ELECTRICAL ENGINEERING SIMULATION LABORATORY

LAB MANUAL

Academic Year :   2018 - 2019
Course Code   :   AEE105
Regulations   :   IARE - R16
Class   :   II Year I Semester

Department of Electrical and Electronics Engineering

INSTITUTE OF AERONAUTICAL ENGINEERING

(Autonomous)

Dundigal – 500 043, Hyderabad

INSTITUTE OF AERONAUTICAL ENGINEERING

(Autonomous)

Dundigal, Hyderabad - 500 043

ELECTRICAL AND ELECTRONICS ENGINEERING

	Program Outcomes
PO1	**Engineering knowledge:** Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.
PO2	**Problem analysis:** Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences
PO3	**Design/development of solutions:** Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.
PO4	**Conduct investigations of complex problems:** Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.
PO5	**Modern tool usage:** Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.
PO6	**The engineer and society:** Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.
PO7	**Environment and sustainability:** Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.
PO8	**Ethics:** Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.
PO9	**Individual and team work:** Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.
PO10	**Communication:** Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.
PO11	**Life-long learning:** Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.
PO12	**Project management and finance:** Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects and in multidisciplinary environments.

Program Specific Outcomes	
PSO1	**Professional Skills:** Able to utilize the knowledge of high voltage engineering in collaboration with power systems in innovative, dynamic and challenging environment, for the research based team work.
PSO2	**Problem - Solving Skills:** To explore the scientific theories, ideas, methodologies and the new cutting edge technologies in renewable energy engineering, and use this erudition in their professional development and gain sufficient competence to solve the current and future energy problems universally.
PSO3	**Successful Career and Entrepreneurship:** To be able to utilize of technologies like PLC, PMC, process controllers, transducers and HMI and design, install, test, and maintain power systems and industrial applications.

INDEX

ATTAINMENT OF PROGRAM OUTCOMES & PROGRAM SPECIFIC OUTCOMES

Exp. No.	Experiment	Program Outcomes Attained	Program Specific Outcomes Attained
1	Measurement of three phase active and reactive power for balanced and unbalanced loads	PO1,PO5	PSO2
2	Plot the locus diagrams of series RL and RC circuits	PO1,PO2,PO5	PSO2
3	To calculate and verify 'Z' parameters and 'Y' parameters of two-port network.	PO1,PO2,PO5	PSO2
4	To calculate and verify 'ABCD' parameters and 'H' parameters of two-port network.	PO4,PO5	PSO2
5	Fourier analysis of square wave, half wave rectified and full wave rectified sine wave using MATLAB.	PO1,PO2,PO5	PSO2
6	Draw the electrical symbols using VISSIO software.	PO1,PO2,PO5	PSO2
7	To study and plot the transient response of series and parallel RL and RC circuits using MATLAB.	PO2,PO3,PO5	PSO2
8	To study and plot the transient response of series and parallel RLC circuit using MATLAB.	PO2,PO3,PO5	PSO2
9	Simulation of low pass and high pass filters using digital simulation.	PO2,PO3,PO5	PSO2
10	Editing and building a VI, creating a sub VI.	PO2,PO3,PO4,PO5	PSO2
11	Using FOR loop, WHILE loop, charts and arrays, graph and analysis VIs.	PO2,PO3,PO4,PO5	PSO2
12	Signal generation, display of wave form, minimum and maximum values of wave form, modulation.	PO3,PO4,PO5	PSO2
13	Three phase sine wave generation and display.	PO3,PO4	PSO2
14	Frequency measurement using Lissajous figures in LabVIEW.	PO1,PO3,PO4	PSO2

ELECTRICAL CIRCUITS AND VIRTUAL INSTRUMENTATION LABORATORY

OBJECTIVE:

The objective of the electrical circuits and virtual instrumentation laboratory is to expose the students to apply different techniques used in circuit analysis to calculate circuit parameters and two port network parameters, applications of Fourier transforms in electric circuits and design filters and analyze through digital simulation. It also aims to introduce MATLAB, PSPICE a circuit simulation software tools. It enables the students to gain sufficient knowledge on the programming and simulation of Electrical circuits.

OUTCOMES:

Upon the completion of electrical circuits and virtual instrumentation laboratory practical course, the student will be able to attain the following:

1. Familiarity with AC circuit analysis techniques.
2. Analyze complicated circuits using different network functions or two port parameters.
3. Acquire skills of using MATLAB, PSPICE software for electrical circuit studies.

EXPERIMENT – 1

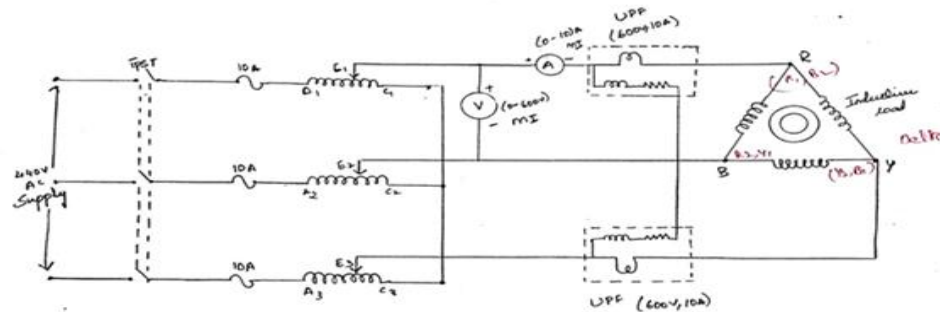# MEASUREMENT OF THREE PHASE ACTIVE AND REACTIVE POWER FOR BALANCED AND UNBALANCED LOADS

1.1 AIM:

Measurement of three phase active power for balanced star and delta load.

1.2 APPARATUS:

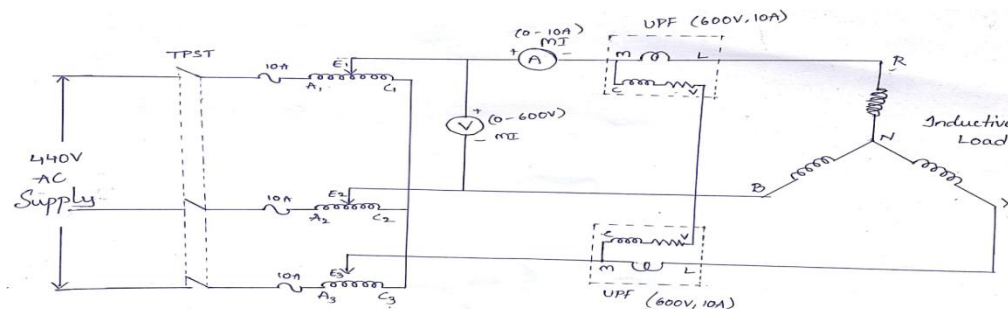| S. No.	Name of the Equipment	Range	Type	Quantity
1	Voltmeter			
2	Ammeter			
3	Three Phase Auto -Transformer			
4	Wattmeters			
5	Connecting Wires			

1.3 CIRCUIT DIAGRAM:

1.3.1 Delta Connected Load:

Fig – 1.1 Measurement of Active Power of Balanced Delta Connected Load

1.3.2 Star Connected Load

Fig – 1.2 Measurement of active power of balanced Star connected load

1.4   PROCEDURE:

1. Give the connections as per the circuit diagram.
2. Switch on   the three-phase supply by closing TPST switch.
3. Slowly apply the three-phase input voltage of 440V using three-phase auto-transformer.
4. Now apply the star connected load and note down the readings of ammeter, voltmeter and wattmeters.
5. Repeat the step 4 for different loads.
6. Calculate the three-phase apparent power and reactive power using below formulae.
7. Repeat the steps 3,4,5,6 for delta connected load.

1.5   THEORETICAL CALCULATIONS:

1.5.1 ACTIVE POWER:

Generally, the reading of wattmeter itself indicates the active power. There by the three-phase power using two-wattmeter method is given as $= W_1 + W_2$.Where, W1 --- first wattmeter reading. W2 --- second wattmeter reading Active power $= W_1 + W_2$

1.5.2 REACTIVE POWER:

Generally, the reading of wattmeter itself indicates the active power. There by the three-phase power using two-wattmeter method is given as $= W1 + W2$. Where, W1 --- first wattmeter reading W2 --- second wattmeter reading

Hence $W_1 + W_2 = 3*V_L*I_L *COS\Phi$. ---------------------------1

$$\Phi = tan^{-1} \{1.732*(W_1 - W_2) / (W_1 + W_2)\}-------------2$$

Then three-phase reactive power is give   as $= 1.732* V_L*I_L.SIN\Phi$ ---------3

From equation 1 and 3 the apparent power is given as

$= \sqrt{(\text{active power})^2 + (\text{reactive power})^2}$

NOTE:

If any wattmeter shows reverse reading interchange M and L terminals of that wattmeter.

1.6   TABULAR COLUMN:

1.6.1 FOR STAR CONNECTED LOAD

Load Number	Line Current	Line Voltage	Watt meter 1(W_1)	Watt meter2 (W_2)	Active Power W_1+W_2	Φ	Reactive Power (1.732* $V_L*I_L.SIN\Phi$)

1.6.2 FOR DELTA CONNECTED LOAD

Load Number	Line Current	Line Voltage	Wattmeter 1(W_1)	Watt meter2 (W_2)	Active Power W_1+W_2	Φ	Reactive Power (1.732* V_L*I_L.SINΦ)

1.7   PRECAUTIONS:

1   Check that TPST switch is properly closed are not.

2   Check that there should not be any loose connections.

1.8   RESULT:

1.9   PRE-LAB VIVA QUESTIONS:

ACTIVE POWER

1. Define active power.
2. Define reactive power.
3. Define apparent power.
4. Define power factor.

REACTIVE POWER

1. Write expression reactive power.
2. Define power factor.
3. Why reactive power is zero for a purely resistive load?
4. Write relationship between active, reactive, apparent power.

1.10   LAB ASSIGNMENT:

1   Find out relation between phase voltage and line voltage in star connected load?
2   Find out relation between phase voltage and line voltage in delta connected load?

1.11   POST LAB VIVA QUESTIONS:

1. Give the expression for active power.
2. Give the expression for reactive power.
3. Give the expression for apparent power.
4. Give the relationship between active, reactive, apparent power.
5. What is power triangle?
6. What is the purpose of taking balanced loads?
7. What are the factors on which Reactive power depends?

<h1 style="text-align:center">EXPERIMENT - 2</h1>

<h2 style="text-align:center">SERIES RL AND RC CIRCUIT CURRENT LOCUS DIAGRAMS</h2>

2.1   AIM:

To draw the current locus diagram for the series RL and RC circuits by varying the resistance.

