Student ID:	22301689	**Lab Section:**	28
Name:	TASNIM RAHMAN MOUMITA	**Lab Group:**	03

Experiment No. 9

Familiarization with the Alternating Current (AC) Waves Using Software (LTSpice) Simulation.

Objective

In this experiment, we shall study some aspects of a sinusoidal waveform, and correlate these with practically measurable values such as peak value, phase angle, and time period. We shall apply a sinusoidal wave input to an RC circuit and observe the effect of changing the frequency on the phase angles.

Theory

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.

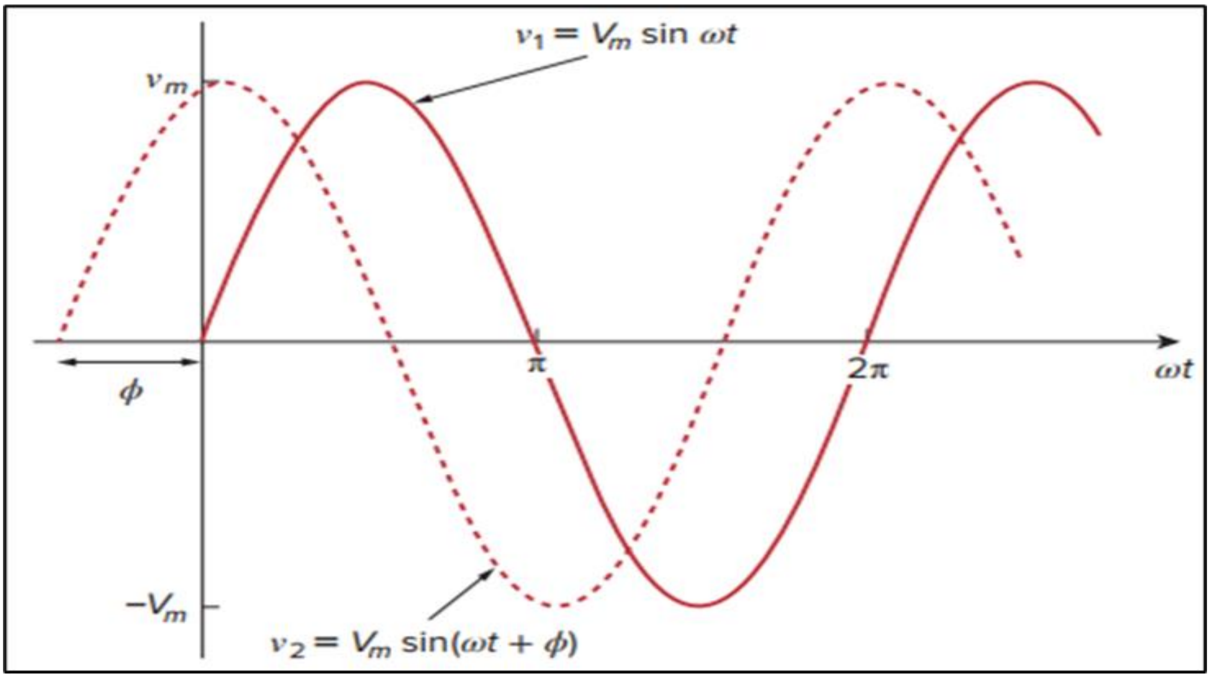


Figure 1: A Sine wave

Here, the time period = **T**

Frequency, **f** = $\frac{1}{T}$ m

$V_1(t) = V_m \sin(2\pi ft) = V \sin(\omega t)$ and, **$V_2(t) =$**

$V_m \sin(2\pi ft + \phi) = V_m \sin(\omega t + \phi)$

Effective Value

The general equation of the RMS value of any function (voltage, current, or any other physical quantity for which rms. calculation is meaningful) is given by the equation,

$$v_{rms} = \sqrt{\frac{1}{T}\int_0^T v^2 dt}$$

Now, for sinusoidal functions, using the above equation we get the RMS value by dividing the peak value (V_m) by the square root of 2. That is,

$$V_{rms} = \sqrt{\frac{1}{T}\int_0^T (V_m \sin(2\pi ft))^2 dt} = \sqrt{\frac{1}{2\pi}\int_0^{2\pi} (V_m \sin\theta)^2 d\theta} = \frac{V_m}{\sqrt{2}}$$

Similarly, for currents, **I = Im/(√2)**. These RMS values can be used directly for power calculation. The formula for average power is given by $P_{avg} = \sqrt{\frac{1}{T}\int_0^T (VI)\, dt}$. And for sinusoids this leads to $P_{avg} = VI \cos\theta$. Here, V and I are rms values and θ is the phase angle between voltage wave and current wave. The phase angle is explained in the next section.

Phase Angle

The phase difference between two ac sinusoidal waveforms is the difference in the electrical angle between two identical points of the two waves. In **figure 2**, the voltage and current are represented by the red and blue lines respectively. Their equations are given as:

$V(t) = V_m \sin(2\pi ft)$

$$I(t) = I_m \sin(2\pi f t - \theta)$$

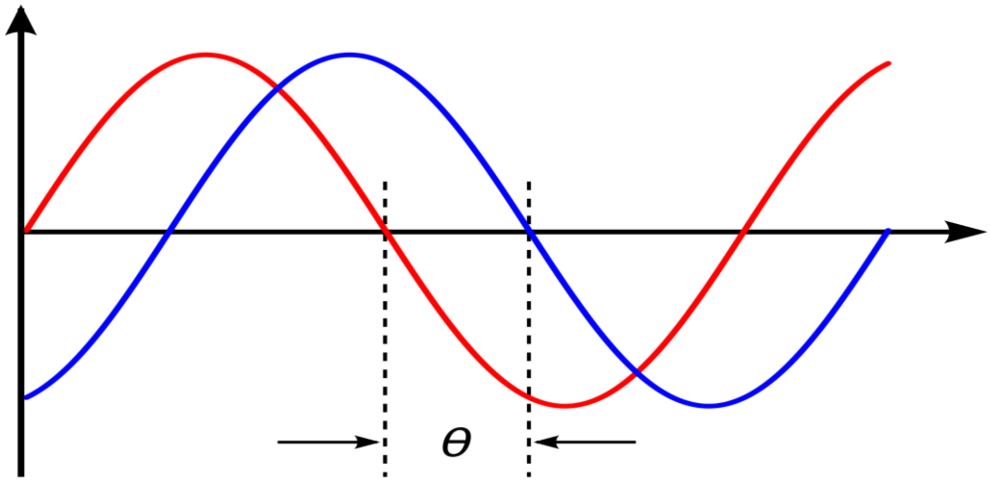


Figure 2: Phase Angle between two signals

It is worth noting that one of the signals would be leading the other one. In other words, the other signal would be lagging the first one. The phase difference can be determined by observing the difference between the time/phase corresponding to the peaks (either both positive or both negative) of the signals, or, alternatively, the zero crossing times from the same side.

