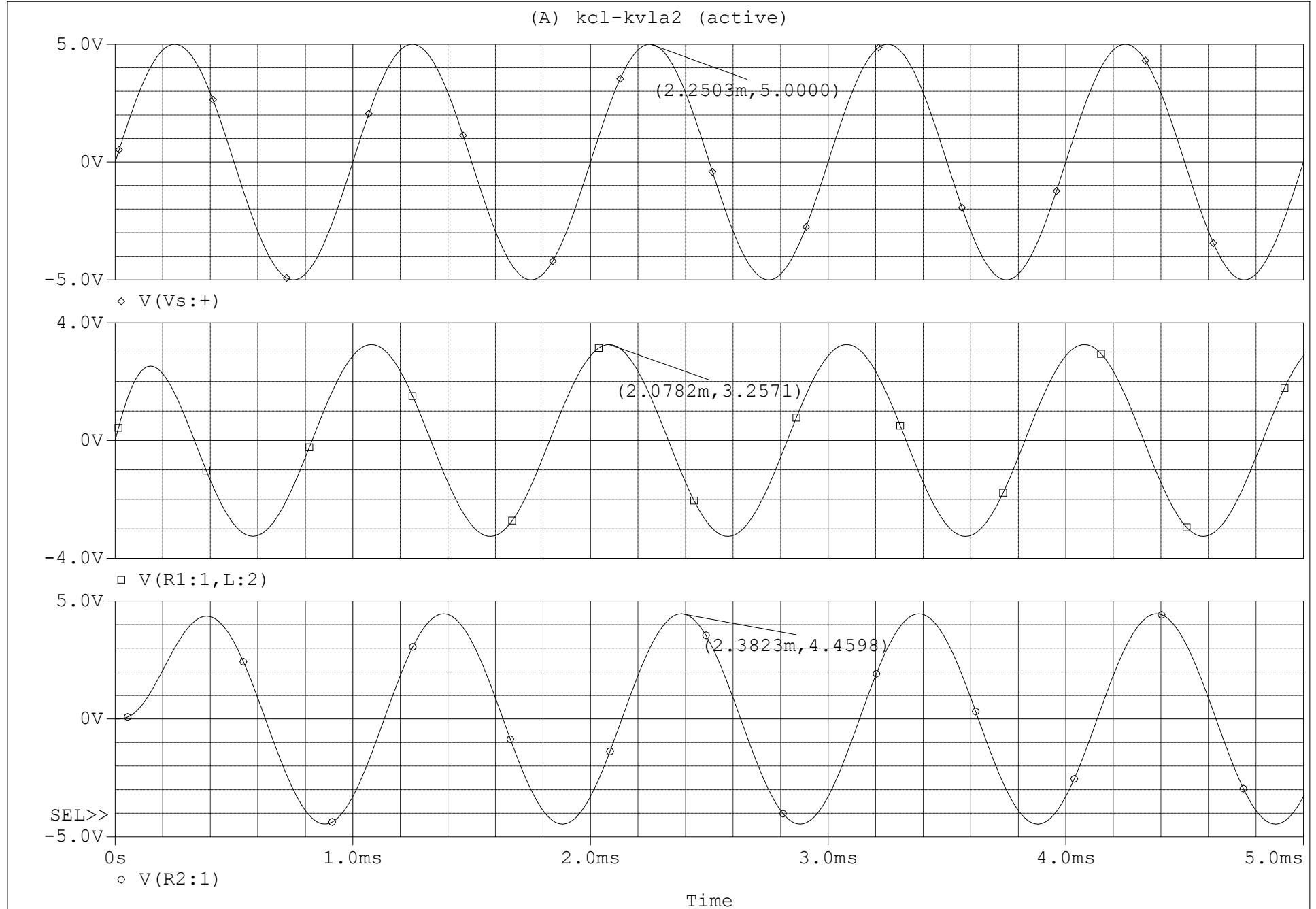
After enabling cursor peak (Probe Cursor Window)

## 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.







### Data:

**Table for Vs:**

Vsa	$\angle V_s$
5	0

**Table for V1:**

Peak Voltage (Vsa)	Sign	t(ms)	f(kHz)	$360f \nabla t$	$\angle V_1 (0)$
3.2574	+	0.1723	1	61.956	$\angle 61.956$

**Table for V2:**

Peak Voltage (Vsa)	Sign	t(ms)	f(kHz)	$360f \nabla t$	$\angle V_2 (0)$
4.5022	-	0.11	1	39.5	$\angle -39.5$

**Table for V1+V2**

$ V_1+V_2 $	$\angle V_1+V_2$
5.0636	0.056

### Calculation:

**Table for V1** =  $|V_{sa}| = 3.2574$  and  $t = 0.1723$ ,  $f = 1$  so  $360f \nabla t = 61.956$  and  $\angle V_1 = +61.956$

**Table for V2** =  $|V_{sa}| = 4.5022$  and  $t = 0.11$ ,  $f = 1$  so  $360f \nabla t = 39.5$  and  $\angle V_2 = -39.5$

**Table for V1+V2** =  $|V_1+V_2| = 5.0636$  so  $\angle V_1+V_2 = \angle 0.056$

### 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 KVL 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 KVL of the given circuit.

**Schematics Drive Link:** [EEE203L-EXP2](#)