2.2   APPARATUS:

S.No.	Equipment	Range	Type	Quantity
1	Decade resistance box			
2	Decade inductance box			
3	Decade capacitance box			
4	Voltmeter			
5	Ammeter			
6	Auto transformer			
7	Connecting Wires			

2.3   CIRCUIT DIAGRAM:

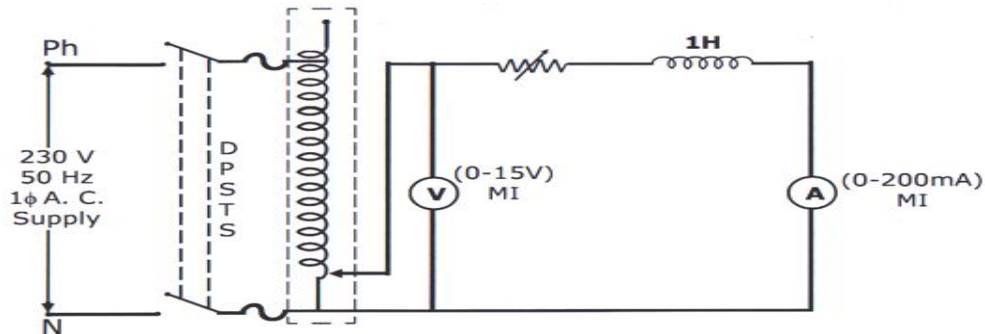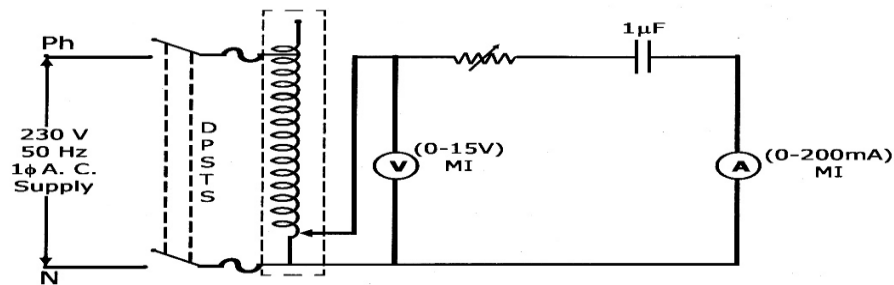<p style="text-align:center">**Fig – 2.1 Series RL Circuit**</p>

2.4   PROCEDURE:

1. The circuit is connected as shown in figure.
2. The supply voltage is set to 10V.
3. Maintain the voltage constant throughout the experiment.
4. Set the value of inductance to 1H.
5. Vary the resistance in steps and note down the current readings.
6. Find the phase angle 'ϕ' using the formula Phase angle 'ϕ' = tan-1 XL / R.
7. Draw the locus diagram between current I and phase angle 'ϕ'.

2.5   TABULAR COLUMN:

S.No	Resistance	Current	$\phi' = \tan^{-1} X_c / R$
1			
2			
3			
4			
5			

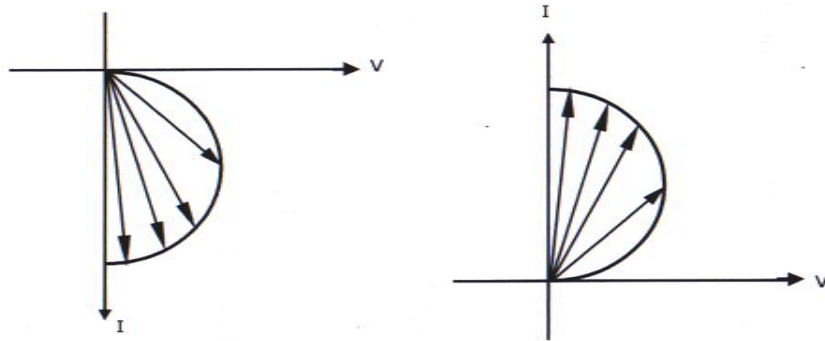## 2.6   CIRCUIT DIAGRAM:

Fig – 2.2 Series RC Circuit

2.7   PROCEDURE:

1. The circuit is connected as shown in figure
2. The supply voltage is set to 10 V.
3. Maintain the voltage constant throughout the experiment.
4. Set the value of capacitance to 1µF.
5. Vary the resistance in steps and note down the current readings.
6. Find the phase angle 'ϕ' using the formula.
7. Phase angle 'ϕ' = $\tan^{-1} Xc / R$.
8. Draw the locus diagram between current I and phase angle 'ϕ'.

2.8   TABULAR COLUMN:

S.No	Resistance	Current	$\phi' = \tan^{-1} Xc / R$
1			
2			
3			
4			
5			

2.9   MODEL GRAPH:

Fig - 2.3 Locus Diagrams for Series RL and RC Circuit

2.10   RESULT:

2.10   PRE LAB VIVA QUESTIONS:

1. Define Locus diagram.
2. How does the voltage varies with respective frequency across inductor?
3. How to draw locus Diagram for Series RL Circuit?
4. How to draw the locus Diagram for Series RC Circuit?

2.11   POST VIVA LAB QUESTIONS:

1. Define Impedance.
2. What is the current expression for series RL circuit?
3. What is the current expression for series RC circuit?

EXPERIMENT – 3

IMPEDANCE (Z) AND ADMITTANCE(Y) PARAMETERS

3.1   AIM:

To find the Z and Y parameters of a two-port network.

3.2   APPARATUS:

S. No.	Name of the Equipment	Range	Type	Quantity
1	Resistors			
2	Ammeter			
3	Voltmeter			
4	R.P.S			
5	Bread Board			
6	Connecting wires			

3.3   CIRCUIT DIAGRAM:
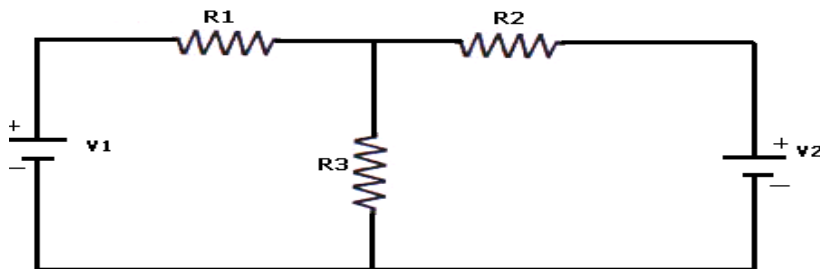
Fig – 3.1 Basic Circuit

3.4   PROCEDURE:

Z – Parameters:

1. Connect the circuit as shown in fig.
2. Open circuit port2 that is (i.e $I_2 = 0$) and measure I_1 and V_2 and   calculate Z_{11} & Z_{21} using the formulas

$$Z_{11} = \frac{V_1}{I_1} | I_2 = 0 \quad Z_{21} = \frac{V_2}{I_1} | I_2 = 0$$

3. To Measure Z_{12} and Z_{22}, open circuit port1 (i.e. $I_1 = 0$) and measure V_1 and I_2 and calculate Z_{12} & Z_{22} using the formulas

$$Z_{12} = \frac{V_1}{I_2} | I_1 = 0 \quad Z_{22} = \frac{V_2}{I_2} | I_1 = 0$$

Y – Parameters:

1. Connect the circuit as shown in fig.
2. Short circuit port 2 (i.e. $V_2 = 0$) and measure V_1, I_1 & I_2 and calculate Y_{11} & Y_{12} using the formulas.

$$Y_{11} = \frac{I_1}{V_1} \big|_{V_2} = 0 \quad Y_{21} = \frac{I_2}{V_1} \big|_{V_2} = 0$$

3. To Measure Y_{12} and Y_{22}, short circuit port 1 (i.e. $V_1 = 0$) and measure V_2, I_1 and I_2 and calculate Y_{12} & Y_{22} using the formulas.

$$Y_{11} = \frac{I_1}{V_2} \big|_{V_1} = 0 \quad Y_{21} = \frac{I_2}{V_2} \big|_{V_1} = 0$$

3.5   TABULAR COLUMN:

When $I_2 = 0$

S.No	V_1(V)	I_1(mA)	V_2(V)
1			
2			
3			

When $I_1 = 0$

S.No	V_1(V)	V_2(V)	I_2(mA)
1			
2			
3			

When $V_2 = 0$

S.No	V_1(V)	I_1(mA)	I_2(mA)
1			
2			
3			

When $V_1 = 0$

S.No	I_1(mA)	I_2(mA)	V_2(V)
1			
2			
3			

3.6   RESULT:

3.7   PRE-LAB VIVA QUESTIONS

1. What are Z parameters?
2. What are Y parameters?
3. What is the other name of Z parameter?
4. What is the other name of Y parameter?
5. Give the reciprocity and symmetry condition in terms of Z parameters.
6. Give the reciprocity and symmetry condition in terms of Y parameters.

3.8   LAB ASSIGNMENT:

1. Find out Z parameters for Π network and T network?
2. Find out Y parameters for Π network and T network?

3.9   POST LAB VIVA QUESTIONS:

1. Give the relation between Y and Z parameters.
2. What are the different parameters used to represent two port networks?
3. Represent Z parameter in terms of Y parameter.

EXPERIMENT – 4

TRANSMISSION AND HYBRID PARAMETERS

4.1   AIM:

To find the ABCD and h - parameters of a two-port network.

4.2   APPARATUS:

S. No.	Name of the Equipment	Range	Type	Quantity
1.	Resistors			
2.	Ammeter			
3.	Voltmeter			
4.	R.P.S			
5.	Bread Board			
6.	Connecting wires			

4.3   CIRCUIT DIAGRAM:
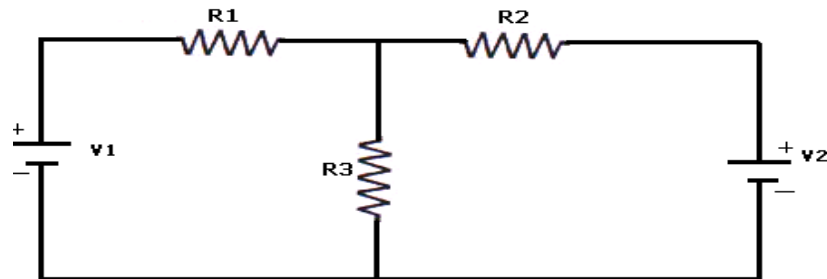
Fig – 4.1 Basic Circuit

4.4   PROCEDURE:

ABCD – Parameters

1. Connect the circuit as shown in fig.
2. Connect the circuit as shown in figure make $I_2=0$ and note down the values of V_1, I_1 & V_2.

$$A = V_1/V_2 \quad |I_2=0$$

$$C = I_1/V_2 \quad |I_2=0$$

3. Make $V_2=0$ and note down the values of V_1,I_1 & I_2.

$$B = -V_1/I_2 \quad |V_2=0$$

$$D = -I_1/I_2 \quad |V_2=0$$

4. Find the values of A, B, C, D and compare them with the theoretical values.

H – Parameters

1. Connect the circuit as shown in fig.

2. Connect the circuit as shown in figure make $I_1=0$ and note down the values of V_1, I_2 & V_2

$$h_{11} = V_1/V_1 \qquad |V_2=0$$

$$h_{21} = I_2/I_1 \qquad |V_2=0$$

3. Make $V_2=0$ and note down the values of V_1, I_1 & I_2.

$$h_{11} = V_1/V_1 \qquad |V_2=0$$

$$h_{22} = I_2/V_2 \qquad |I_1=0$$

4. Find the values of h11, h22, h12, h21 and compare them with the theoretical value

4.5   TABULAR COLUMN:

When $I_2=0$

S.No	$V_1(V)$	$I_1(mA)$	$V_2(V)$

When $I_1=0$

S.No	$V_1(V)$	$V_2(V)$	I2(mA)

When $V_2=0$

S.No	$V_1(V)$	$I_1(mA)$	I2(mA)

4.6   RESULT:

4.7   PRE LAB VIVA QUESTIONS:

1   What are ABCD parameters?
2   What are h parameters?
3   What is the other name of ABCD parameter?
4   What is the other name of h parameter?
5   Give the reciprocity and symmetry condition in terms of ABCD parameters.
6   Give the reciprocity and symmetry condition in terms of h parameters.

4.8   LAB ASSIGNMENT:

1   Find out ABCD parameters for Π network and T network?
2   Find out H parameters for Π network and T network?

4.9   POST LAB VIVA QUESTIONS:

1.   Give the relation between ABCD and H parameters.
2.   What are the different parameters used to represent two port networks?
3.   Represent ABCD parameter in terms of H parameter.

FOURIER ANALYSIS OF SQUARE WAVE, HALF WAVE RECTIFIED AND FULL WAVE RECTIFIED SINE WAVE USING MATLAB

5.1   AIM:

To obtain Fourier transform of square wave, half wave rectified and full wave rectified sine wave using MATLAB.