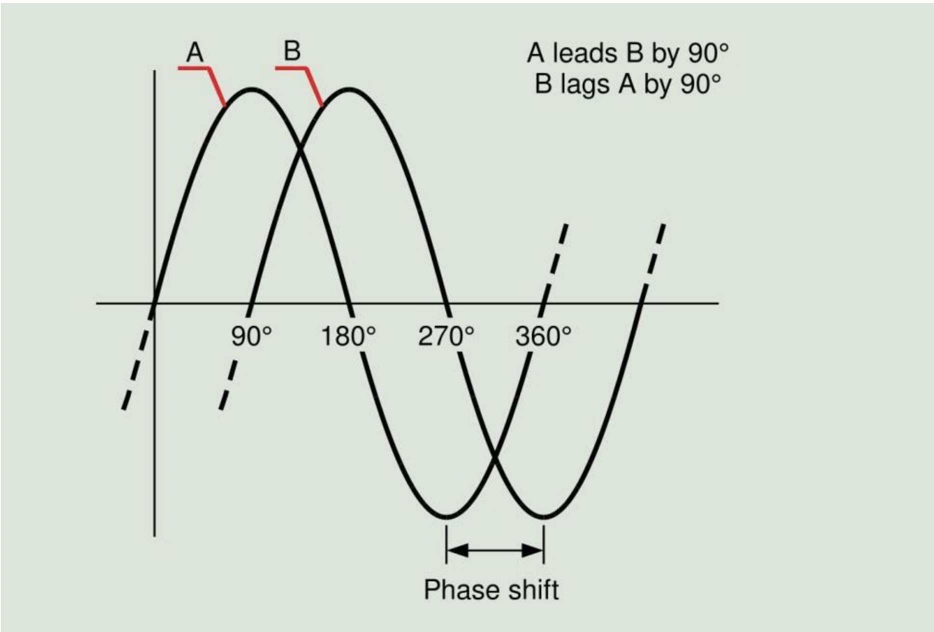


Figure 3: Concept of Leading & Lagging

If the time difference between the two positive peaks/negative peaks/zero crossings is found to be $\mathbf{t_{diff}}$, Then the **Phase Difference** $= \mathbf{t_{diff} * f * 360^{\circ}}$, where, **f** = frequency.

Impedance

For AC circuit analysis, impedance plays the same role as resistance plays in DC circuit analysis. It can be stated safely, that the concept of impedance is the most important thing that makes AC analysis so much popular among engineers. As you will see in your later courses, any other periodic forms of time-varying voltages or currents, are converted into an equivalent series consisting of sines and cosines (much like any function can be expanded by the power series of the independent variable using the Taylor series), only because the analysis of sinusoidal voltages becomes very much simple due to the impedance technique.

What is impedance anyway? Putting it simply, it is just the ratio of RMS voltage across the device to the RMS current through it. That is:

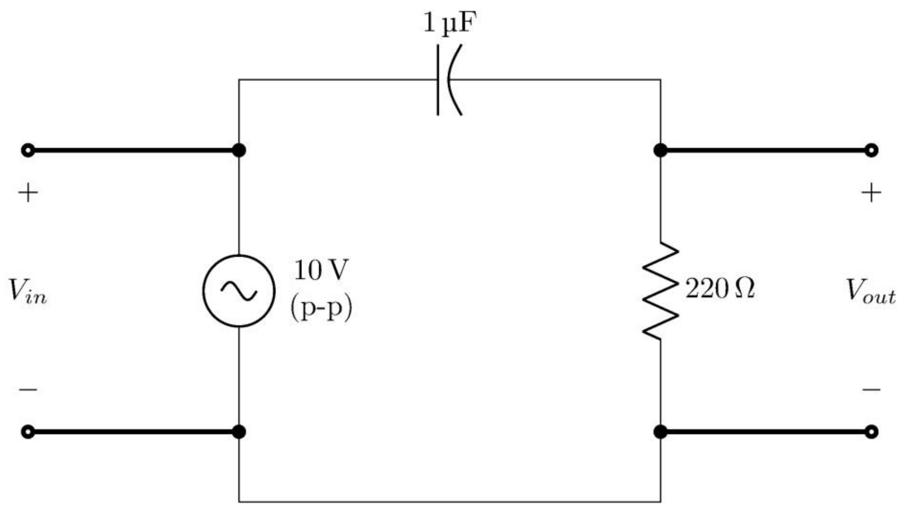
$$\text{Impedance, } Z = I\angle\boldsymbol{\theta}\underline{V} = I_m V\overline{\angle\boldsymbol{\theta}_m}$$

Its unit is Ohms (**Ω**).

Procedure

Simulation using LTspice

We will familiarize you with the Alternating Current (AC) Waves in LTspice by simulating the simple circuit shown below (**Circuit 1**). Let us visualize V_{in} and V_{out} and find out the phase difference introduced by the capacitor in this circuit..



Circuit 1

Let us simulate this circuit step by step in LTSpice as described below:

- ➢ Open a new schematic window by clicking ***File → New Schematic***. Draw the circuit shown below in **Figure 1.** Modify the components with their values and name the nodes. To name the nodes, ***Right-click on the wire/node → Label Net***. Do not forget to add a ground to the circuit.

