DESIGN OF ELECTRICAL CIRCUITS USING ENGINEERING SOFTWARE TOOLS

Year	**:**	**2023 - 2024**
Subject Code	**:**	**R20C209**
Regulations	**:**	**R20**
Class	**:**	**II B.Tech I Semester**
Branch	**:**	**EEE**

Prepared by

Mr.P.T.Krishna Sai
(Assistant Professor)

DEPARTMENT OF ELECTRICAL & ELECTRONICS ENGINEERING

DHANEKULA INSTITUTE OF ENGINEERING & TECHNOLOGY
(Approved by AICTE, Affiliated by JNTU, Kakinada)
GANGURU :: VIJAYAWADA – 521 139

DHANEKULA INSTITUTE OF ENGINEERING & TECHNOLOGY

Department of Electrical and Electronics Engineering

POs/PSOs

List Program Outcomes

1.Engineering knowledge :apply the knowledge of mathematics ,science ,engineering fundamentals ,and an engineering specialization to the solution of complex engineering problems.

2.ProblemAnalysis :identify ,formulate, review research literature, and analyze complex engineering problems reaching sustained conclusions using first principles of mathematics ,natural sciences, and engineering sciences.

3.Design/Development Of Solutions :design solutions for complex engineering problems and design system components or process that meet the specified needs with appropriate consideration for the public health and safety, and the cultural ,societal ,and environmental considerations.

4.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.

5.Modern Tool Usage :create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modelling to complex engineering activities with an understanding of the limitations.

6.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.

7.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.

8.Ethics:apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.

9.Individual And Team Work: function effectively as an individual, and as a member or a leader in diverse teams, and in multidisciplinary settings.

10.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.

11.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.

12.Life- Long Learning: recognize the need for, and have the preparation and ability to engage in independent and life- long learning in broadest context of technological change.

List Program Specific Outcomes

PSO 1: Ability to design solutions for identified problems by using latest engineering tools like MATLAB, Simulink, PSPICE, plc etc.

PSO 2: Able to design and develop the Green Electrical systems.

INDEX

S. No.	List of Experiments	CO No.	Page No.
1	Verification of Kirchhoff's current law and voltage law digital simulation.	R20CO209.1	6
2	Verification of mesh analysis digital simulation.	R20CO209.1	10
3	Verification of nodal analysis digital simulation.	R20CO209.1	13
4	Determination of average value, rms value, form factor, peak factor of sinusoidal wave, square wave digital simulation.	R20CO209.2	16
5	Verification of super position theorem using digital simulation.	R20CO209.2	20
6	Verification of reciprocity theorem using digital simulation.	R20CO209.2	23
7	Verification of maximum power transfer theorem using digital simulation	R20CO209.2	27
8	Verification of Thevenin's theorem using digital simulation	R20CO209.2	30
9	Verification of Norton's theorem using digital simulation	R20CO209.2	34
10	Verification of compensation theorem using digital simulation	R20CO209.2	37
11	Verification of Milliman's theorem using digital simulation	R20CO209.2	41
12	Verification of series resonance using digital simulation	R20CO209.3	45
13	Verification of parallel resonance using digital simulation	R20CO209.3	51
14	Verification of self inductance and mutual inductance by using hard ware	R20CO209.4	57
15	Generation of various signals and sequences (Periodic and Aperiodic), such as unit Impulse, Step, Square, Saw tooth, Triangular, Sinusoidal, Ramp, Sinc. using digital simulation	R20CO209.4	63
16	Operations on signals and sequences such as Addition, Multiplication, Scaling, Shifting, Folding, Computation of Energy,and Average Power using digital simulation	R20CO209.4	76

ATTAINMENT OF PROGRAM OUTCOMES & PROGRAM SPECIFIC OUTCOMES

Exp. No.	Experiment	Program Outcomes Attained	Program Specific Outcomes Attained
1	Verification of Kirchhoff's current law and voltage law digital simulation.	PO1,PO5	PSO1
2	Verification of mesh analysis digital simulation.	PO1,PO2,PO5	PSO1
3	Verification of nodal analysis digital simulation.	PO1,PO2,PO5	PSO1
4	Determination of average value, rms value, form factor, peak factor of sinusoidal wave, square wave digital simulation.	PO4,PO5	PSO1
5	Verification of super position theorem using digital simulation.	PO1,PO2,PO5	PSO1
6	Verification of reciprocity theorem using digital simulation.	PO1,PO2,PO5	PSO1
7	Verification of maximum power transfer theorem using digital simulation	PO2,PO3,PO5	PSO1
8	Verification of Thevenin's theorem using digital simulation	PO2,PO3,PO5	PSO1
9	Verification of Norton's theorem using digital simulation	PO2,PO3,PO5	PSO1
10	Verification of compensation theorem using digital simulation	PO2,PO3,PO4,PO5	PSO1
11	Verification of Milliman's theorem using digital simulation	PO2,PO3,PO4,PO5	PSO1
12	Verification of series resonance using digital simulation	PO3,PO4,PO5	PSO1
13	Verification of parallel resonance using digital simulation	PO3,PO4	PSO1
14	Verification of self inductance and mutual inductance by using hard ware	PO1,PO3,PO4	PSO1
15	Generation of various signals and sequences (Periodic andAperiodic), such as unit Impulse, Step, Square, Saw tooth, Triangular, Sinusoidal, Ramp, Sinc. using digital simulation	PO1, PO2,PO5	PSO1
16	Operations on signals and sequences such as Addition, Multiplication, Scaling, Shifting, Folding, Computation of Energy,and Average Power	PO1, PO2,PO5	PSO1

DESIGN OF ELECTRICAL CIRCUITS USING ENGINEERING SOFTWARE TOOLS

COURSE OBJECTIVES

At the end of the Course/Subject, the students will:

Course	Objectives	
DECS Lab	R20CObj209.1	To Learn the fundamentals of MATLAB Tools and to verify and simulate various electrical circuits using Mesh and Nodal Analysis.
	R20CObj209.2	To verify and simulate various theorems.
	R20CObj209.3	To verify and simulate RLC series and parallel resonance.
	R20CObj209.4	To determine self and mutual inductance of a magnetic circuit, parameters of a given coil and to generate various waveform signals and sequences

COURSE OUTCOMES

At the end of the Course/Subject, the students will be able to :

CO No	Description	Blooms Taxonomy Level
R20C209.1	Familiarity with DC and AC circuit analysis techniques using MATLAB/SIMULINK	Evaluation
R20C209.2	Analyze complicated circuits using different network theorems and simulate various wave form for determination of wave form parameters	Evaluation
R20C209.3	Acquire skills of using MATLAB software for simulation of RLC series and parallel resonance.	Evaluation
R20C209.4	Determine the self and mutual inductance of coupled coils, generate and operate various waveform signals and sequences	Evaluation

Course Out Come	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12
R20C209.1	3	3	3	-	3	-	-	-	3	3	-	3
R20C209.2	3	3	3	-	3	-	-	-	3	3	-	3
R20C209.3	3	3	3	-	3	-	-	-	3	3	-	3
R20C209.4	3	3	3	-	3	-	-	-	3	3	-	3
Level of mapping	**3**	**3**	**3**	**-**	**3**	**-**	**-**	**-**	**3**	**3**	**-**	**3**

Justification of Mapping of Course Outcome with Program Outcomes:

1. All the outcomes are highly correlated to PO1 except CO3 and CO5 because in order to understand the instruction set and C programming for PIC with an example basic mathematical knowledge is required. Course outcomes 3 & 5 are related to interfacing so moderately mapped with PO1.

2. CO1, CO2, CO4 & CO5 are highly mapped with PO2 because problem analysis is needed to understand various registers and instruction set to write program. Outcome 3 related to interface so moderately mapped with PO2.
3. PO3 is design related so CO3 and CO5 are moderately mapped with PO3. Interfacing aims to design of embedded system so CO3 is moderately mapped with PO3.
4. PO5 defines utilization of Modern tool, which is related to CO5. CO5 is highly mapped with PO5, for programming will use Kiel, Ride and some other new software's will be used.
5. By applying programming and interfacing knowledge student will be able to design embedded circuits by considering the societal health, safety issues so CO 3 is moderately mapped with PO6.
6. In order to develop embedded system, basic subject is in need i.e., architecture of microprocessors & microcontrollers so all the outcomes except CO2 are moderately mapped with PO12.

Course Outcome vs PSOs Mapping:

Course Out Come	PSO1	PSO2
R20C320.1	3	0
R20C320.2	3	0
R20C320.3	3	0
R20C320.4	3	0
R20C320.5	3	0
Level of mapping	3	0

Justification of Mapping of Course Outcome with Program Specific Outcomes: NIL

EXPERIMENT - 1

VERIFICATION OF KVL AND KCL USING DIGITAL SIMULATION.

AIM:

To verify Kirchhoff’s Voltage Law (KVL) and Kirchhoff’s Current Law (KCL) using digital simulation.

APPARATUS:

S. No	SOFTWARE USED	DESK TOP QUANTITY
1	MATLAB R2016a	01

CIRCUIT DIAGRAMS:

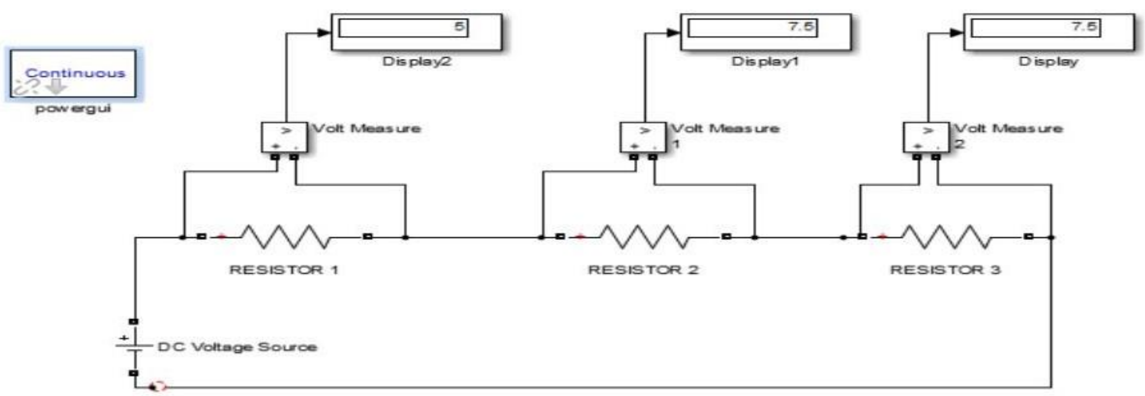


Figure – 1.3 Verification of KVL

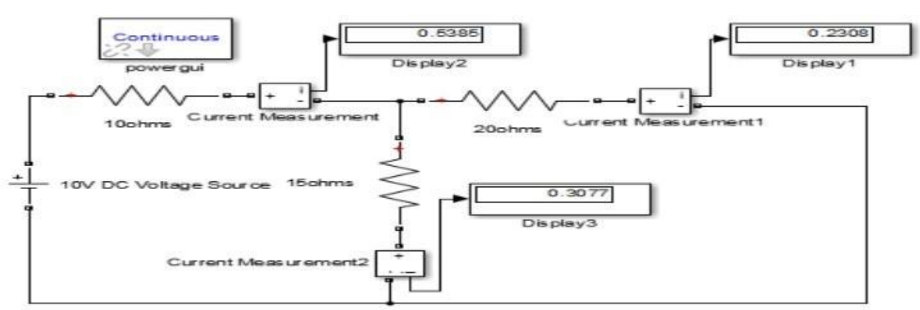


Figure – 1.4 Verification of KCL

PROCEDURE:

1. Make the connections as shown in the circuit diagram by using MATLAB Simulink.
2. Measure the voltages and currents in each resistor.
3. Verify the KVL and KCL.

OBSERVATIONS:

For KVL

Applied Voltage V (volts)	V_1 (volts)		V_2 (volts)		V_3 (volts)		$V_1+V_2+V_3$ (volts)	
	Theoretical	Practical	Theoretical	Practical	Theoretical	Practical	Theoretical	Practical

For KCL

Applied Voltage V (volts)	I (A)		I_1 (A)		I_2 (A)		I_1+I_2 (A)	
	Theoretical	Practical	Theoretical	Practical	Theoretical	Practical	Theoretical	Practical

RESULT:

PRE LAB VIVA QUESTIONS:

1. Define current.
2. Define voltage.
3. What is resistance?
4. Define ohm's law.
5. State KCL and KVL.

POST LAB VIVA QUESTIONS:

1. What do you mean by junction?
2. Derive current division rule.
3. Explain the sign conventions.
4. Explain the color coding of resistors.

EXPERIMENT - 2

MESH ANALYSIS USING DIGITAL SIMULATION

AIM: To verify mesh analysis using digital simulation.

APPARATUS:

S. No	SOFTWARE USED	DESKTOP QUANTITY
1	MATLAB R2016a	01

SIMULATION DIAGRAMS:

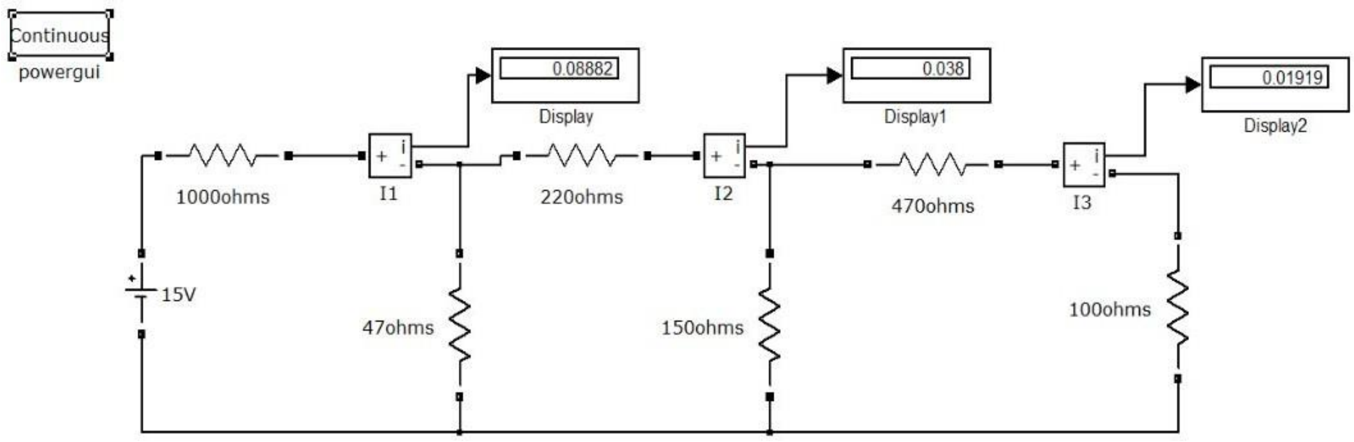


Figure – 2.2 Mesh analysis in MATLAB

PROCEDURE:

1. Make the connections as shown in the circuit diagram by using MATLAB Simulink.
2. Measure current in each resistor.
3. Verify the mesh analysis.

OBSERVATIONS:

Applied Voltage V (volts)	Loop current(I_1)		Loop current (I_2)		Loop current(I_3)	
	Theoretical	Practical	Theoretical	Practical	Theoretical	Practical

RESULT

PRE LAB VIVA QUESTIONS:

1. On which law is the mesh analysis based?
2. What is mesh analysis?
3. When do we go for super mesh analysis?
4. What is the equation for determining the number of independent loop equations in mesh current method?

POST LAB VIVA QUESTIONS:

1. How do we calculate branch currents from loop currents?
2. How do we calculate branch voltages from loop currents?

EXPERIMENT - 3

(A) NODAL ANALYSIS

AIM

The study of nodal analysis is the objective of this exercise, specifically its usage in multi-source DC circuits. Its application in finding circuit node voltages will be investigated.

APPARATUS:

S.No.	Equipment	Range	Type	Quantity
1.	Resistors	-	-	
2.	Voltmeter			
3.	R.P.S			
4.	Bread Board	-	-	
5.	Connecting Wires			required

THEORY:

In electric circuits analysis, nodal analysis, node-voltage analysis, or the branch current method is a method of determining the voltage (potential difference) between "nodes" (points where elements or branches connect) in an electrical circuit in terms of the branch currents.

CIRCUIT DIAGRAM:

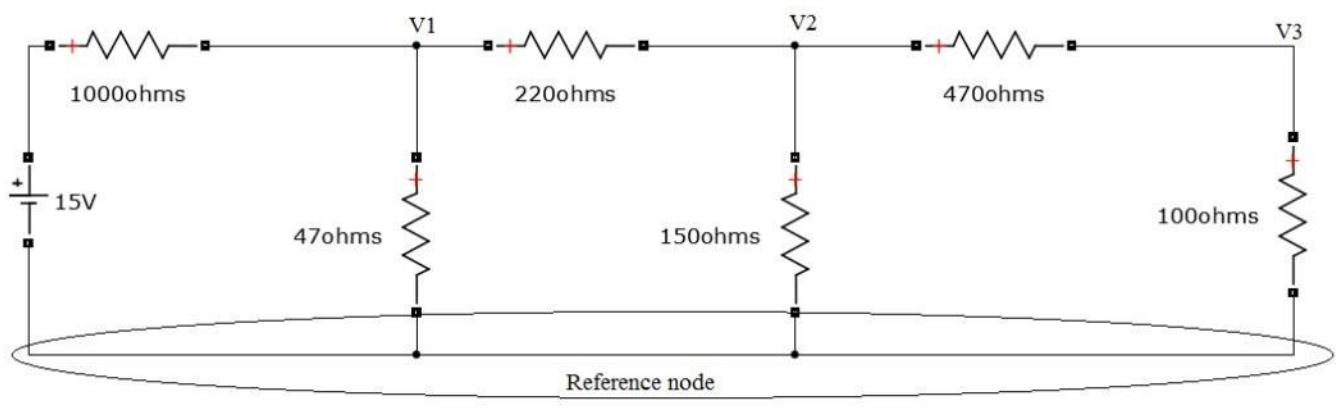


Figure – 3.1 Nodal analysis

PROCEDURE

.

1. Connect the circuit diagram as shown in Figure 3.1.
2. Switch ON the supply to RPS.
3. Apply the voltage (say 15v) and note the voltmeter readings.
4. Gradually increase the supply voltage in steps.
5. Note the readings of voltmeters.
6. Verify the practical results obtained with theoretical results.

OBSERVATIONS:

Applied Voltage V (volts)	Node voltage(v_1)		Node voltage(v_2)		Node voltage(v_3)	
	Theoretical	Practical	Theoretical	Practical	Theoretical	Practical

PRECAUTIONS:

1. Check for proper connections before switching ON the supply
2. Make sure of proper color coding of resistors
3. The terminal of the resistance should be properly connected.

RESULT

(B) NODAL ANALYSIS USING DIGITAL SIMULATION

AIM:

To verify nodal analysis using digital simulation.

APPARATUS:

S. No	SOFTWARE USED	DESKTOP QUANTITY
1	MATLAB R2016a	01

SIMULATION DIAGRAMS:

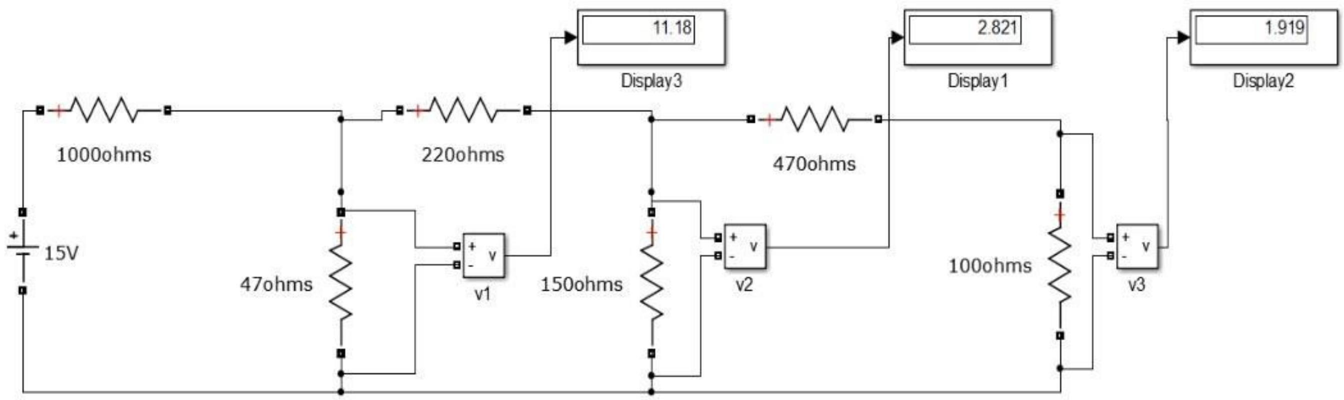


Figure – 3.2 Nodal analysis in MATLAB

PROCEDURE:

1. Make the connections as shown in the circuit diagram by using MATLAB Simulink.
2. Measure the voltage across each node using voltage measurement.
3. Verify with the theoretical results obtained with practical results

OBSERVATIONS:

Applied Voltage V (volts)	Node voltage(V_1)		Node voltage (V_2)		Node voltage (V_3)	
	Theoretical	Practical	Theoretical	Practical	Theoretical	Practical

RESULT:

PRE LAB VIVA QUESTIONS:

1. On which law is the nodal analysis based?
2. What is nodal analysis?
3. When do we, go for super-node analysis?

POST LAB VIVA QUESTIONS:

1. Define node.
2. Is nodal analysis is applicable to both dc and ac supply?
3. How do we calculate branch currents from node voltages?
4. How do we calculate branch voltages from node voltages?

EXPERIMENT - 4

AVERAGE VALUE, RMS VALUE, FORM FACTOR, PEAK FACTOR OF SINUSOIDAL WAVE, SQUARE WAVE

AIM:

To determine the average value, RMS value, form factor, peak factor of sinusoidal wave, square wave.

APPARATUS

S. No	Name	Range	Quantity
1	Resistors	100Ω	2 Nos
2	Inductor	1 mH	1 No
3	Function Generator		1 No
4	Multimeter		1 No
5	CRO		1 No

THEORY:

In alternating current (AC, also ac) the movement (or flow) of electric charge periodically reverses direction. An electric charge would for instance move forward, then backward, then forward, then backward, over and over again. In direct current (DC), the movement (or flow) of electric charge is only in one direction.

Average value: Average value of an alternating quantity is expressed as the ratio of area covered by wave form to distance of the wave form.

Root Mean Square (RMS) Value: The RMS value of an alternating current is expressed by that steady DC current which when flowing through a given circuit for given time produces same heat as produced by that AC through the sane circuit for the same time period. In the common case of alternating current when I(t) is a sinusoidal current, as is approximately true for mains power, the RMS value is easy to calculate from the continuous case equation above. If we define Ip to be the peak current, then in general form

$$I_{\mathrm{RMS}} = \sqrt{\frac{1}{T_2 - T_1}\int_{T_1}^{T_2} (I_{\mathrm{p}}\sin(\omega t))^2 dt}.$$

Where t is time and ω is the angular frequency ($\omega = 2\pi/T$, where T is the period of the wave).
For a sinusoidal voltage,

$$V_{\mathrm{rms}} = \frac{V_{\mathrm{peak}}}{\sqrt{2}}.$$

The factor is called the crest factor, which varies for different waveforms. For a triangle wave form centered about zero.

$$V_{\text{rms}} = \frac{V_{\text{peak}}}{\sqrt{3}}.$$

For a square wave form centered about zero

RMS (Root Mean Square) value of an ac wave is the mean of the root of the square of the voltages at different instants. For an ac wave it will be 1/ √2 times the peak value.

CIRCUIT DIAGRAM:

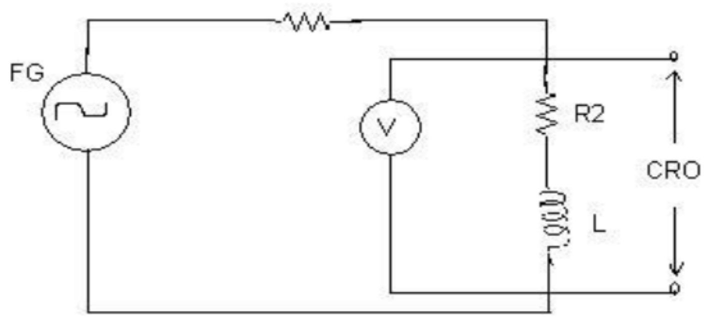


Fig – 4.1 Basic Circuit

PROCEDURE:

1. Connect the circuit as shown in the circuit diagram of fig. 4.1.
2. Set the value of frequency say 100 Hz in the function generator.
3. Adjust the ground of channel 1 and 2 of Cathode Ray Oscilloscope and then set it into DC mode.
4. Connect CRO across the load in DC mode and observe the waveform. Adjust the DC offset of function generator.
5. Note down the amplitude and frequency.
6. Set the multimeter into AC mode and measure input voltage and voltage across point AB. This value gives RMS value of sinusoidal AC.
7. Calculate the average value.
8. Repeat experiment for different frequency and different peak to peak voltage.
9. Measure the RMS and Average value of DC signal also where instead of function generator you can use DC supply.

OBSERVATIONS & CALCULATIONS:

Peak value (V)	RMS value (V)	Average value (V)

PRECAUTIONS:

1. Check for proper connections before switching ON the supply
2. Make sure of proper color coding of resistors
3. The terminal of the resistance should be properly connected

RESULT:

(B) AVERAGE VALUE, RMS VALUE, FORM FACTOR, PEAK FACTOR OF SINUSOIDAL WAVE, SQUARE WAVE USING DIGITAL SIMULATION

AIM:

To Determine the average value, RMS value, form factor, peak factor of sinusoidal wave, square wave.

APPARATUS:

S. No	SOFTWARE USED	DESKTOP QUANTITY
1	MATLAB R2016a	01

CIRCUIT DIAGRAM:

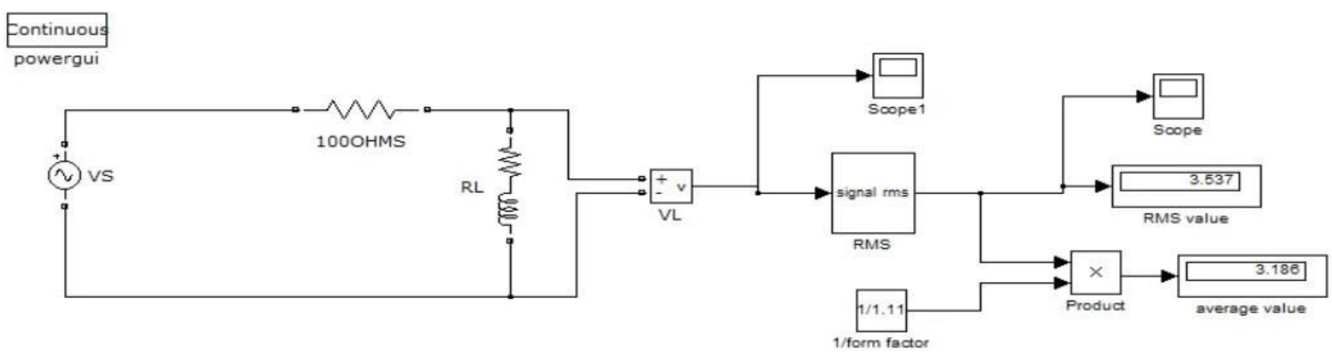


Fig – 4.2 MATLAB Simulink circuit

PROCEDURE:

1. Make the connections as shown in the circuit diagram by using MATLAB Simulink.
2. Measure the Peak value of the voltage obtained
3. Verify with the practical results obtained with theoretical results

OBSERVATIONS & CALCULATIONS:

Peak value (V)	RMS value (V)	Averagevalue (V)

RESULT:

PRE LAB VIVA QUESTIONS:

1. What is complex wave?
2. Define Instantaneous value.
3. Why RMS value is not calculated for DC quantity?
4. Define RMS Value.
5. What is the expression for form factor and peak factor?

POST LAB VIVA QUESTIONS:

1. What is RMS value of Sin wave?
2. Why RMS value is specified for alternating Quantity?
3. Why average value is calculated for half cycle for an sine wave?
4. Define form factor and peak factor for an alternating wave.

EXPERIMENT - 5

(A) VERIFICATION OF SUPERPOSITION THEOREM

AIM:

To Verify principle of Superposition theoretically and practically.

STATEMENT:

In an linear, bilateral network the response in any element is equal to sum of individual responses While all other sources are non-operative.

APPARATUS:

S.No.	Equipment	Range	Type	Quantity
1.	Resistors	-	-	
2.	Ammeter			
3.	R.P.S			
4.	Bread Board	-	-	
5.	Connecting Wires			required

CIRCUIT DIAGRAM:

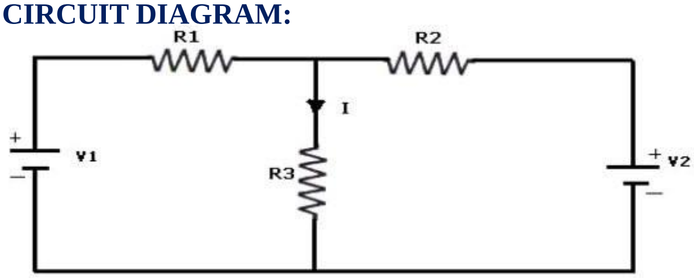


Fig- 5.1 Both Voltage Sources are acting (V_1&V_2)

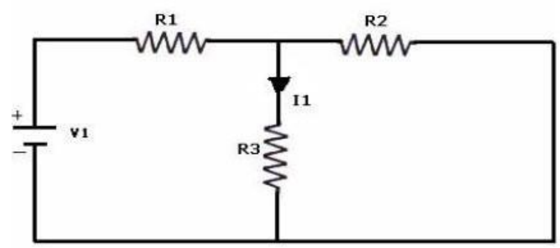


Fig - 5.2 Voltage Source V_1 is acting alone

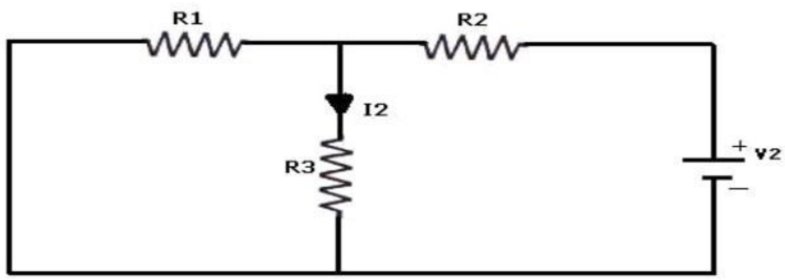


Fig - 5.3 Voltage Source V_2 is acting alone

PROCEDURE:

1. Connect the circuit as shown in figure (5.1) and note down the current flowing through R_3 and let it be I.
2. Connect the circuit as shown in figure (5.2) and note down the ammeter Reading, and let it be I_1.
3. Connect the circuit as shown in figure (5.3) and note down the ammeter reading, and let it be I_2.
4. Verify for $I=I_1+I_2$.

5. Compare the practical and theoretical currents.

TABULAR COLUMN:

PARAMETERS	WHEN BOTH V_1 & $V_2 \neq 0$ (I)	WHEN $V_1 \neq 0$ & $V_2 = 0$ (I_1)	WHEN $V_1 = 0$ & $V_2 \neq 0$ (I_2)
Current through R_3 (Theoretical Values)			
Current through R_3 (Practical Values)			

PRECAUTIONS:

1. Check for proper connections before switching ON the supply
2. Make sure of proper color coding of resistors
3. The terminal of the resistance should be properly connected

RESULT

(B)VERIFICATION OF SUPERPOSITION THEOREM USING DIGITAL SIMULATION.

AIM:

To verify Superposition theorem using digital simulation.

APPARATUS:

S. No	SOFTWARE USED	DESK TOP QUANTITY
1	MATLAB R 2016a	01

CIRCUIT DIAGRAMS:

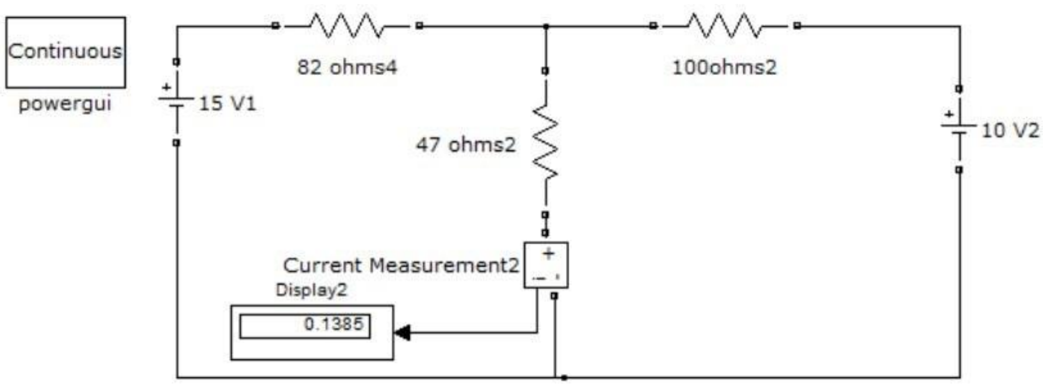


Figure – 5.4 Verification of super position theorem.

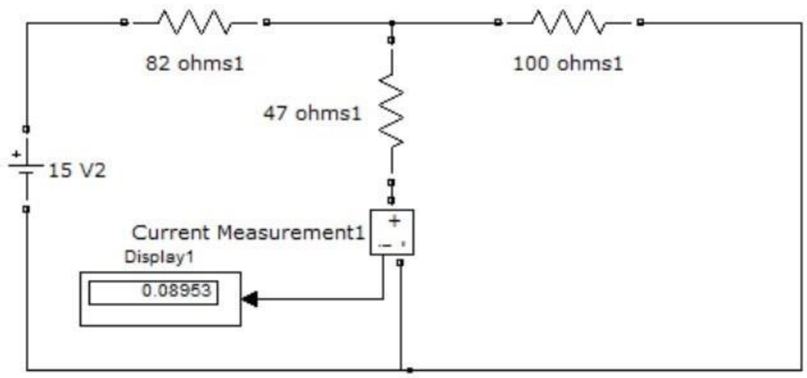


Figure – 5.5. Verification of super position theorem.

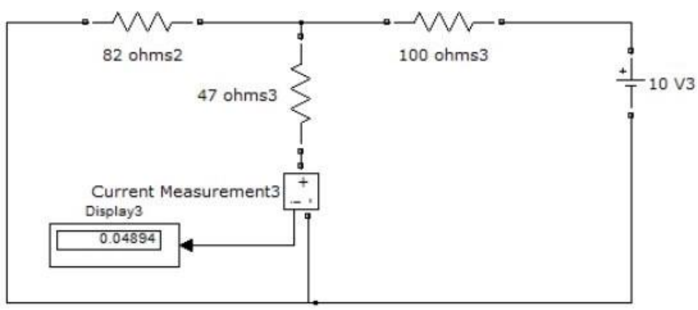


Figure – 5.6. Verification of super position theorem.

PROCEDURE:

1. Make the connections as shown in the circuit diagram by using MATLAB Simulink.
2. Measure the current in each circuit using current measurement.
3. Verify with the theoretical results obtained with practical results

RESULT:

PRE LAB VIVA QUESTIONS:

1. State Superposition theorem.
2. How to find power using Superposition theorem?
3. Write applications of super position theorem.

POST LAB VIVA QUESTIONS:

1. Is it possible to apply Superposition theorem to nonlinear circuit?
2. Is it possible to apply Superposition theorem to ac as well as dc circuit?

EXPERIMENT – 6

(A)VERIFICATION OF RECIPROCITY THEOREM

AIM:

To verify the condition of Reciprocity for an electric network.

STATEMENT

In any linear, bilateral, single source network the ratio of excitation to response is constant even when their positions are inter - changed.

APPARATUS:

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

CIRCUIT DIAGRAM:

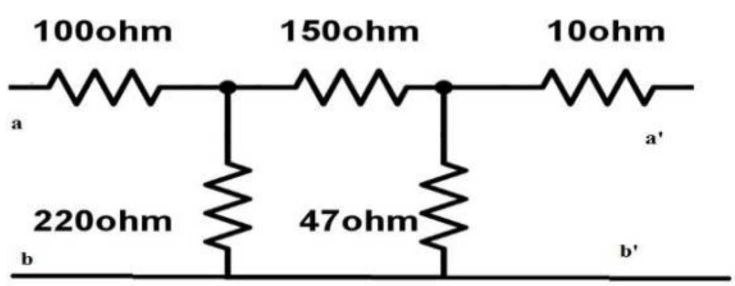


Fig - 6.1 Basic Circuit

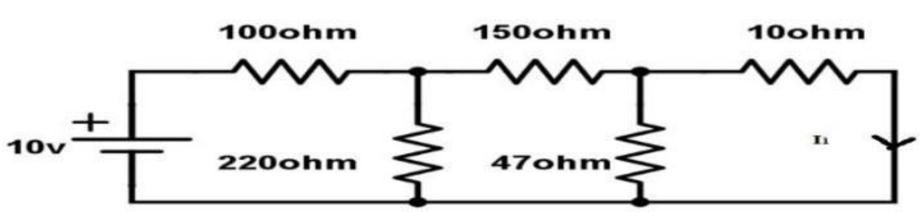


Fig – 6.2 Response due to 10v before interchanging load

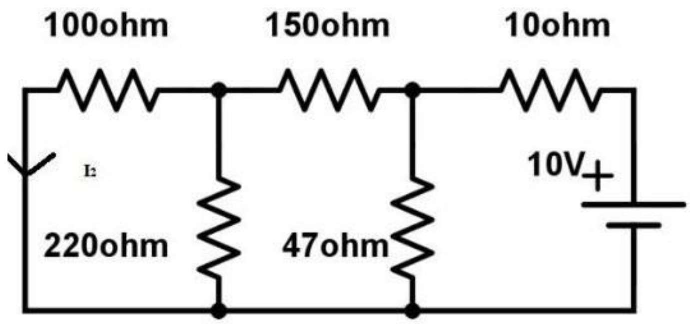


Fig – 6.3 Response due to 10v after interchanging load

PROCEDURE:

1. Connect the circuit as shown in fig 6.2.
2. Measure the current I1 in the branch.
3. Inter - change voltage source and response as shown in fig6.3 and note down the current I2.
4. Observe that the currents I1 and I2 should be same.
5. Measure the ratio of excitation and response and check whether they are equal in both cases are not.

TABULAR COLUMN:

Parameters	Theoretical Values	Practical Values

PRECAUTIONS:

1. Check for proper connections before switching ON the supply
2. Make sure of proper color coding of resistors
3. The terminal of the resistance should be properly connected

RESULT

(B)VERIFICATION OF RECIPROCITY THEOREM USING DIGITAL SIMULATION.

AIM:

To verify Reciprocity theorem using digital simulation.