5.2   APPARATUS:

S. No	SOFTWARE USED	DESKTOP QUANTITY
1	MATLAB	01

5.3   THEORY:

Fourier Transform Theorems:

We may use Fourier series to motivate the Fourier transform as follows. Suppose that f is a function which is zero outside of some interval [□L/2, L/2]. Then for any T ≥ L we may expand f in a Fourier series on the interval [□T/2, T/2], where the "amount" of the wave e2πinx/T in the Fourier series of f is given by

By definition

$$F(w) = \int_{-\infty}^{\infty} f(x)e^{-iwx}dx$$

fourier series of square wave is given by

$$f(x) = \frac{4}{\pi} \sum_{n=1,3,5,\dots}^{\infty} \frac{1}{n} \sin\left(\frac{n\pi x}{L}\right).$$

Fourier series of half wave rectified sine wave is given by

$$f(t) = \frac{A}{\pi} + \frac{A}{2}\sin(\omega_0 t) - \frac{2A}{\pi} \sum_{n=2,4,6,\dots}^{\infty} \frac{\cos(n\omega_0 t)}{(n^2-1)}$$

Fourier series of full wave rectified sine wave is given by

$$f(t) = \frac{2A}{\pi} - \frac{4A}{\pi} \sum_{n=1}^{\infty} \frac{\cos(n\omega_0 t)}{(4n^2-1)}$$

5.4 CIRCUIT DIAGRAM:
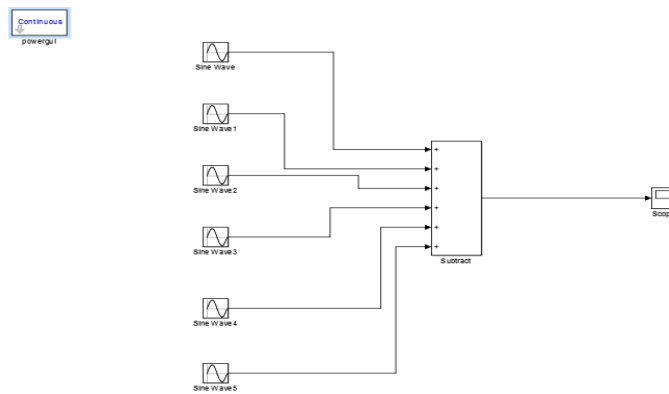
Fig - 5.1 Fourier analysis of square wave
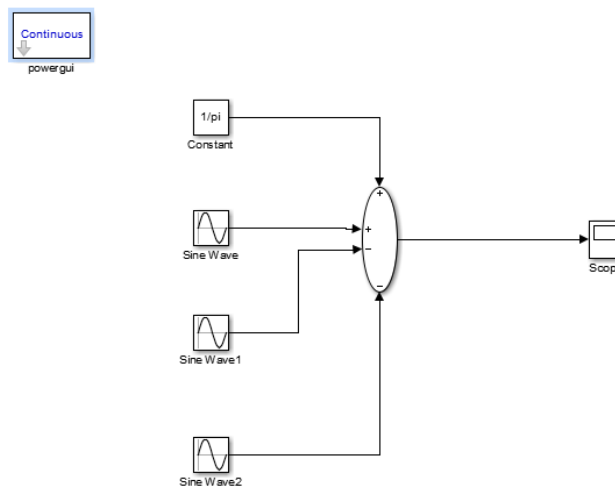
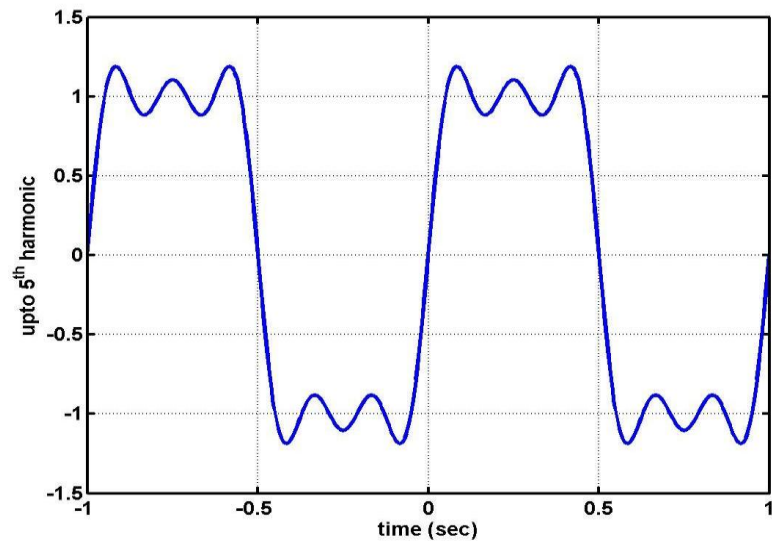**Fig - 5.2 Fourier analysis of half wave rectified sine wave**
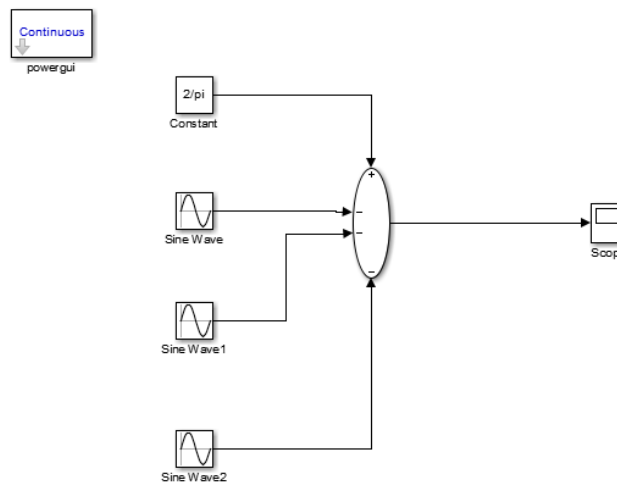
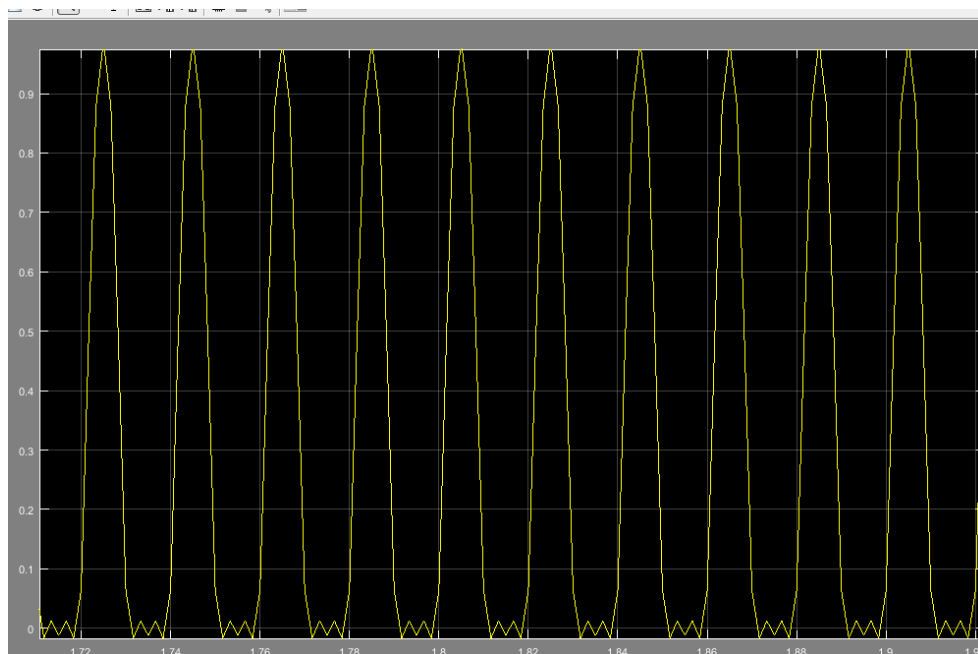**Fig - 5.3 Fourier analysis of full wave rectified sine wave**

5.4   PROCEDURE:

1   Make the connections as shown in the figures 5.1 and 5.2 by using MATLAB Simulink.

2   Set the parameters according to Fourier series.
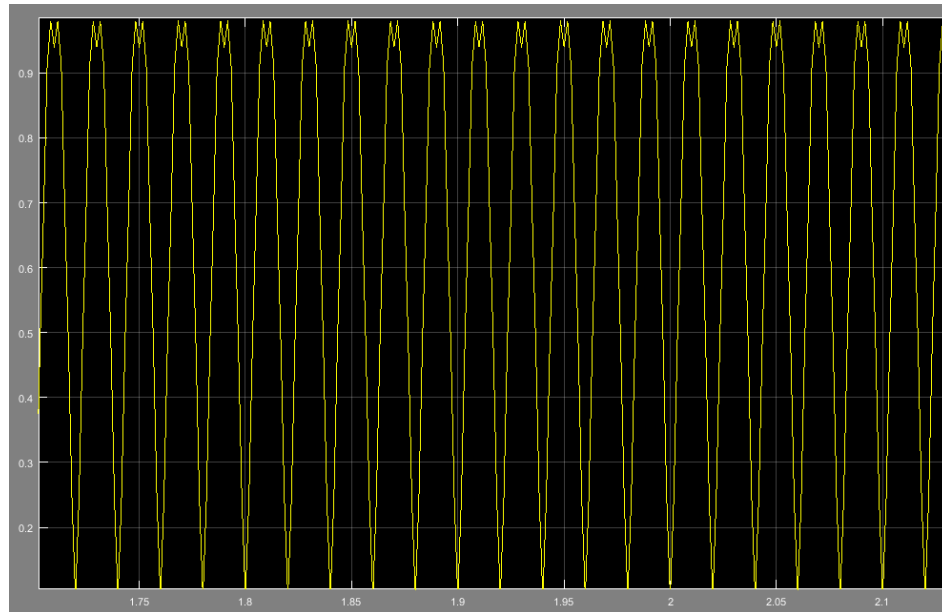
3   Check the scope wave forms in each circuit.

5.5   OUTPUT:

Fig- 5.4 Fourier analysis of square wave

Fig – 5.5 Fourier analysis of half wave rectified sine wave

Fig – 5.6Fourier analysis of full wave rectified sine wave

5.6   RESULT:

5.7   PRE-LABVIVA QUESTIONS:

1. Write the Fourier series of square wave.
2. Write Fourier series of half wave rectified sine wave.
3. Write Fourier series of half wave rectified sine wave.

5.9   POST LAB VIVA QUESTIONS:

1. Define Fourier series?
2. What are the properties of Continuous-Time Fourier Series?
3. What is the sufficient condition for the existence of F.T?
4. Define the F.T of a signal?
5. What is the difference between F.T & F.S?

<div align="center">

EXPERIMENT – 6

ELECTRICAL CIRCUIT MODEL USING VISIO SOFTWARE

</div>

6.1   AIM:

To draw the different electrical circuits using VISSIO software.

6.2   APPARATUS:

S. No	Software Used	Quantity
1	VISIO	01

6.3   CIRCUIT DIAGRAMS:

<div align="center">
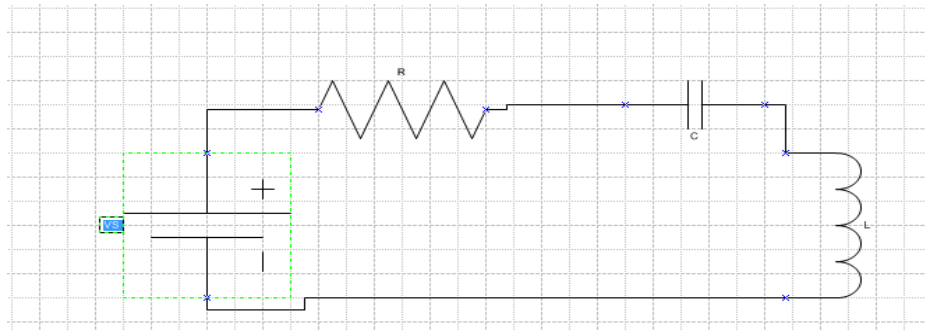
Fig - 6.1   Series RLC circuit

</div>

6.4   PROCEDURE:

1.   Open the VISIO window.

2.   Select engineering option.

3.   Select metrics and click on create.

4.   Now the drag any element required into the page one by one.

5.   Connect all the elements according to required circuit using connector.

6.   Copy the thus formed circuit and paste in the word file.

6.5   RESULT:

6.6   **PRE-LABVIVA QUESTIONS:**

1. Can we simulate electrical circuit using VISIO?
2. Why do we need VISIO software?

6.7   **POST LAB VIVA QUESTIONS:**

1. For what type of circuit, we connect ground?
2. For what type of circuit, we connect neutral?
3. Are poly phase transformers connected using VISIO software?