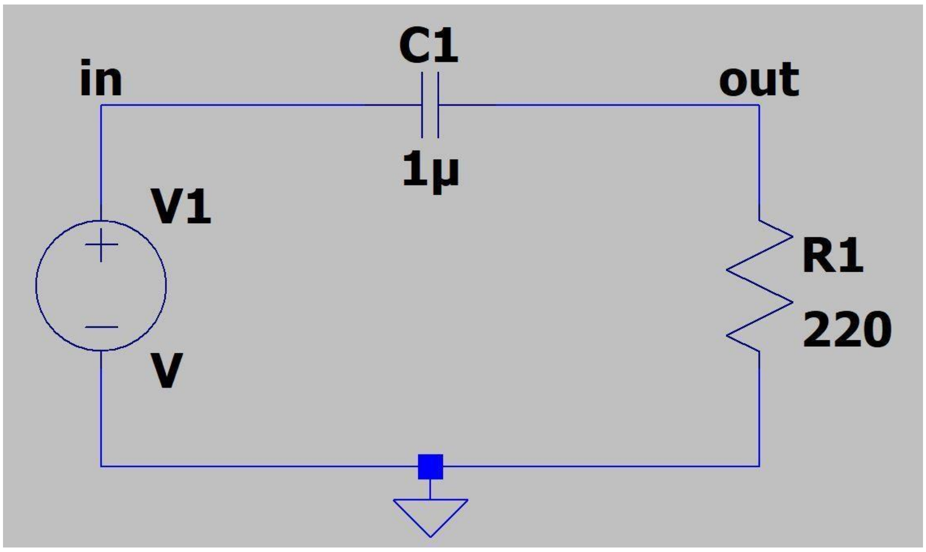
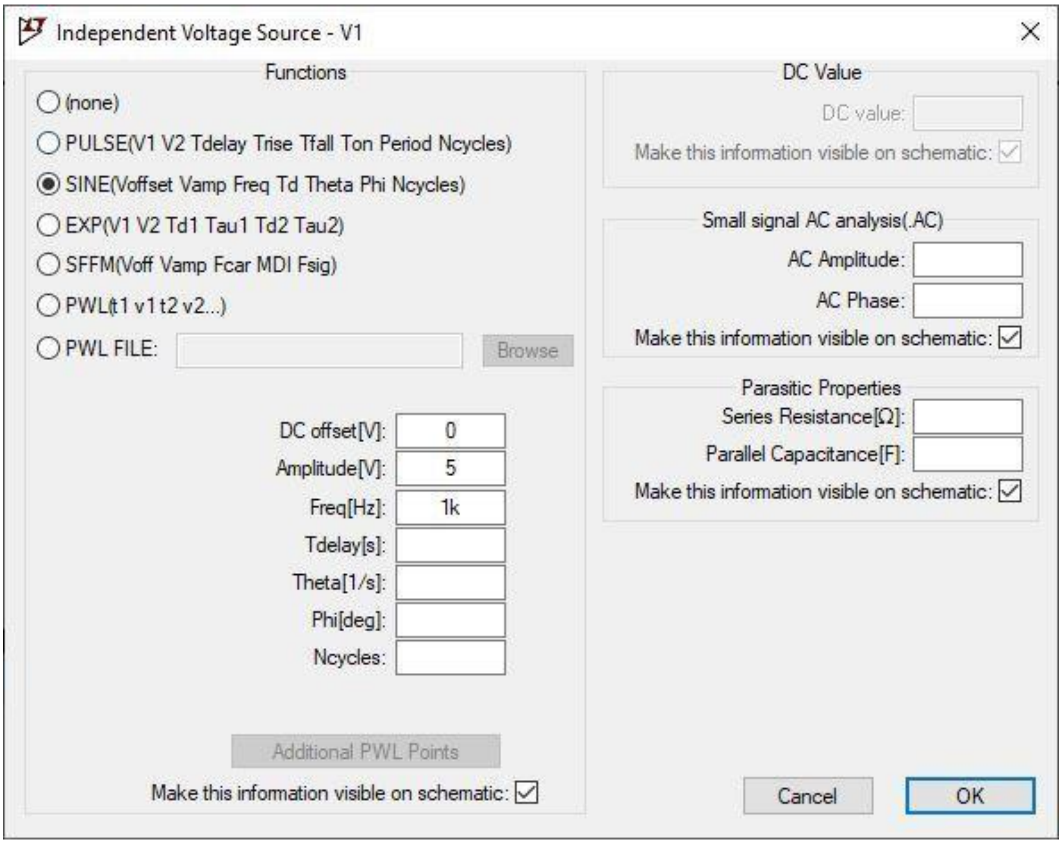


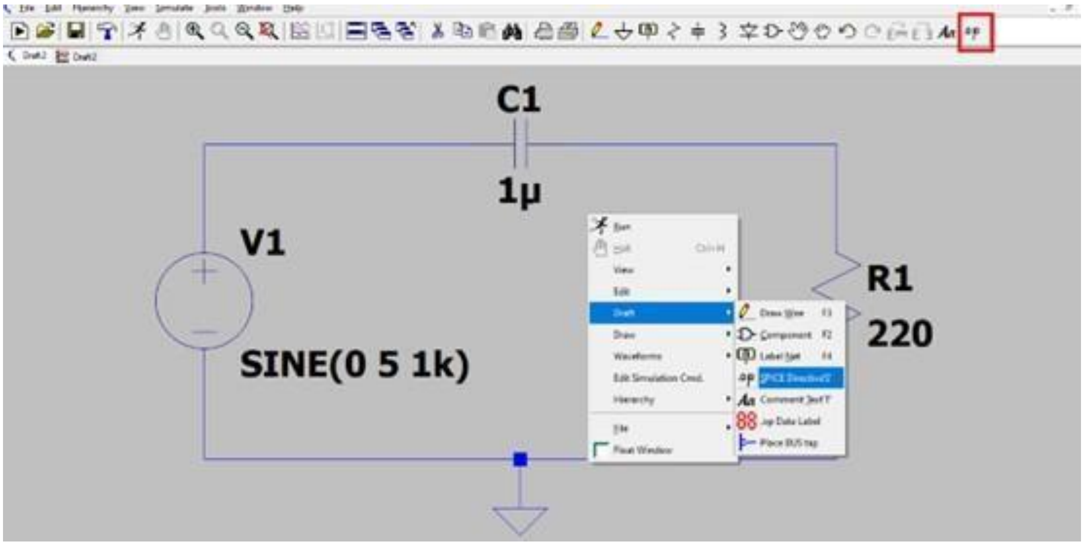
Figure 1

- ➢ To modify the voltage source as an AC voltage source, ***Right-click on the voltage source → Select Advanced → insert the values as below and, click OK***- for a **10V p-p 1 KHz** AC voltage source.

