

NETWORK ANALYSIS LABORATORY

LAB MANUAL

Academic Year : **2019 - 2020**
Course Code : **AEEB12**
Regulation : **IARE - R18**
Class : **III Semester**
Branch : **EEE**



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	Problem Solving: Exploit the knowledge of high voltage engineering in collaboration with power systems in innovative, dynamic and challenging environment, for the research based team work.
PSO2	Professional Skills: Identify 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	Modern Tools in Electrical Engineering: Comprehend the technologies like PLC, PMC, process controllers, transducers and HMI and design, install, test, maintain power systems and industrial applications.

INDEX

S. No.	List of Experiments	Page No.
1	Verification of mesh and nodal analysis using hardware.	
2	Verification of super position and reciprocity theorems using hardware.	
3	Verification of maximum power transfer theorem using hardware.	
4	Verification of Thevenin's and Norton's theorems using hardware.	
5	Verification of compensation and Milliman's theorems using hardware.	
6	To calculate and verify 'Z' parameters and 'Y' parameters of two-port network	
7	To calculate and verify 'ABCD' parameters and 'H' parameters of two-port network.	
8	Virtual instruments (vi) using Labview	
9	Signal generation of triangular wave; saw tooth, square wave and display of wave form, minimum and maximum values of wave form and modulation.	
10	Frequency measurement using Lissajous figures in Lab View.	
11	Using FOR loop, WHILE loop, charts and arrays, graph and analysis VIs.	
12	To determine the equivalent parameters of series, parallel, cascade connection of two port network.	
13	Analysis of given circuit using source transformation technique	
14	To design the resonant frequency, quality factor and band width of a series resonant circuit using hardware.	

ATTAINMENT OF PROGRAM OUTCOMES & PROGRAM SPECIFIC OUTCOMES

Exp. No.	Experiment	Program Outcomes Attained	Program Specific Outcomes Attained
1	Verification of mesh and nodal analysis using hardware.	PO1	PSO1
2	Verification of super position and reciprocity theorems using hardware.	PO1	PSO1
3	Verification of maximum power transfer theorem using hardware.	PO1	PSO1
4	Verification of Thevenin's and Norton's theorems using hardware.	PO1	PSO1
5	Verification of compensation and Milliman's theorems using hardware.	PO1	PSO1
6	To calculate and verify 'Z' parameters and 'Y' parameters of two-port network	PO1,PO3	PSO1
7	To calculate and verify 'ABCD' parameters and 'H' parameters of two-port network.	PO1,PO3	PSO1
8	Virtual instruments (vi) using Labview	PO1,PO2,PO5	PSO1 , PSO3
9	Signal generation of triangular wave; saw tooth, square wave and display of wave form, minimum and maximum values of wave form and modulation.	PO1,PO2,PO5	PSO1 , PSO3
10	Frequency measurement using Lissajous figures in Lab View.	PO1, PO3, PO5	PSO1 , PSO3
11	Using FOR loop, WHILE loop, charts and arrays, graph and analysis VIs.	PO1,PO3,PO5	PSO1 , PSO3
12	To determine the equivalent parameters of series, parallel, cascade connection of two port network.	PO1,PO3	PSO1
13	Analysis of given circuit using source transformation technique.	PO1,PO2	PSO1
14	To design the resonant frequency, quality factor and band width of a series resonant circuit using hardware	PO1,PO3,PO5	PSO1 , PSO3

NETWORK ANALYSIS LABORATORY

OBJECTIVE:

The objective of Network Analysis lab is to expose the students to analyze complex electric circuits and give them experimental skill. The purpose of lab experiment is to continue to build circuit construction skills using different circuit element. It also aims to introduce MATLAB a circuit simulation software tool. It enables the students to gain sufficient knowledge on the programming and simulation of Electrical circuits,

OUTCOMES:

Upon the completion of Electrical Circuit and simulation practical course, the student will be able to attain the following:

- 1 Familiarity with DC and AC circuit analysis techniques.
- 2 Analyze complicated circuits using different network theorems.
- 3 Acquire skills of using MATLAB software for electrical circuit studies.
- 4 Analysis of RLC resonant circuits.

EXPERIMENT - 1

(A) MESH ANALYSIS

1.1.1 AIM

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

1.1.2 APPARATUS:

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

1.1.3 THEORY:

Multi-source DC circuits may be analyzed using a mesh current technique. The process involves identifying a minimum number of small loops such that every component exists in at least one loop. KVL is then applied to each loop. The loop currents are referred to as mesh currents as each current interlocks or meshes with the surrounding loop currents. As a result there will be a set of simultaneous equations created, an unknown mesh current for each loop. Once the mesh currents are determined, various branch currents and component voltages may be derived.

1.1.4 CIRCUIT DIAGRAM:

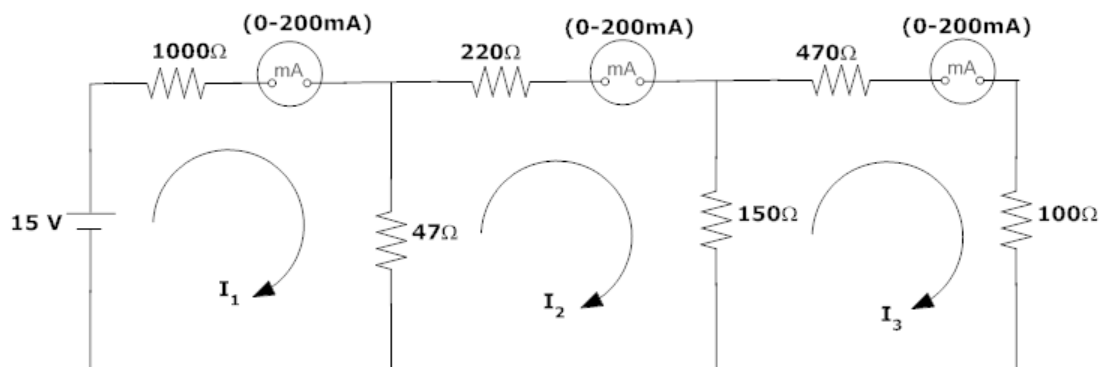


Figure – 1.1.1 Meshanalyses

1.1.5 PROCEDURE

1. Connect the circuit diagram as shown in Figure 1.1.1.
2. Switch ON the supply to RPS.
3. Apply the voltage (say 15V).

4. Gradually increase the supply voltage in steps.
5. Connect ammeters in the loop and find the currents I_1, I_2 and I_3 .
6. Verify with the theoretical results obtained with practical results

1.1.6 OBSERVATIONS:

Applied Voltage V (volts)	Loopcurrent(I_1)		Loopcurrent (I_2)		Loopcurrent(I_3)	
	Theoretical	Practical	Theoretical	Practical	Theoretical	Practical

1.1.7 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.

1.1.8 RESULT:

1.1.9 PRE LABVIVA QUESTIONS:

1. Mesh analysis is based on _____?
2. Explain mesh analysis?
3. Mention the application of super mesh analysis?
4. What is the equation for determining the number of independent loop equations in mesh current method?

1.1.10 POST LAB VIVA QUESTIONS:

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

(B) NODAL ANALYSIS

1.2.1 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.

1.2.2 APPARATUS:

S No.	Equipment	Range	Type	Quantity
1.	Resistors	-	-	
2.	Voltmeter			
3.	Regulated Power Supply Units			
4.	Bread Board	-	-	
5.	Connecting Wires			required

1.2.3 THEORY:

In electric circuit 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.

1.2.4 CIRCUIT DIAGRAM:

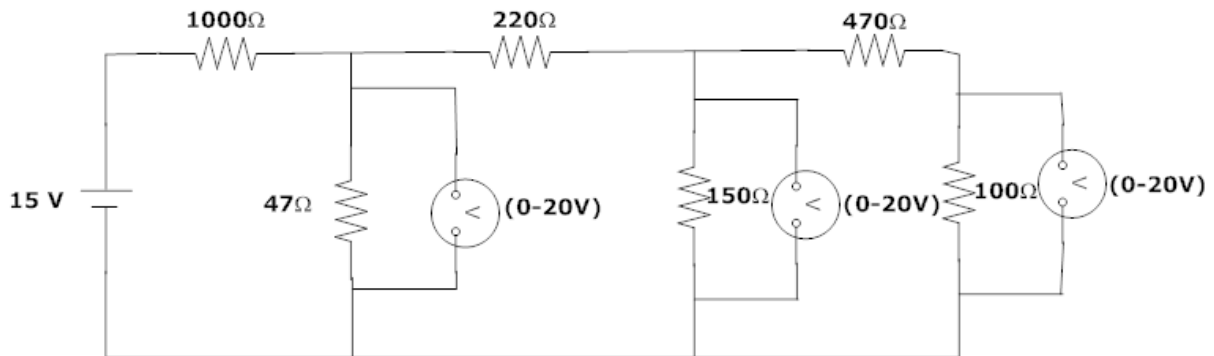


Figure 1.2.1 Nodal Analysis

1.2.5 PROCEDURE

1. Connect the circuit diagram as shown in Figure 1.2.1.
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 with the theoretical results obtained with practical results.

1.2.6 OBSERVATIONS:

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

1.2.7 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.