<h1 style="text-align:center">EXPERIMENT – 7</h1>

<h2 style="text-align:center">TRANSIENT RESPONSE OF SERIES AND PARALLEL RL and RC CIRCUIT USING MATLAB</h2>

7.1   AIM:

To study the current and voltage in the series and parallel RL and RC circuit.

7.2   APPARATUS:

| S. No	Software Used	Quantity
1	MATLAB	01

7.3   CIRCUIT DIAGRAMS:
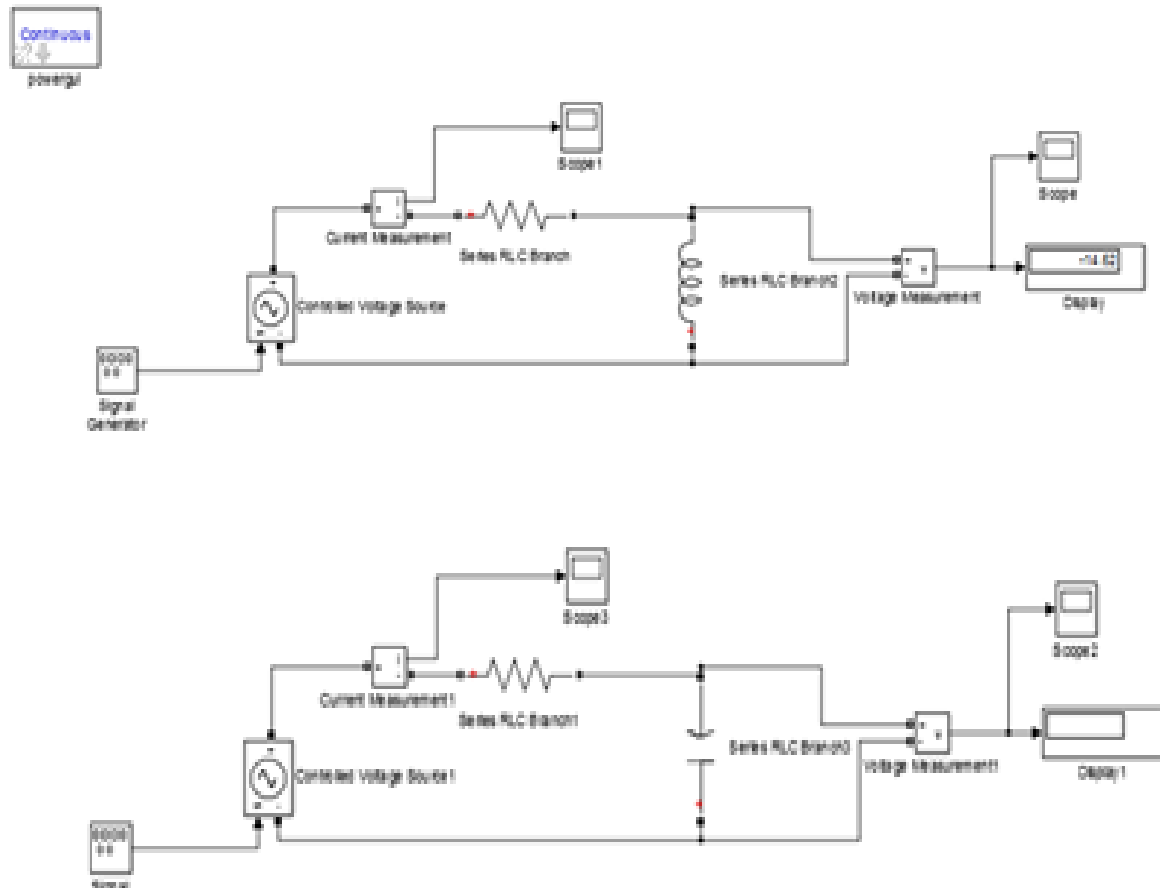
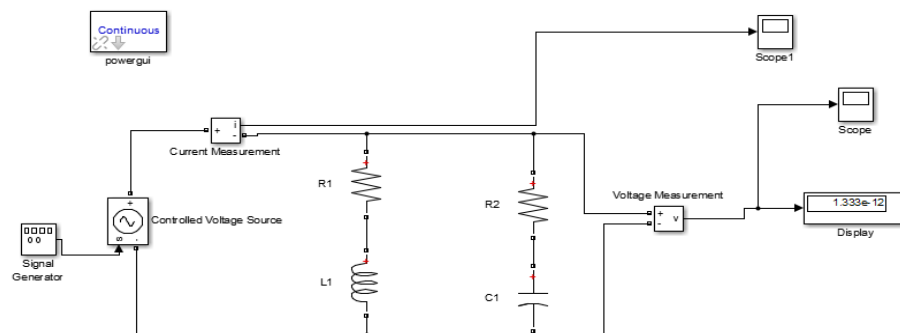**Fig - 7.1   Series RL and RC circuit**
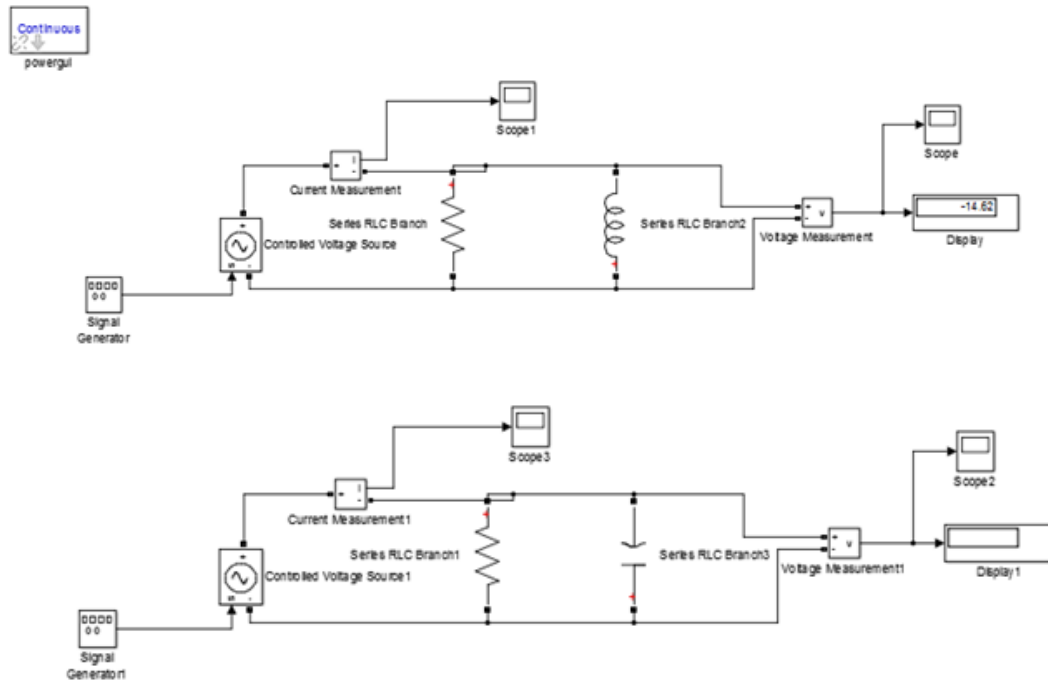
Fig - 7.2 Parallel RL and RC circuit

7.4 PROCEDURE:

1 Make the connections as shown in the figures 7.1 and 7.2 by using MATLAB Simulink.
2 Set the square wave signal in signal generator.
3 Change the simulation configuration parameters.
4 Check the voltages and currents wave forms in each circuit.

7.5 RESULT:

7.6 PRE-LABVIVA QUESTIONS:

1. What is the steady state value of series RL circuit?
2. What is the steady state value of series RC circuit?
3. What is the steady state value of parallel RL circuit?
4. What is the steady state value of parallel RC circuit?

7.7 POST LAB VIVA QUESTIONS:

1. Give the applications of series RLcircuits?
2. Give the applications of parallel RL circuits**?**
3. Give the applications of series RC circuits?
4. Give the applications of parallel RC circuits**?**

EXPERIMENT – 8

SIMULATION OF TRANSIENT RESPONSE OF RLC CIRCUITS USING MATLAB

8.1 AIM:

To study the current and voltage in the series and parallel RLC circuit.

8.2 APPARATUS:

S. No	Software Used	Desktop Quantity
1	MATLAB	01

8.3 CIRCUIT DIAGRAMS:
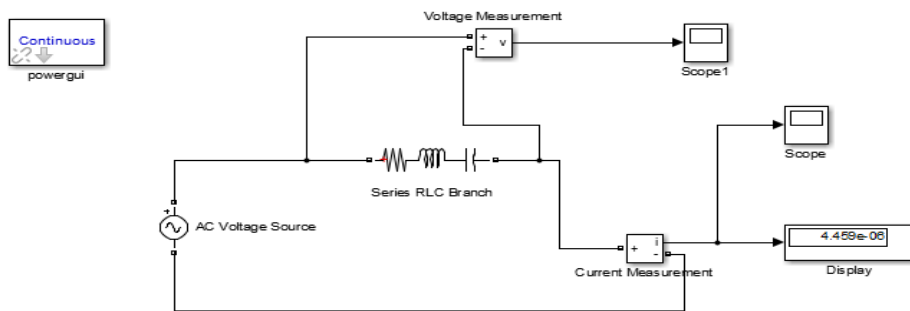
Fig – 8.1 Series RLC circuit

Fig – 8.2 Parallel RLC circuit

8.4 PROCEDURE:

1. Make the connections as shown in the figures 8.1 and 8.2 by using MATLAB Simulink.
2. Check the voltages and currents wave forms in each circuit.
3. Set the square wave signal in signal generator.
4. Change the simulation configuration parameters.

8.5   RESULT:

8.6   PRE-LABVIVA QUESTIONS:

1. Define transient response.
2. Define steady state response.
3. If XL > XC then the circuit is -------------.
4. If XL < XC then the circuit is -------------.

8.7   POST LAB VIVA QUESTIONS:

1. If XL > XC then the nature of response in series circuit is -------------.
2. If XL < XC then the nature of response in parallel circuit is -------------.
3. Application of series RLC circuits
4. Applications of parallel RLC circuits

EXPERIMENT -9

DESIGN OF LOWPASS AND HIGHPASS FILTER USING DIGITAL SIMULATION

9.1   AIM:

To verify the characteristics of Low pass and High pass filter.