APPARATUS:

S. No	SOFTWARE USED	DESK TOP QUANTITY
1	MATLAB R2016a	01

CIRCUIT DIAGRAMS:

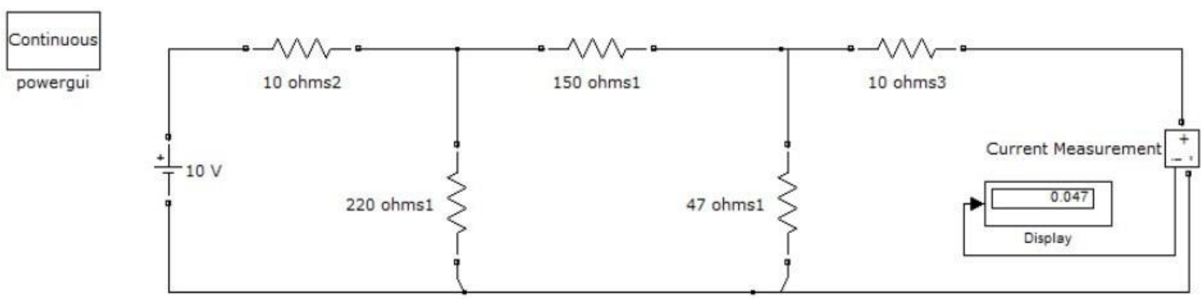


Fig – 6.4 Response due to 10v before interchanging load

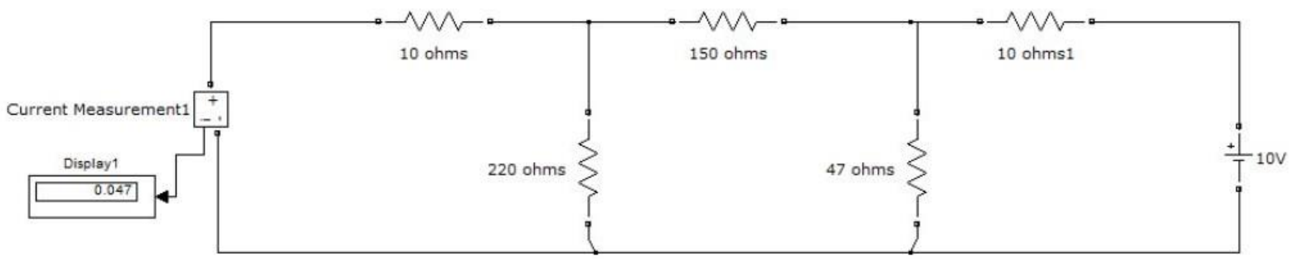


Fig – 6.5 Response due to 10v after interchanging load

PROCEDURE:

1. Make the connections as shown in the circuit-6.4&6.5 diagram by using MATLAB Simulink.
2. Measure response current in the resistor in 10 ohms circuit-6.4.
3. Measure response current in the resistor in 10 ohms circuit-6.5.
4. Verify the reciprocity theorem.

PRE LAB VIVA QUESTIONS:

1. State reciprocity theorem.
2. Is it possible to apply both theorems to ac as well as dc circuit?
3. Is Reciprocity is applicable for unilateral and bilateral networks?

LAB ASSIGNMENT:

1. State and prove reciprocity theorem.
2. State applications of reciprocity theorem.

POST LAB VIVA QUESTIONS:

1. Comment on the applicability of reciprocity theorem on the type of network.
2. Is reciprocity theorem applicable for networks with current source?

EXPERIMENT – 7

(A) VERIFICATION OF MAXIMUM POWER TRANSFER THEOREM

AIM:

To design the load resistor which absorbs maximum power from source.

STATEMENT:

The maximum power transfer theorem states that maximum power is delivered from a source to an load resistance when the load resistance is equal to source resistance. ($R_L = R_S$ is the condition required for maximum power transfer).

CIRCUIT DIAGRAM:

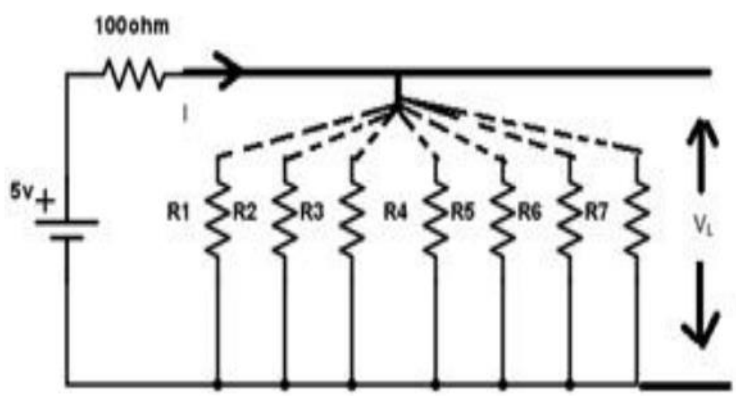


Fig – 7.1 Maximum Power Transfer Circuit

PROCEDURE:

1. Connect the circuit as shown in fig.7.1
2. Vary the load resistance in steps and note down voltage across the load and current flowing through the circuit.
3. Calculate power delivered to the load by using formula P=V*I.
4. Draw the graph between resistance and power (resistance on X- axis and power on Y-axis).
5. Verify the maximum power is delivered to the load when R_L = Rs for DC.

TABULAR COLUMN:

S. No	R_L	V	I	P=VI
1				
2				
3				
4				
5				

MODEL GRAPH:

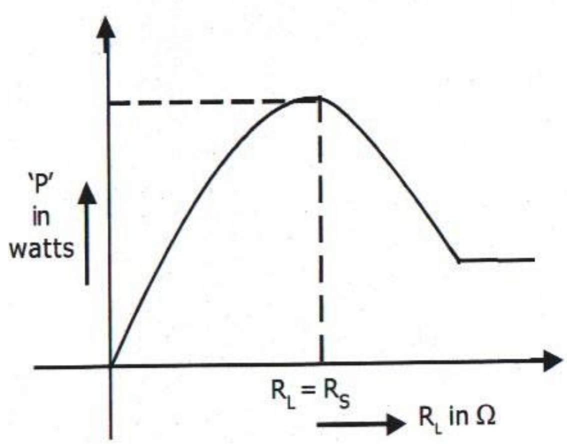


Fig – 7.2 Output Graph of Maximum Power Transfer Theorem

PRECAUTIONS:

1. Check for proper connections before switching ON the supply
2. Make sure of proper color coding of resistors
3. The terminal of the resistance should be properly connected

RESULT

(B) VERIFICATION OF MAXIMUM POWER TRANSFER THEOREM

AIM:

To verify maximum power transfer theorem using digital simulation.

APPARATUS:

S. No	SOFTWARE USED	DESK TOP QUANTITY
1	MATLAB R2016a	01

CIRCUIT DIAGRAMS:

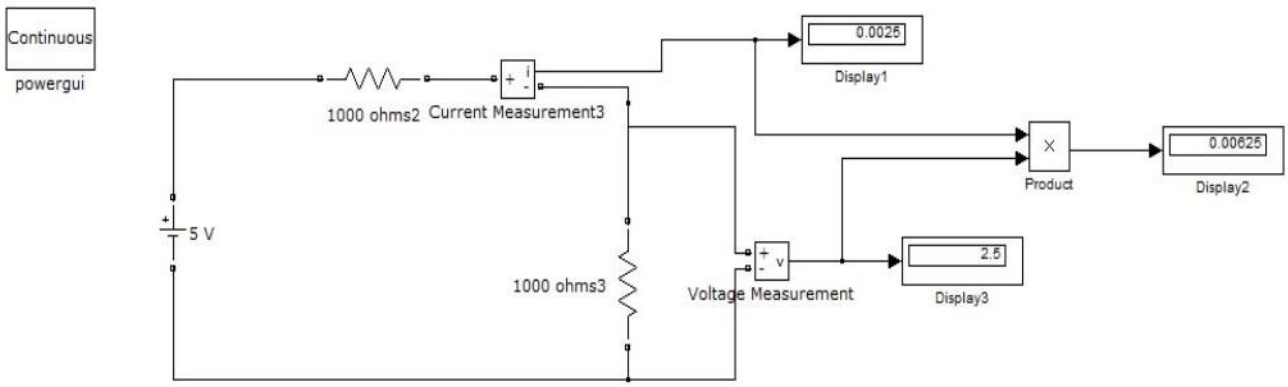


Fig – 7.3 Maximum Power Transfer Circuit

PROCEDURE:

1. Make the connections as shown in the circuit-7.3 diagram by using MATLAB Simulink.
2. Measure the voltage and current through the load resistor using voltage measurement and current measurement
3. Calculate the power .
4. Find the resistance at which maximum power delivered

RESULT:

PRE LAB VIVA QUESTIONS:

1. State maximum power transfer theorem.
2. Is it possible to apply maximum power transfer theorem to ac as well as dc circuit?
3. How to find power using maximum power transfer theorem?

LAB ASSIGNMENT:

1. State and prove maximum power transfer theorem for dc circuit.
2. State and prove maximum power transfer theorem for ac circuit.

POST LAB VIVA QUESTIONS:

1. What are conditions for maximum power transfer theorem?
2. Is it possible to apply maximum power transfer theorem to nonlinear circuit?

EXPERIMENT - 8

(A)VERIFICATION OF THEVENIN'S THEOREM

AIM:

To Verify Thevenin's theorem.

APPARATUS:

S.No.	Equipment	Range	Type	Quantity
1	Ammeter			
2	Voltmeter			
3	R.P.S			
4	Bread Board			
5	Resistors			
6	Connecting Wires			As required

STATEMENT:

Any linear, bilateral network having a number of voltage, current sources and resistances can be replaced by a simple equivalent circuit consisting of a single voltage source in series with a resistance, where the value of the voltage source is equal to the open circuit voltage and the resistance is the equivalent resistance measured between the open circuit terminals with all energy sources replaced by their ideal internal resistances

CIRCUIT DIAGRAM:

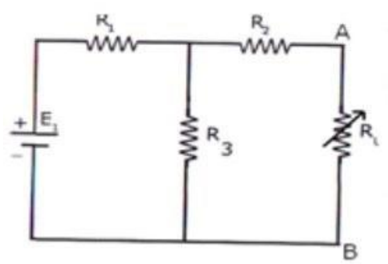


Fig-8.1 Measurement of V_{TH} or V_{OC}

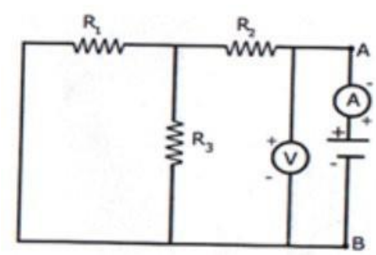


Fig – 8.2 Measurement of R_{TH}

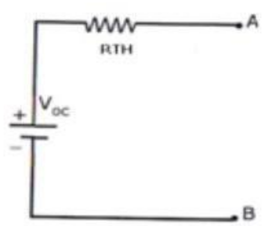


Fig – 8.3 Measurement of I_L (I_L = V_{TH} or V_{OC}/ R_{TH} +R_L)

PROCEDURE:

1. Connect the circuit diagram as shown in fig.8.1
2. Measure current in R_L.
3. Connect the circuit as shown in fig8.2.
4. Measure open circuit voltage Voc by open circuiting terminals i.e, V_{TH}
5. Draw the Thevenin's equivalent circuit as shown in fig8.3
6. Measurement current in R_L

8.6.1 TABULAR COLUMN:

Parameters	Theoretical Values	Practical Values
V_{oc}		
R_{TH}		
I_L		

PRECAUTIONS:

1. Check for proper connections before switching ON the supply
2. Make sure of proper color coding of resistors
3. The terminal of the resistance should be properly connected.

RESULT:

(B)VERIFICATION OF THEVENIN'S THEOREM USING DIGITAL SIMULATION.

AIM:

To verify Thevenin's theorem using digital simulation.

APPARATUS:

S. No	SOFTWARE USED	DESK TOP QUANTITY
1	MATLAB R2016a	01

CIRCUIT DIAGRAMS:

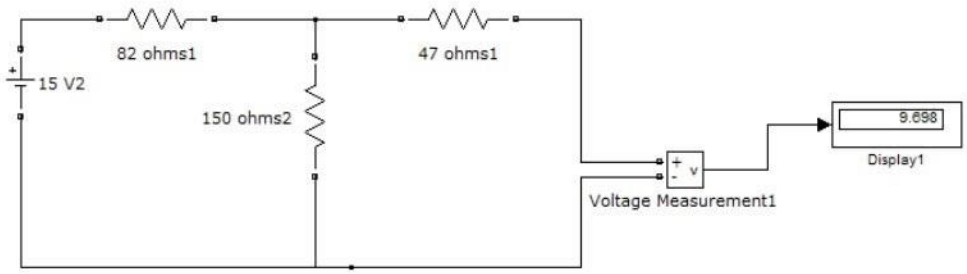


Fig – 8.4 Measurement of V_{TH} or V_{OC}

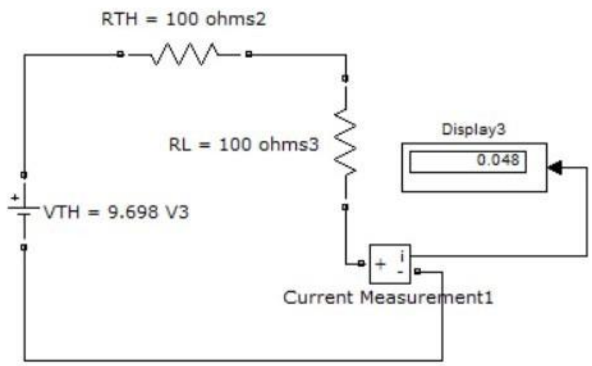


Fig – 8.5 Measurement of I_L (I_L= V_{TH} or V_{OC}/ R_{TH} +R_L)

PROCEDURE:

1. Make the connections as shown in the circuit-8.4 diagram by using MATLAB Simulink.
2. Measure the open circuit voltage across the load terminals using voltage measurement.
3. Connect circuit fig 8.5 Thevenin's equivalent circuit in MATLAB and find the load current.

RESULT:

PRE LAB VIVA QUESTIONS:

1. What is load resistance?
2. How will you calculate Thevenin's resistance R_{TH}?
3. How will you calculate Thevenin's voltage V_{TH}?
4. How will you calculate load current I_L?

LAB ASSIGNMENT:

1. Solve the theoretical value of Thevenin’s theorem for different circuits.
2. Solve the theoretical value of Thevenin’s resistance for different circuits.

POST LAB VIVA QUESTIONS:

1. Write the applications of Thevenin’s theorem.
2. Write the limitations of Thevenin’s theorem.

EXPERIMENT - 9

(A)VERIFICATION OF NORTON 'S THEOREM

AIM:

To Verify Norton's theorem.

STATEMENT

Any linear, bilateral network with current sources, voltage sources and resistances can be replaced by an equivalent circuit consisting of a current source in parallel with a resistance. The value of the current source is the current flowing through the short circuit terminals of the network and the resistance is the equivalent resistance measured between the open circuit terminals of the network with all the energy sources replaced by their internal resistances.

CIRCUIT DIAGRAM:

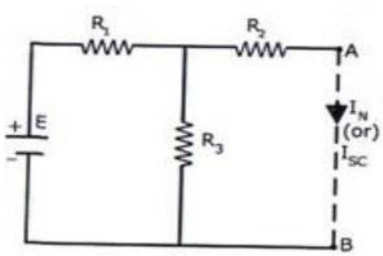


Fig – 9.1 Norton's Current Circuit

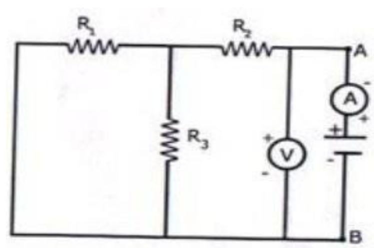


Fig – 9.2 Norton's Equivalent Resistance circuit

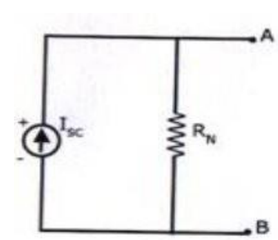


Fig – 9.3 Norton's Equivalent Circuit

PROCEDURE:

1. Connect the circuit diagram as shown in fig 9.1.
2. Measure the current I_{SC} (or) I_N through AB by short-circuiting the resistance between A and B.
3. Connect the circuit diagram as shown in fig 9.2.
4. The resistance between A and B are obtained by using. Voltmeter, ammeter method and the ratio of V and I gives R_N.
5. Draw Norton's equivalent circuit by connecting I_N & R_N in parallel as shown in fig9.3 and find load current.

TABULAR COLUMN:

Parameters	Theoretical Values	Practical Values
I_{SC}/ I_N		
R_N		
I_L		

RESULT:

(B)VERIFICATION OF NORTON'S THEOREM USING DIGITAL SIMULATION

AIM:

To verify Norton's theorem using digital simulation.

APPARATUS:

S. No	SOFTWARE USED	DESK TOP QUANTITY
1	MATLAB R2016a	01

CIRCUIT DIAGRAMS:

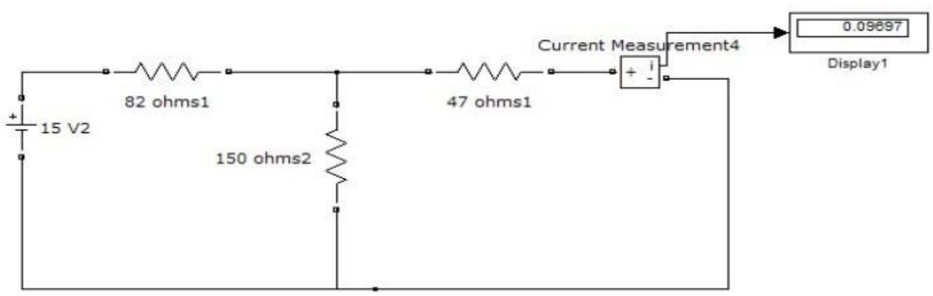


Fig-9.4 Norton's current in MATLAB

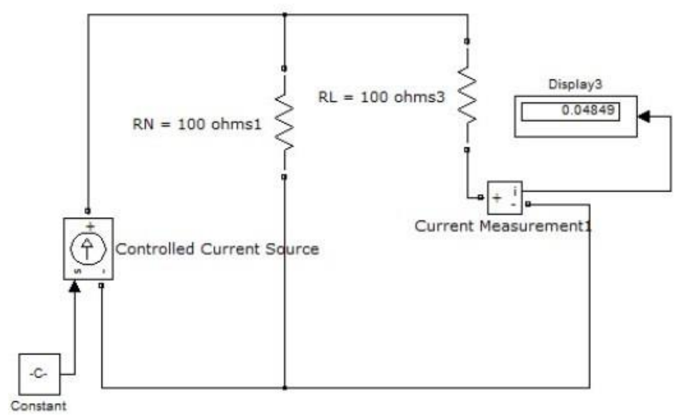


Fig-9.5 Load current in MATLAB

PROCEDURE:

1. Make the connections as shown in the circuit-9.4 diagram by using MATLAB Simulink.
2. Measure the short circuit current through the load terminals using current measurement.
3. Connect circuit fig 9.5 Norton's equivalent circuit in MATLAB and find the load current.

PRECAUTIONS:

1. Check for proper connections before switching ON the supply
2. Make sure of proper color coding of resistors
3. The terminal of the resistance should be properly connected

RESULT:

PRE LAB VIVA QUESTIONS:

1. State Norton's theorem.
2. Define R_N.
3. Define I_N.

LAB ASSIGNMENT:

1. State and prove Norton's theorem.
2. Derive the value of R_N.
3. Find Norton's equivalent resistance from the circuit having dependent source?

POST LAB VIVA QUESTIONS:

1. Convert Thevenin's equivalent into Norton's equivalent.
2. Is it possible to apply Norton's theorem ac as well as dc circuit?
3. What are the applications of Norton's theorem?

EXPERIMENT – 10

(A)VERIFICATION OF COMPENSATION THEOREM

AIM:

To verify the compensation theorem and to determine the change in current.

APPARATUS:

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

STATEMENT

Compensation theorem states that any element in electrical network can be replaced by its equivalent voltage source, whose value is equal to product of current flowing through it and its value. (Compensation theorem got the importance of determining the change in current flowing through element or circuit because of change in the resistance value).

CIRCUIT DIAGRAM:

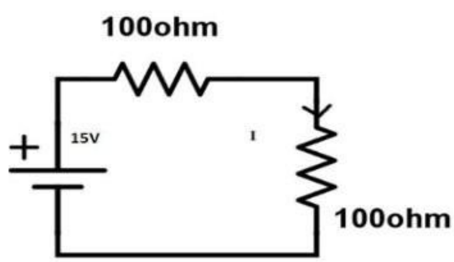


Fig – 10.1 Basic Circuit

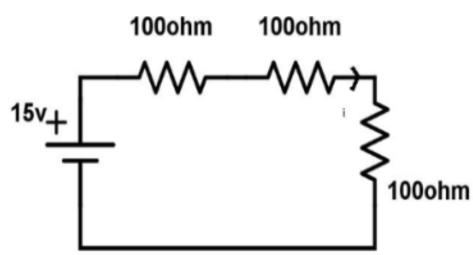


Fig – 10.2 After change in resistance circuit

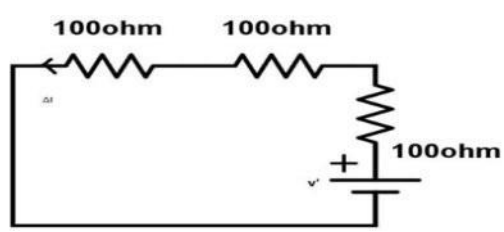


Fig -10.3 Compensation Theorem Circuit

PROCEDURE:

1. Connect the circuit as shown in fig10.1.
2. Measure the current I..
3. Connect the circuit as shown in fig10.2. by increasing the circuit resistance(ΔR), Measure the current I_1.
4. The change in current in the circuit can be found by connecting a voltage source equal to $I_1\Delta R$ as shown in fig10.3.
5. Measure the current I" i.e., the change in current.
6. Observe that I"= I- I_1.

TABULAR COLUMN:

S.No.	Parameters	Theoretical value	Practical value
1			
2			
3			

RESULT:

(B) VERIFICATION OF COMPENSATION THEOREM USING DIGITAL SIMULATION

AIM:

To verify compensation theorem using digital simulation.

APPARATUS:

S. No	SOFTWARE USED	DESK TOP QUANTITY
1	MATLAB R 2016a	01

CIRCUIT DIAGRAMS:

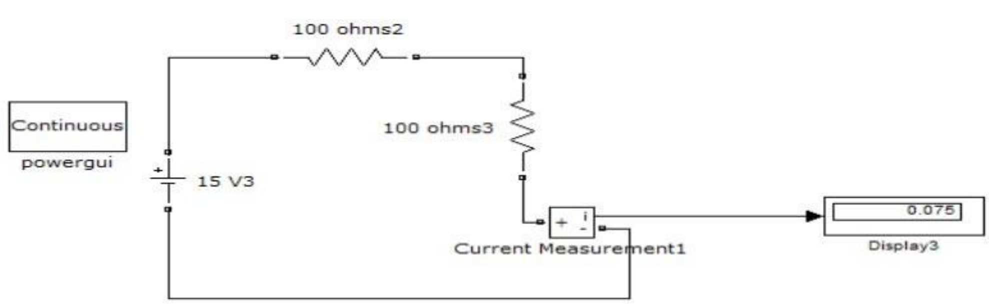


Fig -10.4 Basic circuit in MATLAB.

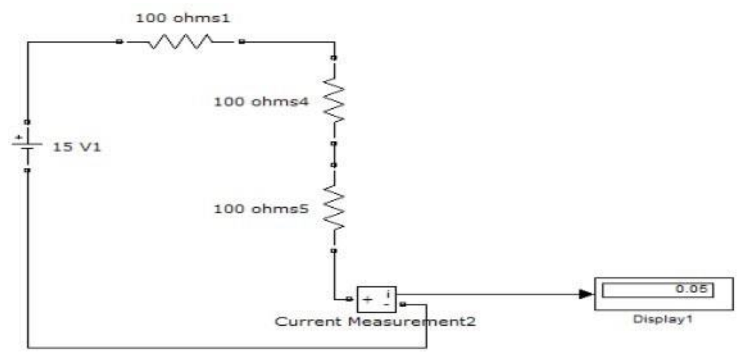


Fig -10.5 After compensation of resistance circuit in MATLAB

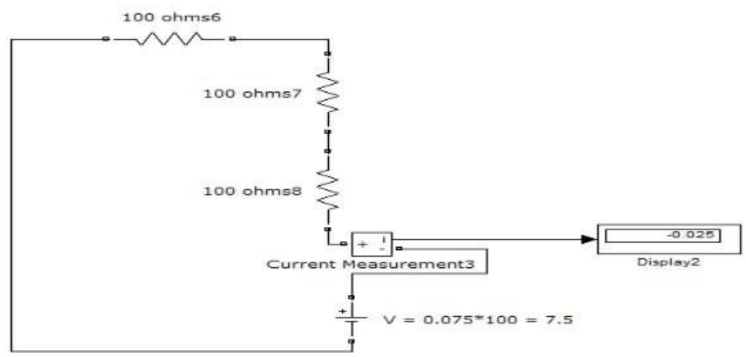


Fig -10.6 Compensation Theorem Circuit in MATLAB

PROCEDURE:

1. Make the connections as shown in the circuit diagrams by using MATLAB Simulink.
2. Using fig.10.4 find the current flowing through original circuit.
3. Using fig 10.5 find current flowing through the change in resistance.
4. Using fig10.6 find the changed current.

PRECAUTIONS:

1. Check for proper connections before switching ON the supply
2. Make sure of proper color coding of resistors
3. The terminal of the resistance should be properly connected

RESULT:

PRE LAB VIVA QUESTIONS:

1. What is Compensation theorem?
2. Is it possible to apply compensation theorem to ac as well as dc circuit?
3. Is Compensation theorem applicable for unilateral and bilateral networks?

LAB ASSIGNMENT:

1. State and prove Compensation theorem.
2. Give the importance of Compensation theorem.

POST LAB VIVA QUESTIONS:

1. Which condition is required to apply the Compensation theorem for the circuit?
2. Comment on the applicability of Compensation theorem on the type of network.
3. Give the importance of compensation theorem.

EXPERIMENT – 11

A) VERIFICATION OF MILLIMAN'S THEOREM

AIM:

To verify the Milliman's Theorem.

STATEMENT:

This theorem states that in any network, if the voltage sources $V_1, V_2, \ldots\ldots V_n$ in series with their internal resistances $R_1, R_2 \ldots R_n$ respectively are in parallel, then these sources may be replaced by a single voltage source V' in series-with R_1.

V' = V1G1+V2G2+-----VnGn / G1+G2 --------Gn

R_1 = 1 / G1+G2 ------------- Gn

APPARATUS:

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

CIRCUIT DIAGRAM:

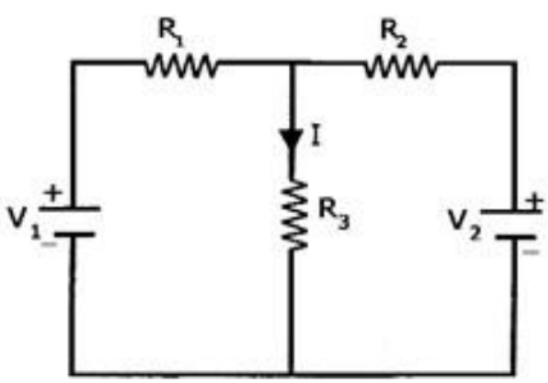


Fig – 11.1 Basic Circuit

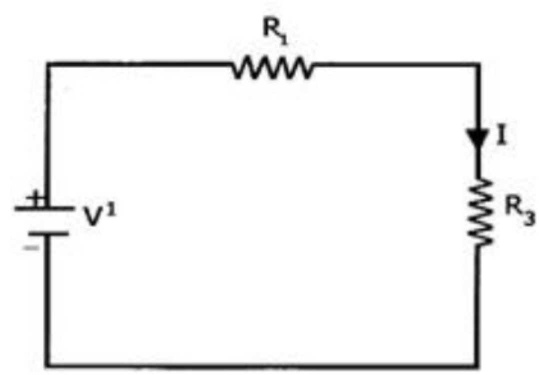


Fig – 11.2 Millman's Circuit

PROCEDURE:

1. Connect the circuit as shown in fig.11.1
2. Measure the current through the resistor R3.
3. Connect the circuit as shown in fig.11.2 and measure the current through R3.
4. Observe that the two currents are same.

TABULAR COLUMN:

S.No.	Parameters	Theoretical value	Practical value
1			
2			

RESULT:

(B)VERIFICATION OF MILLIMAN'S THEOREM USING DIGITAL SIMULATION

AIM:

To verify Milliman's theorem using digital simulation.

APPARATUS:

S. No	SOFTWARE USED	DESK TOP QUANTITY
1	MATLAB R2016a	01

CIRCUIT DIAGRAMS:

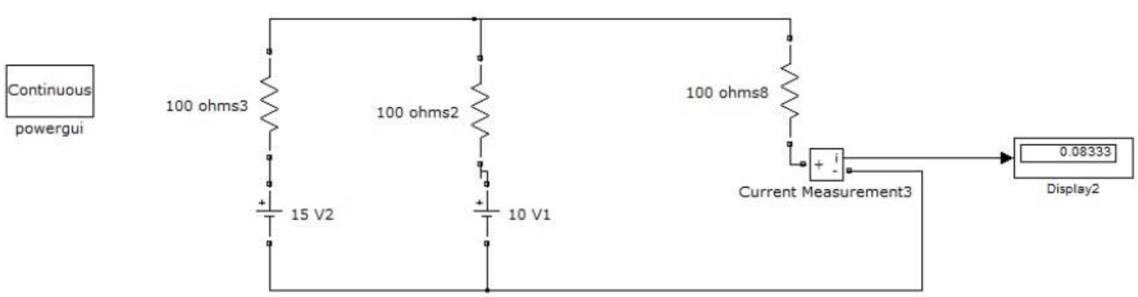


Fig – 11.3 Simulation Basic Circuit

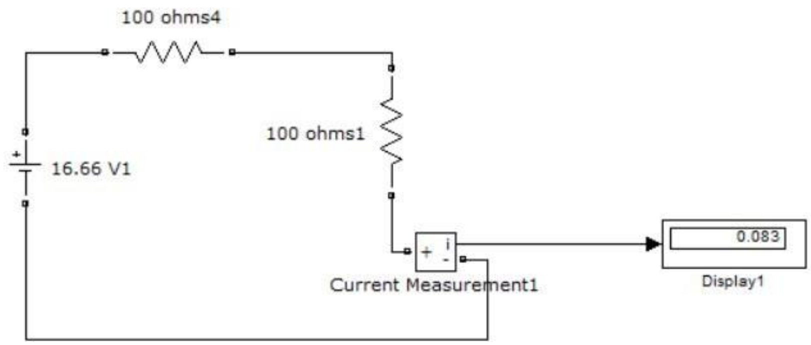


Fig – 11.4 Simulation Millman's Circuit

PROCEDURE:

1. Connect the circuit as shown in fig 11.3 using MAT LAB Simulink.
2. Measure the current through the resistor R3.
3. Connect the circuit as shown in fig.11.4 and measure the current through R3.
4. Observe that the two currents are same.

RESULT:

PRE LAB VIVA QUESTIONS:.

1. State Millman's theorem.
2. Is it possible to apply both theorems to ac as well as dc circuit?
3. Is Millman's is applicable for unilateral and bilateral networks?

LAB ASSIGNMENT:

1. State and prove Millman's theorem.
2. State application of Millman's theorem.

POST LAB VIVA QUESTIONS:

1. Which condition is required to apply the Milliman's theorem for the circuit?
2. Comment on the applicability of Milliman's theorem on the type of network.

EXPERIMENT – 12

(A) VERIFICATION OF SERIES RESONANCE

AIM:

To design the resonant frequency, quality factor and band width of a series resonant circuit.

APPARATUS:

S. No.	Name of the Equipment	Range	Type	Quantity
1	Signal generator			
2	Required resistors			
3	Required Inductors			
4	Required capacitors			
5	CRO probes			
6	Connecting wires			

CIRCUIT DIAGRAM:

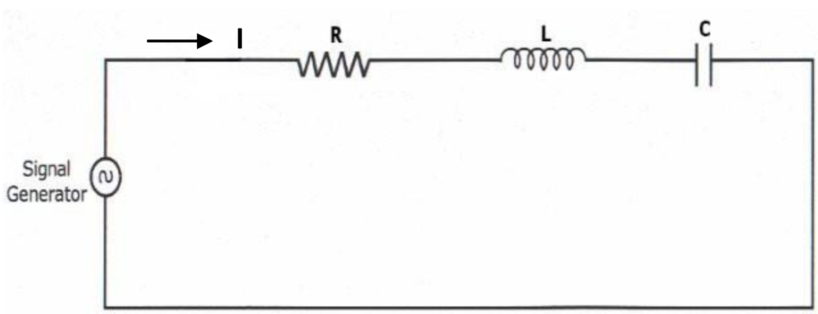


Fig – 12.1 Series Resonance

PROCEDURE:

1. Connect the circuit as shown in fig.12.1 for series resonant circuit
2. Set the voltage of the signal from function generator to 10V.
3. Vary the frequency of the signal in steps and note down the magnitude of response on CRO respectively.(response wave form is observed across element R)
4. Form the observation table between the frequency and magnitude of response in CRO for series resonance circuit.
5. Draw a graph between frequency and magnitude of response on the semi-log sheet and determine the resonant frequency, quality factor and bandwidth of series RLC circuit\.

THEORETICAL CALCULATIONS:

Series Resonance

Resonant Frequency (f_r) = 1/(2∏√LC) Lower

cut off frequency (f_1) = fr-R/4∏L

Upper cut off frequency (f_2) = fr+R/4∏L Quality

factor $Q_r = \omega_r L/R = 1/\omega_r RC$

Band Width f_2-f_1 = R/2∏L

TABULAR COLUMN:

S.No.	Frequency *(Hz)*	Magnitude of response
1		
2		
3		
4		
5		
6		
7		
8		
9		
10		

MODEL GRAPH:

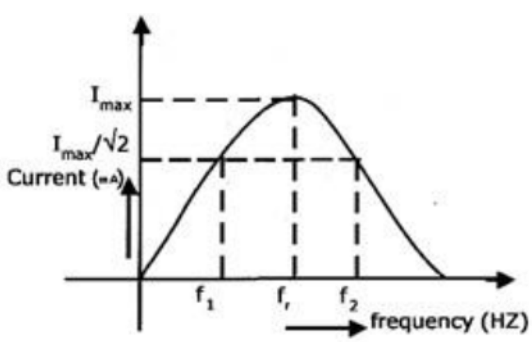


Fig – 12.2 Series Resonance

RESULT:

(B) DESIGN AND SIMULATION OF SERIES RESONANCE CIRCUIT.

AIM:

To plot the magnitude curve for various frequencies for the given RLC series circuit.

SOFTWARE REQUIRED:

S. No	SOFTWARE USED	DESK TOP QUANTITY
1	MATLAB R 2016a	01

THEORY:

A circuit is said to be in resonance when applied voltage V and current I are in phase with each other. Thus at resonance condition, the equivalent complex impedance of the circuit consists of only resistance (R) and hence current is maximum. Since V and I are in phase, the power factor is unity.

The complex impedance

$Z = R + j\,(XL - XC)$

Where $XL = \omega L$

$XC = 1/\omega C$

At resonance, XL= XC and hence Z= R

Bandwidth of a Resonance Circuit:

Bandwidth of a circuit is given by the band of frequencies which lies between two points on either side of resonance frequency, where current falls through 1/1.414 of the maximum value of resonance. Narrow is the bandwidth, higher the selectivity of the circuit. As shown in the model graph, the bandwidth AB is given by f2 – f1. f1 is the lower cut off frequency and f2 is the upper cut off frequency

CIRCUIT DIAGRAM:

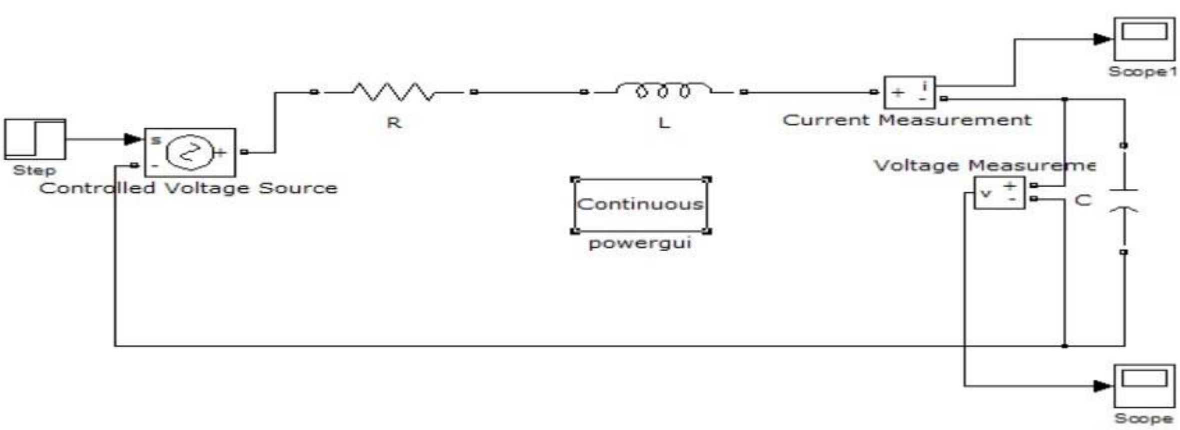


Fig – 12.3 Series Resonance simulation diagram

PROCEDURE:

1. Open a new MATLAB/SIMULINK model
2. Connect the circuit as shown in the figure12.3
3. Debug and run the circuit
4. By double clicking the powergui plot the value of current for the different values of frequencies

MODEL GRAPH FOR SERIES RESONANCE

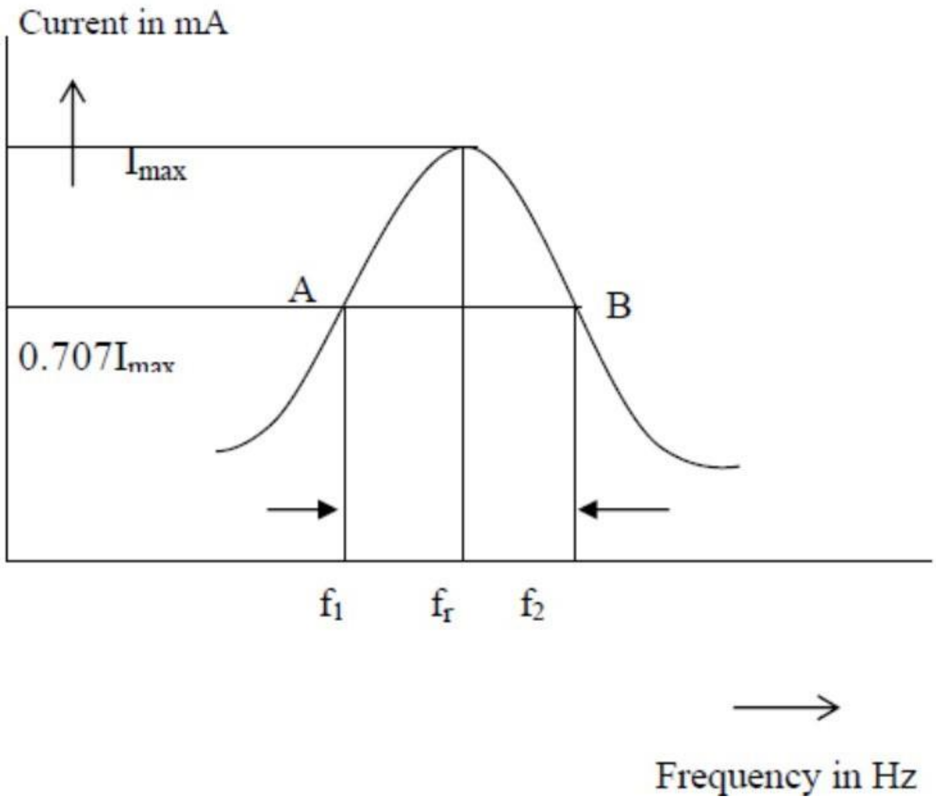


Fig – 12.4 Series Resonance Model graph

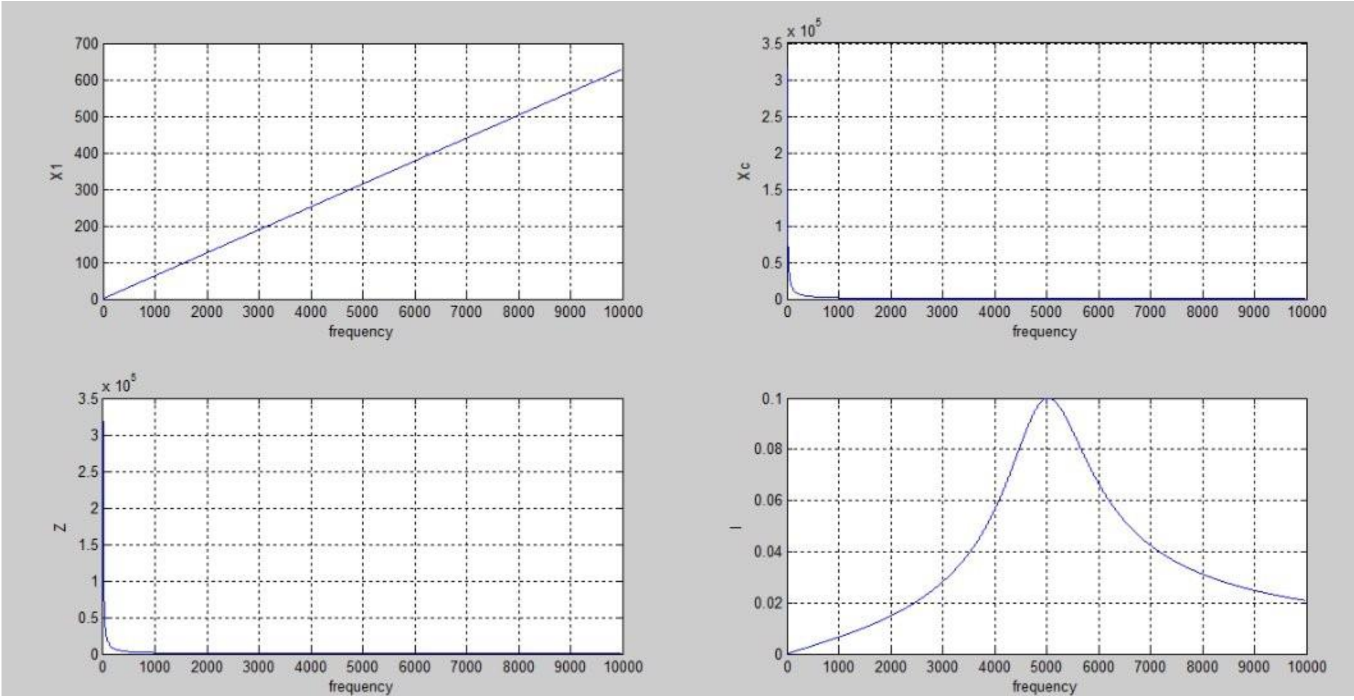
%PROGRAM TO FIND THE SERIES RESONANCE

```
clc;
clear all;
close all;
r=input('enter the resistance value----->');
l=input('enter the inductance value----- >');
c=input('enter the capacitance value ---- >');
v=input('enter the input voltage------->');
f=5:2:300;
xl=2*pi*f*l;
xc=(1./(2*pi*f*c));
x=xl-xc;
```

```
z=sqrt((r^2)+(x.^2));
i=v./z;
%plotting the graph
subplot(2,2,1);
plot(f,xl);
grid;
xlabel('frequency');
ylabel('X1');
subplot(2,2,2);
plot(f,xc);
grid;
xlabel('frequency');
ylabel('Xc');
subplot(2,2,3);
plot(f,z);
grid;
xlabel('frequency');
ylabel('Z');
subplot(2,2,4);
plot(f,i);
grid;
xlabel('frequency');
ylabel('I');
```

PROGRAM RESULT:

```
enter the resistance value ---->100
enter the inductance value------>10e-3
enter the capacitance value----->0.1*10^-6
enter the input voltage------->10
```



PRE LAB VIVA QUESTIONS:

1. Define resonance.
2. Give condition for series resonance.
3. Define band width.
4. Define quality factor.
5. What is the importance of quality factor?

LAB ASSIGNMENT:

1. Define quality factor and give expression for it.
2. Give the application of series resonance circuit.

POST LAB VIVA QUESTIONS:

1. Define series resonance.
2. Define magnification.
3. What is the power factor under resonant condition?

EXPERIMENT – 13

(A) VERIFICATION OF PARALLEL RESONANCE

AIM:

To design the resonant frequency, quality factor and band width of a parallel resonant circuit.

APPARATUS:

S. No.	Name of the Equipment	Range	Type	Quantity
1	Signal generator			
2	Required resistors			
3	Required Inductors			
4	Required capacitors			
5	CRO probes			
6	Connecting wires			

CIRCUIT DIAGRAM:

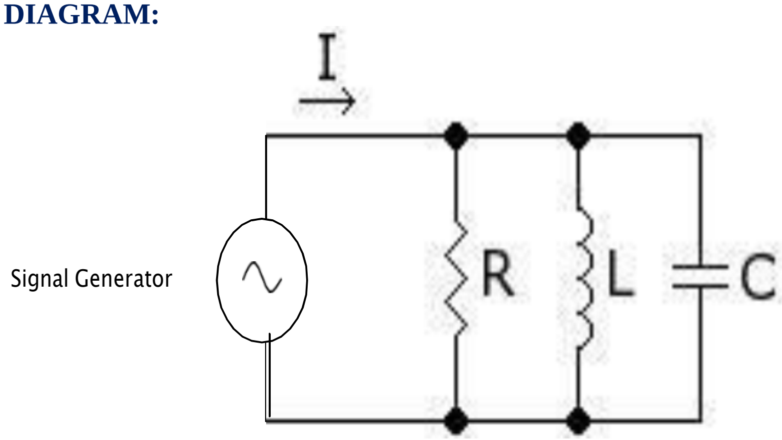


Fig – 13.1 Parallel Resonance

PROCEDURE:

1. Connect the circuit as shown in fig.2 for parallel resonant circuit.
2. Set the voltage of the signal from function generator to 10V.
3. Vary the frequency of the signal in steps and note down the magnitude of response on CRO respectively.(response wave form is observed across element R)
4. Form the observation table between the frequency and magnitude of response in CRO firstly for parallel resonance circuit.
5. Draw a graph between frequency and magnitude of response on the semi-log sheet and determine the resonant frequency, quality factor and bandwidth of parallel RLC circuit.

THEORETICAL CALCULATIONS:

Parallel Resonance

Resonant Frequency $(f_r) = 1/(2\prod\sqrt{LC})$

Lower cut off frequency(f_1) $=f_r-1/4\prod RC$ Upper

cut off frequency $(f_2) = f_r+1/4\prod RC$ Quality

factor $Q_r = \omega_r CR = f_r/B.W$

Band Width $f_2-f_1 = 1/2\prod RC$

TABULAR COLUMN:

S.No.	Frequency (Hz)	Magnitude of response
1		
2		
3		
4		
5		
6		
7		
8		
9		
10		

MODEL GRAPH:

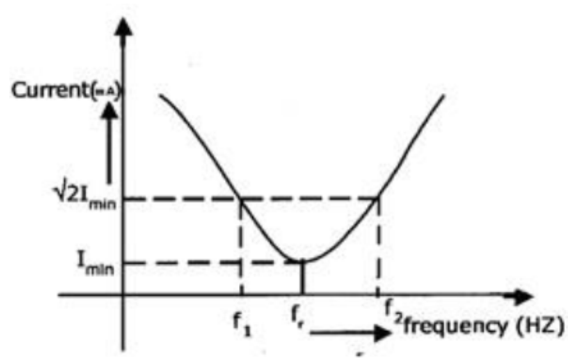


Fig – 13.2. Parallel Resonance

RESULT

(B) DESIGN AND SIMULATION OF PARALLEL RESONANCE CIRCUIT.

AIM:

To plot the magnitude of current for various frequencies for the given RLC parallel circuit.

SOFTWARE REQUIRED:

S. No	SOFTWARE USED	DESK TOP QUANTITY
1	MATLAB R2016a	01

THEORY:

A circuit is said to be in resonance when applied voltage V and current I are in phase with each other. Thus at resonance condition, the equivalent complex impedance of the circuit consists of only resistance (R) and hence current is maximum. Since V and I are in phase, the power factor is unity. The complex impedance

$Z = R + j\,(XL - XC)$

Where $XL = \omega L$

$XC = 1/\omega C$

At resonance, XL= XC and hence Z= R

Bandwidth of a Resonance Circuit:

Bandwidth of a circuit is given by the band of frequencies which lies between two points on either side of resonance frequency, where current falls through 1/1.414 of the maximum value of resonance. Narrow is the bandwidth, higher the selectivity of the circuit. As shown in the model graph, the bandwidth AB is given by f2 – f1. f1 is the lower cut off frequency and f2 is the upper cut off frequency.

PROCEDURE:

1. Open a new MATLAB/SIMULINK model.
2. Connect the circuit as shown in the figure 13.3.
3. Debug and run the circuit.
4. By double clicking the powergui plot the value of current for the different values of frequencies.

SIMULATION DIAGRAM FOR PARALLEL RESONANCE:

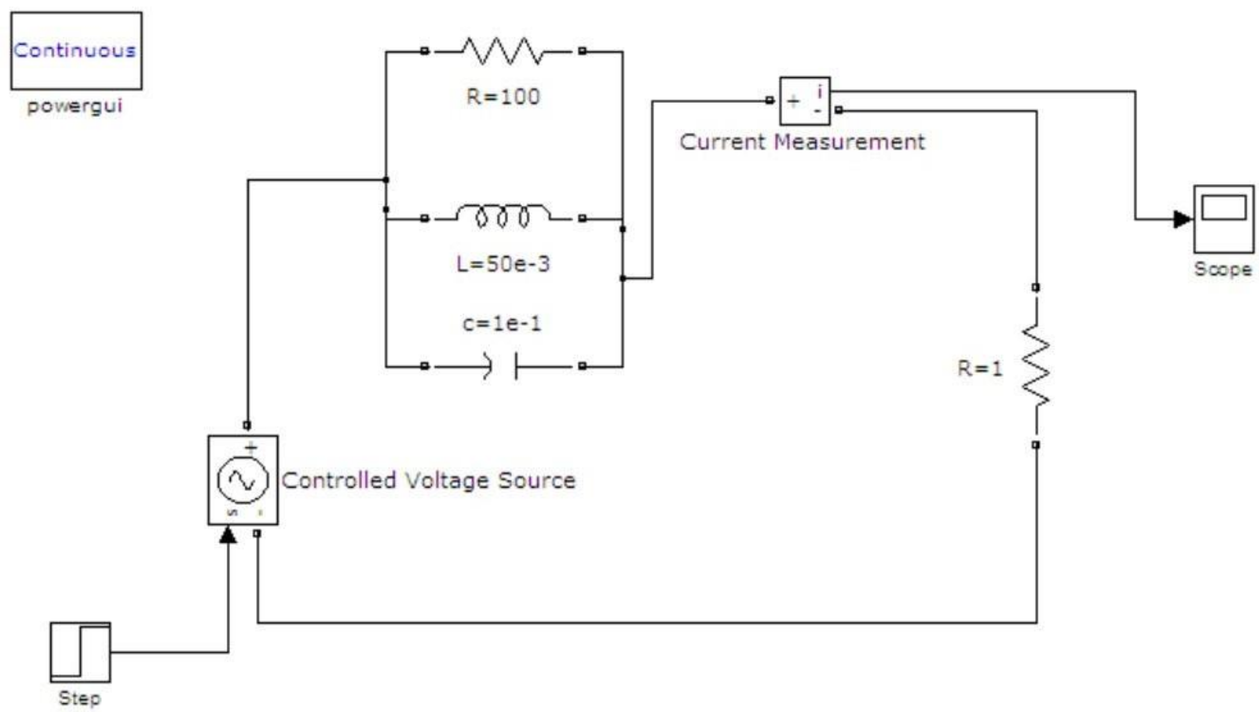
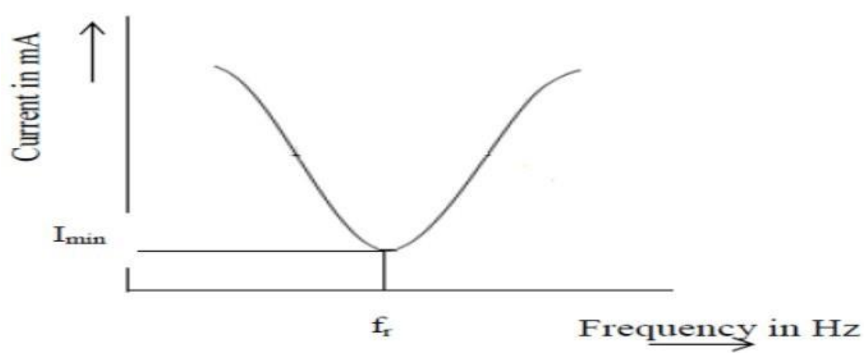


Fig – 13.3 Parallel Resonance

MODEL GRAPH FOR PARALLEL RESONANCE



To obtain the graphs of frequency vs. BL , frequency vs. BC , frequency vs. admittance and frequency vs. current vary frequency in steps for the given circuit and find the resonant frequency and check by theoretical calculations.

R = 1000W, C = 400 m F, L = 1 H, V = 50V vary frequency in steps of 1 Hz using Matlab.

%PROGRAM TO FIND THE PARALLEL RESONANCE

```
clc;
clear all;
close all;
r=input('enter the resistance value----->');
l=input('enter the inductance value----- >');
c=input('enter the capacitance value ---- >');
v=input('enter the input voltage------->');
f=0:2:50;
xl=2*pi*f*l;
xc=(1./(2*pi*f*c));
b1=1./xl;
bc=1./xc;
b=b1-bc;
g=1/r;
y=sqrt((g^2)+(b.^2));
i=v*y;
%plotting the graph
subplot(2,2,1);
plot(f,b1);
grid;
xlabel('frequency');
ylabel('B1');
subplot(2,2,2);
plot(f,bc);
grid;
xlabel('frequency');
ylabel('Bc');
subplot(2,2,3);
plot(f,y);
grid;
xlabel('frequency');
ylabel('Y');
subplot(2,2,4);
plot(f,i);
grid;
xlabel('frequency');
```

```
ylabel('I');
```

PRE LAB VIVA QUESTIONS:

1. Define parallel resonance.
2. Give condition for parallel resonance.
3. Define band width.
4. Define quality factor.
5. What is importance of quality factor?

LAB ASSIGNMENT:

1. Define quality factor and give expression for it.
2. Give the application of parallel resonance circuit.

POST LAB VIVA QUESTIONS:

1. What is the difference between series and parallel resonance?
2. What do you observe from the parallel resonance graphs?

EXPERIMENT – 14

DETERMINATION OF SELF, MUTUAL INDUCTANCE AND COEFFICIENT OF COUPLING

AIM:

To determine self, mutual inductance and coefficient of coupling of a mutually coupled circuit.

APPARATUS:

S. No.	Name of the Equipment	Range	Type	Quantity
1	Ammeter			
2	Voltmeter			
3	1-phase Transformer			
4	1-phase Variac			
5	1-ph A.C. Supply			
6	Connecting Wires			

CIRCUIT DIAGRAM

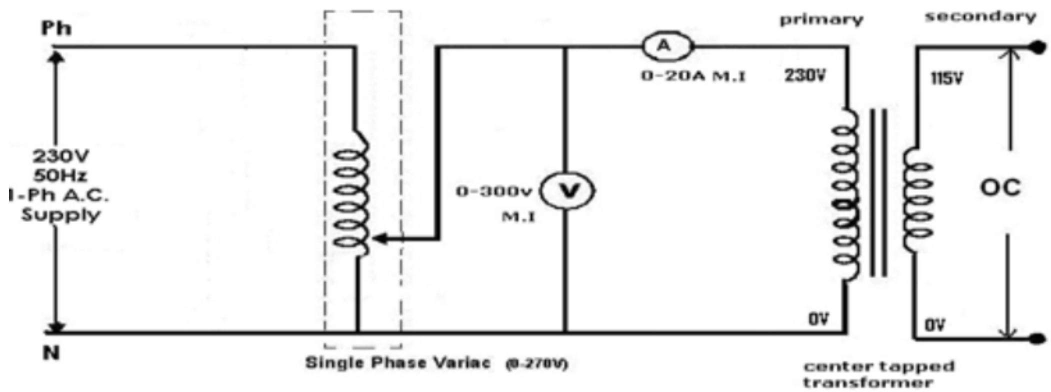


Fig – 14.1 Circuit Diagram of self Inductance of Coil 1

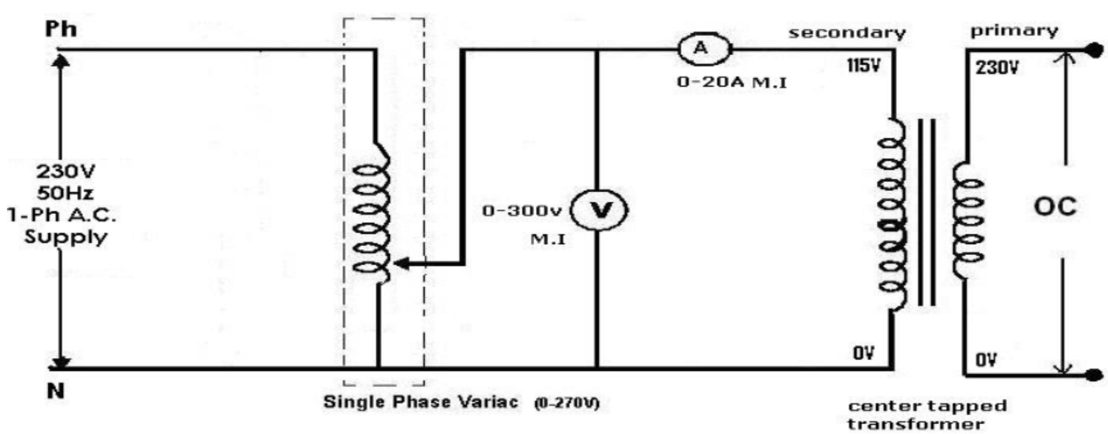


Fig – 14.2 Circuit Diagram of self Inductance of Coil 2

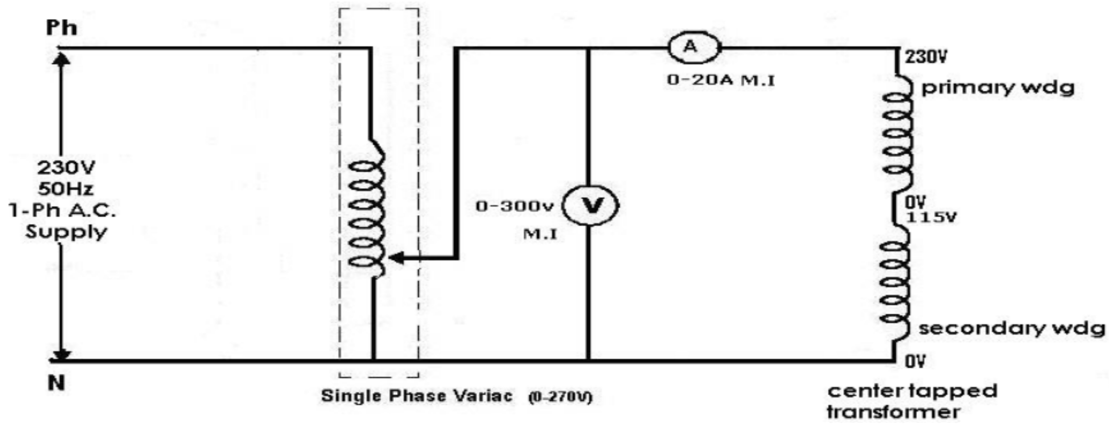


Fig – 14.3 Circuit Diagram of Equivalent Inductance

PROCEDURE:

1. Connect the circuit as shown in fig.14.1 and measure the self inductance of coil 1 i.e. L_1 by noting the voltmeter and ammeter readings.
2. Connect the circuit as shown in fig.14.2 and measure the self inductance of coil 2 i.e. L_2 by noting the voltmeter and ammeter readings.
3. Connect the circuit as shown fig.14.3 and note down voltmeter (V) and ammeter (A) readings and determine equivalent inductance Leq.
4. Calculate coefficient of coupling.

THEORETICAL CALCULATIONS:

(Neglect winding resistance)

$L_{eq} = L_1 + L_2 \pm 2M$

Mutual Inductance $M = [(L_{eq} - (L_1 + L_2))/2]$

Coefficient of Coupling $K = M/\sqrt{(L_1 L_2)}$

Where L_1 and L_2 are determined as follows

Determination of L_1

From fig.14.1

X_{L1}= voltmeter reading /ammeter reading

$X_{L1} = \omega L_1 = 2\Pi f L_1$

$L_1 = X_{L1}/2\Pi f$ (Henry)

Determination of L_2

From fig 14.2

X_{L2} = Voltmeter reading /Ammeter reading

$X_{L2} = \omega L_2 = 2\Pi f L_2$

$L_2 = X_{L2}/2\Pi f$ (Henry)

Determination of L_{eq}

From fig 14.3

X_{Leq} = Voltmeter reading /Ammeter reading

$X_{Leq} = \omega L_{eq} = 2\Pi f L_{eq}$

$L_{eq} = X_{Leq}/2\Pi f$ (Henry)

TABULAR COLUMN: For

X_{L1}

S. No.	Voltmeter Reading	Ammeter Reading	X_{L1} =V/I
1			
2			
3			
4			

For X_{L2}

S. No.	Voltmeter Reading	Ammeter Reading	X_{L2} =V/I
1			
2			
3			
4			

For X_{eq}

S. No	Voltmeter Reading	Ammeter Reading	X_{Leq} =V/I
1			
2			
3			
4			

RESULT:

PRE LAB VIVA QUESTIONS:

1. What is self inductance?
2. What is mutual inductance?
3. Define coefficient of coupling.
4. What is the formula for coefficient of coupling?
5. Define self induced emf.
6. Define mutually induced emf.

LAB ASSIGNMENT:

1. Derive the expression between self inductance of two coils, mutual inductance between them andcoefficient of coupling.
2. State and explain Faraday's law of electromagnetic induction.
3. Two coils of self inductances L_1 and L_2 are connected in series and M is the mutual inductancebetween them, derive the expression for the net inductance of the coil.

POST LAB VIVA QUESTIONS:

1. Using dot convention, discuss the coupling in a simple magnetic circuit?
2. What is statically induce emf and dynamically induce emf?
3. Compare electric and magnetic circuit.
4. What is Lenz's law?

EXPERIMENT – 15

GENERATION OF VARIOUS SIGNALS&SEQUENCES

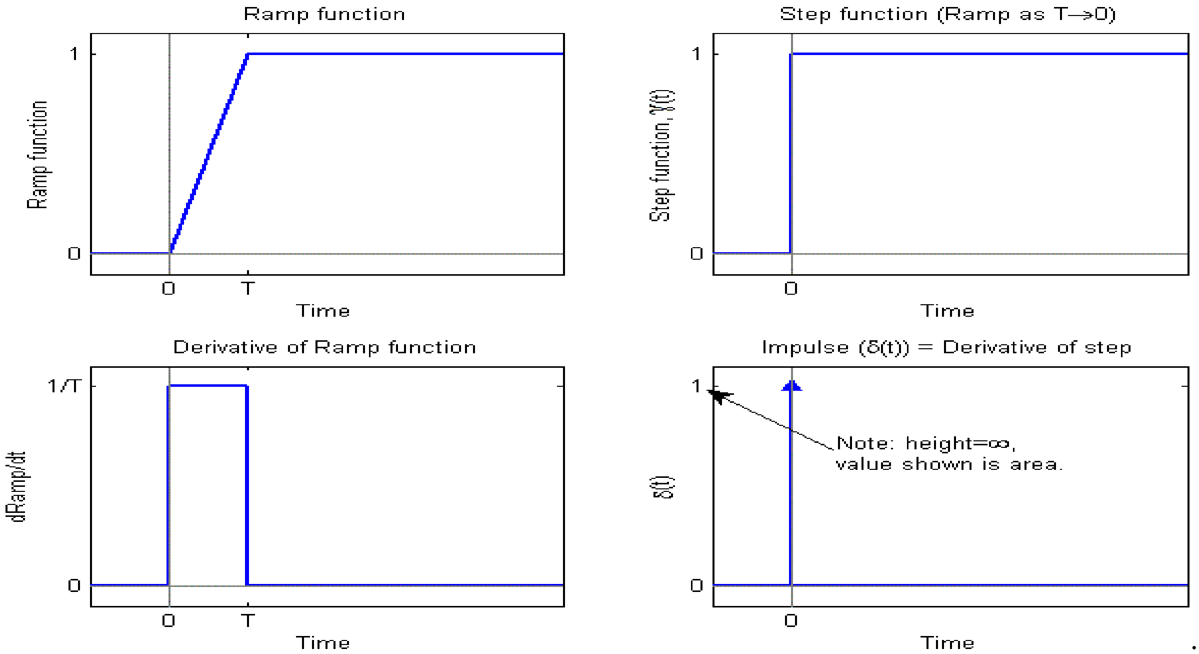
AIM:- To write a "MATLAB" Program to generate various signals and sequences,such asunit impulse, unit step, unit ramp, sinusoidal,square,saw tooth,triangular,sinc signals.

SOFTWARE REQURIED:-

S. No	SOFTWARE USED	DESK TOP QUANTITY
1	MATLAB R2016a	01

THEORY:-

One of the more useful functions in the study of linear systems is the "unit impulse function."An ideal impulse function is a function that is zero everywhere but at the origin, where it is infinitely high. However, the *area* of the impulse is finite. This is, at first hard to visualize butwe can do so by using the graphs shown below



Key Concept: Sifting Property of the ImpulseIf b>a, then

$$\int_a^b \delta(t-T)\cdot f(t)dt = \begin{cases} f(T), & a < T < b \\ 0, & \text{otherwise} \end{cases}$$

Example: Another integral problem

Assume a<b, and evaluate the integral

$$\int_a^b \delta(t)\cdot f(t-T)dt$$

Solution:

We now that the impulse is zero except at t=0 so

$$\delta(t)\cdot f(t-T) = \delta(t)\cdot f(0-T) = \delta(t)\cdot f(-T)$$

And

$$\int_a^b \delta(t)\cdot f(t-T)dt = \int_a^b \delta(t)\cdot f(-T)dt$$

$$= f(-T)\cdot\int_a^b \delta(t)dt$$

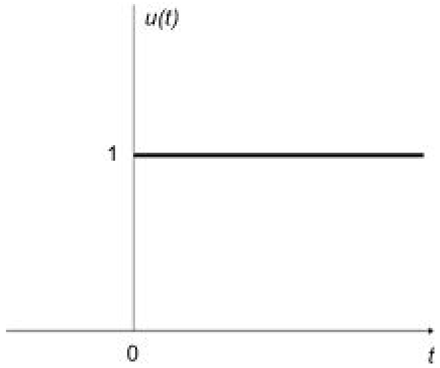
$$= \begin{cases} f(-T), & a<0<b \\ 0, & \text{otherwise} \end{cases}$$

Unit Step Function

The unit step function and the impulse function are considered to be fundamental functions in engineering, and it is strongly recommended that the reader becomes very familiar with both ofthese functions.

The unit step function, also known as the Heaviside function, is defined as such:

$$u(t) = \begin{cases} 0, & \text{if } t<0 \\ 1, & \text{if } t>0 \\ \frac{1}{2}, & \text{if } t=0 \end{cases}$$



Sometimes, u(0) is given other values, usually either 0 or 1. For many applications, it is irrelevant what the value at zero is. u(0) is generally written as undefined.

Derivative

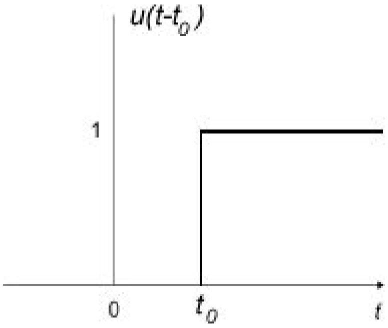
The unit step function is level in all places except for a discontinuity at t = 0. For this reason, the derivative of the unit step function is 0 at all points t, except where t = 0. Where t = 0, the derivative of the unit step function is infinite.

The derivative of a unit step function is called an impulse function. The impulse function willbe described in more detail next.

Integral

The integral of a unit step function is computed as such:

$$\int_{-\infty}^{t} u(s)ds = \left\{ \begin{matrix} 0, & \text{if } t < 0 \\ \int_0^t ds = t, & \text{if } t \geq 0 \end{matrix} \right\} = tu(t)$$



Sinc Function

There is a particular form that appears so frequently in communications engineering, that wegive it its own name. This function is called the "Sinc fu

The Sinc function is defined in the following manner:

$$\text{sinc}(x) = \frac{\sin(\pi x)}{\pi x} \text{ if } x \neq 0$$

And

Sinc(0)=1

The value of sinc(x) is defined as 1 at x = 0, since

$$\lim_{x \to 0} \text{sinc}(x) = 1$$

This fact can be proven by noting that for x near 0,

$$1 > \frac{\sin(x)}{x} > \cos(x)$$

Then, since

Then, since cos(0) = 1, we can apply the approaches one as x goes to zero. Thus, defining s continuous.

Also, the Sinc function approaches zero as x goes towards infinity, with the envelope of sinc(x)tapering off as 1/x.

Rect Function

The Rect Function is a function which produces a rectangular centered at t = 0. The Rect function pulse also has a height of 1. The Sinc function and the rectangular function form a Fourier transform pair.A

Rect function can be written in the form:

$$\operatorname{rect}\left(\frac{t-X}{Y}\right)$$

where the pulse is centered at X and has width Y. We can define the impulse function above in terms of the rectangle function by centering the pulse at zero (X = 0), setting it's height to 1/A and setting the pulse width to A, which approaches zero:

$$\delta(t) = \lim_{A\to 0} \frac{1}{A} \operatorname{rect}\left(\frac{t-0}{A}\right)$$

We can also construct a Rect function out of a pair of unit step functions

$$\operatorname{rect}\left(\frac{t-X}{Y}\right) = u(t-X+Y/2) - u(t-X-Y/2)$$

Here, both unit step functions are set a distance of Y/2 away from the center point of (t - X).

<u>SAW TOOTH:-</u>

The sawtooth wave (or saw wave) is a kind of non-sinusoidal waveform. It is named a sawtooth based on its resemblance to the teeth on the blade of a saw. The convention is that a sawtooth wave ramps upward and then sharply drops. However, thereare also sawtooth waves in which the wave ramps downward and then sharply rises. The latter type of sawtooth wave is called a 'reverse sawtooth wave' or 'inverse sawtooth wave'. As audio signals, the two orientations of sawtooth wave sound identical. The piecewise linear function based on the floor function of time t, is an example of a sawtooth wave with period 1.

$$x(t) = 2\left(\frac{t}{a} - \operatorname{floor}\left(\frac{t}{a} + \frac{1}{2}\right)\right)$$

Triangle wave

A triangle wave is a non-sinusoidal waveform named for its triangular shape.A bandlimited triangle wave pictured in the time domain (top) and frequency domain (bottom). The fundamental is at 220 Hz (A2).Like a square wave, the triangle wave contains only odd harmonics. However, the higher harmonics roll off much faster than ina square wave (proportional to the inverse square of the harmonic number as opposed to just the inverse).It is possible to approximate a triangle wave with additive synthesis by adding odd harmonics of the fundamental, multiplying every (4n+1)th harmonic by 1 (or changing its phase by w), and rolling off the harmonics by the inverse

$$x_{\text{triangle}}(t) = \frac{8}{\pi^2}\sum_{k=0}^{\infty}(-1)^k \frac{\sin((2k+1)\omega t)}{(2k+1)^2}$$
$$= \frac{8}{\pi^2}\left(\sin(\omega t) - \frac{1}{9}\sin(3\omega t) + \frac{1}{25}\sin(5\omega t) - \cdots\right)$$

where ω is the angular frequency.

square of their relative frequency to the fundamental. This infinite Fourier series converges to the trianglewave:

Sinusoidal Signal Generation

The sine wave or sinusoid is a mathematical function that describes a smooth repetitive oscillation. It occurs often in pure mathematics, as well as physics, signal processing, electrical engineering and many other fields. Its most basic form as a function of time (t)is:

where:

- A, the amplitude, is the peak deviation of the function from its center position.
- f , the angular frequency, specifies how many oscillations occur in a unit time interval, in radians per second
- Ɵ, the phase, specifies where in its cycle the oscillation begins at t = 0.

A sampled sinusoid may be written as:

$$x(n) = A\sin(2\pi \frac{f}{f_s} n + \vartheta)$$

where f is the signal frequency, fs is the sampling frequency, Ɵ is the phase and A is theamplitude of the signal.

PROCEDURE:-

1. Open MATLAB
2. Open new M-file
3. Type the program
4. Save in current directory
5. Compile and Run the program
6. For the output see command window\ Figure window

PROGRAM:-

```
%unit impulse function%
clc;
clear all;
close all;
t=-10:1:10;
x=(t==0);
subplot(2,1,1);
plot(t,x,'g');
xlabel('time');
```

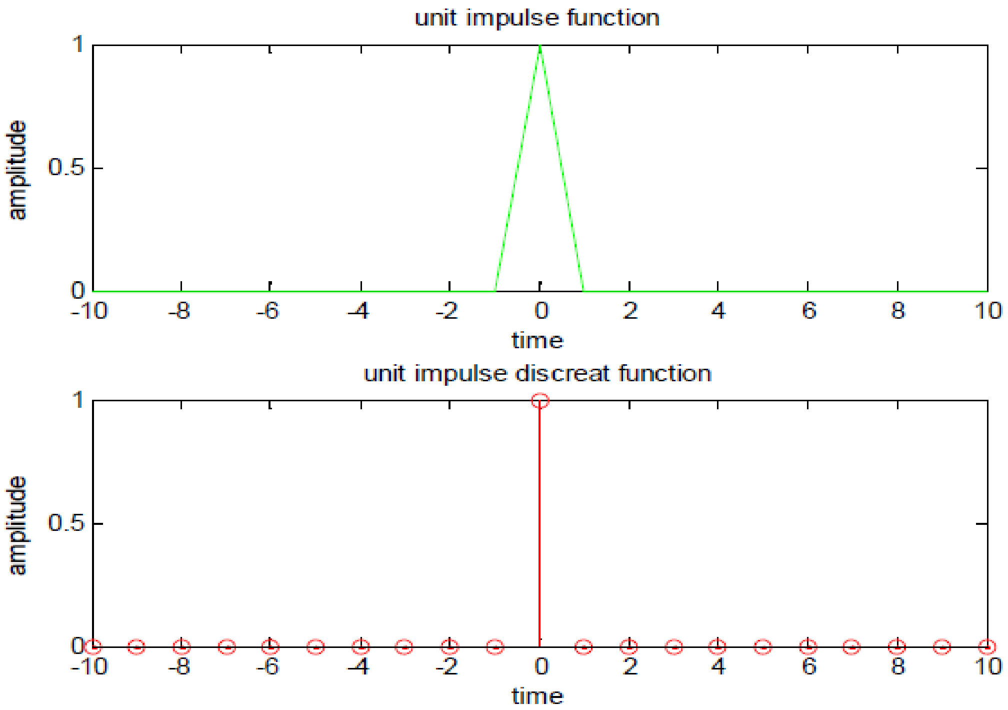
```
ylabel('amplitude');
title('unit impulse function');subplot(2,1,2);
stem(t,x,'r');
xlabel('time');
ylabel('amplitude');
title('unit impulse discreat function');
%unit step function%
clc;
clear all;
close all;
N=100;
t=1:100;
x=ones(1,N);
subplot(2,1,1);
plot(t,x,'g');
xlabel('time');
ylabel('amplitude');
title('unit step function');
subplot(2,1,2);
stem(t,x,'r');
xlabel('time');
ylabel('amplitude');
title('unit step discreat function');
%unit ramp function%
%unit ramp function%
clc;
clear   all;
close   all;
t=0:20;
x=t;
subplot(2,1,1);
plot(t,x,'g');
xlabel('time'); ylabel('amplitude');
title('unit ramp function');
subplot(2,1,2);
```

```
stem(t,x,'r');
xlabel('time');
ylabel('amplitude');
title('unit ramp discreat function');
%sinusoidal function%
clc;
clear all;
close all;
t=0:0.01:2;
x=sin(2*pi*t);
subplot(2,1,1);
plot(t,x,'g');
xlabel('time');
ylabel('amplitude');
title('sinusoidal signal');
subplot(2,1,2);
stem(t,x,'r');
xlabel('time');
ylabel('amplitude');
title('sinusoidal sequence');
%square function%
clc;
clear all;
close all;
t=0:0.01:2;
x=square(2*pi*t);
subplot(2,1,1);
plot(t,x,'g');
xlabel('time');
ylabel('amplitude');
title('square signal');
subplot(2,1,2);
stem(t,x,'r');
xlabel('time');
label('amplitude');
title('square sequence');
```

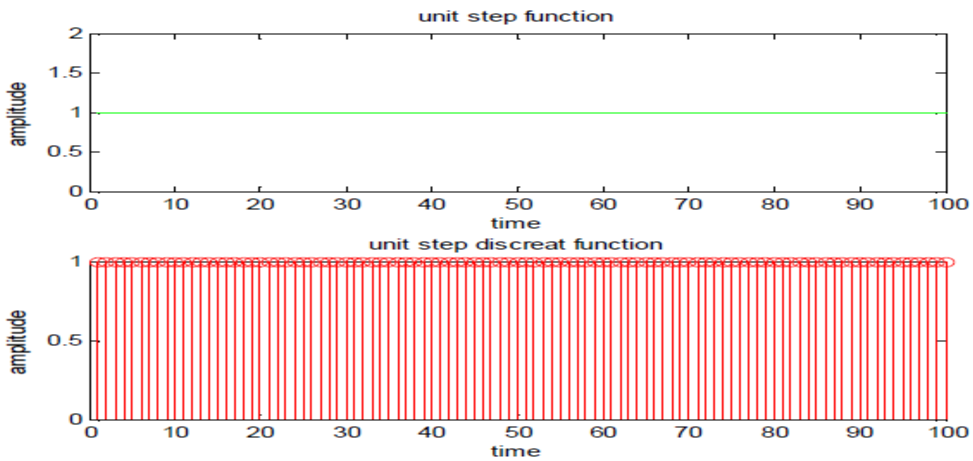
```
%sawtooth function%
clc;
clear all;
close all;
t=0:0.01:2;
x=sawtooth(2*pi*5*t);
subplot(2,1,1);
plot(t,x,'g');
xlabel('time');
ylabel('amplitude');
title('sawtooth signal');
subplot(2,1,2);
stem(t,x,'r');
xlabel('time');
ylabel('amplitude');
title('sawtooth sequence');
%trianguler function%
clc;
clear all;
close all;
t=0:0.01:2;
x=sawtooth(2*pi*5*t,0.5);
subplot(2,1,1);
plot(t,x,'g');
xlabel('time');
ylabel('amplitude');
title('trianguler signal');
subplot(2,1,2);
stem(t,x,'r');
xlabel('time');
ylabel('amplitude');
title('trianguler sequence');
%sinc function%
clc;
clear all;
close all;
```

```
t=linspace(-5,5);
x=sinc(t);
subplot(2,1,1);
plot(t,x,'g');
xlabel('time');
ylabel('amplitude');
title('sinc signal');
subplot(2,1,2);
stem(t,x,'r');
xlabel('time');
ylabel('amplitude');
title('sinc sequence');
```

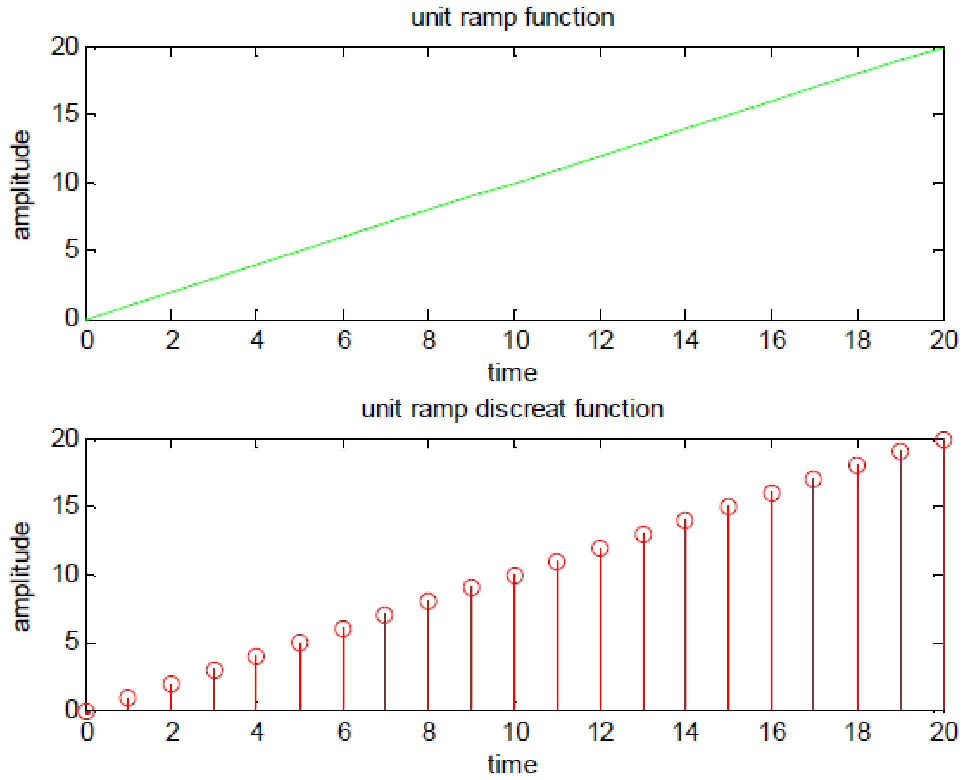
unit impulse function



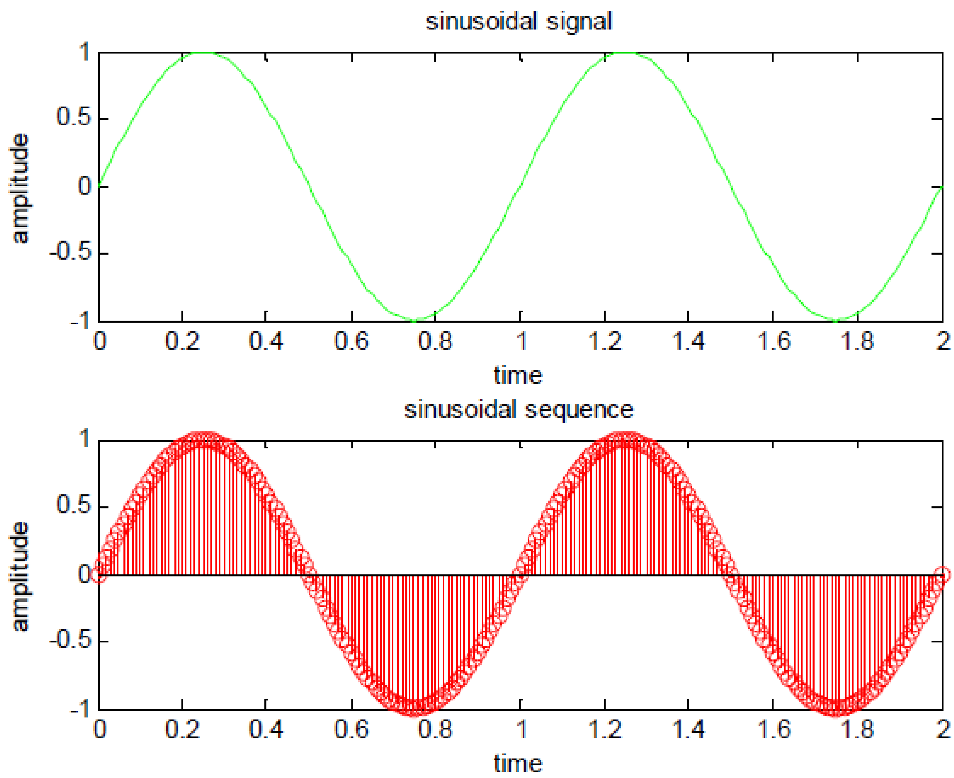
unit step function



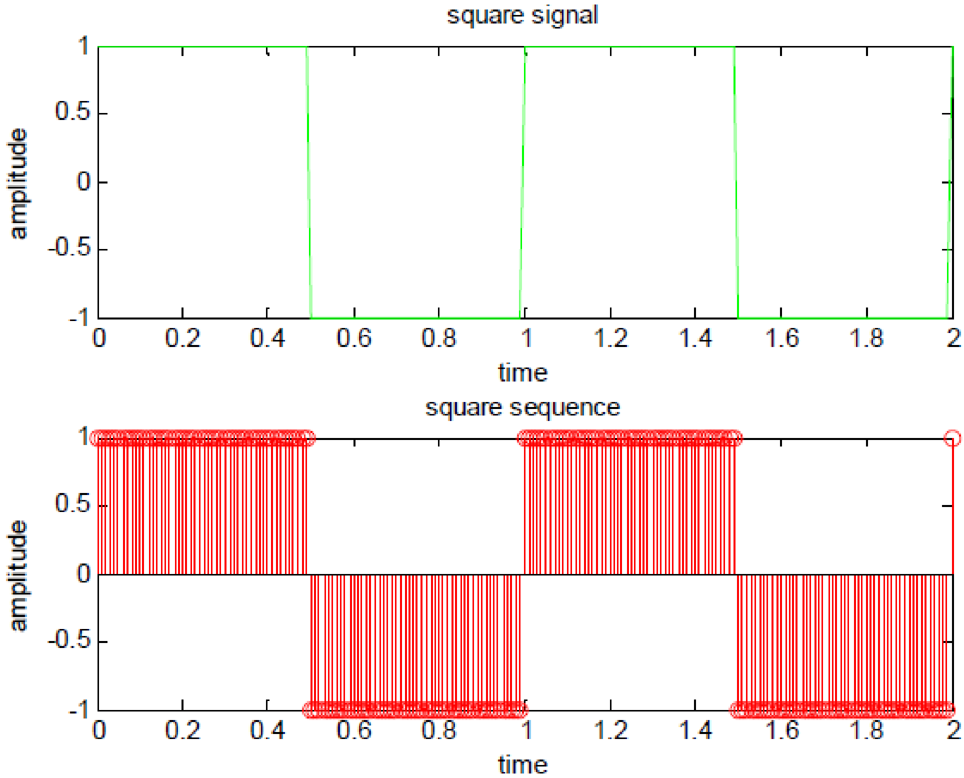
Unit ramp function



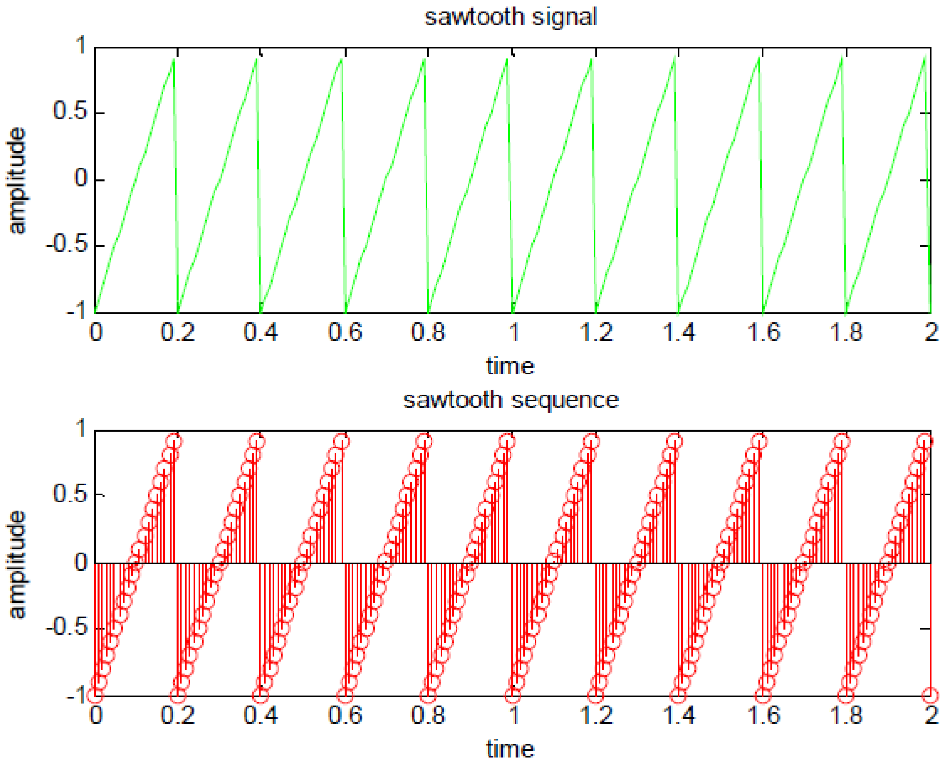
sinusoidal function



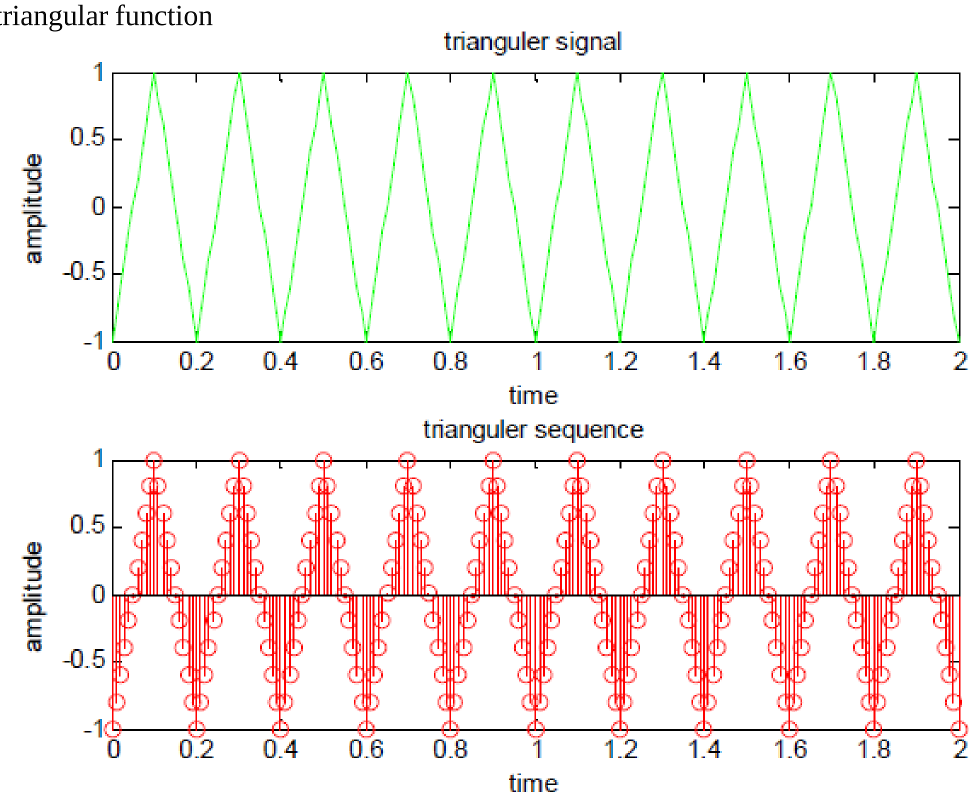
square function



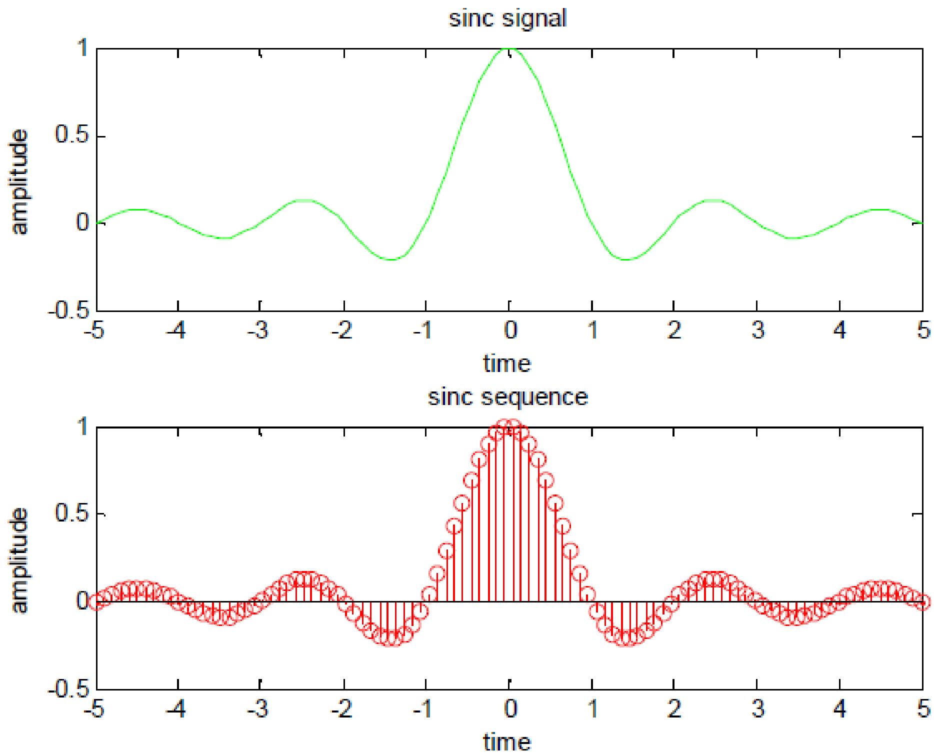
sawtooth function



triangular function



sinc function



RESULT:-

Thus the Generation of continuous time signals like unit step, saw tooth, triangular, sinusoidal, ramp and sinc functions are successfully completedby using MATLAB.

VIVA QUESTIONS:-

1. Define Signal?
2. Define deterministic and Random Signal?
3. Define Delta Function?
4. What is Signal Modeling?
5. Define Periodic and a periodic Signal?

EXPERIMENT – 16

OPERATIONS ON SIGNALS AND SEQUENCES

To performs operations on signals and sequences such as addition, multiplication, scaling, shifting, folding, computation of energy and average power.

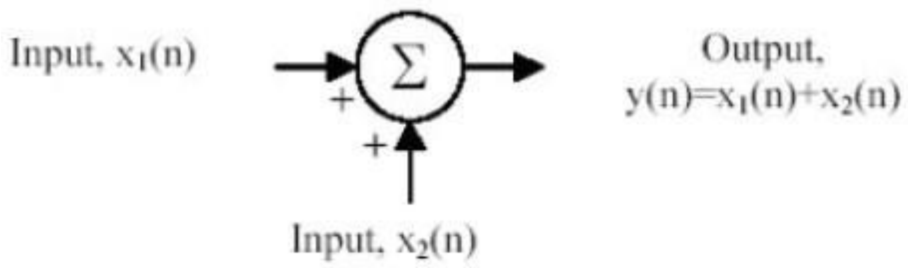
SOFTWARE REQURIED:-

S. No	SOFTWARE USED	DESK TOP QUANTITY
1	MATLAB R2016a	01

THEORY:-**Basic Operation on Signals**

Time shifting: y(t)=x(t-T)The effect that a time shift has on the appearance of a signalIf T is a positive number, the time shifted signal, x (t -T) gets shifted to the right, otherwise it gets shifted left.

Signal Shifting and Delay:

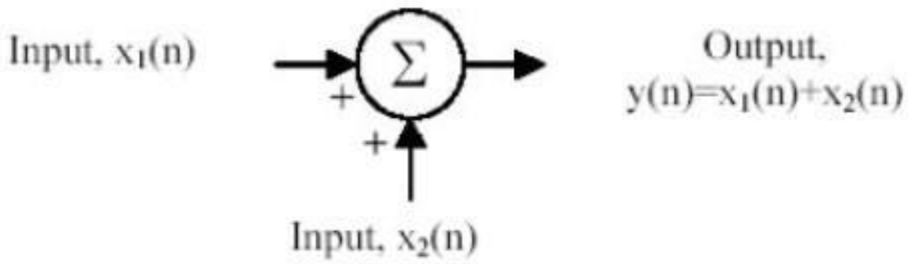


Shifting : y(n)={x(n-k)} ; m=n-k; y=x;

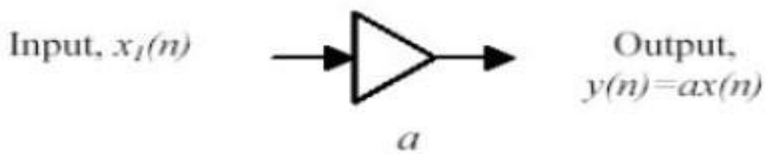
Time reversal: Y(t)=y(-t) Time reversal _ips the signal about t = 0 as seen inFigure 1.

Signal Addition and Substraction :

Addition: any two signals can be added to form a third signal,z (t) = x (t) + y (t)



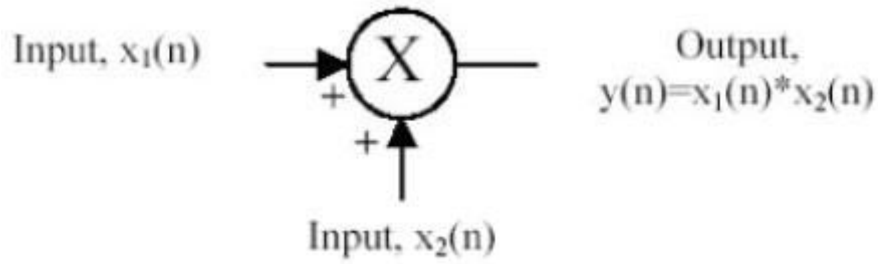
Signal Amplification/Attuation :



Multiplication/Divition :

of two signals, their product is also a signal.z (t)

= x (t) y (t)



folding:

y(n)={x(-n)} ; y=fliplr(x); n=-fliplr(n);

PROCEDURE:-

1. Open MATLAB
2. Open new M-file
3. Type the program
4. Save in current directory
5. Compile and Run the program
6. For the output see command window\ Figure window

PROGRAM:-

```
%Addition and multiplication of two signals%clc;
clear all;
close all;
t=0:0.001:2;
s1=6*sin(2*pi*5*t);
subplot(4,1,1);
plot(t,s1,'g');
xlabel('time');
ylabel('amplitude');
title('first signal');
s2=8*sin(2*pi*5*t);
subplot(4,1,2);
plot(t,s2,'r');
xlabel('time');
ylabel('amplitude');
title('second signal');
```

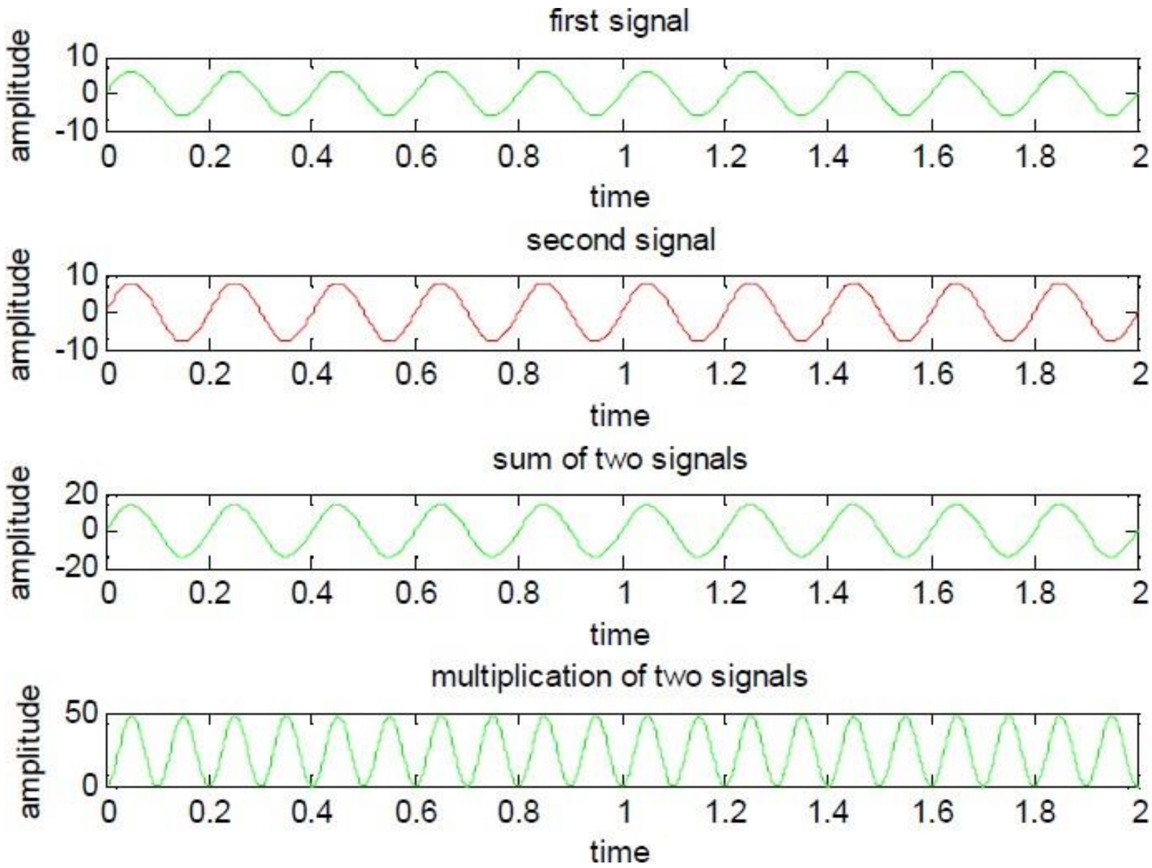
```
s3=s1+s2;
subplot(4,1,3);
plot(t,s3,'g');
xlabel('time');
ylabel('amplitude');
title('sum of two signals');
s4=s1.*s2; subplot(4,1,4);
plot(t,s4,'g');
xlabel('time');
ylabel('amplitude');
title('multiplication of two signals');
%Amplitude scaling for signals%
clc;
clear all;
close all;
t=0:0.001:2;
s1=6*sin(2*pi*5*t);
subplot(3,1,1);
plot(t,s1,'g');
xlabel('time');
ylabel('amplitude');
title('sinusoidal signal');
s2=3*s1; subplot(3,1,2);
plot(t,s2,'r');
xlabel('time');
ylabel('amplitude');
title('amplified signal');
s3=s1/3; subplot(3,1,3);
plot(t,s3,'g');
xlabel('time');
ylabel('amplitude');
title('attenuated signal');
%Time scaling for signals%
clc;
clear all;
close all;
```

```
t=0:0.001:2;
s1=6*sin(2*pi*5*t);
subplot(3,1,1);
plot(t,s1,'g');
xlabel('time');
ylabel('amplitude');
title('sinusoidal signal');
t1=3*t;
subplot(3,1,2);
plot(t1,s1,'r');
xlabel('time');
ylabel('amplitude');
title('compressed signal');
t2=t/3;
subplot(3,1,3);
plot(t2,s1,'g');
xlabel('time');
ylabel('amplitude');
title('enlarged signal');
%Time shifting of a signal%
clc;
clear all;
close all;
t=0:0.001:3;
s1=6*sin(2*pi*5*t);
subplot(3,1,1);
plot(t,s1,'g');
xlabel('time');
ylabel('amplitude');
title('sinusoidal signal');
1=t+10;
subplot(3,1,2);
plot(t1,s1,'r');
xlabel('time');
ylabel('amplitude');
title('right shift of the signal');
```

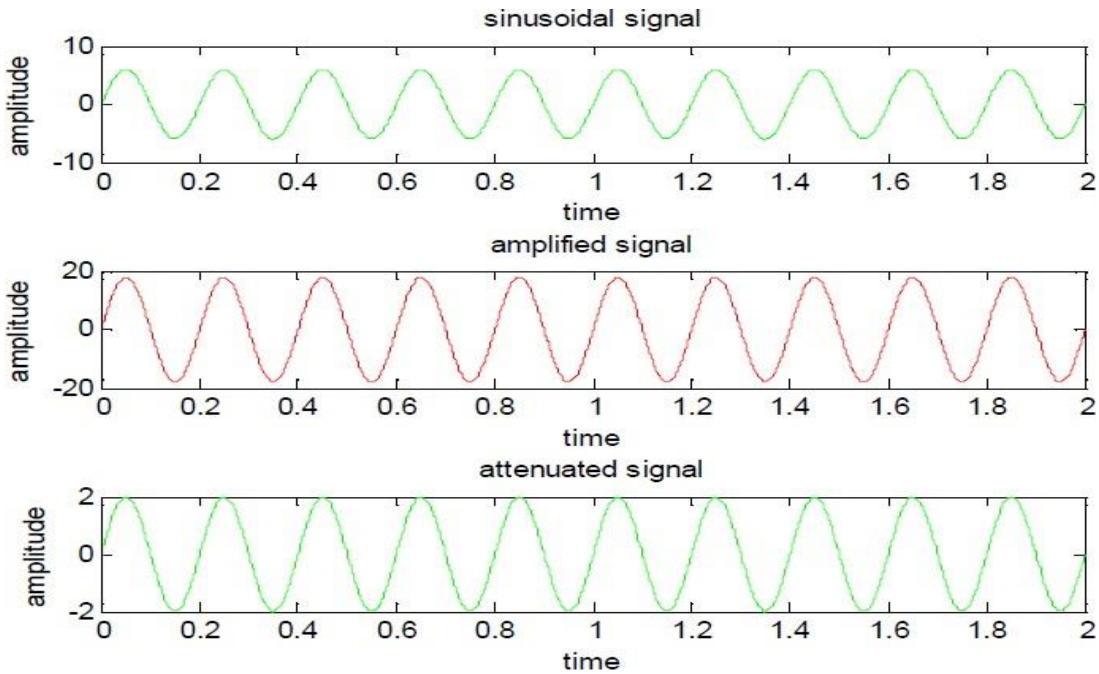
```
t2=t-10;
subplot(3,1,3);
plot(t2,s1,'g');
xlabel('time');
ylabel('amplitude');
title('left shift of the signal');
%Time folding of a signal%
clc;
clear all;
close all;
t=0:0.001:2;
s=sin(2*pi*5*t);
m=length(s); n=[-
m:m];
y=[0,zeros(1,m),s];
subplot(2,1,1);
plot(n,y,'g');
xlabel('time');
ylabel('amplitude');
title('original signal');
y1=[fliplr(s),0,zeros(1,m)];
subplot(2,1,2);
plot(n,y1,'r');
xlabel('time');
ylabel('amplitude');
title('folded signal');
```

OUTPUT:-

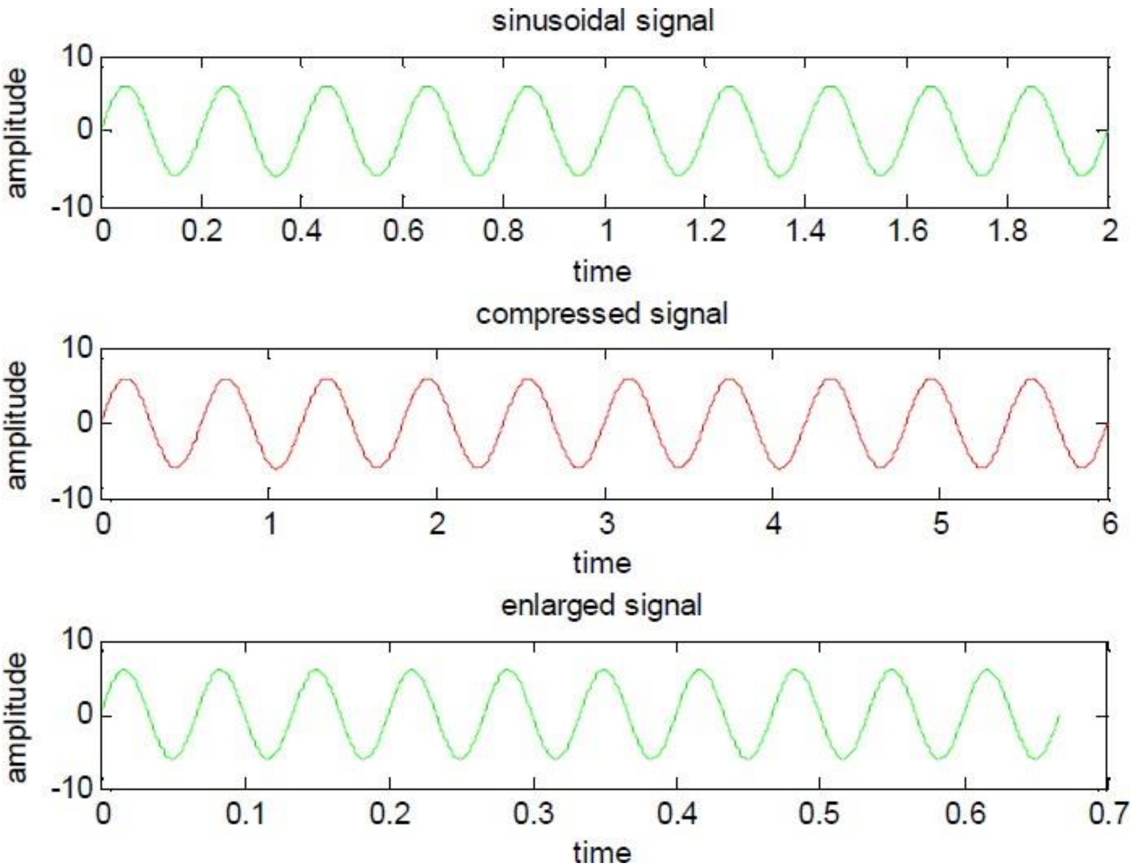
Addition and multiplication of two signals



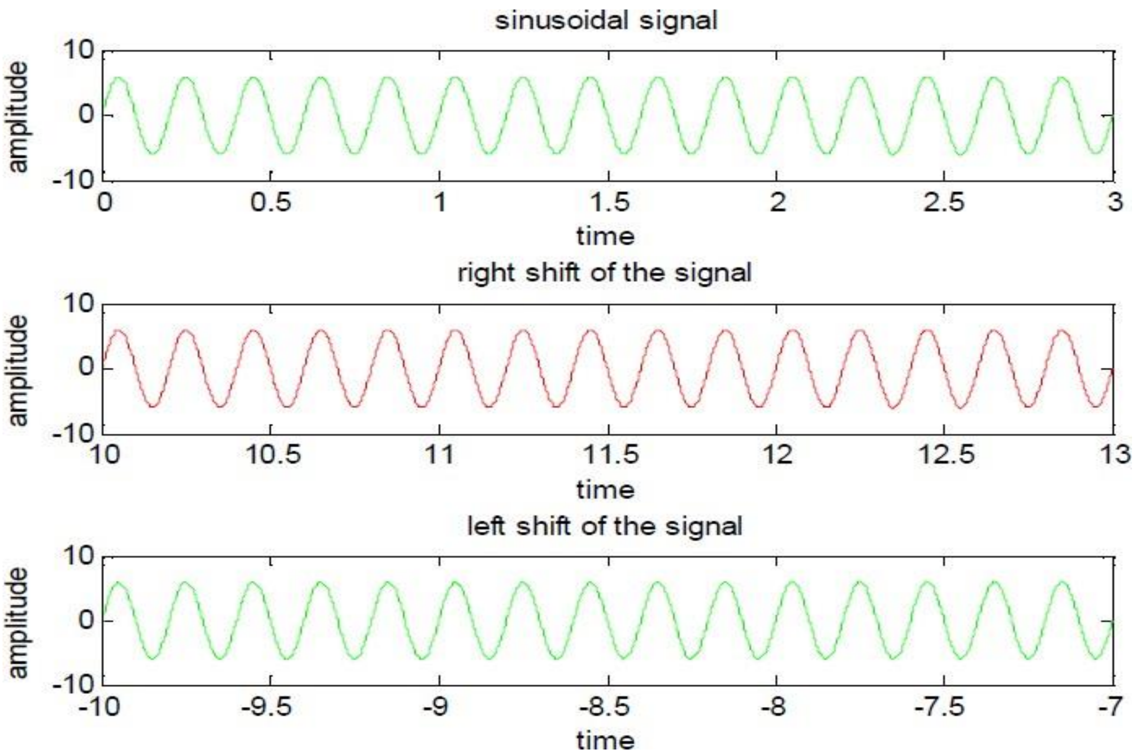
Amplitude scaling for signals



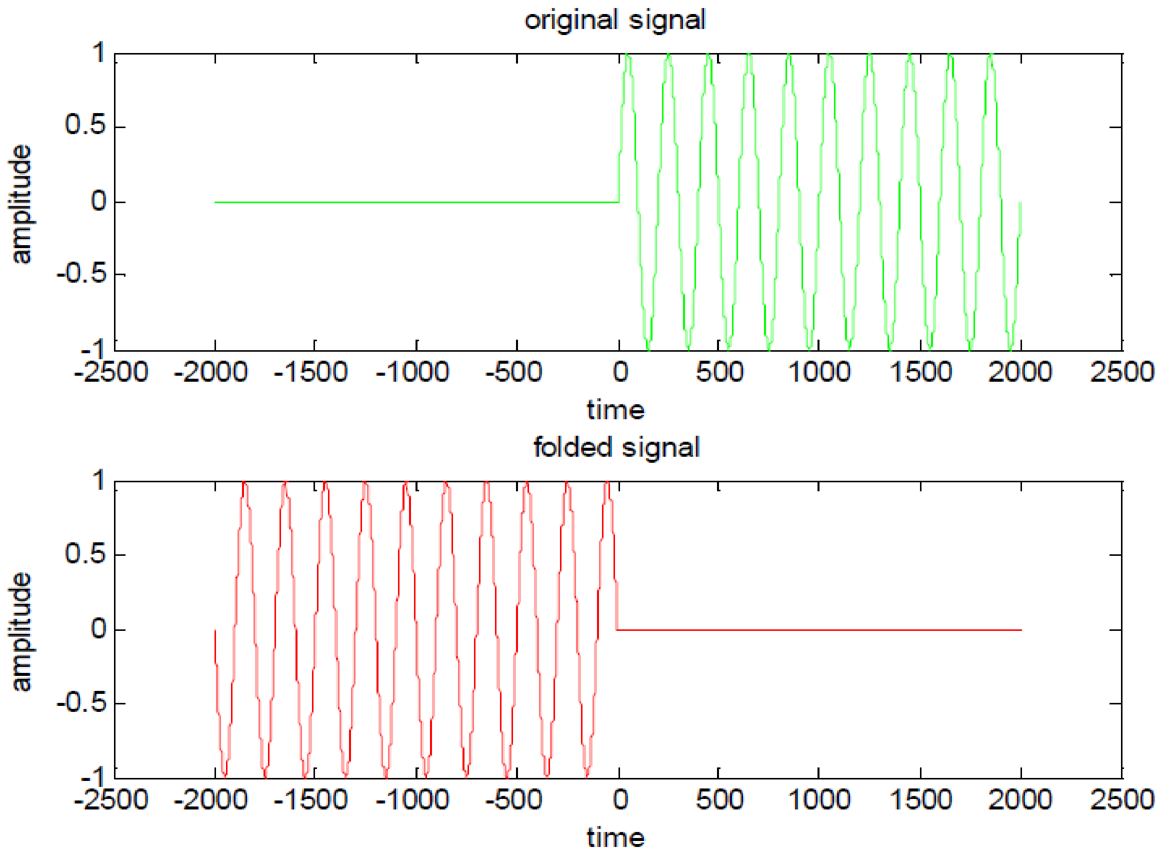
Time scaling for signals



Time shifting of a signal



Time folding of a signal



RESULT:-

In this experiment the various operations on signals have been Performed Using MATLAB have been demonstrated.

VIVA QUESTIONS:-

1. Define Symmetric and Anti-Symmetric Signals?
2. Define Continuous and Discrete Time Signals?
3. What are the Different types of representation of discrete time signals?
4. What are the Different types of Operation performed on signals?
5. What is System?

Microgrid Control functions

Control at Blocks	Function Assignments
Device-Level Control	Voltage/Frequency Control, Current/Power Control, Reactive Power Control, Generation Control, Load Control, Energy Storage Control, Islanding Detection, Fault Detection and Protection
Local Area Control	Load Management, Energy Management, Automatic Generation Control, Fast Load Shedding, Disconnection, Resynchronization, Disturbance Recording
Supervisory Control	Generation and Load Dispatch, Optimization (Voltage Profile, Economic), Spinning Reserve, Reconfiguration, Black Start, Protection Coordination, Forecasting, Data Management and Visualization
Grid-Interactive Control	Area Electric Power System Control, Electricity Markets, DMS Interaction, Distribution System Interaction, SCADA