

# 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: Familiarization with alternating current (AC) waves (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

**Brac University**  
**Department of Electrical & Electronic Engineering (EEE)**  
**EEE203L – Electrical Circuits II Laboratory**

**Experiment 1**

**Name of the Experiment:**

**Familiarization with the alternating current (AC) waves**

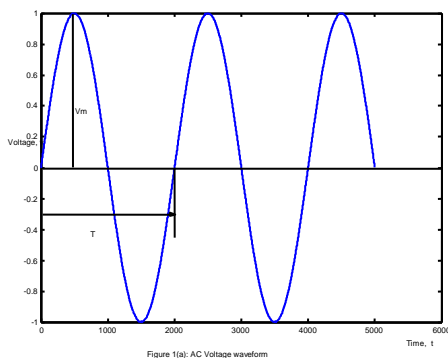
**Objective:**

In this experiment, we shall study some aspects of sinusoidal waveform, and correlate these with practically measurable values such as- rms. value (also called effective value), phase angle and time period. Also an exposure to simple ac circuit and some circuit elements are made. Try to familiarize yourself with

- Oscilloscope
- How to measure peak value, phase angle and time period (or frequency) using oscilloscope
- The methods of measuring rms. value both using oscilloscope and multimeter
- Difference between AC & DC setting of multimeter & oscilloscope
- Capacitor, resistor and breadboard

**Introduction:**

Any periodic variation of current or voltage where the current (or voltage), when measured along any particular direction goes positive as well as negative, is defined to be an AC quantity. Sinusoidal AC wave shapes are the ones where the variation (current or voltage) is a sine function of time.



Here, Time period =  $T$ , Frequency,  $f = 1/T$   
 $v(t) = V_m \sin(2\pi ft)$

## **Experiment 01 (Simulation):**

**Objective:** This software experiment will be performed to learn the simulation steps for being familiar with the alternating current (AC) waves and learn its properties by using the Pspice software