9.2   APPARATUS:

S. No	Software Used	Desktop Quantity
1	MATLAB	01

9.3   CIRCUIT DIAGRAM:
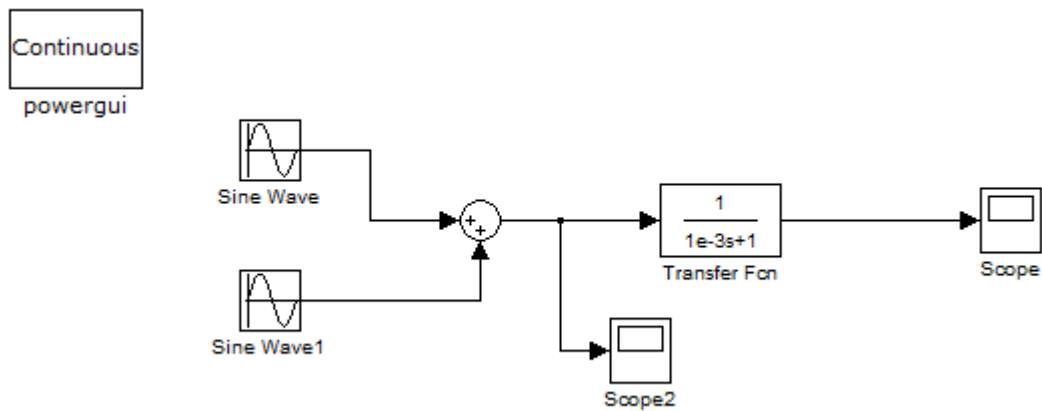
Fig - 9.1 Circuit diagram for analysis of low pass filter
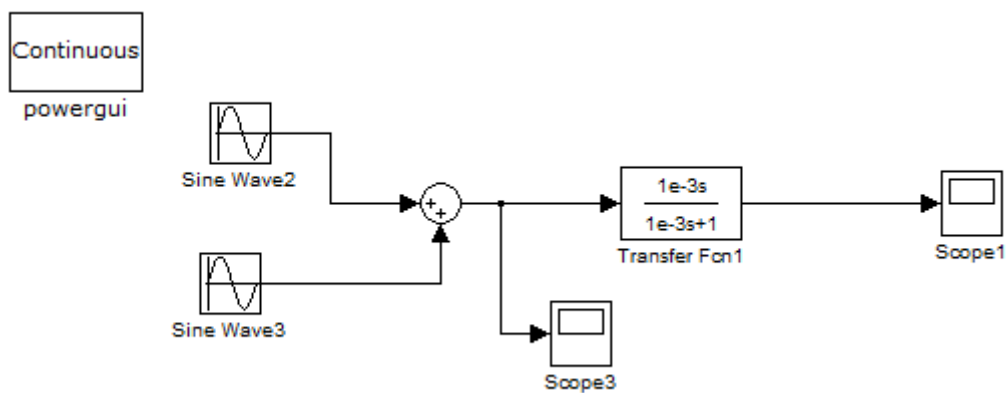
**Fig - 9.2 Circuit diagram for analysis of high pass filter**

29 | P a g e

9.4 PROCEDURE:

1. Make the connections as shown in the figures 9.1 and 9.2 by using MATLAB Simulink.
2. Change the parameters of transfer function block.
3. Check the waveform in scope.
4. In low pass filter circuit, you will observe it allows only low frequency sine wave and for high pass filter circuit it allows only high pass frequency sine wave.

9.5 OUTPUT WAVEFORMS:
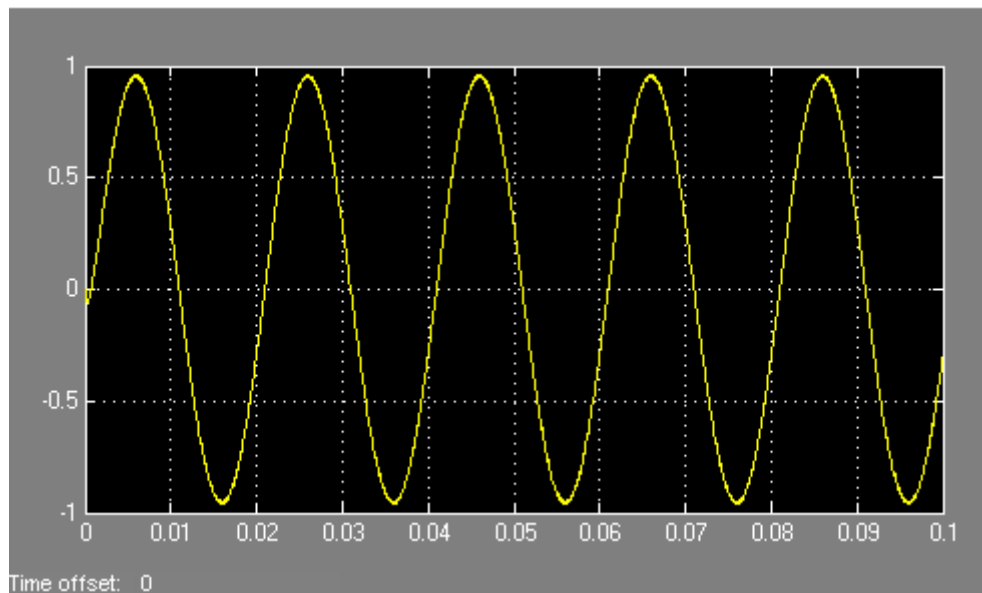
Fig 9.3 output waveform for analysis of low pass filter
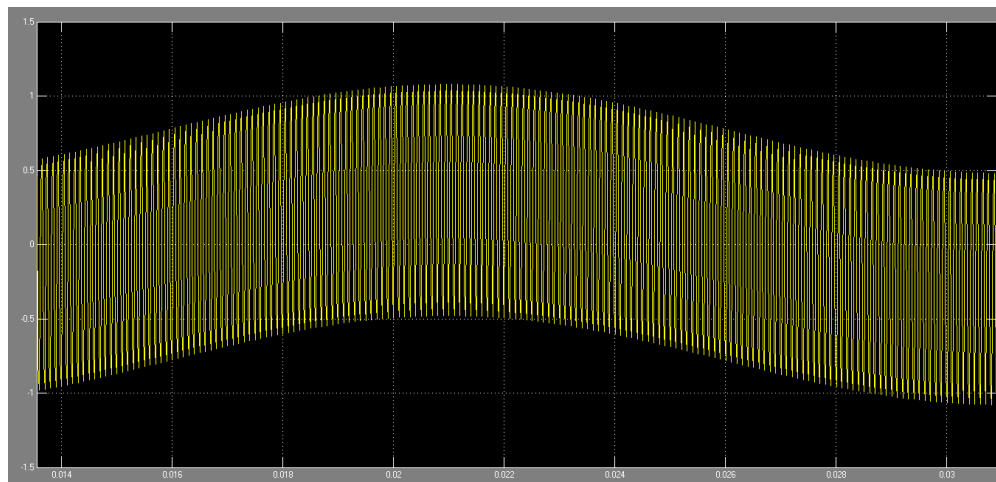
Fig 9.4 output waveform for analysis of high pass filter

9.6   RESULT:

9.7   PRE-LAB VIVA QUESTIONS:

1. What is high pass filter?
2. What is low pass filter?
3. What is expression for cut of frequency for line pass or high pass filter?
4. What is frequency response curve?

9.8   POSTLAB VIVA QUESTIONS:

1. When does cut of frequency occur?
2. What does high pass filter generally comprise of?
3. The region between pass band and stop band is called?
4. The rate of change of amplitude with frequency in a filter is?

EXPERIMENT: 10

VIRTUAL INSTRUMENTS (VI) USING LabVIEW

10.1   AIM:

Editing and Building a VI, Creating a VI

10.2   APPARATUS:

S. No.	Name of the Equipment	Range	Type	Quantity
1	LABVIEW SOFTWARE	-	-	-

10.3   INTRODUCTION:

LabVIEW programs are called virtual instruments, or VIs, because their appearance and operation imitate physical instruments, such as oscilloscopes and multimeters. Lab VIEW contains a comprehensive set of tools for acquiring, analyzing, displaying, and storing data, as well as tools to help you troubleshoot code you write.

In LabVIEW, you build a user interface, or front panel, with controls and indicators. Controls are knobs, push buttons, dials, and other input mechanisms. Indicators are graphs, LEDs, and other output displays. After you build the front panel, you add code using VIs and structures to control the front panel objects. The block diagram contains this code. You can use Lab VIEW to communicate with hardware such as data acquisition, vision, and motion control devices, as well as GPIB, PXI, VXI, RS232, and RS485 instruments.

VIRTUAL INSTRUMENT:

A virtual instrument (VI) is a program in the graphical programming language G. Virtual instrument front panels often have a user interface similar to physical instruments. G also has built-in functions that are similar to VIs, but do not have front panels or block diagrams as VIs do.
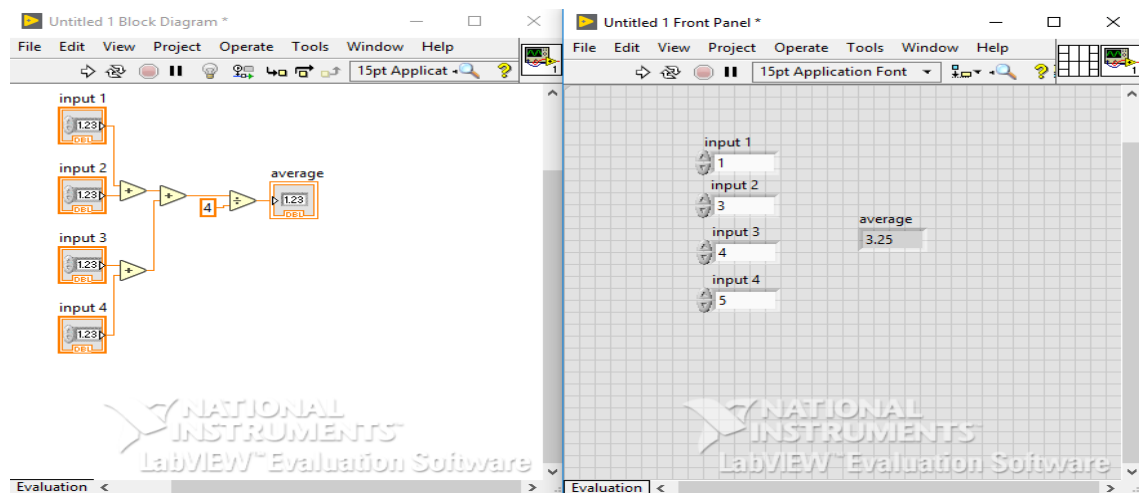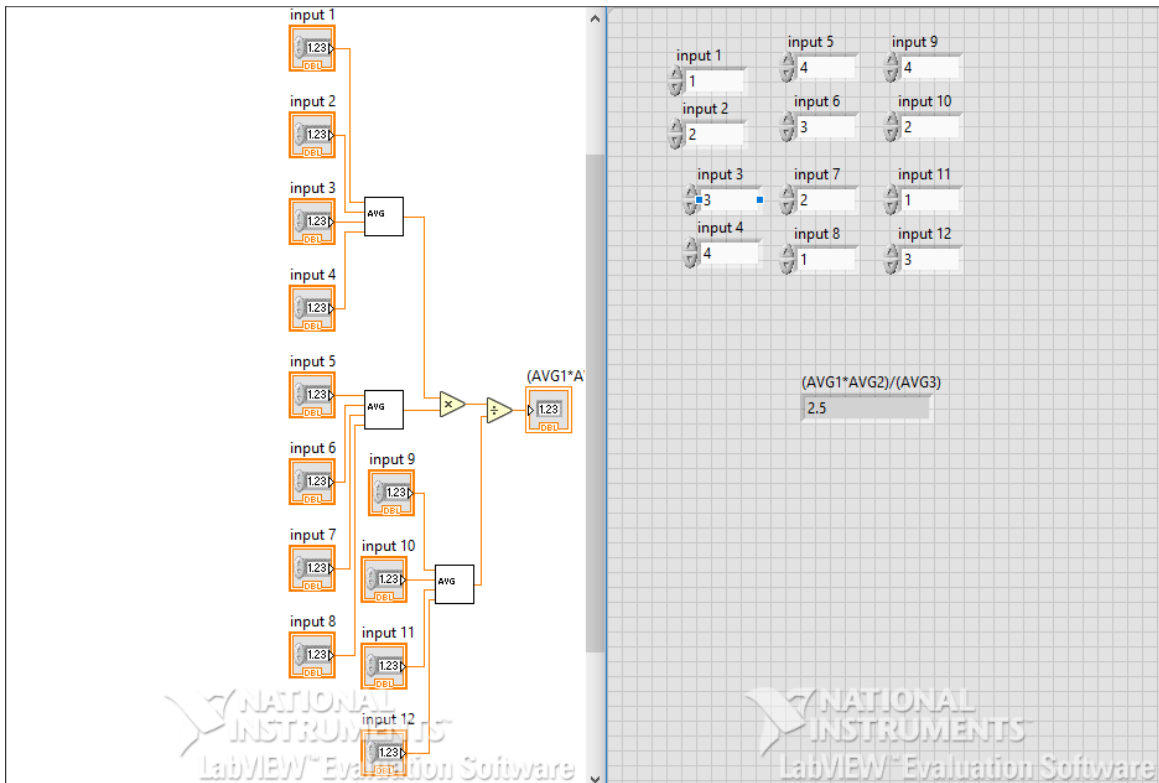
10.4   BLOCK DIAGRAM:

Fig 10.1 Block Diagram and Front Panel for Creating VI

Fig - 10.2 Block Diagram for using sub VI

10.5   PROCEDURE:

10.5.1 CREATING A VI

1. Open new VI by selecting File» New, select New VI from the Lab VIEW dialog box. Then you can see block diagram panel and front panel.
2. Then right click on the block diagram panel and select adder block from functions>>numeric. Then select and drag it to the block diagram panel.
3. Copy paste the adder block as we need four inputs for the average of four numbers.
4. Then right click on the adder block and create a control, which displays the input in the front panel. Likewise create controls for the four inputs.
5. Then copy paste another adder block and connect the outputs of first two adder blocks to it.
6. Then right click on the block diagram panel and select divider block from functions>>numeric. Select the divider block and drag it to block diagram panel.
7. And connect the output of 3rd adder block to 1st input and create a constant to second input by right clicking on divider block and give its value as 4.
8. Go to front panel and label every block as shownin figure, and give the values for input blocks.
9. Run the VI. Save the VI as 'AVG' from file menu.

10.5.2 CREATING A SUB VI:

(To use a VI as a sub VI, you must create an icon to represent it on the block diagram of another VI, and a connector pane to which you can connect inputs and outputs. Lab VIEW provides several tools with which you can create or edit an icon for your VI's.)

1. Open AVG.vi from file menu.
2. From the front panel, pop up on the icon in the top right corner and select Edit Icon. You also can double click on the icon to invoke the icon editor.
3. Erase the default icon. With the Select tool, which appears as a dotted rectangle, click and drag over the section you want to delete. You also can double click on the shaded rectangle in the tool box to erase the icon.
4. Create the text with the Text tool. To change the font, double-click on the Text tool. Your icon should look similar to the following illustration.
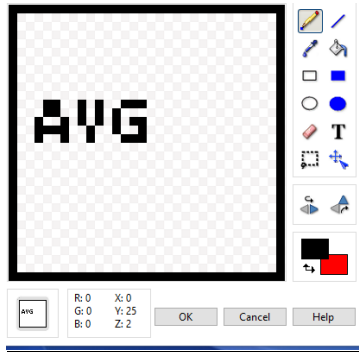
5. Close the Icon Editor by clicking on OK. The new icon appears in the icon pane.
6. Define the connector terminal pattern by popping up in the icon pane on the front panel and choosing Show Connector.
7. Assign the terminals to inputs and output.

    a. Click on the top terminal in the connector. The cursor automatically changes to the Wiring tool, and the terminal turns black.

    b. Click on the input 1 block. The selected terminal changes to a color consistent with the data type of the control/indicator selected. Likewise do to all the inputs and outputs.
8. Save the VI by choosing File» Save.
9. Close the VI by choosing File» Close.

10.5.3 EDITING AND RUNNING VI:

1. Open a new VI by selecting File» New.
2. Then right click on the block diagram and select already existing VI from functions>>select VI.
3. Then select AVG.VI from the folder and drag it to the block diagram panel.

4. Then copy paste two times. Then right click and select multiply and divide block from functions>>numeric.

5. Then wire the blocks as shown in the diagram

6. Create controls to inputs and indicators to outputs as shown.

7. Save the VI from file>>save.

10.7 RESULTS:

10.8 PRE-LAB QUESTIONS:

1. What is expansion of LabVIEW?
2. What is VI?
3. What is front panel?
4. What is block diagram?

10.9 POST LAB QUESTIONS:

1. What do you mean by controls and indicators?
2. How do you create a sub VI?
3. How do you use functions from function pallete.
4. What are various color coding for data types?
5. What is sequence of executing LabVIEW program?

EXPERIMENT: 11

STRUCTURES USING LAB VIEW

11.1 AIM:

Using FOR loop, WHILE loop, Charts and Arrays, Graphs and analysis VI s.

11.2 APPARATUS:

S. No.	Name of the Equipment	Range	Type	Quantity
1	LABVIEW SOFTWARE	-	-	-

11.3 INTRODUCTION:

A structure is a program control element. Structures control the flow of data in a VI. G has five structures: the While Loop, the For Loop, the Case structure, the Sequence structure, and the Formula Node.

While loop: A while loop is a control flow statement you use to execute a block of the sub diagram code repeatedly until a given Boolean condition is met. First, you execute the code within the sub diagram, and then the conditional terminal is evaluated. Unlike a for loop, a while loop does not have a set iteration count; thus, a while loop executes indefinitely if the condition never occurs.

For loop: A for loop is a control flow statement you use to execute a block of the sub diagram code a set number of times, but a while loop stops executing the sub diagram only if the value at the conditional terminal exists.

Charts: A chart is a numeric plotting indicator which is updated with new data periodically. You can find two types of charts in the Controls» Graph palette: waveform chart and intensity chart. You can customize charts to match your data display requirements or to display more information. Features available for charts include: a scrollbar, a legend, a palette, a digital display, and representation of scales with respect to time.

Array: An array is a collection of data elements that are all the same type. An array has one or more dimensions and up to 231 – 1 elements per dimension, memory permitting. You access each array element through its index. The index is in the range 0 to n – 1, where n is the number of elements in the array.

Graph: A graph is a two-dimensional display of one or more data arrays called plots. There are three types of graphs in the Controls» Graph palette:

• XY graph
• Waveform graph
• Intensity graph

The difference between a graph and a chart is that a graph plots data as a block, whereas a chart plots data point by point, or array by array.
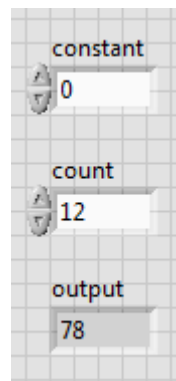
11.4   PROCEDURE:

11.4.1   USING WHILE LOOP:

1. Open a new VI. You can open a blank VI by selecting File» New VI.
2. Then you can see block diagram panel and front panel.
3. Right click on block diagram panel and select while loop from structures>> while loop and drag it to the block diagram panel.
4. Select adder and greater than or equal to blocks from numeric block and drag it to block diagram panel.
5. Go to front panel and two controls and an indicator by right clicking on the front panel selecting from numeric block and label them as constant, output and count.
6. Wire the blocks as shown in block diagram.
7. Give the value to count and run the VI.

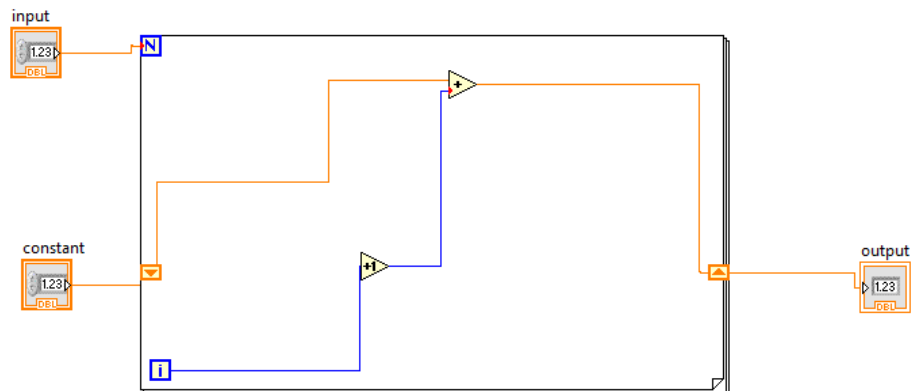### 11.4.1.2 BLOCK DIAGRAM:
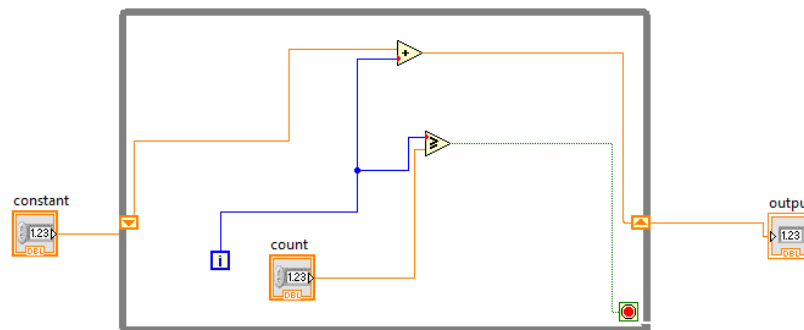
Fig 11.1 Block Diagram of While Loop
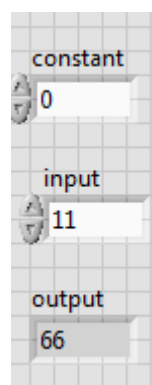
11.4.1.3 FRONT PANNEL:

11.4.2 USING FOR LOOP:

1. Open a new VI. You can open a blank VI by selecting File» New VI.
2. Then you can see block diagram panel and front panel.
3. Right click on the block diagram panel and select for loop from structures >> for loop and drag it to block diagram panel.
4. Select adder and increment blocks from numeric block and drag it to the block diagram panel.
5. Go to front panel and add two controllers and one indicator and label them as constant, input, output.
6. Wire the blocks as shown in block diagram.
7. Give the value to input in front panel and run the VI.

11.4.2.1 BLOCK DIAGRAM:
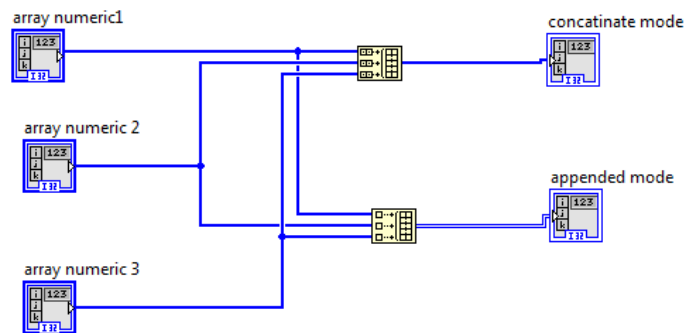

Fig 11.2 Block Diagram of For Loop

11.4.2.2 FRONT PANEL:
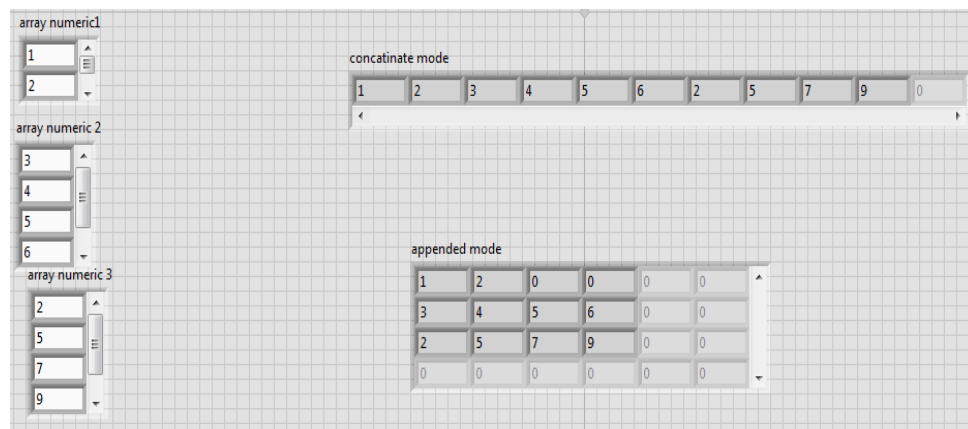
11.4.3 USING ARRAY:

1. Open a new VI. You can open a blank VI by selecting File» New VI.
2. Then you will see front panel and block diagram panel.
3. Got to front panel and right click and select array from array block.
4. Right click in the array block and add numeric control to it, follow the same for 5 array blocks.
5. Then for one array block give size as 2D by right clicking on the array block and selecting properties and then size column, this is used for appending mode.
6. Then go to block diagram panel and add build array block by right clicking on block diagram panel and then selecting array block.
7. For the first build array block right click and select concatenate mode and the other automatically becomes appending mode.
8. Wire the blocks as shown in block diagram.
9. Give values to input arrays and run the VI.

11.4.3.1 BLOCK DIAGRAM:

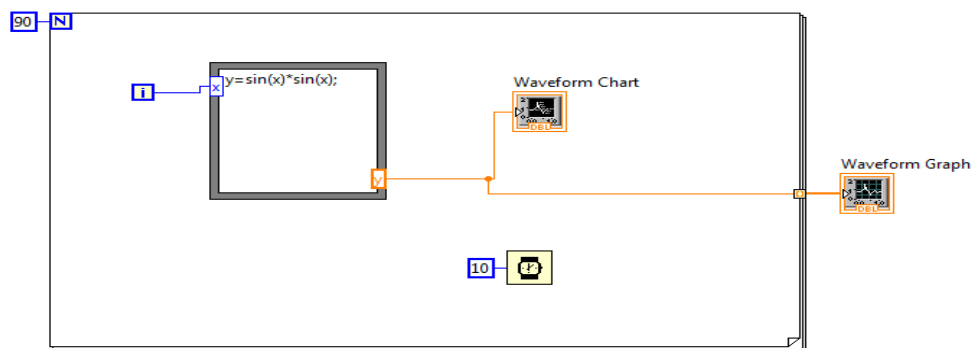**Fig - 11.3 Block Diagram of Array**

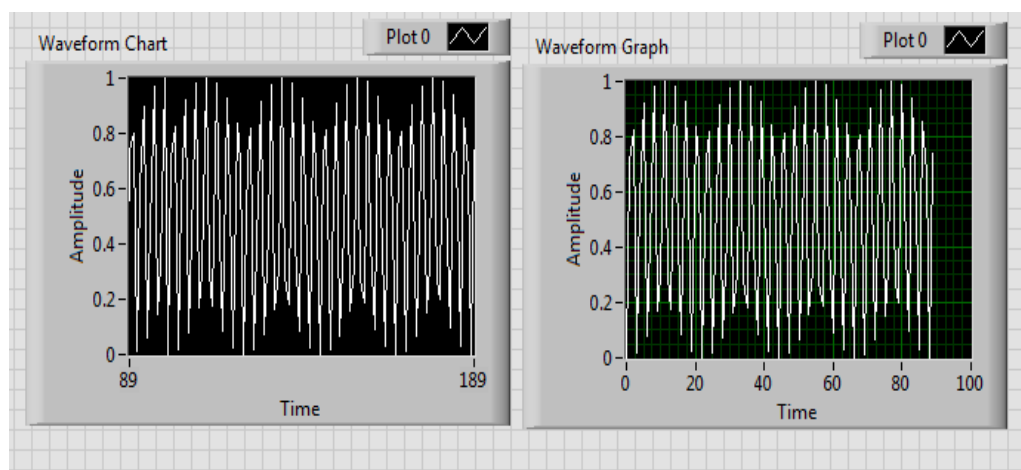### 11.4.3.2 FRONT PANEL:

11.4.4 USING CHARTS AND WAVEFORM:

1. Open a new VI. You can open a blank VI by selecting File» New VI.
2. Then you will see the block diagram panel and front panel.
3. Right click on the block diagram panel and select the for loop block from structure block.
4. Add the formula node block and timer block in to the for loop block.
5. Create constants for timer block.
6. Go to front panel and add waveform graph and waveform chart by right clicking on front panel.
7. Wire the blocks as shown in block diagram panel.
8. Run the VI.

11.4.4.1 BLOCK DIAGRAM:

**Fig - 11.4 Block Diagram of Waveform and Chart**

11.4.4.2 FRONT PANEL:

Fig - 11.5 Waveform for Charts and Graphs

11.4.5 USING ANALYSIS:

1. Open a new VI. You can open a blank VI by selecting File» New VI.
2. Then you will see block diagram panel and front panel.
3. Right click on block diagram panel and add simulate signal from express block.
4. Copy another simulate signal block.
5. Add spectral measurement from signal analysis block and also add multiplier to block diagram panel.
6. Add waveform graph to front panel by right clicking on front panel.
7. Wire the blocks as shown in block diagram panel.

11.4.5.1 BLOCK DIAGRAM:
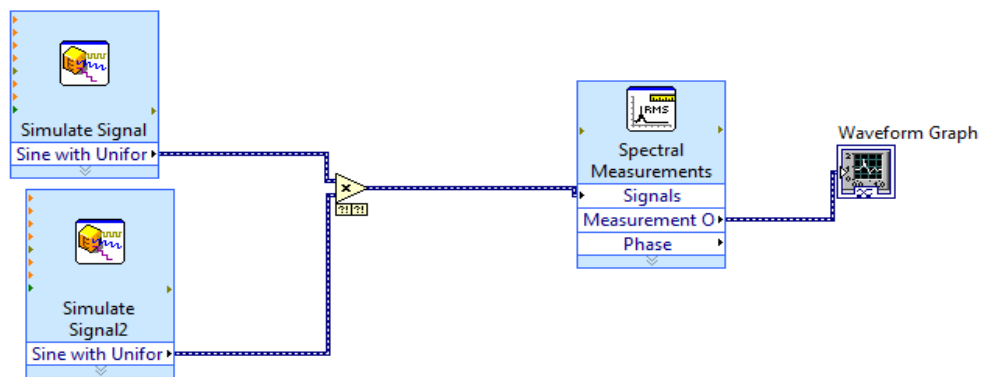
Fig - 11.6 block diagram of analysis

11.4.5.2 FRONT PANEL:
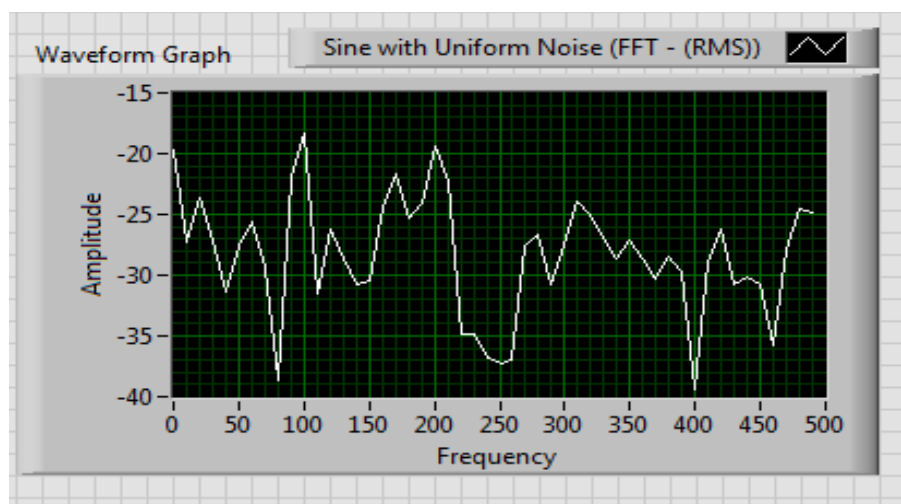
Fig - 11.7 Waveform for Analysis

11.5   RESULT:

11.6   PRE LAB QUESTIONS:

1. What are the structures available in lab view?
2. How are arrays represented in lab view?
3. What are the various analysis blocks available in labVIEW?
4. What is difference between charts and graphs?

11.7   POST LAB QUESTIONS:

1. Explain in detail use of for loop.
2. Explain in detail use of while loop.
3. Explain in detail the use of charts and graphs.
4. Explain use of analysis block.

EXPERIMENT- 12

GENERATION OF COMMON WAVEFORMS USING LAB VIEW

12.1   AIM:

To perform signal generation and display wave form, minimum and maximum values of waveform and modulation.

12.2   APPARATUS:

S. No.	Name of the Equipment	Range	Type	Quantity
1	LABVIEW SOFTWARE	-	-	-

12.3   INTRODUCTION:

In the analog world, a signal frequency is measured in Hz or cycles per second. But the digital system often uses a digital frequency, which is the ratio between the analog frequency and the sampling frequency:

Digital frequency = analog frequency / sampling frequency

This digital frequency is known as the normalized frequency. Its units are cycles/sample. Some of the Signal Generation VIs use an input frequency control, f, that is assumed to use normalized frequency units of cycles per sample. When you use a VI that requires the normalized frequency as an input, you must convert your frequency units to the normalized units of cycles/sample. You must use these normalized units with the following VIs.

• Sine Wave
• Square Wave
• Saw tooth Wave
• Triangle Wave
• Arbitrary Wave
• Chirp Pattern

If you are used to working in frequency units of cycles, you can convert cycles to cycles/sample by dividing cycles by the number of samples generated.
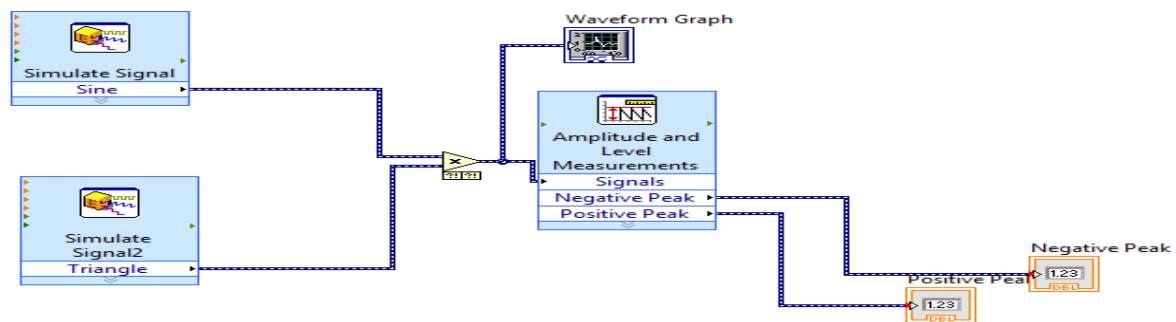
12.4   BLOCK DIAGRAM:

Fig - 12.1Block Diagram for Generating Common Waveform

12.5 PROCEDURE:

1. Open a new VI. You can open a blank VI by selecting File» New VI.
2. Then you will see front panel and block diagram panel.
3. Right click on block diagram panel and add simulate signal blocks and copy paste another simulate signal block.
4. Add a multiplier to the block diagram panel to perform modulation.
5. Add the amplitude level measurement block by right clicking on the block diagram panel to measure the positive negative peaks of a waveform.
6. Create indicators to the positive and negative peaks.
7. Place a waveform graph block in the front panel to show the modulation graph and common waveform graph.
8. By double clicking on the simulate signal block change the wave forms to display different waveforms.
9. Wire the blocks as shown in block diagram panel.

12.6 FRONT PANEL:
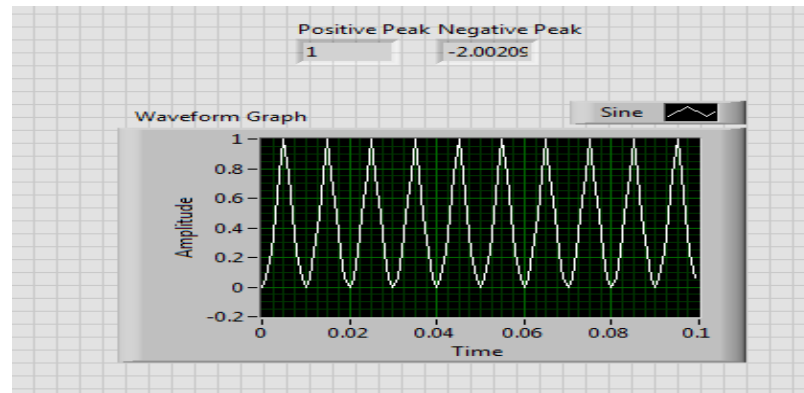
Fig -12.2 Graph for Generating Common Waveform

12.7 RESULT:

12.8 PRE LAB QUESTIONS:

1. How many types of waveforms can be generated in labVIEW?
2. How are indicators created in labVIEW?
3. How many types of waveform blocks available in labVIEW.

12.9 POST LAB QUESTIONS:

1. Why is amplitude and level block used?
2. What is meant by modulation of signals?
3. What is the function of simulate signal block.
4. How are positive and negative peak displayed in front panel.

<h1 style="text-align:center">EXPERIMENT: 13</h1>

<h1 style="text-align:center">SINE WAVE GENERATION USING LAB VIEW</h1>

13.1 AIM:

To generate three phase sine wave using lab view.

13.2 APPARATUS:

S. No.	Name of the Equipment	Range	Type	Quantity
1	LABVIEW SOFTWARE	-	-	-

13.3 INTRODUCTION:

The three phase balanced supply voltages and currents have a constant magnitude and are displaced by 120 degrees each. The order in which all the three phase waveforms reach their maximum value is called phase sequence.
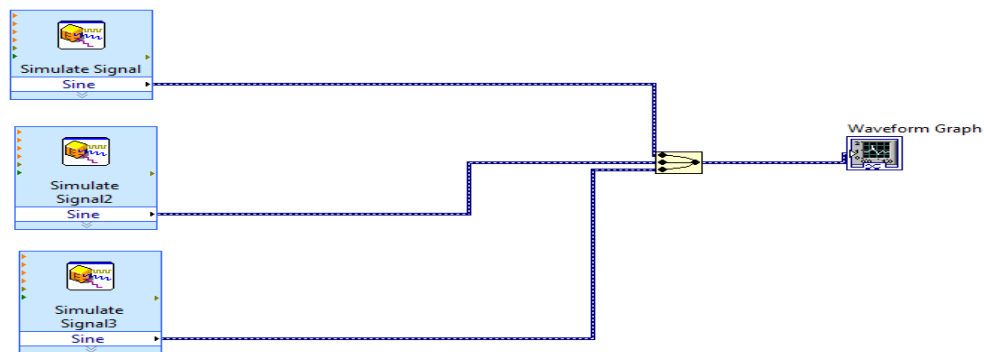
13.4 BLOCK DIAGRAM:

Fig 13.1Block Diagram for Generating Three Phase Sine Wave

13.5 PROCEDURE:

1. Open new VI, you will see a front panel and block diagram panel.
2. Right click on block diagram panel and you will see the function palette.
3. From the function palette select the simulate signal block and drag it to the block diagram panel and copy paste it two times.
4. Set the magnitude, phase angle, and frequency to the simulate signals.
5. Then select the merge signal block and drag it to the block diagram panel.
6. Now right click on front panel and you will find control and indicator palette.
7. Select graph and then select waveform graph.

8. Then connect the blocks in the block diagram panel with the wires, delete the bad wires if needed.

9. Run the VI.

10. Save the VI as waveform.vi in the LabVIEW/Activity directory.

13.6   FRONT PANEL:
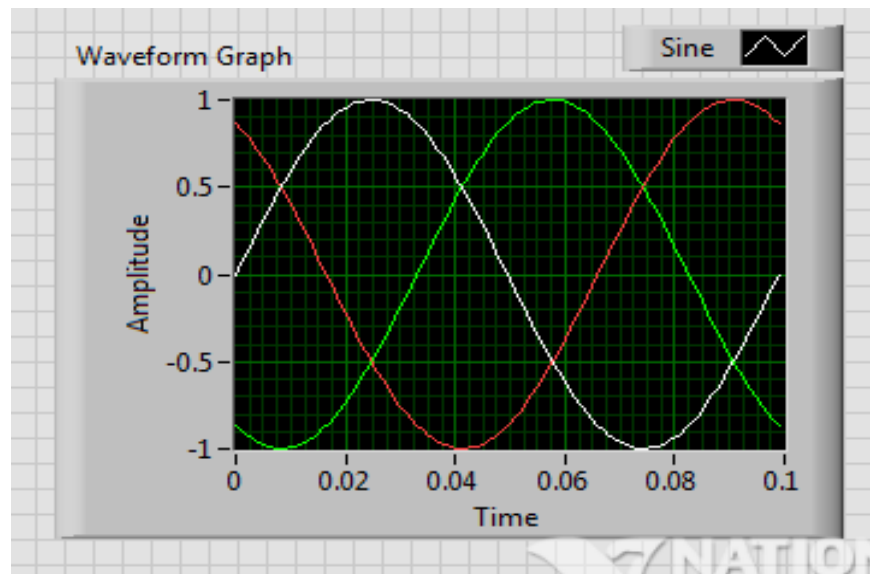
Fig - 13.2 Grpah for three phase sine wave

13.7   RESULT:

13.8   PRE LAB QUESTIONS:

1. Define phase sequence.
2. What is line voltage and phase voltage.
3. What is balanced three phase system?
4. Write the relationship between phase and line voltage in star connected three phase system.

13.9   POST LAB QUESTIONS:

1. Why is simulate signal block used.
2. How do you merge the signals in LabVIEW?
3. Write the relationship between phase and line voltage in delta connected three phase system.
4. Write the relationship between phase and line current in star connected three phase system.

EXPERIMENT: 14

FREQUENCY MEASURMENT USING LabVIEW

14.1   AIM:

To measure frequency using Lissajous figures in LabVIEW.

14.2   APPARATUS:

S. No.	Name of the Equipment	Range	Type	Quantity
1	LABVIEW SOFTWARE	-	-	-

14.3   INTRODUCTION:

A Lissajous figure is produced by taking two sine waves (one with known frequency and the other whose frequency has to be measured) displaying them on an XY graph. Lissajous patterns are displayed by LabVIEW which may be used for accurate measurement of frequency.
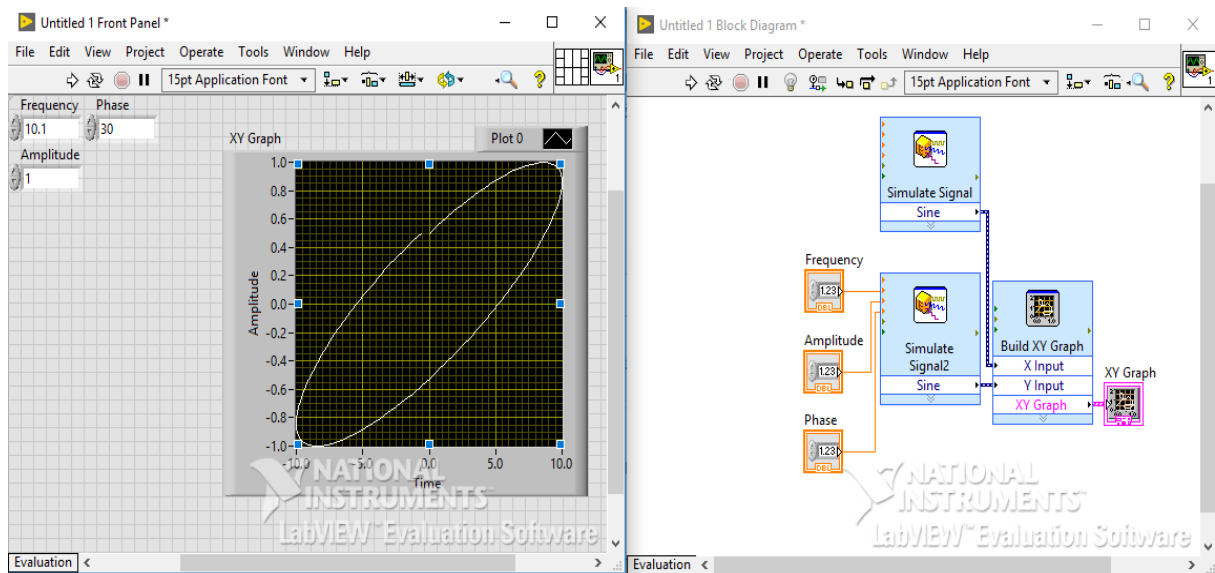
14.4   BLOCK DIAGRAM:

Fig 14.1Block Diagram for Lissajous Pattern

14.5   PROCEDURE:

1. Open new blank VI from file>> new VI, then you will get two panels named block diagram panel and front panel.
2. Right click on the blank space on the front panel. Then you will get control palette. Click on the XY Graph icon to select the Express XY Graph object and drag it onto the Front Panel workspace.
3. Right-click in the Controls palette and place two Numeric Control objects to the left of the XY Graph object in the Front Panel workspace.

4. Select the Block Diagram workspace to activate the Functions palette, Right-click in the Functions palette and move the cursor over the Input icon to open the Input sub palette.
5. Click on the Simulate Sig icon to select the Simulate Signal object and drag it onto the block diagram panel and copy paste it for another simulate signal block.
6. Then connect the blocks in the block diagram panel with the wires, delete the bad wires if needed.
7. A number of conclusions can be drawn from the above block diagram when sinusoidal voltages of same frequencies are applied.

   (i) A straight-line result when the two voltages are equal and are either in phase with each other or 180^0 out of phase with each other.
   (ii) An ellipse appears when magnitude of two signals are equal and has phase difference of either 30^0 or 330^0 and 150^0 or 220^0.
   (iii) A circle is thus formed only when magnitude of two signals are equal and has phase difference of either 90^0 or 270^0.

8. The Lissajous pattern for other frequencies also can be drawn, when the voltages are equal and the frequency ratios are (2:1), (3:1), (4:1), (1:2), (1:3) and so on.
9. Use Save All in the File menu of either workspace to save your work.
10. Run VI

14.6   EXPECTED GRAPHS:
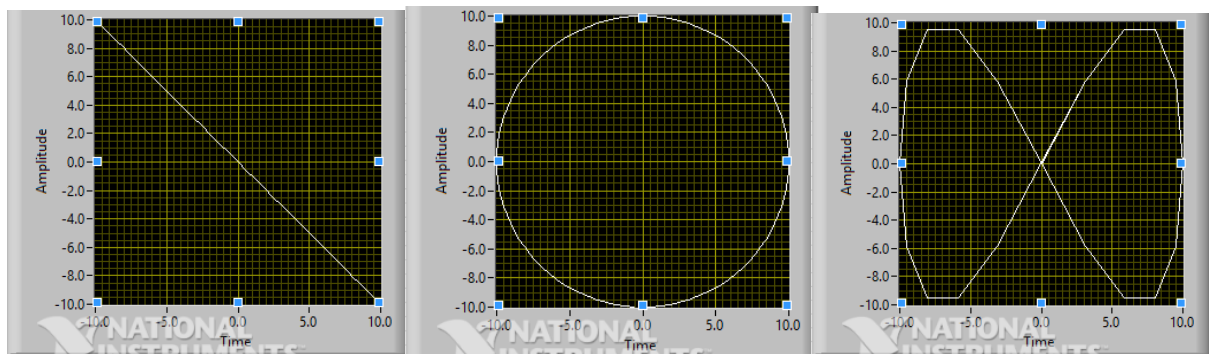
Fig - 14.2 Graphs for Lissajous Figures

14.7   RESULT:

14.8   PRE LAB VIVA QUESTIONS:

1. How to measure phase angle from Lissajous figures.
2. How to measure frequency from Lissajous figures.
3. If two signals are having same frequency what is the pattern of Lissajous figure.
4. If phase angle difference between two waves is 0 or 360 degrees then what is Lissajous pattern.

14.9   POST LAB VIVA QUESTIONS:

1. How to get the Lissajous pattern.
2. If phase angle difference between two waves is 90 or 270 degrees then what is Lissajous pattern.
3. If two ac voltages are perfectly in phase with each other then what is pattern of Lissajous figures.
4. When you get the circle pattern of Lissajous figure.