1.2.8 RESULT

1.2.9 PRE LABVIVA QUESTIONS:

1. Name the laws on which nodal analysis based?
2. Explain nodal analysis?
3. Give the necessary conditions for applying the super node analysis?

1.2.10 POST LAB VIVA QUESTIONS:

1. Define node.
2. Is nodal analysis is applicable to both DC and AC supply?
3. How to calculate branch currents from nodal voltages?
4. How to calculate branch voltages from nodal voltages?

EXPERIMENT – 2

(A) VERIFICATION OF SUPERPOSITION THEOREM

2.1.1 AIM:

To verify principle of Superposition theorem for an electrical circuit theoretically and practically.

STATEMENT:

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

2.1.2 APPARATUS:

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

2.1.3 CIRCUIT DIAGRAM:

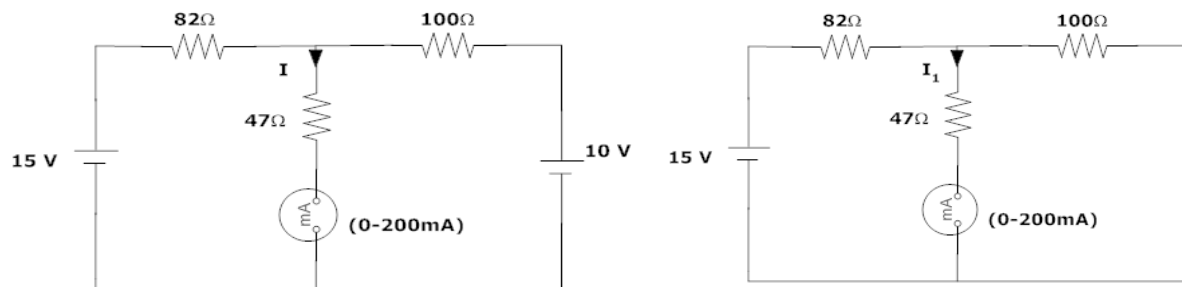


Fig- 2.1.1 Both Voltage Sources are acting (V_1 & V_2) Fig -2.1.2 Voltage Source V_1 is acting alone

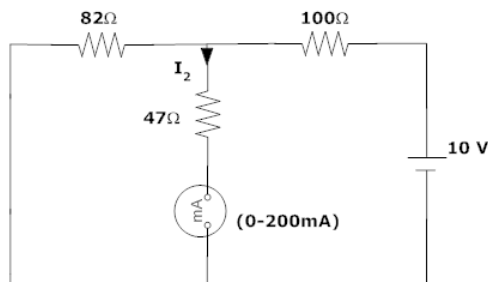


Fig – 2.1.3 Voltage Source V_2 is acting alone

2.1.4 PROCEDURE:

1. Connect the circuit as shown in figure (8.1) and note down the current flowing through R_3 and let it be I .
2. Connect the circuit as shown in figure (8.2) and note down the ammeter Reading, and let it be I_1 .
3. Connect the circuit as shown in figure (8.3) 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.

2.1.5 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)			

2.1.6 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

2.1.7 RESULT

2.1.8 PRE LAB VIVA QUESTIONS:

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

2.1.9 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?

B) VERIFICATION OF RECIPROcity THEOREM

2.2.1 AIM:

To verify the condition of Reciprocity for an electric network theoretically and practically.

2.2.2 STATEMENT

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

2.2.3 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			

CIRCUITDIAGRAM:

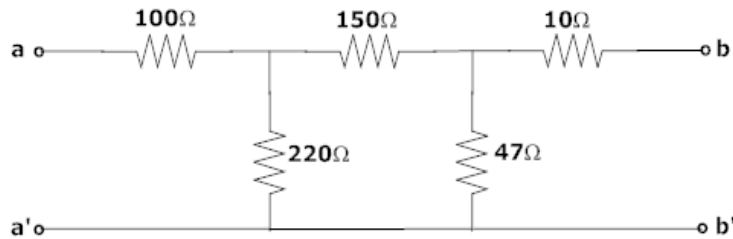


Fig - 2.2.1 Basic Circuit

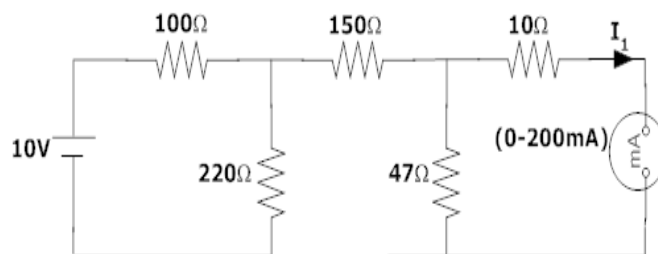


Fig - 2.2.2 Response due to 10v before interchanging load

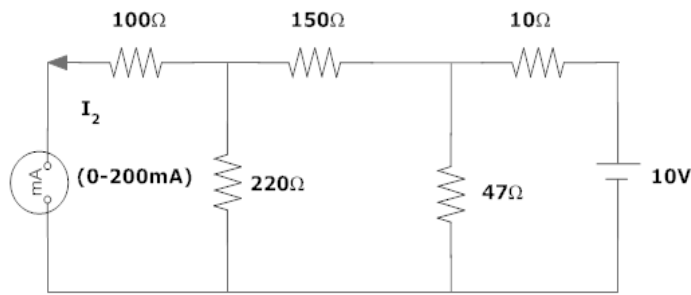


Fig – 2.2.3 Response due to 10v after interchanging load

2.2.4 PROCEDURE:

1. Connect the circuit as shown in fig 2.2.2.
2. Measure the current I_1 in the branch.
3. Inter - change voltage source and response as shown in fig 9.3 and note down the current I_2 .
4. Observe that the currents I_1 and I_2 should be same.
5. Measure the ratio of excitation and response and check whether they are equal in both cases are not.

2.2.5 TABULAR COLUMN:

Parameters	Theoretical Values	Practical Values

2.2.6 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

2.2.7 RESULT

2.2.8 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 applicable for unilateral and bilateral networks?

2.2.9 LAB ASSIGNMENT:

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

2.2.10 POST LAB VIVA QUESTIONS:

1. Comment on the applicability of reciprocity theorem on the type of network.
2. Is reciprocity theorem applicable to nonlinear circuits?

EXPERIMENT – 3

VERIFICATION OF MAXIMUM POWER TRANSFER THEOREM

3.1 AIM:

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

3.2 STATEMENT:

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

3.3 CIRCUITDIAGRAM:

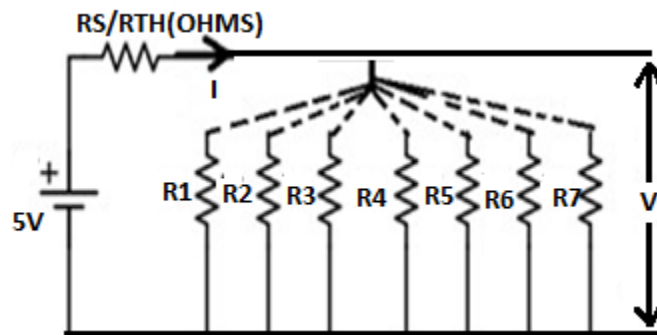


Fig – 3.1 Maximum Power Transfer Circuit

3.4 PROCEDURE:

1. Connect the circuit as shown in fig.3.1
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 delivered to the load when $R_L = R_S$ for DC.

3.5 TABULAR COLUMN:

S. No	R	V	I	P
1				
2				
3				
4				
5				

3.6 MODELGRAPH:

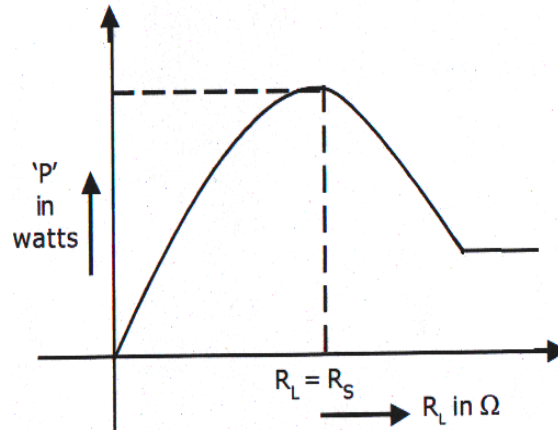


Fig – 3.2 Output Graph of Maximum Power Transfer Theorem

3.7 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.

3.8 RESULT

3.9 PRE LAB VIVAQUESTIONS:

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?

3.10 LABASSIGNMENT:

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

3.11 POST LAB VIVAQUESTIONS:

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

EXPERIMENT - 4

(A) VERIFICATION OF THEVENIN'S THEOREM

4.1.1 AIM:

To Verify Thevenin's theorem for an electrical circuit theoretically and practically.