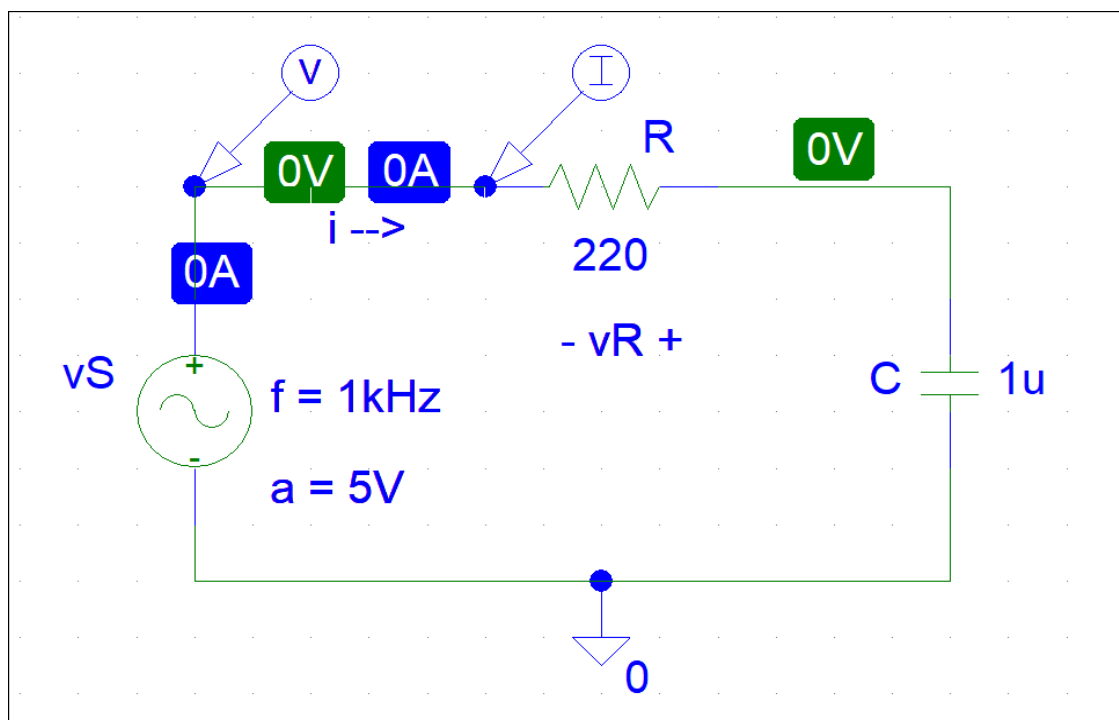
### **Equipments required:**

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

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

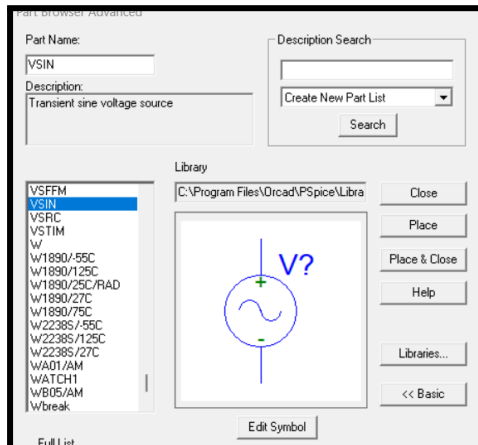
- Vsin voltage source
- Resistor (R)
- Capacitor (C)
- Ground (GND-Analog)

### **Circuit diagram:**

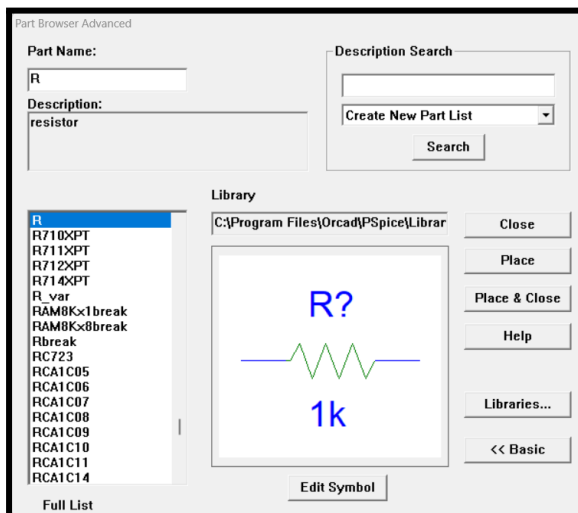


*Figure: Circuit diagram for determining sine wave simulated in Pspice Schematics*

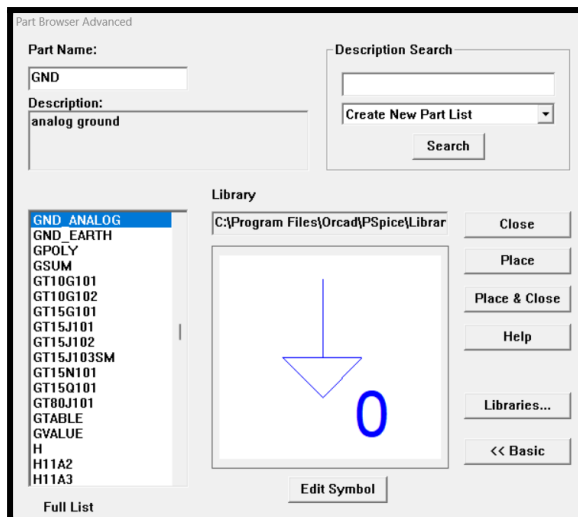
## Tools, values and parameter setup menu:



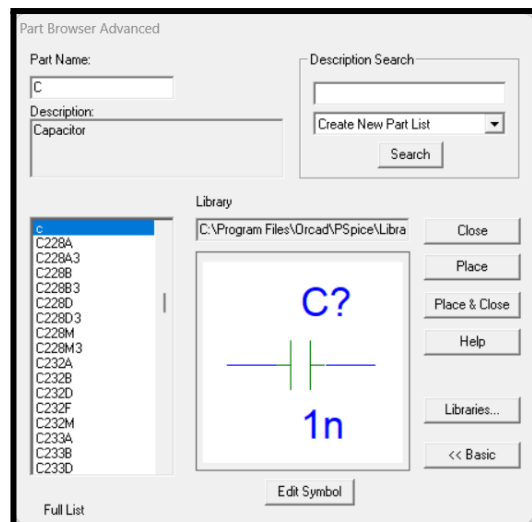
Selection of AC Source (Sin wave)