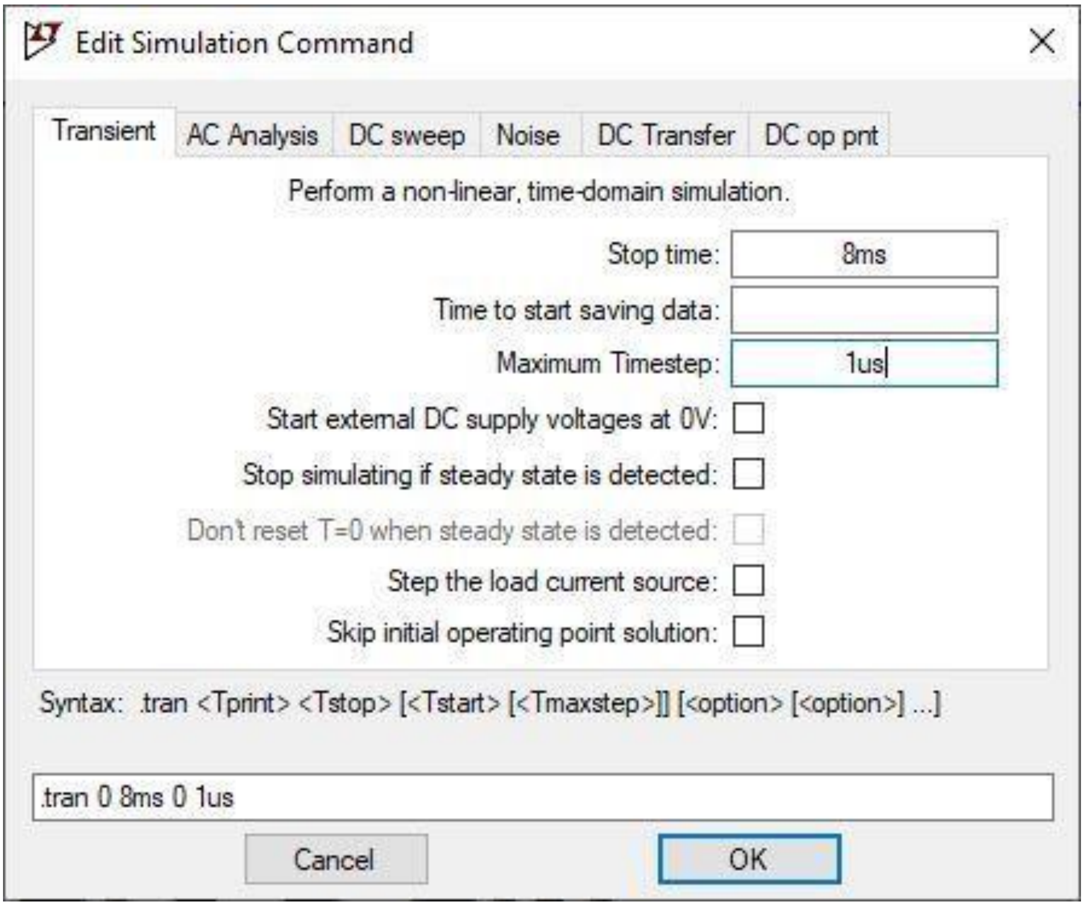


- ➢ To see the responses/waveshapes we must do ***'Transient Analysis'***. The transient analysis calculates a circuit's response over a period.

To run the transient analysis, we have to write the analysis command. Find ***the 'Spice Directives'*** option by ***Right-clicking on the schematic → Draft → Spice Directives or, find it from the toolbar above.***

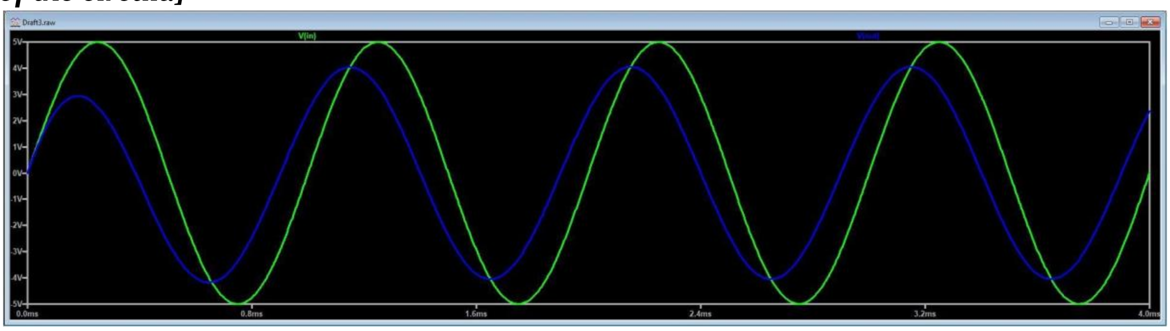


➢ After clicking the ***'Spice Directives'***, the ***'Edit Text on the Schematic'*** window will appear. Now ***Right-click on the blank space on this window → Select 'Help me Edit' → Analysis Command.*** A window titled ***'Edit Simulation Command'*** will appear. Insert values in the boxes as below and click OK. It will generate a transient analysis command. Place the command somewhere on the schematic. *[Notice the '.tran' syntax for transient analysis.]*