4.1.2 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

4.1.3 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

4.1.4 CIRCUIT DIAGRAM:

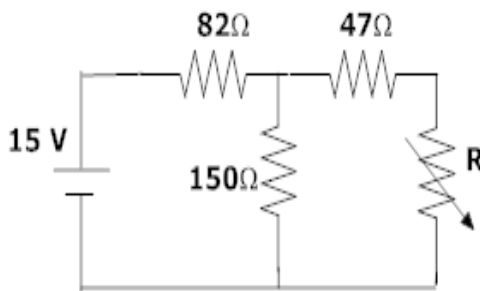


Fig-4.1.1 Basic Circuit

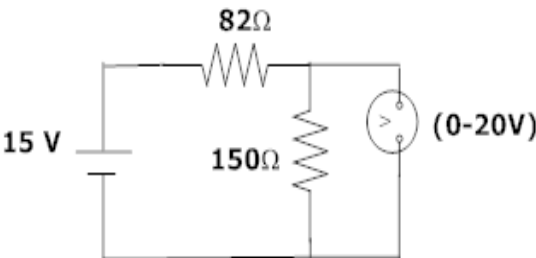


Fig-4.1.2 Measurement of V_{TH} or V_{OC}

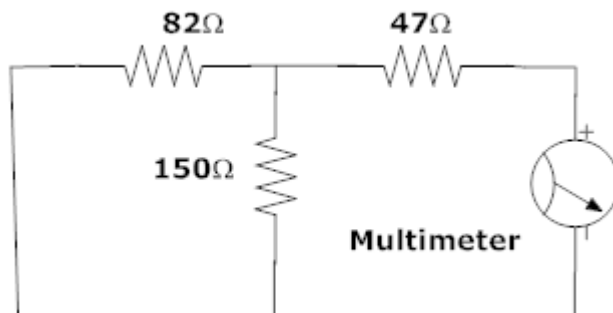


Fig - 4.1.3 Measurement of R_T

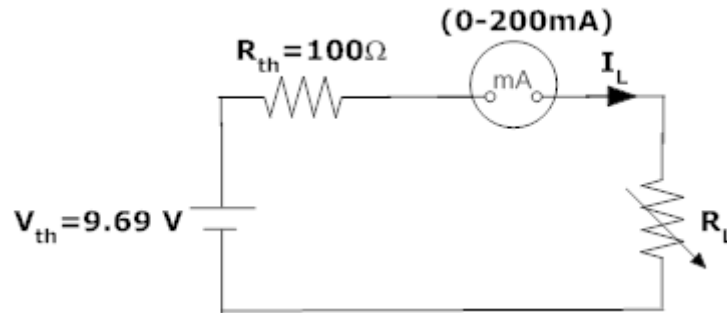


Fig – 4.1.4 Measurement of I_L ($I_L = V_{TH} / R_{TH} + R_L$)

4.1.5 PROCEDURE:

1. Connect the circuit as shown in fig 4.1.1
2. Measure current in R_L
3. Connect the circuit as shown in fig4.1.2.
4. Measure open circuit voltage V_{oc} by open circuiting terminals i.e, V_{TH}
5. Draw the Thevenin's equivalent circuit as shown in fig 4.1.3
6. Measurement current in R_L

4.1.6 TABULAR COLUMN:

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

4.1.7 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

4.1.8 RESULT:

4.1.9 PRE LAB VIVA QUESTIONS:

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

4.1.10 LAB ASSIGNMENT:

1. Solve the theoretical value of Thevenin's voltage for different circuits
2. Solve the theoretical value of Thevenin's resistance for different circuits

4.1.11 POST LAB VIVA QUESTIONS:

1. Is Thevenin's theorem applicable to both AC and DC supply?
2. State Thevenin's theorem.

(B) VERIFICATION OF NORTON'S THEOREM

4.2.1 AIM:

To Verify Norton's theorem for electrical circuit theoretically and practically

4.2.2 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.

4.2.3 CIRCUIT DIAGRAM:

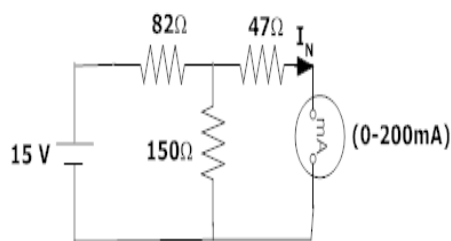


Fig -4.2.1 Basic Circuit

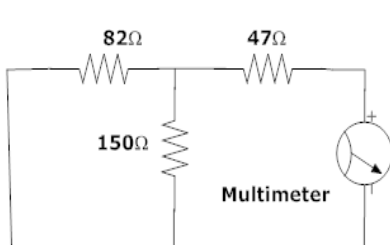


Fig -4.2.2 Norton's Equivalent Resistance Measurement

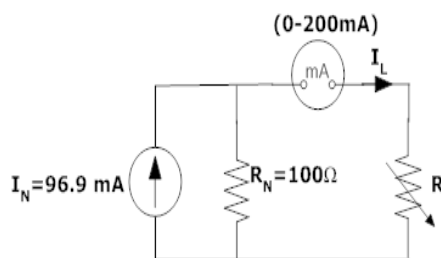


Fig - 4.2.3 Norton's load current measurement

4.2.4 PROCEDURE:

1. Connect the circuit diagram as shown in fig 12.1.
2. Measure the current I_{SC} (or) I_N through short circuited terminal.
3. Connect the circuit diagram as shown in fig 12.2.
4. Find the resistance between open circuited terminals by using multimeter.
5. Draw Norton's equivalent circuit by connecting I_N & R_N in parallel as shown in fig 12.3 and find load current.

4.2.5 TABULAR COLUMN:

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

4.2.6 RESULT:

4.2.7 PRE LAB VIVAQUESTIONS:

1. State Norton's theorem.
2. Define Norton's resistance R_N .
3. Explain the procedure for finding the Norton's current I_N .

4.2.8 LABASSIGNMENT:

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?

4.2.9 POST LAB VIVAQUESTIONS:

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 – 5

(A) VERIFICATION OF COMPENSATION THEOREM

5.1.1 AIM:

To verify the compensation theorem and to determine the change in current for an electrical circuit theoretically and practically.

5.1.2 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			

5.1.3 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).

5.1.4 CIRCUIT DIAGRAM:

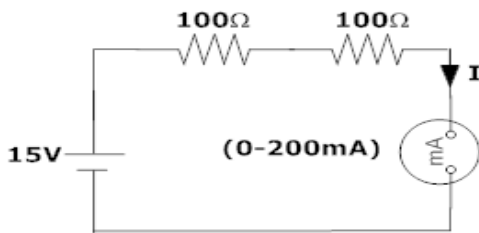


Fig – 5.1.1 Basic Circuit

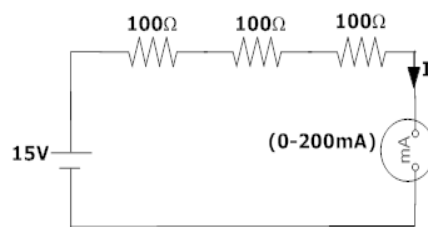


Fig – 5.1.2 addition of resistance ($\Delta R = 100\Omega$) to find current I'

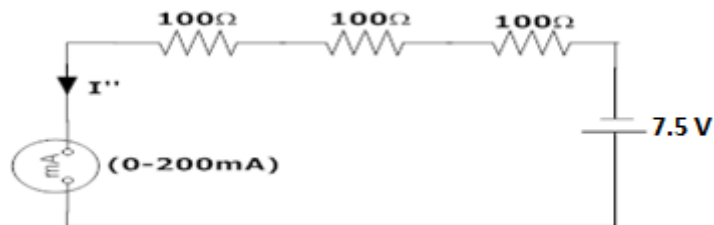


Fig -5.1.3 Compensation Theorem Circuit

5.1.5 PROCEDURE:

1. Connect the circuit as shown in fig5.1.1.
2. Measure the current I.
3. Connect the circuit as shown in fig5.1.2 by increasing the circuit resistance ($\Delta R=100\Omega$) Measure the current I'.
4. The change in current in the circuit can be found by connecting a voltage source equal to $I'\Delta R$ as shown in fig13.3.
5. Measure the current I" i.e., the change in current.
6. Observe that $I''=I-I'$.

5.1.6 TABULAR COLUMN:

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

5.1.7 RESULT:

5.1.8 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?

5.1.9 LAB ASSIGNMENT:

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

5.1.10 POST LAB VIVA QUESTIONS:

1. Which condition is required to apply the Compensation theorem for the circuit?
2. Is compensation theorem is applicable to linear and non-linear circuits?

B) VERIFICATION OF MILLIMAN'S THEOREM

5.2.1 AIM:

To verify the Milliman's Theorem for an electrical circuit theoretically and practically.