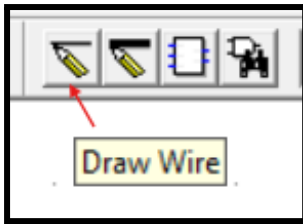
Selection of Resistor



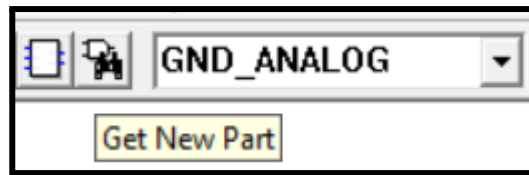
Selection of Ground



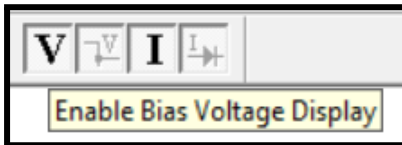
Selection of Capacitor



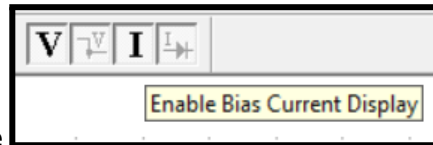
Wire tool



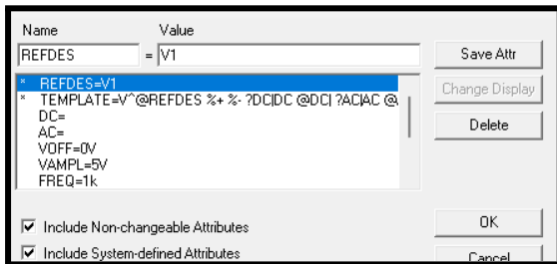
Parts menu



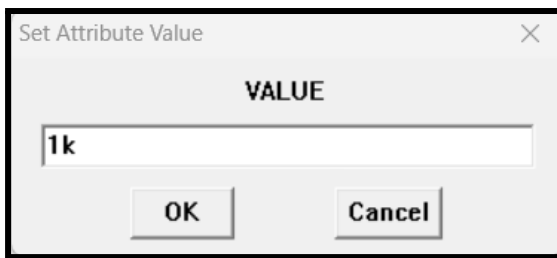
Bias Voltage



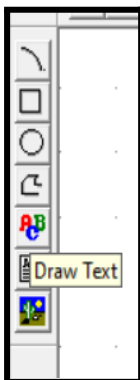
Bias Current



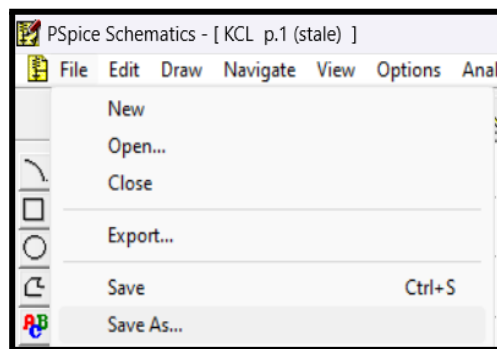
Values set in VSIN



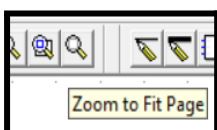
Resistor set (R value set)