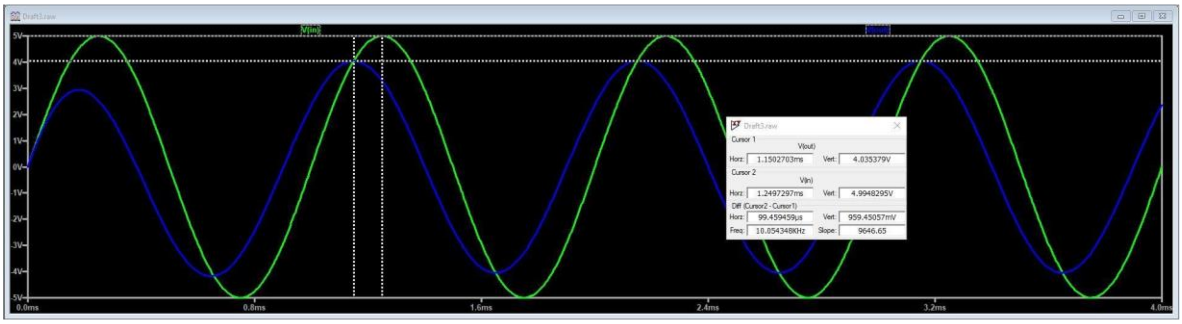
➢ To run the simulation, click ***'Run'***. Find the ***'Run'*** button from the above toolbar or by just Right-clicking on the schematic.

➢ After clicking the ***'Run'*** button a plot window will appear. In this window we can see responses and waveshapes of voltage and currents with respect to time. To see a plot, ***Right-click on the plot window → Add trace → Select any voltage or current → OK. [We can also add trace by simply using markers on the schematic. When the run is complete a cursor will appear if we place the mouse cursor on a wire or component of the circuit.]***



The axes properties (Range) can be changed by **Right-clicking on the horizontal (x-axis) and the vertical (y-axis).**

- To extract data from a plot/response, use the data cursor. **A cursor for a particular trace will appear by clicking on the name of that trace. One click will produce one cursor, clicking twice will produce two. Two produce two cursors for two different traces, first, click twice on the name of one trace. Then, click once on the name of the other trace.** The data point of the cursor can be moved by the arrow keys from the keyboard.



- A window will appear on the bottom right corner containing the values corresponding to the cursors. Note that it also shows the difference between the two cursors (data points) for both the vertical and the horizontal axes. Use this to find **Time difference, t_{diff}** between V_{in} & V_{out} (Notice that V_{out} is the representative of supply current in this circuit in mA). Use t_{diff} to determine **Phase Difference**.

- Save the Schematic by clicking ***File*** → ***Save as*** → ***'Name.asc'*** and the plots by clicking ***File*** → ***Save plot settings*** → ***'Name.plt'*** for future use and analysis.

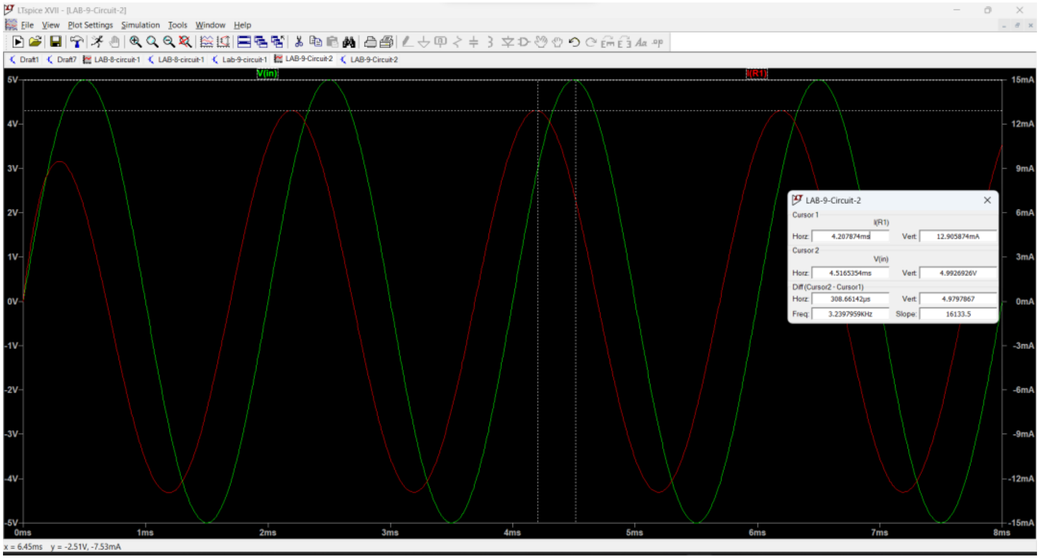
Lab Work

1. Measure the **Phase Difference** between V_{in} & V_{out} from the plot in degrees.

From the Circuit we simulated, the **Phase Difference =** 36.0021 degrees

2. Perform similar analysis to Measure the **Phase Difference** between V_{in} & V_{out} when the frequency of the supply voltage is 500 Hz. Do the same for 2 KHz. Observe the effect of changing the frequency on the Phase Difference.