5.2.2 STATEMENT:

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

$$V' = \frac{V_1 G_1 + V_2 G_2 + V_3 G_3 + \dots + V_n G_n}{G_1 + G_2 + G_3 + \dots + G_n} \dots \dots \dots (1)$$

$$R' = \frac{1}{G_1 + G_2 + G_3 + \dots + G_n} \dots \dots \dots (2)$$

5.2.3 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			

5.2.4 CIRCUIT DIAGRAM:

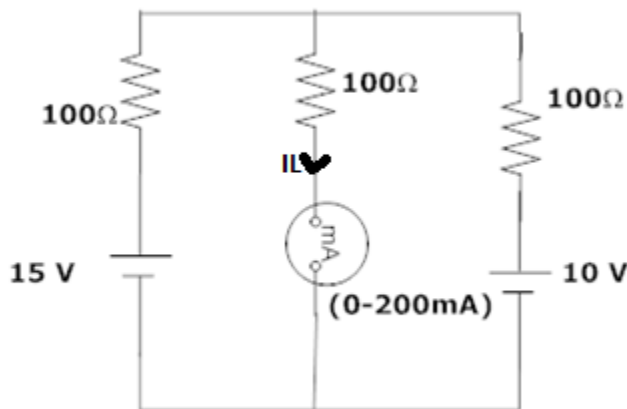
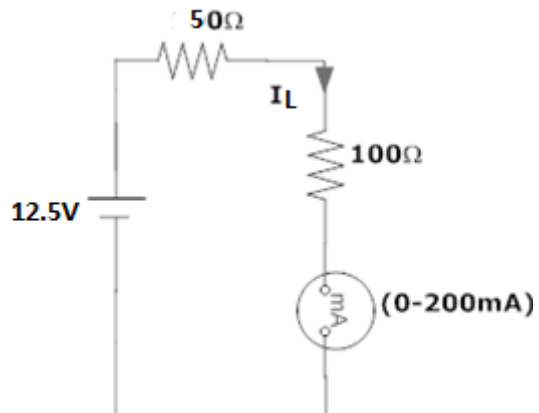


Fig – 5.2.1 Basic Circuit



Fig– 5.2.2 Millman's Circuit

5.2.5 PROCEDURE:

1. Connect the circuit as shown in fig.5.2.1
2. Measure the current through the resistor 100Ω .
3. Connect the circuit as shown in fig.5.2.2 and measure the current through 100Ω .
4. Observe whether the two currents are equal.

5.2.6 TABULAR COLUMN:

S.No.	Parameters	Theoretical value	Practical value
1			
2			

5.2.7 RESULT:

5.2.8 PRE LAB VIVA QUESTIONS:.

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

5.2.9 LAB ASSIGNMENT:

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

5.2.10 POST LAB VIVA QUESTIONS:

1. Which condition is required to apply the Millman's theorem for the circuit?
2. Is Millman's theorem applicable for linear and non-linear networks?

EXPERIMENT – 6

IMPEDANCE (Z) AND ADMITTANCE(Y) PARAMETERS

6.1 AIM:

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

6.2 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			

6.3 CIRCUIT DIAGRAM:

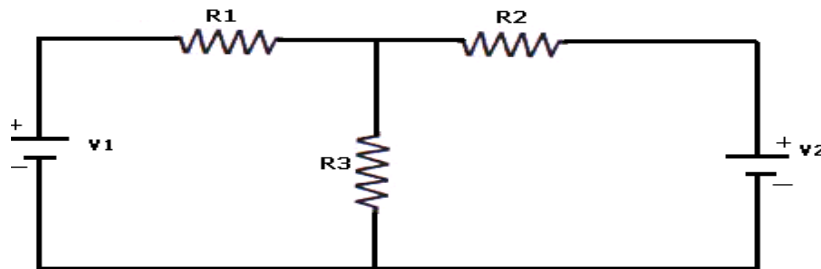


Fig – 6.1 Basic Circuit

6.4 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.

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

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.

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

6.5 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			

6.6 RESULT:

6.7 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.

6.8 LAB ASSIGNMENT:

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

6.9 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 – 7

TRANSMISSION AND HYBRID PARAMETERS

7.1 AIM:

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

7.2 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			

7.3 CIRCUIT DIAGRAM:

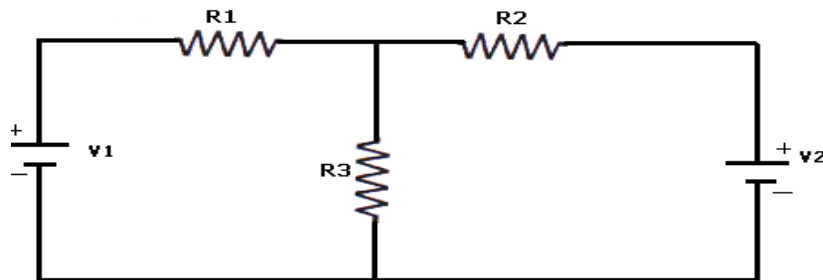


Fig – 7.1 Basic Circuit

7.4 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

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

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

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

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

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

4. Find the values of h_{11} , h_{22} , h_{12} , h_{21} and compare them with the theoretical value

7.5 TABULAR COLUMN:

When $I_2=0$

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

When $I_1=0$

S.No	$V_1(V)$	$V_2(V)$	$I_2(mA)$

When $V_2=0$

S.No	$V_1(V)$	$I_1(mA)$	$I_2(mA)$

7.6 RESULT:

7.7 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.

7.8 LAB ASSIGNMENT:

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

7.9 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.

EXPERIMENT: 8

VIRTUAL INSTRUMENTS (VI) USING LabVIEW

8.1 AIM:

Editing and Building a VI, Creating a VI

8.2 APPARATUS:

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

8.3 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.