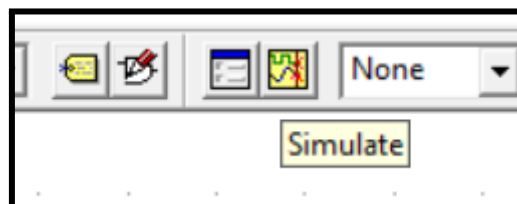
Text tools



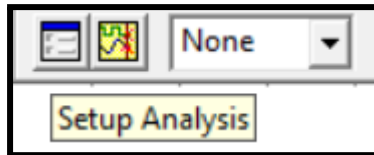
File saving



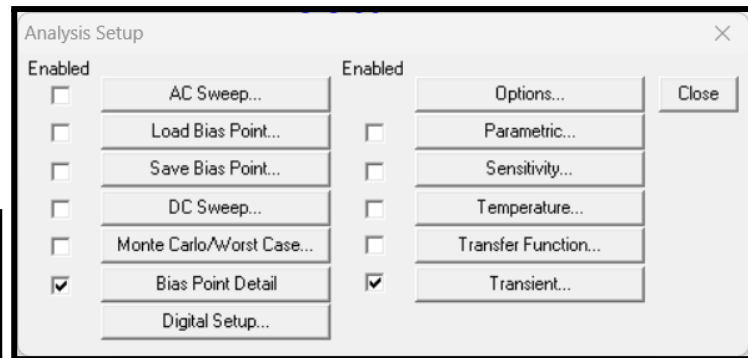
Zoom



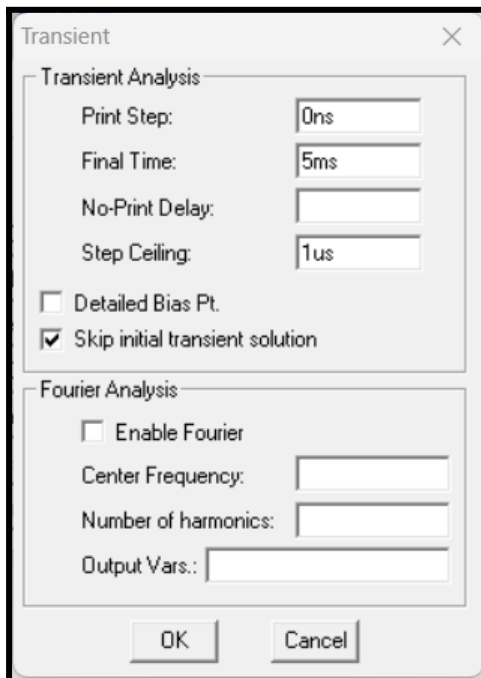
Begin Simulation



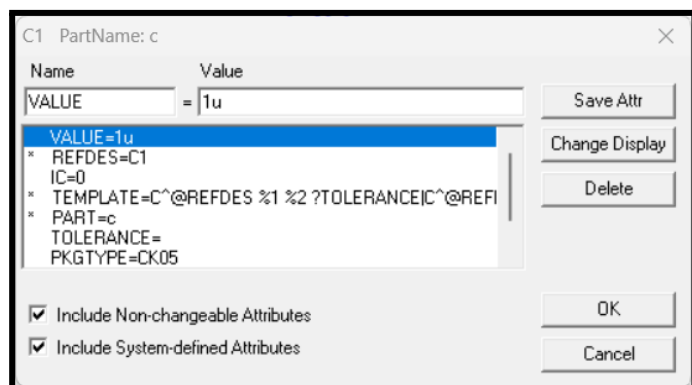
Setup Analysis Icon



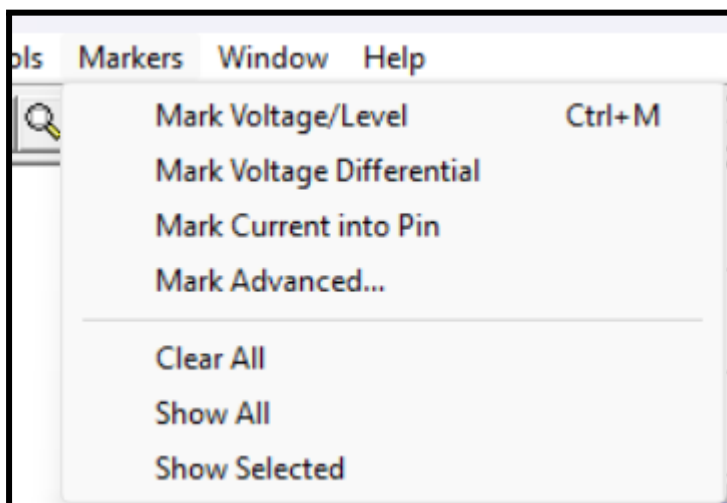
Analysis Setup Menu (Transient On)



