BRAC UNIVERSITY
DEPT. OF COMPUTER SCIENCE AND ENGINEERING
COURSE NO.: CSE250
Circuits and Electronics Laboratory

Name of the Experiment: Familiarization with the alternating current (AC) waves.

OBJECTIVE:

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

- Oscilloscope
- How to measure peak value, phase angle, and time period (or frequency) using an oscilloscope
- The methods of measuring rms. value both using an 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.
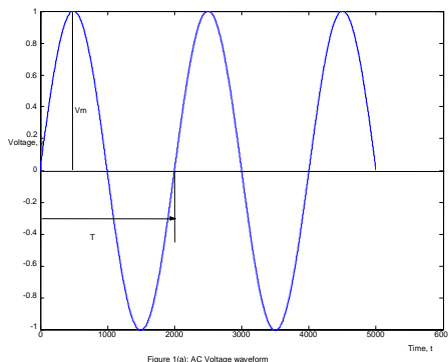

Figure 1(a): AC Voltage waveform

Here, the time period = T

Frequency, $f = \dfrac{1}{T}$

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

Effective value:

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

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,

$$V = \sqrt{\frac{1}{T}\int_0^T (V_m \sin(2\pi ft))^2\, dt}$$

$$= \sqrt{\frac{1}{2\pi}\int_0^{2\pi} (V_m \sin(\theta))^2\, d\theta} = \frac{V_m}{\sqrt{2}}$$

Similarly, for currents, $I = \dfrac{Im}{\sqrt{2}}$. These rms. values can be used directly for power calculation. The formula for average power is given by $P_{avg} = \frac{1}{T}\int_0^T (vi)\,dt$. 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:

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 equations are given as:

$V = V_m \sin(2\pi ft)$
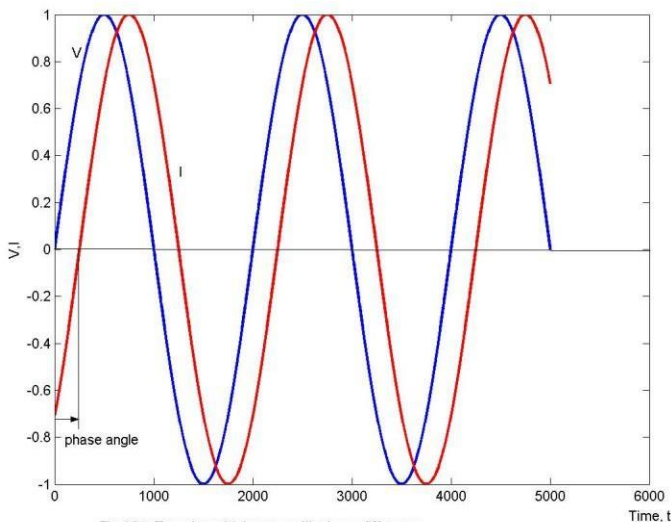$I = I_m \sin(2\pi f.t - \theta)$

Fig 1(b): Two sinusoidal waves with phase difference

Impedance:

For, ac circuit analysis, impedance plays the same role as resistance plays in dc circuit analysis. It can be stated fairly safely that, the concept of impedance is the most important thing that makes the ac analysis so much popular to the 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 are 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:

$$Z = \frac{V}{I \angle \theta} = \frac{V_m}{I_m \angle \theta}$$

Its unit is ohms.

CIRCUIT DIAGRAM:
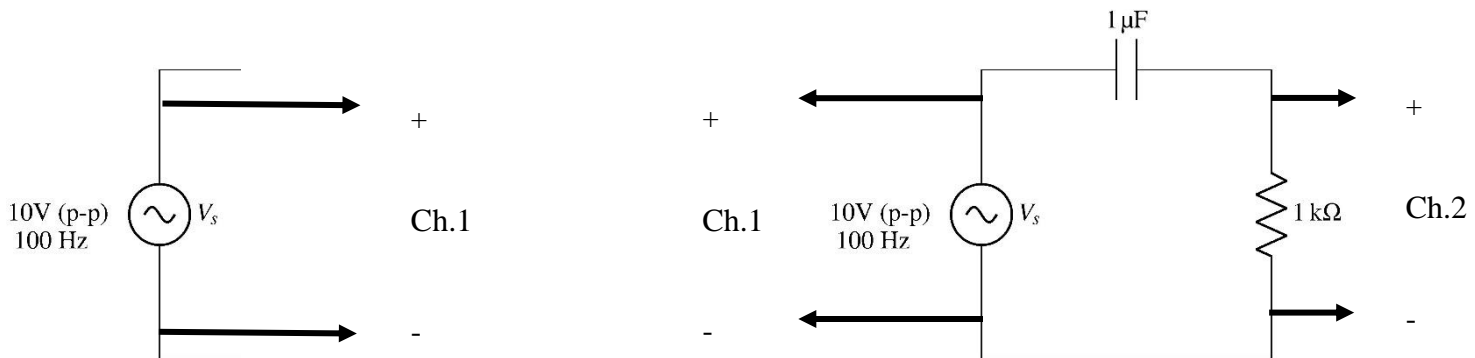
CIRCUIT SIMULATION USING LTspice:

1. Draw the circuit shown in Fig. 1 in a new schematic in LTspice. 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.**

2. 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.
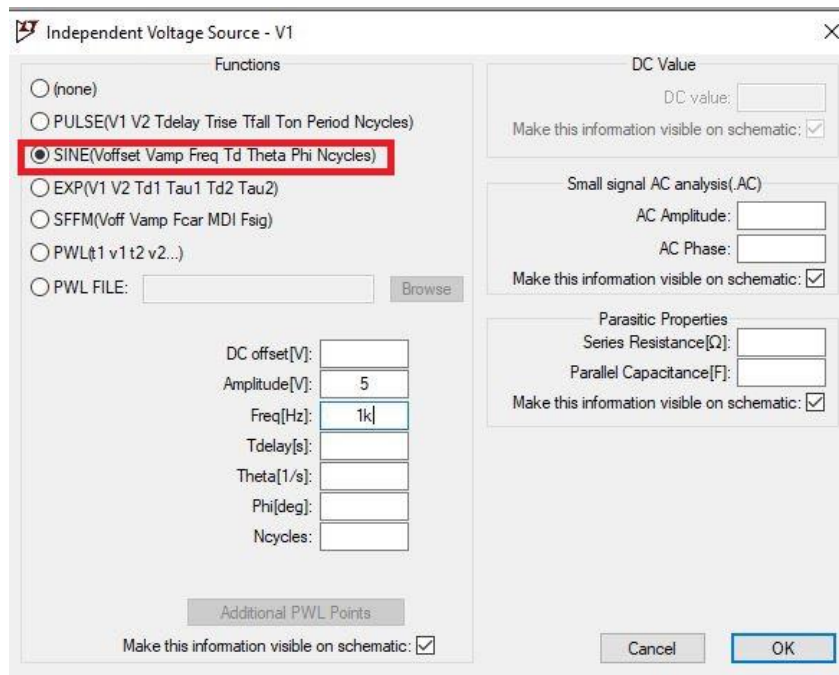
3. To see the responses/waveshapes we have to do *'Transient Analysis'*. The transient analysis calculates a circuit's response over a period of time.

   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.***
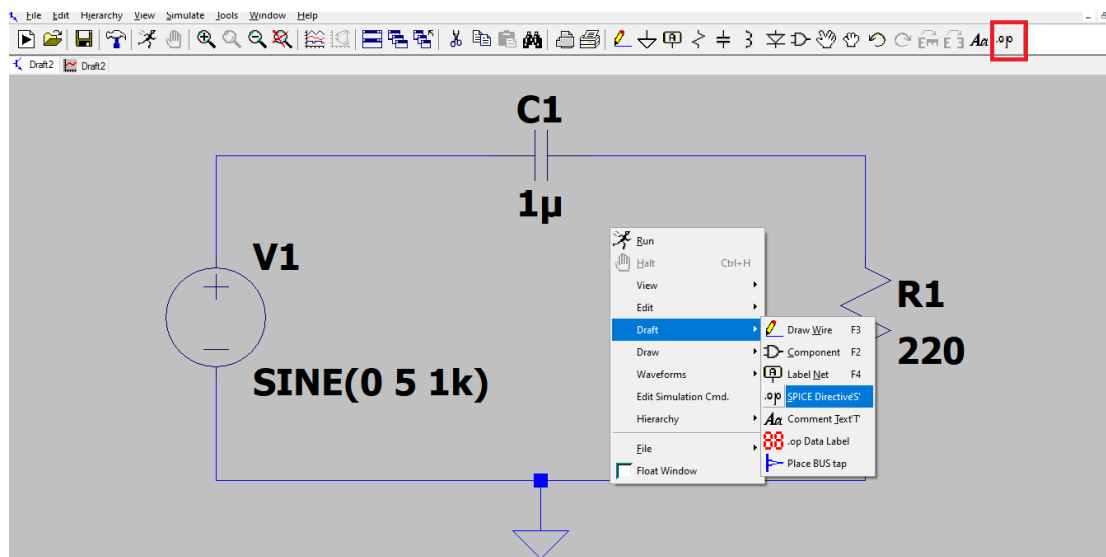
4. After clicking the *'Spice Directives'*, the *'Edit Text on the Schematic'* window will appear. Now *Right-click on the blank space on this window → Select 'Help me Edit' → Analysis Command.* A window titled *'Edit Simulation Command'* will appear. Insert values in the boxes as below and click OK. It will generate a transient analysis command. Place the command somewhere on the schematic. [Notice the '.tran' syntax for transient analysis.]
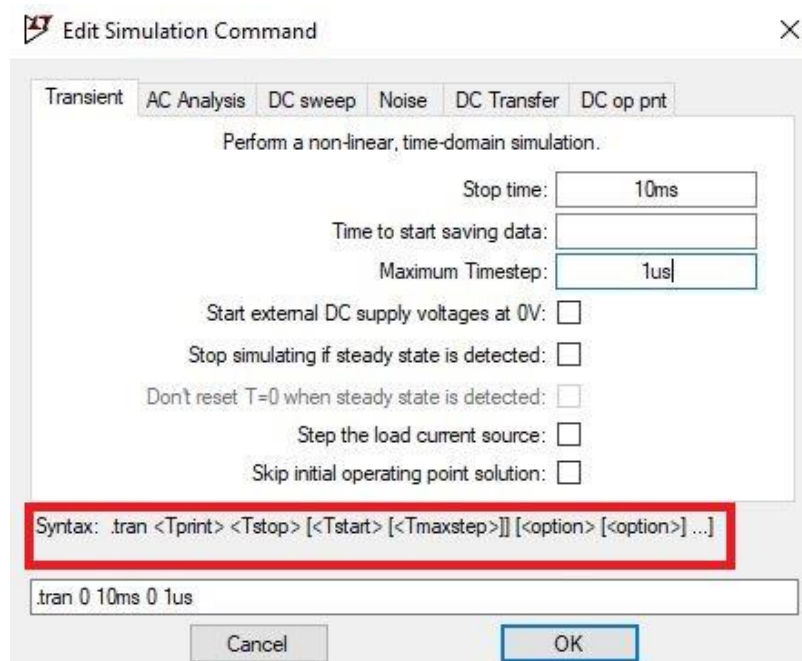
5. To run the simulation, click *'Run'*. Find the *'Run'* button from the above toolbar or by just Right-clicking on the schematic.

6. 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 marker 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.]

7. 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. The data point of the cursor can be moved by the arrow keys from the keyboard.
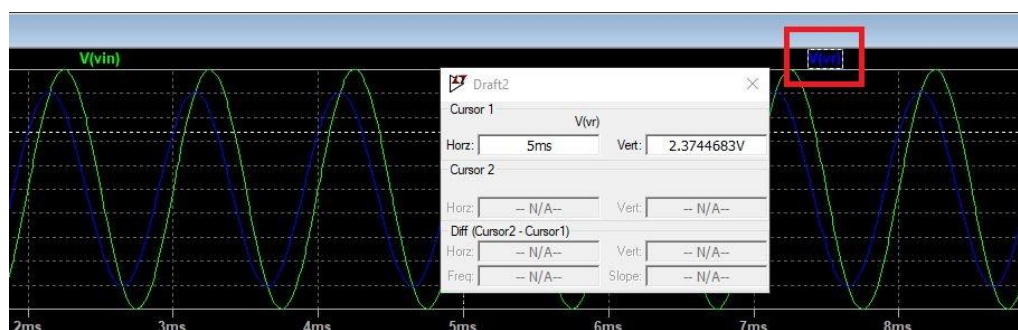
The axes properties (Range) can be changed by Right-clicking on the horizontal (x-axis) and   the vertical (y-axis).

8.   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 TASKS:

1.   Built and simulate the circuit shown in Fig. 1 in LTspice.

2.   Observe the plots for Vin, V$_R$ **,** Current I and note the peak to peak amplitudes, frequency and, time periods.

3.   Measure the phase difference between Vin and I. The phase difference is given by **360f.t degree**, where 't' is the **time delay** between the two waves.

4.   Calculate the impedance using these values of voltage, currents and, phase difference.

REPORT:

1.   Show the data measured in Lab Tasks in a tabular form.

2.   Repeat the Lab Tasks for 10V p-p AC source of 500 Hz and 2 KHz separately and show the results in a tabular form.

3.   Attach screenshots of the plots and circuits in your report.