8.4 BLOCK DIAGRAM:

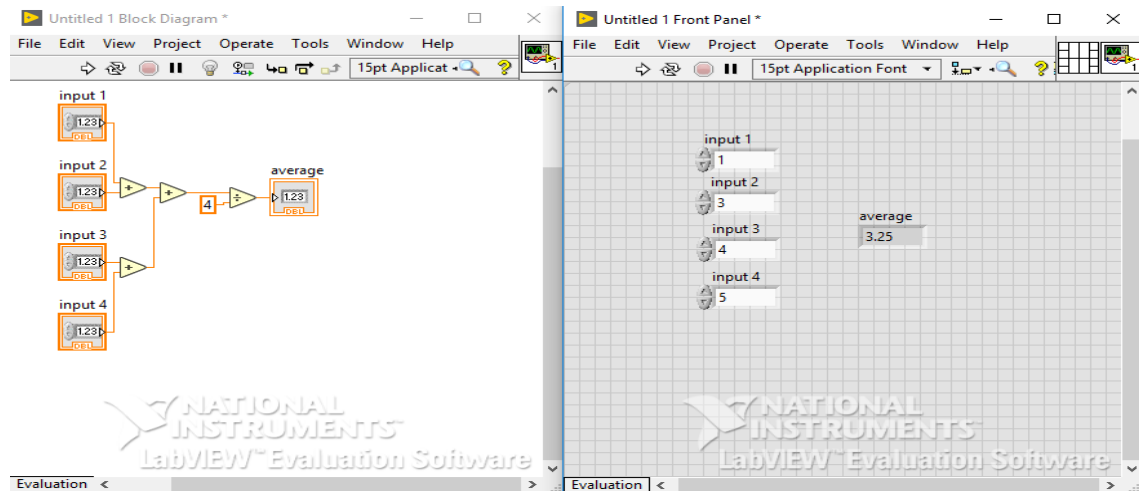


Fig 8.1 Block Diagram and Front Panel for Creating VI

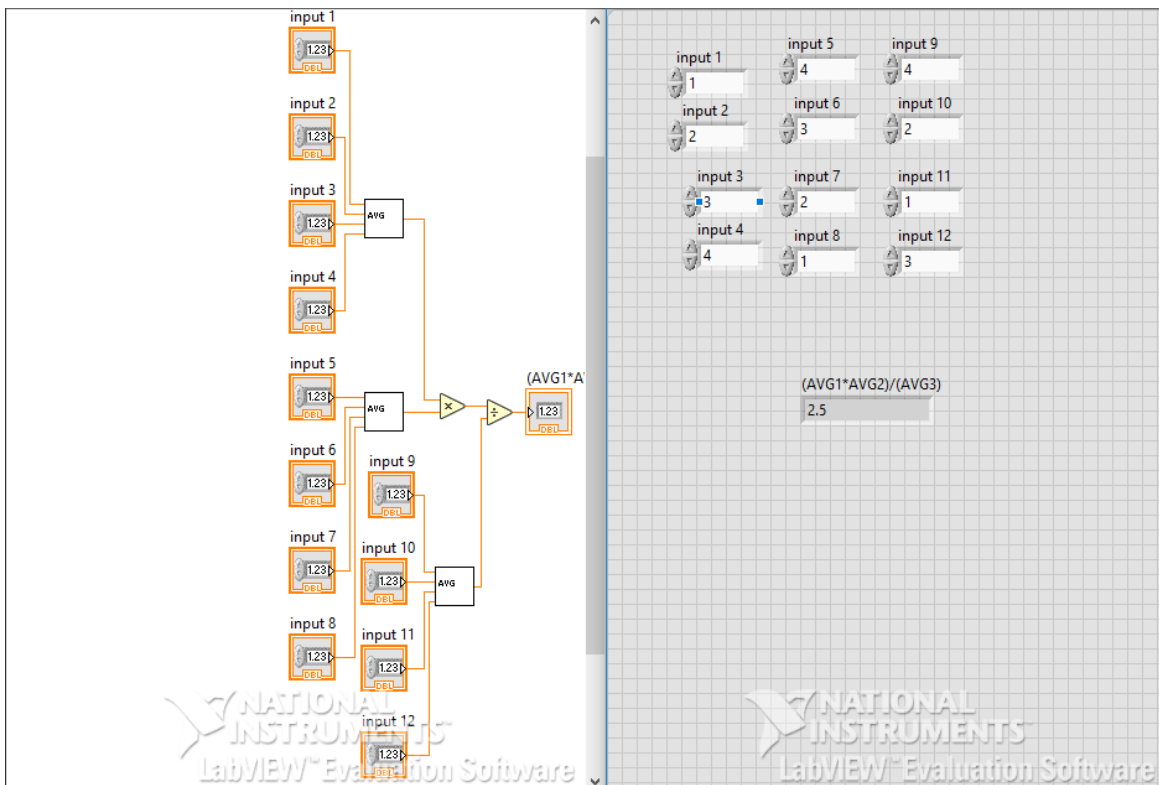


Fig - 8.2 Block Diagram for usingsub VI

8.5 PROCEDURE:

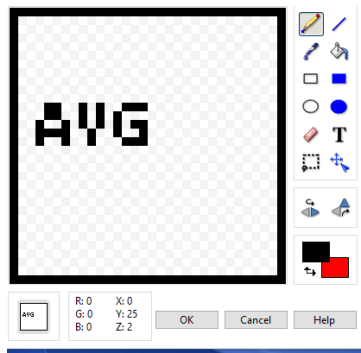
8.5.1 CREATING A VI

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 shown in figure, and give the values for input blocks.
9. Run the VI. Save the VI as 'AVG' from file menu.

8.5.2 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.

8.5.3 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.

8.7 RESULTS:

8.8 PRE-LAB QUESTIONS:

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

8.9 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- 9

GENERATION OF COMMON WAVEFORMS USING LAB VIEW

9.1 AIM:

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

9.2 APPARATUS:

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

9.3 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.

9.4 BLOCK DIAGRAM:

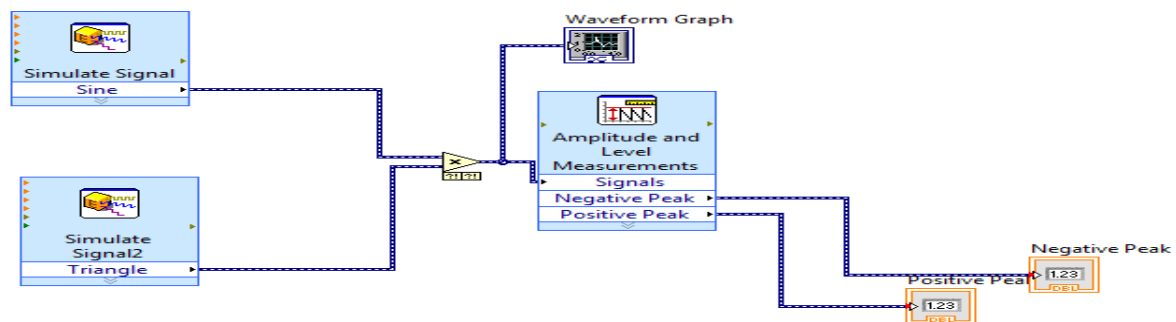


Fig - 9.1 Block Diagram for Generating Common Waveform

9.5 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.

9.6 FRONT PANEL:

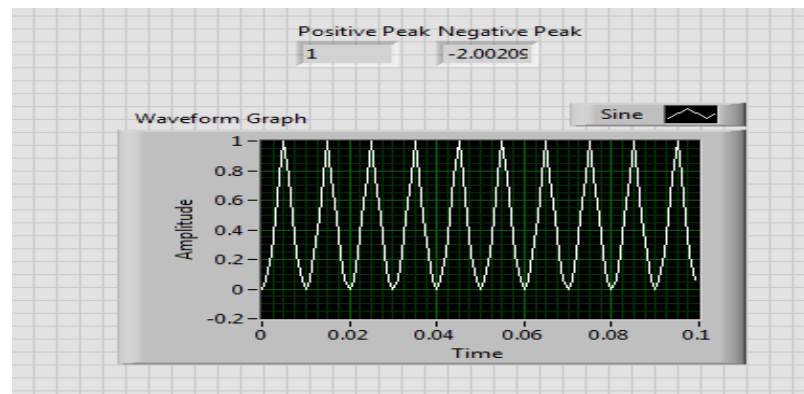


Fig -9.2 Graph for Generating Common Waveform

9.7 RESULT

9.8 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.

9.9 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.

EXPERIMENT: 10

FREQUENCY MEASUREMENT USING LabVIEW

10.1 AIM:

To measure frequency using Lissajous figures in LabVIEW.

10.2 APPARATUS:

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

10.3 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.

10.4 BLOCK DIAGRAM:

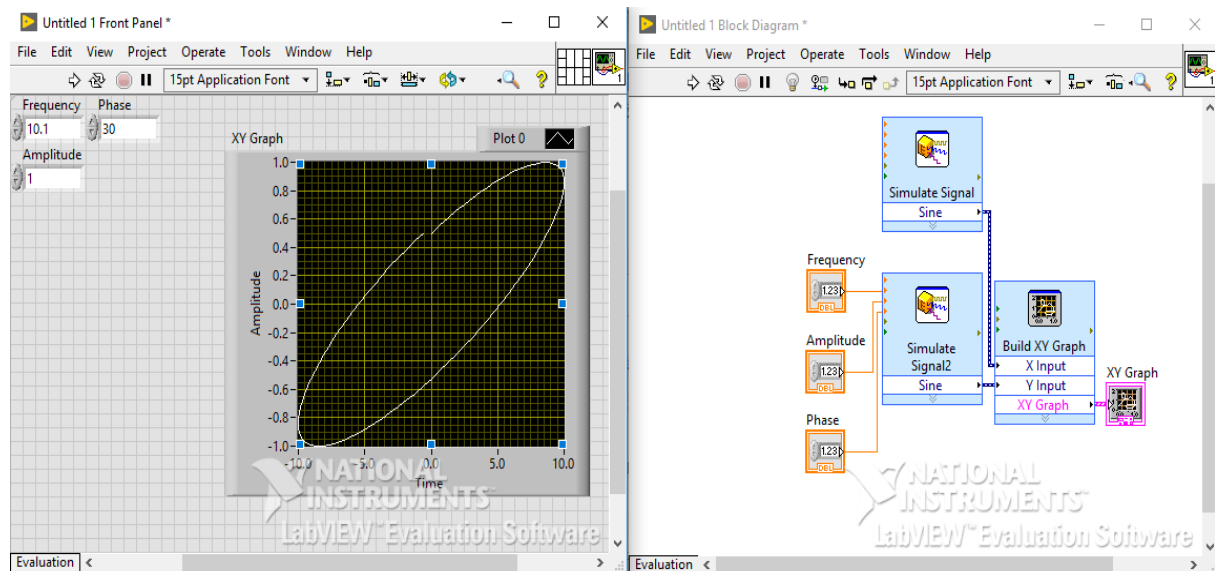


Fig 10.1 Block Diagram for Lissajous Pattern

10.5 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° out of phase with each other.
 - (ii) An ellipse appears when magnitude of two signals are equal and has phase difference of either 30° or 330° and 150° or 220° .
 - (iii) A circle is thus formed only when magnitude of two signals are equal and has phase difference of either 90° or 270° .
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

10.6 EXPECTED GRAPHS:

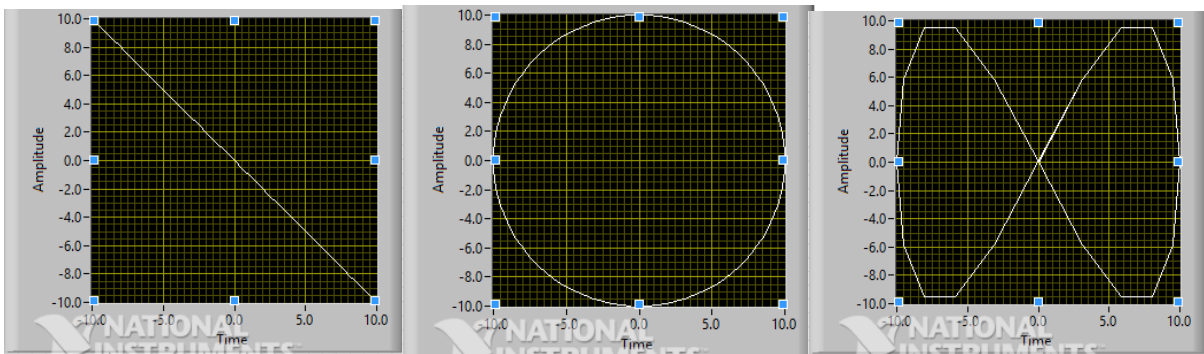


Fig - 10.2 Graphs for Lissajous Figures

10.7 RESULT:

10.8 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.

10.9 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.