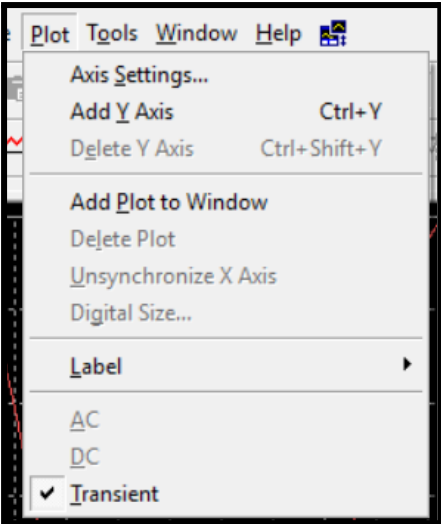
Values set in Transient



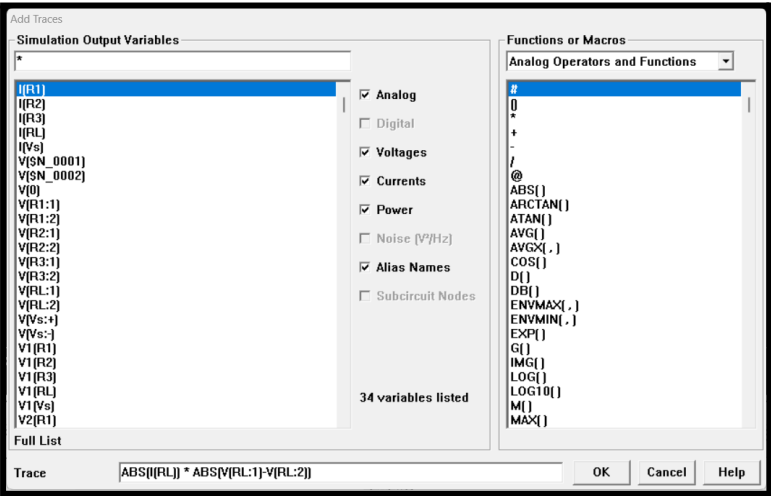
Values set in Capacitor



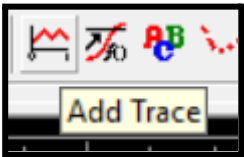
Marker Menu



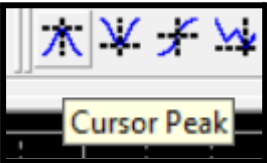
Adding new plot



Add trace value



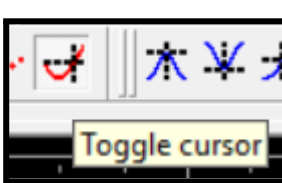
Add trace tool



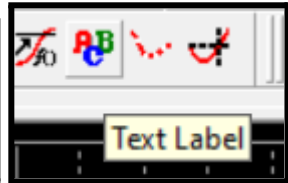
Cursor Peak tool



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 another plot to the window and set the voltage trace value
16. Remove the voltage trace value from the graph of I
17. Use toggle cursor to mark peak of both graphs
18. Calculate the necessary information and fill the data tables.



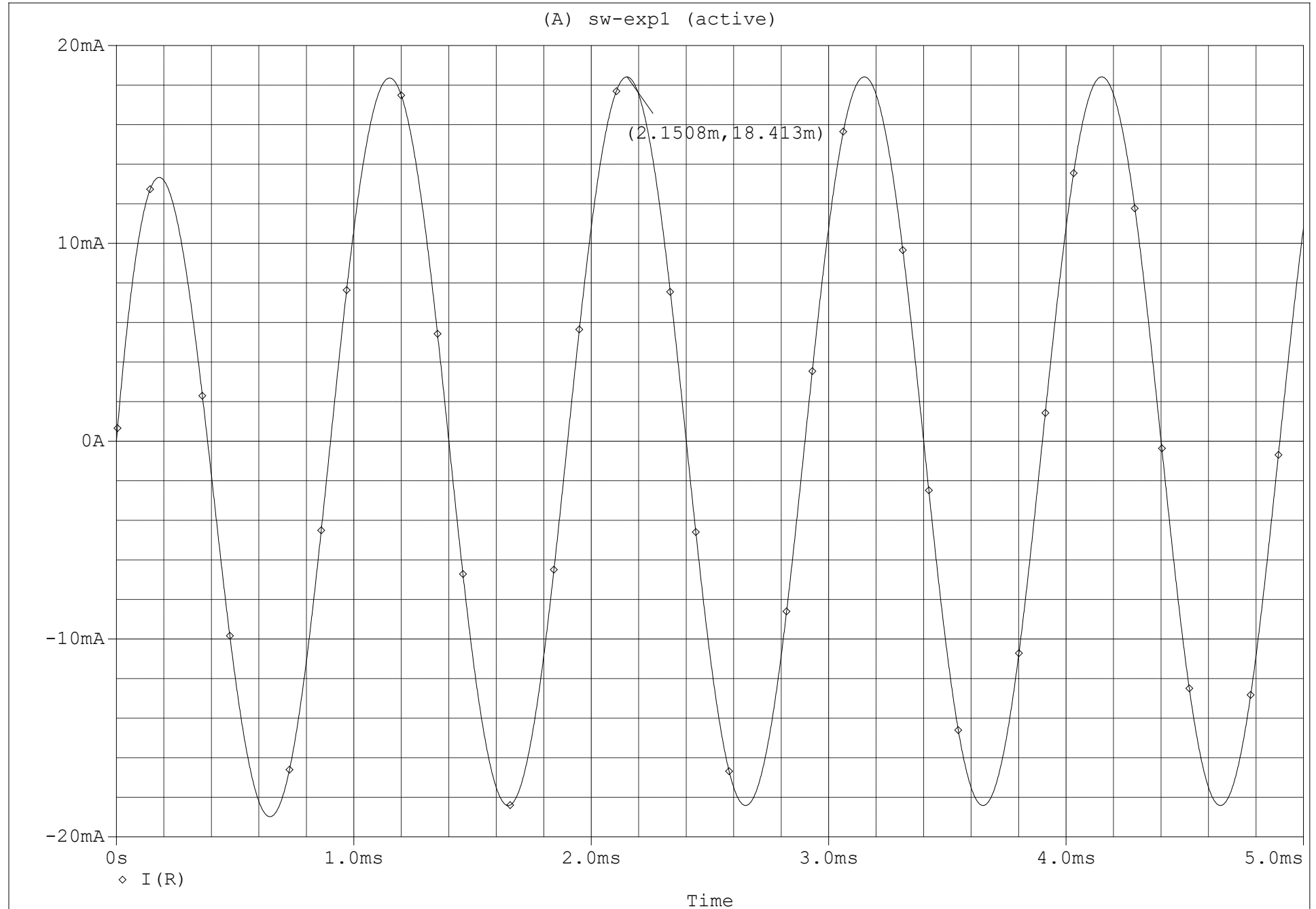


Fig.1 Graph of I simulated in Pspice Schematics

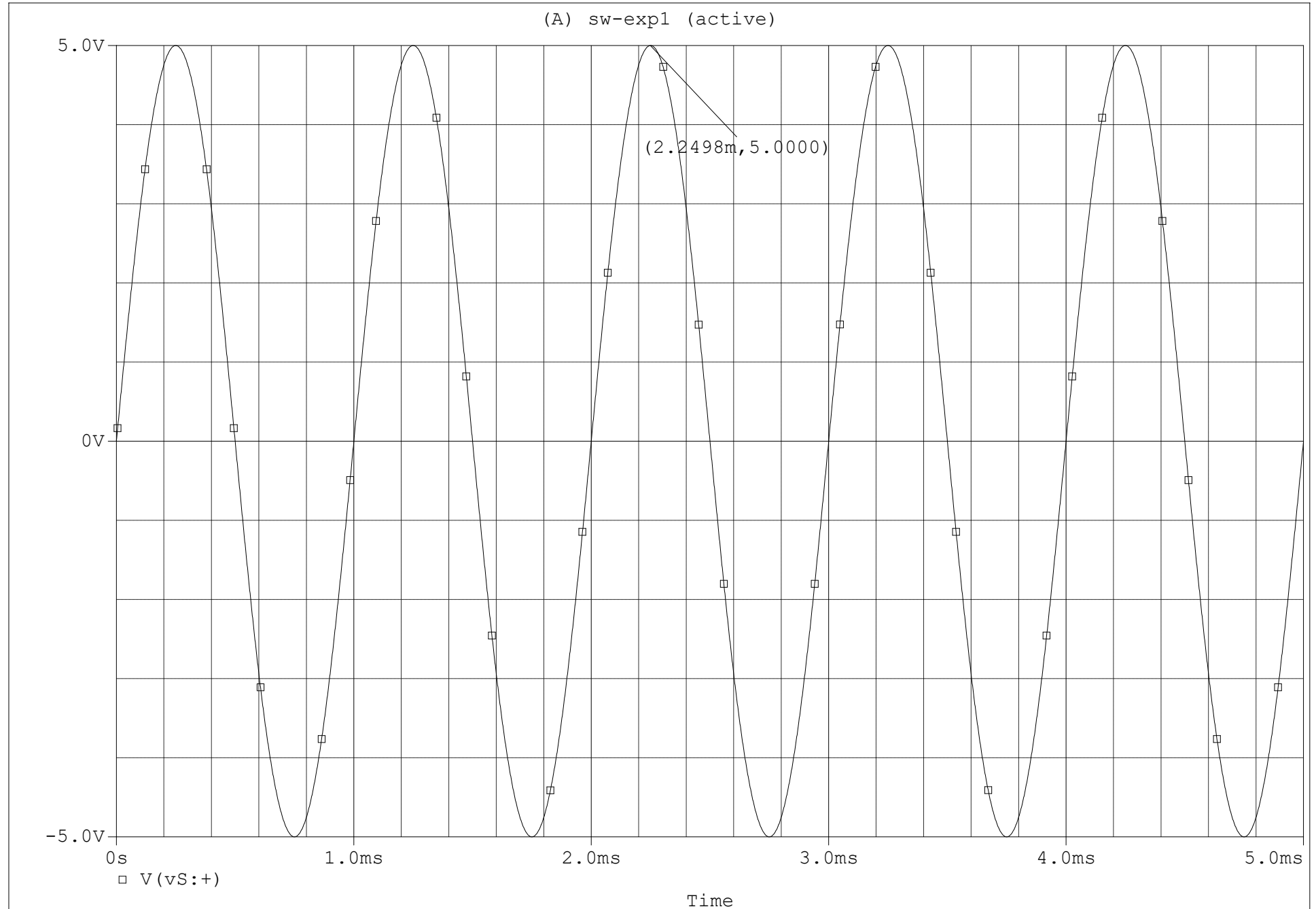


Fig.2: Graph of V simulated in Pspice schematics

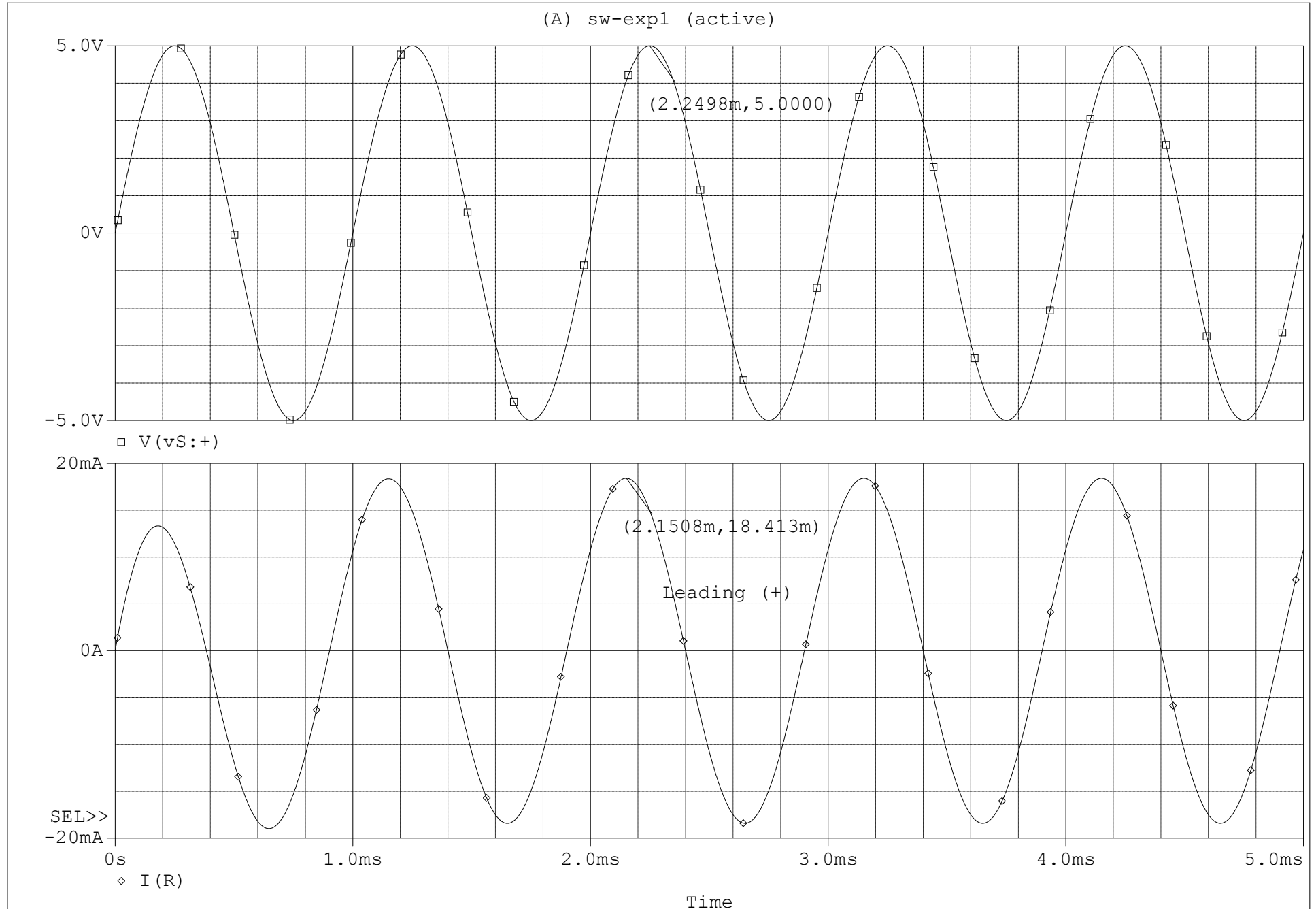


Fig.3: Graph of I and V simulated in Pspice Schematics

Position the two cursors, such that they both are at the adjacent zero crossings of the two waveshapes, so as to be able to measure the time difference between them. The difference in the x axis appears on the left hand column of the “Probe Cursor” window, beside ‘dif’.

Using this information and the formula  $2\pi f t$ , find the phase difference between the two waveshapes, where, ‘t’ is the time difference, ‘f’ is the source frequency, and ‘pi’ is equal to 3.142.

Change the source (from Vsin) frequency to 2 kHz, and then to 500 Hz, and for both the frequencies obtain the output and input waveshapes together. Using the cursors, determine the phase difference as well.

Include the schematic circuit design, the waveshapes for all the three frequencies and the calculated phase differences in a new document and submit the hard copy. (See the file- “How to submit Pspice Assignment” for submission guidelines).

**Data:**

Table for $V_s$	
$ V_s $ (V)	$\angle V_s$ (o)
5V	0

Table for I					
$ I $ (mA)	Sign (+/-)	$\Delta t$ (ms)	f (kHz)	$360f\Delta t$ (o)	$\angle I$ (o)
13.019	+	0.099	1	35.64°	35.64°

Table for Z	
$ Z  = \frac{ V_s }{ I }$ (k $\Omega$ )	$\angle Z = \angle V_s - \angle I$ (o)
0.3841	-35.64°

## **Calculation:**

### **Table for I**

$$|I|(\text{mA}) = 18.413/\sqrt{2} = 13.019 \text{ mA}$$

$$\Delta t (\text{ms}) = 2.2498 - 2.1508 = 0.099$$

$$360f\Delta t (^\circ) = 360 \times 1 \times 0.099 = 35.64^\circ$$

### **Table for |Z|**

$$|Z| = |V_s| / |I| (\text{k-ohm}) = |5| / |13.019| = 0.3841$$

## **Discussion:**

We were able to build and observe a series 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 capacitors in a simulated circuit was also established. From the simulation graph we successfully determined that current was leading in this experiment. All steps necessary to complete the circuit and experiment were mentioned in the instructions. In conclusion, we were able to emulate a sine wave form with an AC source, resistor and capacitor and familiarization was achieved successfully.

**Schematics Drive Link:** [EEE203L-EXP1](#)