For the plot of 500Hz,



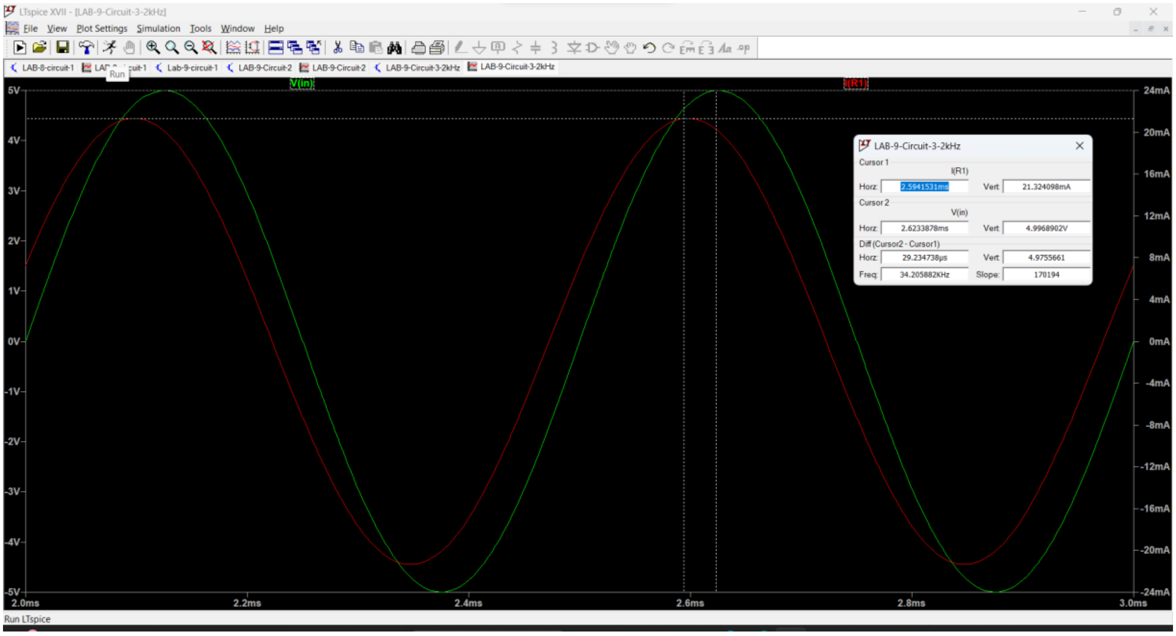
Here, phase difference, Phase Difference = tdiff * f * 360°

$$= 308.66/2000\text{sec}*360°$$

$$= 55.56°(\text{degrees})$$

Again,

For the plot of 2kHz,



Here, phase difference, Phase Difference = tdiff * f * 360°

= (30/50)*360°

= 21.6°(degrees)

So, from the above calculation and graph plot, it can be said that, when the frequency gets increased, the phase difference gets decreased at a remarkable rate.

3. State your opinion on your observations on the effect of changing the frequency on the Phase Difference. Does it make any difference?

After obesrving the graphs,

It is clearly visible that whenever the frequency gets increased, the phase difference gets decreased at a remarkable rate.

In other words, The phase difference between the voltage and current also shifted when we increased the frequency.

Yes,it makes diffrence. Here, it modifies the way that voltage and current interact. It's pretty cool that knowing this makes it easier for us to predict and regulate how AC circuits will behave.

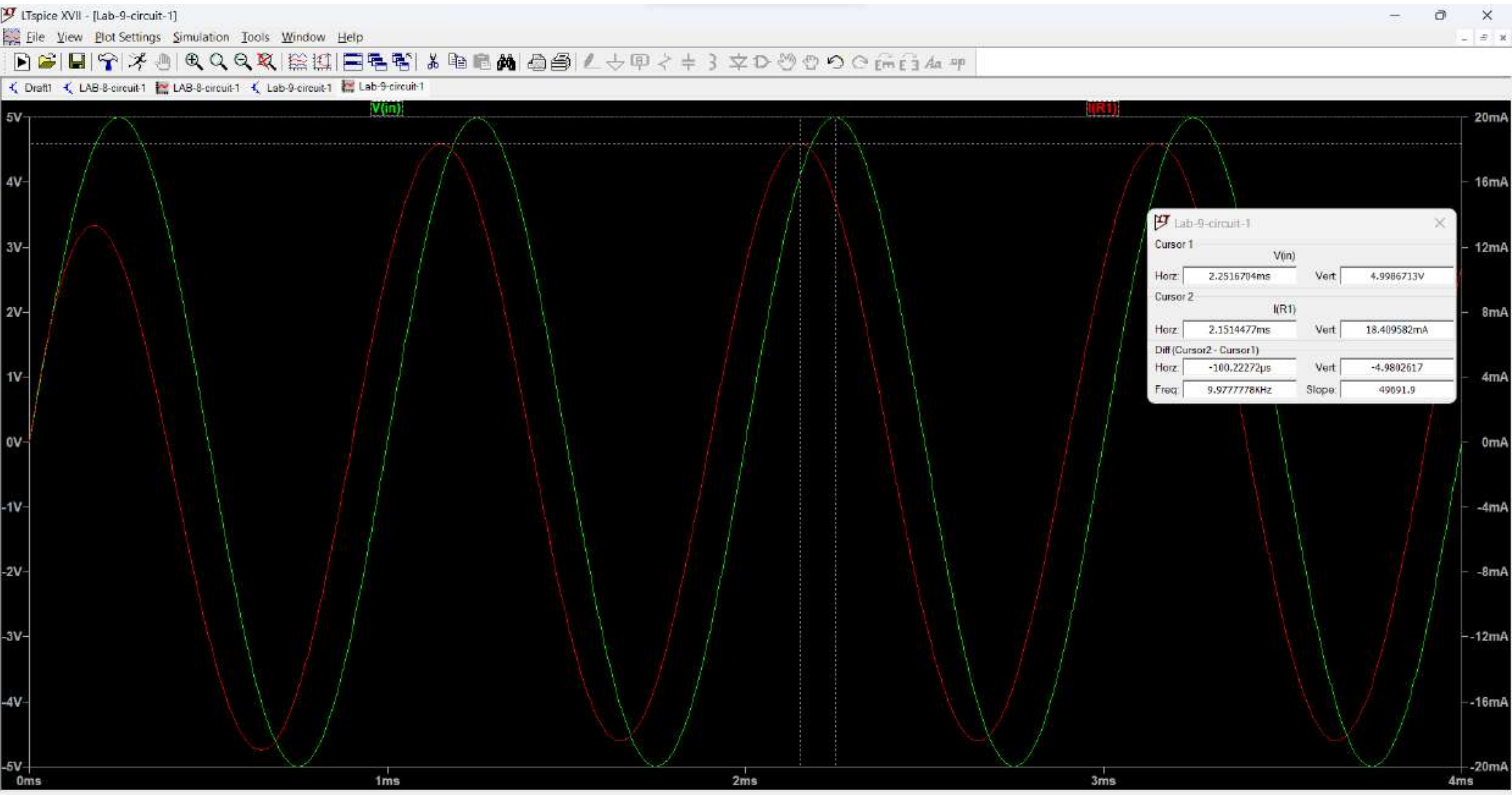
Report

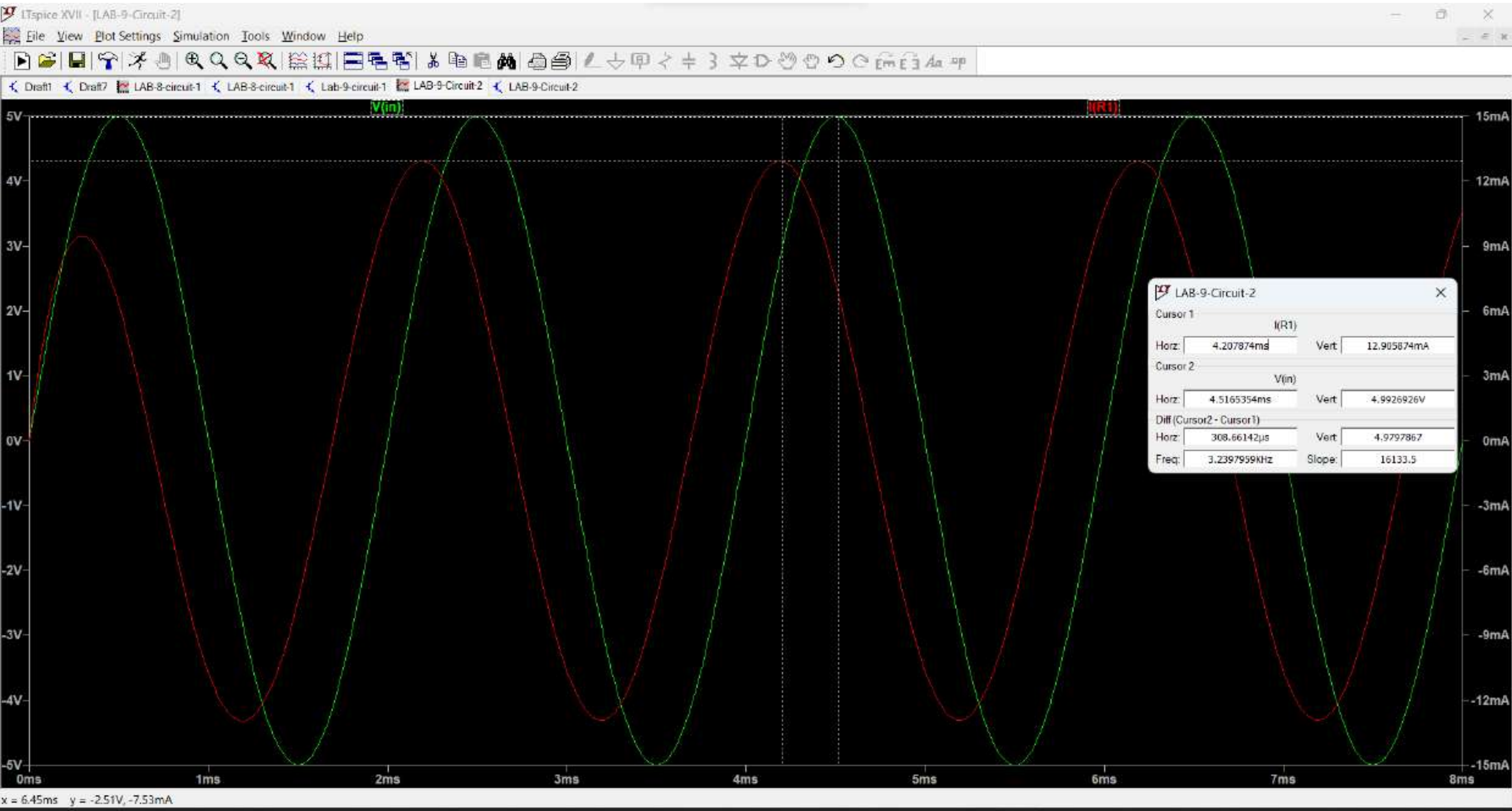
1. Answer to questions and Complete the Lab work sections.
2. Save all your .asc and .plt files and make a zip file. You need to submit it with the report.
3. Discussion *[comment on the obtained results and discrepancies]*. Add pages if necessary.

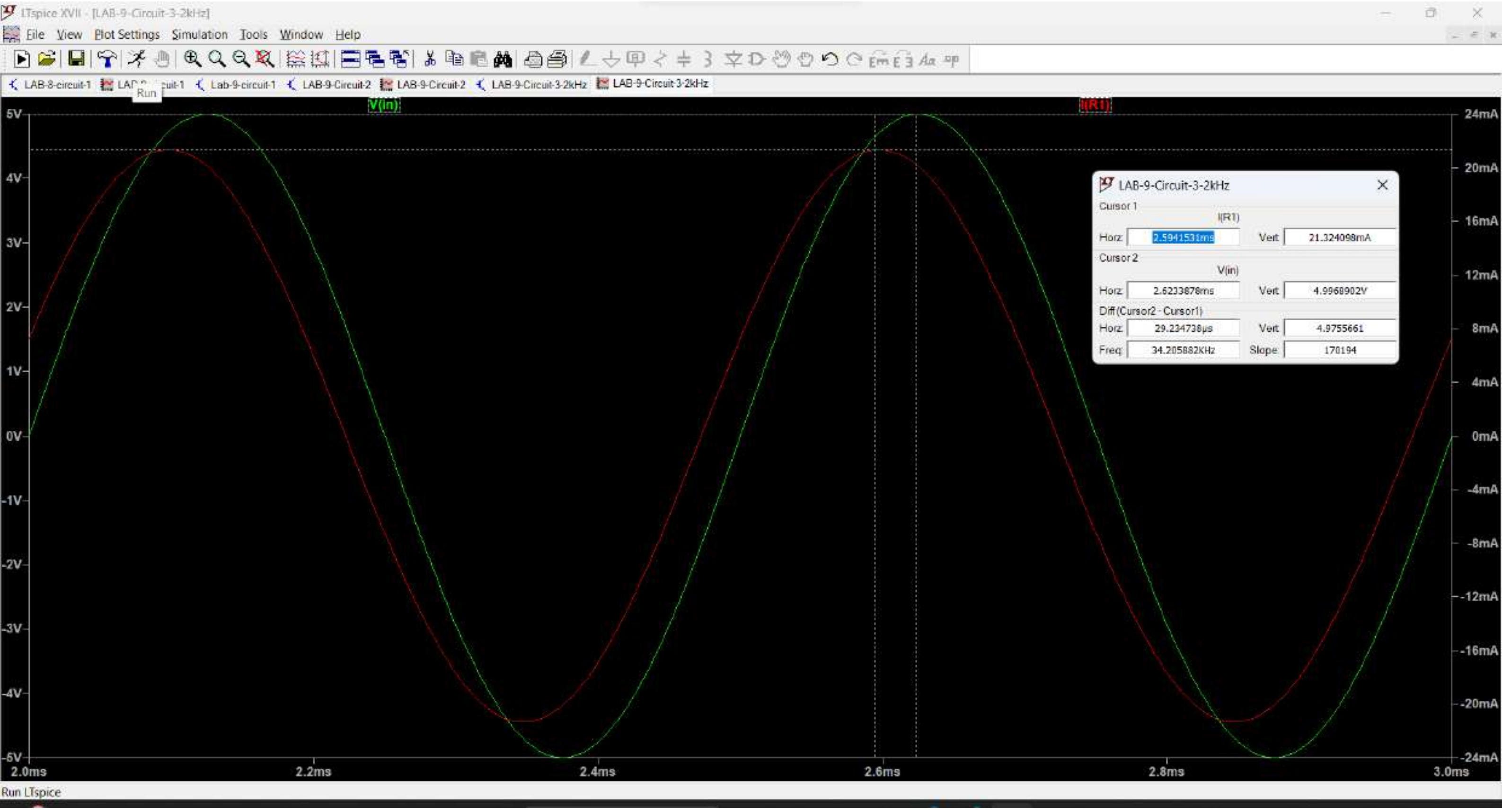
Discussion :

In this experiment,

Changing the frequency in an AC circuit using LTSpice showed some interesting results. The objective of the lab report "Lightweight Current (AC) Waves Using LTSpice Simulation" was to find out how frequency variation impacted phase difference. The experiment demonstrated that the phase difference between two or more waveforms can fluctuate when the frequency of an AC wave is changed. An alternating waveform's phase difference can range from 0 to its maximum duration, T, over a full cycle. Depending on the angular units used, this can occur anywhere along the horizontal axis between $\Phi = 0$ and 2π (radians) or $\Phi = 0$ and $360°$.


LTspice XVII - [Lab-9-circuit-1]
File View Plot Settings Simulation Tools Window Help
Draft1 LAB-8-circuit-1 LAB-8-circuit-1 Lab-9-circuit-1 Lab-9-circuit-1
V(in)
I(R1)
Lab-9-circuit-1
Cursor 1
V(in)
Horz: 2.2516704ms Vert: 4.9986713V
Cursor 2
I(R1)
Horz: 2.1514477ms Vert: 18.409582mA
Diff (Cursor2 - Cursor1)
Horz: -100.22272µs Vert: -4.9802617
Freq: 9.9777778KHz Slope: 49691.9
5V 4V 3V 2V 1V 0V -1V -2V -3V -4V -5V
20mA 16mA 12mA 8mA 4mA 0mA -4mA -8mA -12mA -16mA -20mA
0ms 1ms 2ms 3ms 4ms


LTspice XVII - [LAB-9-Circuit-2]
File View Plot Settings Simulation Tools Window Help
Draft1 Draft7 LAB-8-circuit-1 LAB-8-circuit-1 Lab-9-circuit-1 LAB-9-Circuit-2 LAB-9-Circuit-2
V(in)
I(R1)
5V
4V
3V
2V
1V
0V
-1V
-2V
-3V
-4V
-5V
15mA
12mA
9mA
6mA
3mA
0mA
-3mA
-6mA
-9mA
-12mA
-15mA
0ms
1ms
2ms
3ms
4ms
5ms
6ms
7ms
8ms
LAB-9-Circuit-2
Cursor 1
I(R1)
Horz: 4.207874ms
Vert 12.905874mA
Cursor 2
V(in)
Horz: 4.5165354ms
Vert 4.9926926V
Diff (Cursor2 - Cursor1)
Horz: 308.66142µs
Vert 4.9797867
Freq: 3.2397959KHz
Slope: 16133.5
x = 6.45ms y = -2.51V, -7.53mA



LTspice XVII - [LAB-9-Circuit-3-2kHz]
File View Plot Settings Simulation Tools Window Help
LAB-8-circuit-1
Lab-9-circuit-1
LAB-9-Circuit-2
LAB-9-Circuit-2
LAB-9-Circuit-3-2kHz
LAB-9-Circuit-3-2kHz
Run
V(in)
I(R1)
5V
4V
3V
2V
1V
0V
-1V
-2V
-3V
-4V
-5V
24mA
20mA
16mA
12mA
8mA
4mA
0mA
-4mA
-8mA
-12mA
-16mA
-20mA
-24mA
2.0ms
2.2ms
2.4ms
2.6ms
2.8ms
3.0ms
LAB-9-Circuit-3-2kHz
Cursor 1
I(R1)
Horz: 2.5941531ms
Vert: 21.324098mA
Cursor 2
V(in)
Horz: 2.6233878ms
Vert: 4.9968902V
Diff (Cursor2 - Cursor1)
Horz: 29.234738µs
Vert: 4.9755661
Freq: 34.205882KHz
Slope: 170194
Run LTspice