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 $2^{31} - 1$ 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» Graphpalette:

- 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:

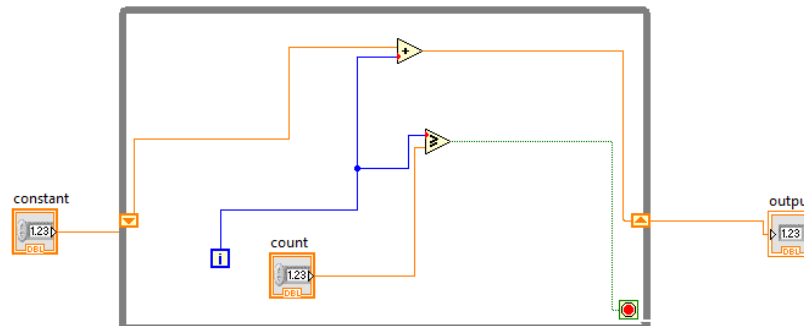
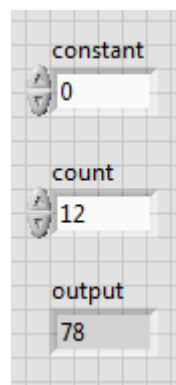


Fig 11.1 Block Diagram of While Loop

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:

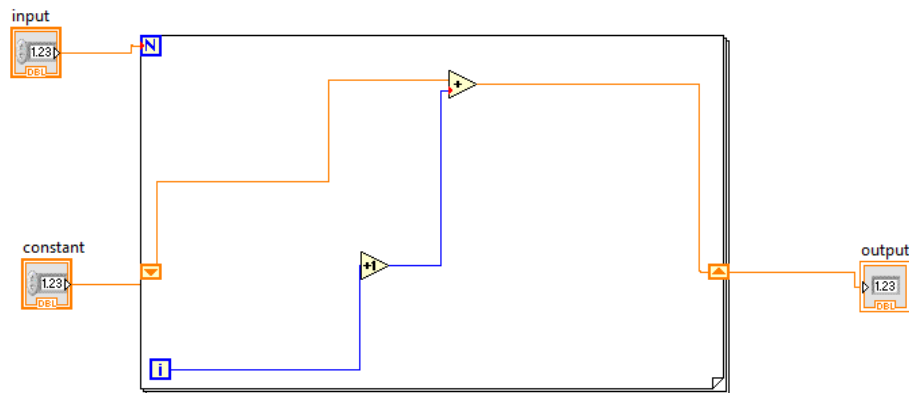
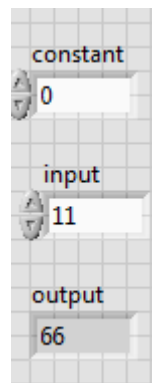


Fig 11.2 Block Diagram of For Loop

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.

- For the first build array block right click and select concatenate mode and the other automatically becomes appending mode.
- Wire the blocks as shown in block diagram.
- Give values to input arrays and run the VI.

11.4.3.1 BLOCK DIAGRAM:

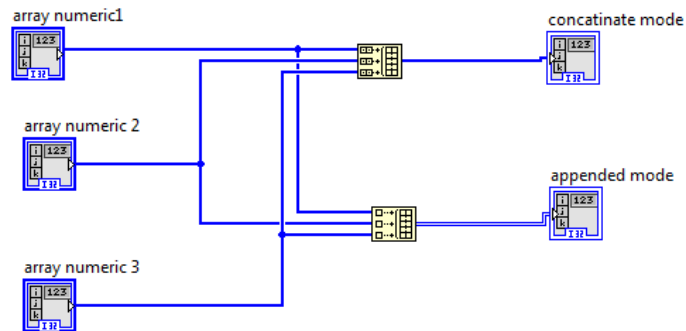
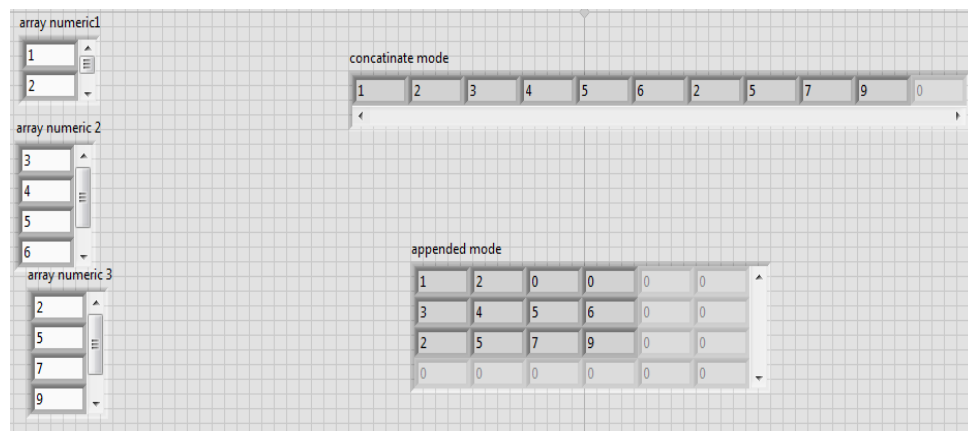


Fig - 11.3 Block Diagram of Array

11.4.3.2 FRONT PANEL:



11.4.4 USING CHARTS AND WAVEFORM:

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

11.4.4.1 BLOCK DIAGRAM:

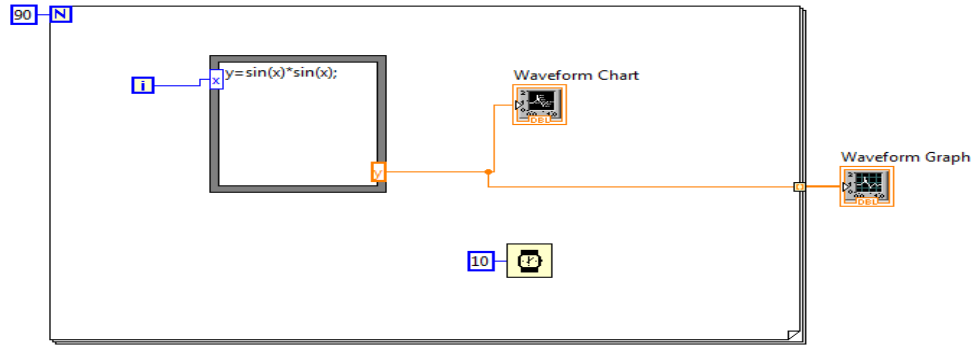


Fig - 11.4 Block Diagram of Waveform and Chart

11.4.4.2 FRONT PANEL:

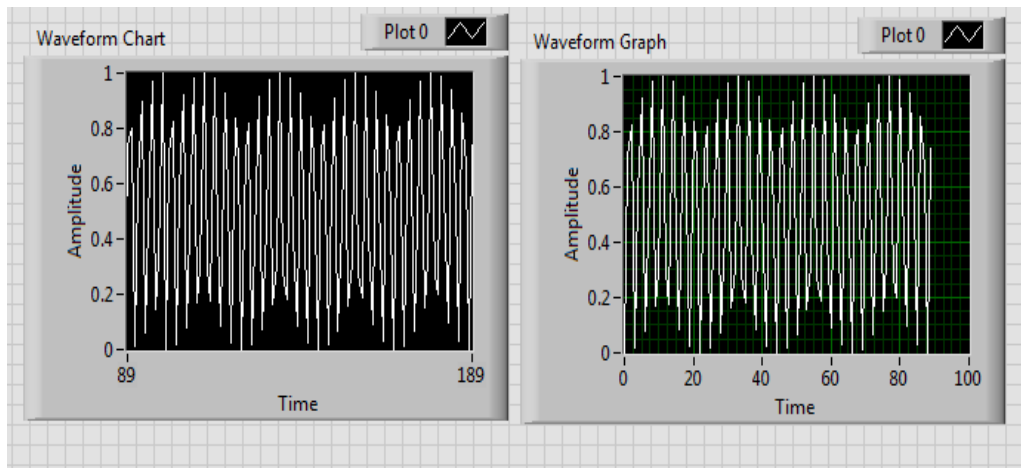


Fig -11.5 Waveform for Charts and Graphs

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:

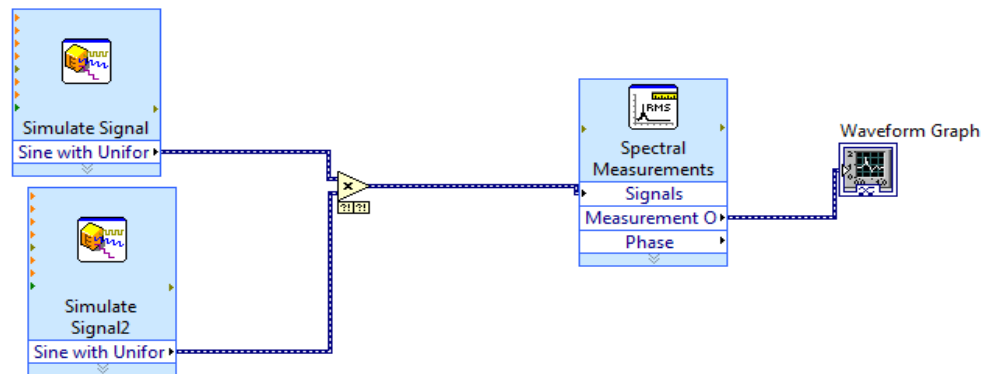


Fig - 11.6 block diagram of analysis

11.4.5.2 FRONT PANEL:

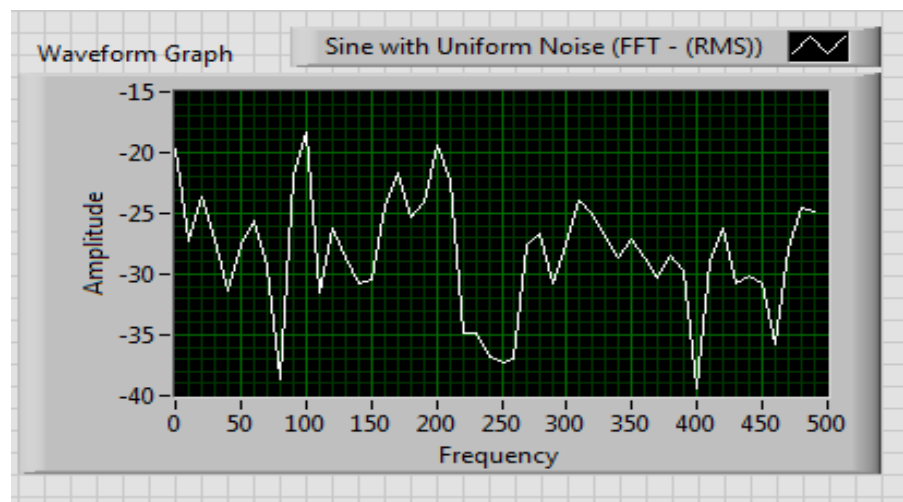


Fig - 11.7 Waveform for Analysis

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 lab VIEW?
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

TO DETERMINE THE EQUIVALENT PARAMETERS OF SERIES, PARALLEL, CASCADE CONNECTION OF TWO PORT NETWORK.

(A) SERIES CONNECTION

12.1 AIM:

To determine the equivalent parameters of series connection of two port network

12.2 APPARATUS:

S No.	Equipment	Range	Type	Quantity
1.	Resistors	-	-	
2.	Voltmeter			
3.	ammeter			
4.	Regulated Power Supply Units			
5.	Bread Board	-	-	
6.	Connecting Wires			required

12.3 INTRODUCTION:

The Series connection is also called as Series-Series connection, Since both input ports & output ports are series connected. Thus,

$$V_1 = Z'_{11}V_2 + Z'_{12}I_2 \quad (1)$$

$$V_2 = Z'_{21}I_1 + Z'_{22}I_2 \quad (2)$$

V_1 & V_2 are dependent variables, I_1 & I_2 are independent variables, Z'_{11} & Z'_{22} is Input & Output driving point impedance. Z'_{12} & Z'_{21} is Reverse & Forward transfer impedance.

12.4 CIRCUIT DIAGRAM:

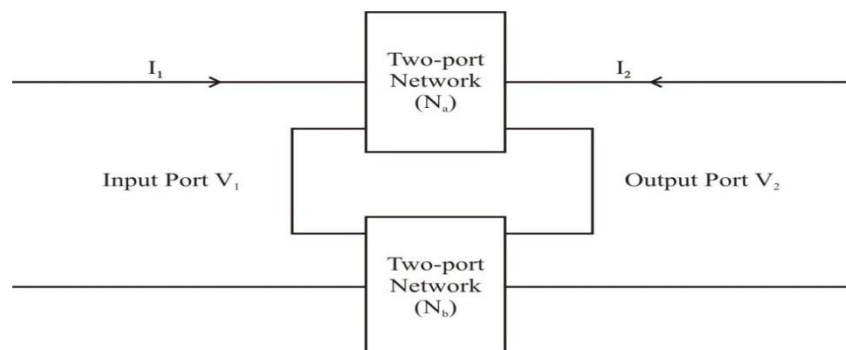


Fig 12.1 circuit diagram for series connection of networks

12.5 PROCEDURE:

1. Connect the circuit as shown in fig and switch 'ON' the experiment board.
2. Open the output port & excite input port with a known voltage source V_s so that $V_1 = V_s$ & $I_2 = 0$. We determine I_1 & I_2 to obtain Z'_{11} & Z'_{21} .
3. Input port is open circuited & Output port is excited with the same voltage source V_s so that $V_2 = V_s$ & $I_1 = 0$. We determine I_2 & V_1 to obtain Z'_{22} & Z'_{12} .
4. Switch OFF the supply after taking the readings

12.6 OBSERVATIONS:

S.N.O	When O/P is open cktd			When I/P is open cktd		
	V_1	I_1	V_2	V_2	V_1	I_2

12.7 PRECAUTIONS:

1. Make the connections according to the circuit diagram. Power supply should be switched off.
2. Connections should be tight.
3. Note the readings carefully

12.8 RESULT:

The Z-parameters of series connection of two-port network has been determined.

12.9 PRE LAB VIVA QUESTIONS:

1. What will be the total impedance if the two networks are connected in series?
2. What will be the total impedance if the two networks are connected in parallel?
3. Which parameter is used for the series representation of connection of two port network?

12.10 POST LAB VIVA QUESTIONS:

1. What is Input driving point impedance
2. What is Output driving point impedance
3. What is Reverse Transfer impedance

(B) PARALLEL CONNECTION

12.2 AIM:

To determine equivalent parameters of parallel connection of two-port network

12.2.1 APPARATUS:

S No.	Equipment	Range	Type	Quantity
1.	Resistors	-	-	
2.	Voltmeter			
3.	ammeter			
4.	Regulated Power Supply Units			
5.	Bread Board	-	-	
6.	Connecting Wires			required

12.2.2 INTRODUCTION:

Consider two port N/Ws connected in parallel so that they have common reference node, then the equation of the N/Ws A&B in terms of Y parameters are given by

$$\begin{aligned}Y_{11} &= Y_{11A} + Y_{11B} \\Y_{12} &= Y_{12A} + Y_{12B} \\Y_{21} &= Y_{21A} + Y_{21B} \\Y_{22} &= Y_{22A} + Y_{22B}\end{aligned}$$

Thus we see that each Y parameter of the parallel N/W is given as the sum of the corresponding parameters of the individual N/Ws.

12.2.3 CIRCUIT DIAGRAM:

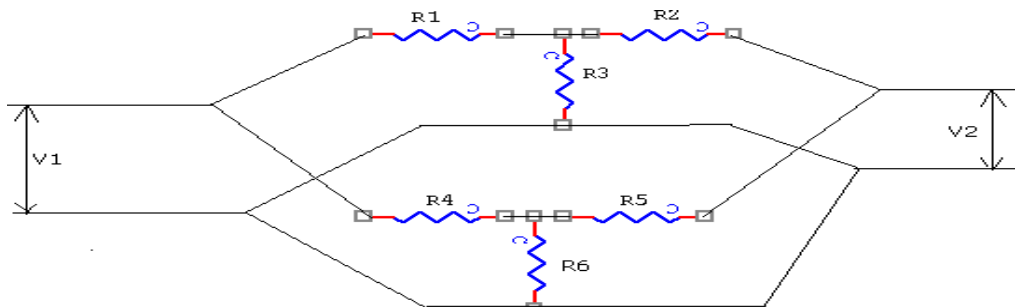


Fig 12.2 circuit diagram for parallel connection of networks

12.2.5 PROCEDURE:

1. Connect the N/Ws A&B separately on the Bread board according to the fig.
2. Take the Reading according to the observation table & calculate Y parameters for both N/Ws & add them.
3. Connect two N/Ws A&B in parallel & take the readings.
4. Calculate the Y parameters of parallel connected N/Ws.
5. Verify that the sum of parameters of A&B N/Ws is equal to the parameters of parallel connected N/Ws.

12.2.6 OBSERVATIONS:

S. No.	When I/P is short ckt			When O/P is short ckt		
	V_2	I_1	I_2	V_1	I_1	I_2

12.2.7 PRECAUTIONS:

1. Make the connections according to the circuit diagram. Power supply should be switched off.
2. Connections should be tight.
3. Note the readings carefully.

12.2.8 RESULT:

12.2.9 PRE LAB VIVA QUESTIONS:

1. What will be the total admittance if the two networks are connected in series?
2. What will be the total admittance if the two networks are connected in parallel?
3. Which parameter is used for the representation of parallel connection of two port network?

12.2.10 POST LAB VIVA QUESTIONS:

1. What is Input driving point Admittance
2. What is Output driving point Admittance
3. What is Reverse Transfer Admittance

(C) CASCADE CONNECTION

12.3 AIM:

To determine the A'B'C'D' parameters of the cascade connection of two-port network

12.3.1 APPARATUS:

S No.	Equipment	Range	Type	Quantity
1.	Resistors	-	-	
2.	Voltmeter			
3.	ammeter			
4.	Regulated Power Supply Units			
5.	Bread Board	-	-	
6.	Connecting Wires			required

12.3.3 INTRODUCTION:

Two port networks are said to be connected in cascade if the output port of the first becomes the input port of the second as shown in fig.

$$V_1 = A'V_2 + B'(-I_2) \quad (1)$$

$$I_1 = C'V_2 + D'(-I_2) \quad (2)$$

V_1 and I_1 are dependent variables; V_2 and I_2 are independent variables A' , D' is Reverse Voltage Ratio & Reverse Current Ratio.
 B' , C' is Reverse Transfer Impedance & Reverse Transfer Admittance.

12.3.3 CIRCUIT DIAGRAM:

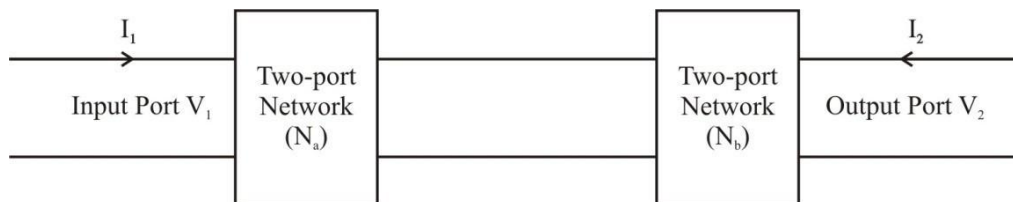


Fig 12.3 circuit diagram for cascade connection of networks

12.3.4 PROCEDURE:

1. Connect the circuit as shown in fig and switch 'ON' the experiment board.
2. In A, B, C, D parameters, open the output port and excite input port with a known voltage source V_s so that $V_1 = V_s$ and $V_2 = 0$. We determine I_1 and V_2 to obtain A' & C'.
3. The output port is short circuited and input port is excited with the same voltage source V_s so that $V_1 = V_s$ & $V_2 = 0$. We determine I_1 & I_2 to obtain B' & D'.
4. Switch OFF the supply after taking the readings.

12.3.5 OBSERVATIONS:

S.N.O	When O/P is open ckt			When O/P is short ckt		
	V_2	I_1	I_2	V_1	I_1	I_2

12.3.6 PRECAUTIONS:

1. Make the connections according to the circuit diagram. Power supply should be switched off.
2. Connections should be tight.
3. Note the readings carefully.

12. 3.7 RESULT:

12.3.8 PRE LAB VIVA QUESTIONS:

1. What do you mean by cascade connection?
2. What is Reverse Voltage Ratio?
3. Which parameter is used for the cascade representation of connection of two port network?

12.3.9 POST LAB VIVA QUESTIONS:

1. What is Reverse Voltage Ratio?
2. What is Reverse Current Ratio?
3. What is Reverse Transfer admittance?

EXPERIMENT - 13

ANALYSIS OF GIVEN CIRCUIT USING SOURCE TRANSFORMATION TECHNIQUE.

13.1 AIM:

Analysis of given circuit using source transformation like Conversion of Voltage to Current source and Conversion of Current to Voltage source

13.2 APPARATUS:

S No.	Equipment	Range	Type	Quantity
1.	Resistors	-	-	
2.	Voltmeter			
3.	ammeter			
4.	Regulated Power Supply Units			
5.	Bread Board	-	-	
6.	Rheostat			
7.	Connecting Wires			required

13.3 THEORY:

An ideal voltage source is energy sources which can supply energy at constant voltage i.e. when the load current is change from minimum to maximum the terminal voltage remain same. To satisfy this requirement the internal resistance of the source should be zero. A non ideal voltage source has non zero internal resistance as a result the terminal voltage decreases with load current.

An ideal current source is an energy source which can supply energy at constant current. To satisfy this requirement the internal resistance of the source should be zero. A non ideal current source has a finite internal resistance as a result the load current changes with change in load current

13.4 CIRCUIT DIAGRAM:

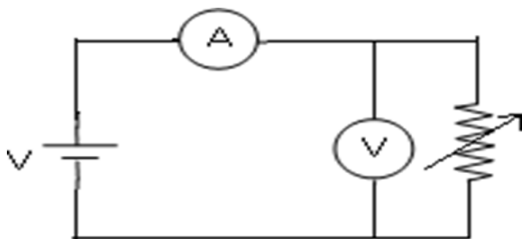


Fig .1

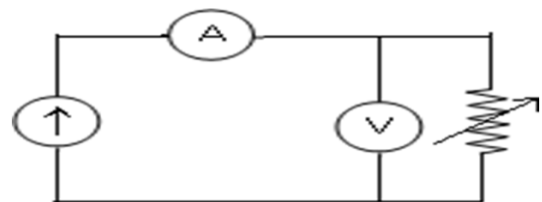


Fig .2

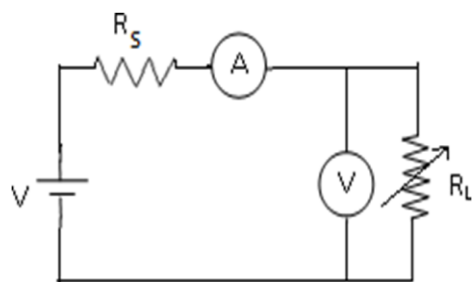


Fig . 3

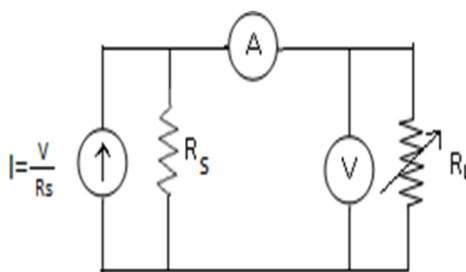


Fig .4

13.5 PROCEDURE:

For Ideal Voltage Source:

1. Connect the circuit as fig1
2. Vary the value of R_L (min - max) in five equal steps.
3. Tabulate the observation in table 1
4. Draw the graph between V_s & I_s

For Ideal Current Source:

1. Connect the circuit as fig 2
2. Vary the value of R_L (min - max) in five equal steps.
3. Tabulate the observation in table 2.
4. Draw the graph between I_s & V_s .

For non- Ideal Voltage Source:

1. Connect the circuit as fig3.
2. Vary the value of R_L (min - max) in five equal steps.
3. Tabulate the observation in table 3.
4. Draw the graph between V_s & I_s .

For non- Ideal Current Source:

1. Connect the circuit as fig4.
2. Vary the value of R_L (min - max) in five equal steps.
3. Tabulate the observation in table 1.
4. Draw the graph between I_s & V_s .

13.6 OBSERVATIONS:

TABLE 1

S. NO	VOLTMETER READING, V (V)	AMMETER READING, I (A)

TABLE 2

S. NO	VOLTMETER READING, V (V)	AMMETER READING, I (A)

TABLE 3

S.NO	VOLTMETER READING V (V)	AMMETER READING I (A)	SOURCE RESISTANCE R_s (Ω)	SOURCE VOLTAGE V_s (V)

TABLE 4

S. NO	VOLTMETER READING V (V)	AMMETER READING I (A)	SOURCE RESISTANCE R_s (Ω)	SOURCE CURRENT I_s (A)

13.7 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.

13.8 RESULT:**13.9 PRE LAB VIVA QUESTIONS:**

1. Why it is not possible to have an ideal source in real world?
2. Why voltage remains constant in ideal voltage source?
3. Why current remains constant in ideal current source?

13.10 POST LAB VIVA QUESTIONS:

1. Why it is not possible to have an ideal source in real world?
2. Why voltage remains not constant in practical voltage source?
3. Why current remains not constant in practical current source.

EXPERIMENT: 14

VERIFICATION OF SERIES RESONANCE USING HARDWARE

14.1 AIM:

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

14.2 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			

14.3 CIRCUIT DIAGRAM:

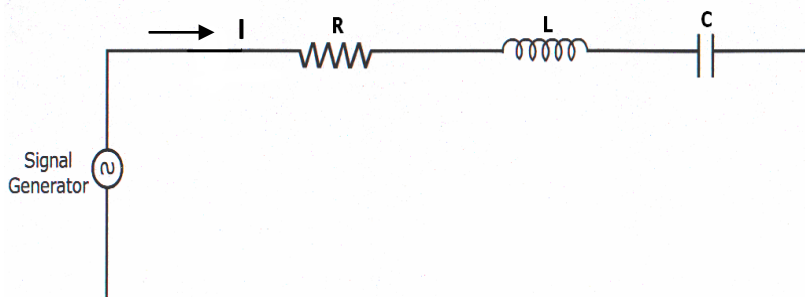


Fig – 13.1 Series Resonance

14.4 PROCEDURE:

1. Connect the circuit as shown in fig.12.1.
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 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 for series circuit.

14.5 THEORETICAL CALCULATIONS:

Series Resonance

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

cut off frequency $(f_1) = f_r - R/4\pi L$

Upper cut off frequency $(f_2) = f_r + R/4\pi L$

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

Band Width $f_2 - f_1 = R/2\pi L$

14.6 TABULAR COLUMN:

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

14.7 MODEL GRAPH:

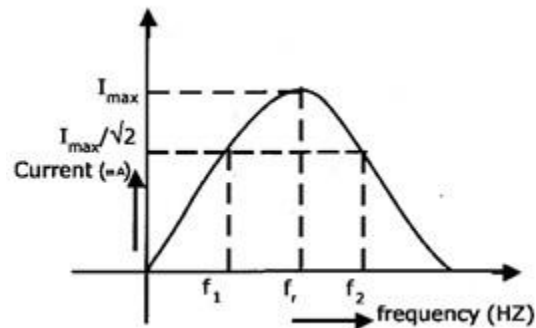


Fig – 13.2 Series Resonance

14.8 RESULT: