SRM VALLIAMMAI ENGINEERING COLLEGE

SRM NAGAR, KATTANKULATHUR – 603203

DEPARTMENT OF ELECTRICAL AND ELECTRONICS ENGINEERING

1905308-ELECTRIC CIRCUITS LABORATORY MANUAL

2nd YEAR – EEE

(REGULATION 2019)

Academic Year: 2021-2022 Odd Semester

Prepared by,

Dr.R.Arivalahan, Associate Professor/EEE

1905308 ELECTRIC CIRCUITS LABORATORY L T P C 0 0 4 2

LIST OF EXPERIMENTS

1. Simulation and experimental verification of electrical circuit problems using Kirchhoff's voltage and current laws.
2. Simulation and experimental verification of electrical circuit problems using Thevenin's theorem.
3. Simulation and experimental verification of electrical circuit problems using Norton's theorem.
4. Simulation and experimental verification of electrical circuit problems using Superposition theorem.
5. Simulation and experimental verification of Maximum Power transfer Theorem.
6. Study of Analog and digital oscilloscopes and measurement of sinusoidal voltage, frequency and power factor.
7. Simulation and Experimental validation of R-C electric circuit transients.
8. Simulation and Experimental validation of frequency response of RLC electric circuit.
9. Design and Simulation of series resonance circuit.
10. Design and Simulation of parallel resonant circuits.
11. Simulation of three phase balanced and unbalanced star, delta networks circuits.
12. Simulation and experimental verification of Millman's theorem.

TOTAL: 60 PERIODS

LABORATORY REQUIREMENTS FOR BATCH OF 30 STUDENTS

1. Regulated Power Supply: 0 – 15 V D.C - 10 Nos / Distributed Power Source.
2. Function Generator (1 MHz) - 10 Nos.
3. Single Phase Energy Meter - 1 No.
4. Oscilloscope (20 MHz)-10 Nos.
5. Digital Storage Oscilloscope (20 MHz) – 1 No.
6. 10 Nos. of PC with Circuit Simulation Software (min 10 Users) (e-Sim Scilab/ Pspice / MATLAB /other Equivalent software Package) and Printer (1 No.)
7. AC/DC - Voltmeters (10 Nos.), Ammeters (10 Nos.) and Multi-meters (10 Nos.)
8. Single Phase Wattmeter – 3 Nos.
9. Decade Resistance Box, Decade Inductance Box, Decade Capacitance Box - 6 Nos each.
10. Circuit Connection Boards - 10 Nos.

Necessary Resistors, Inductors, Capacitors of various quantities (Quarter Watt to 10 Watt).

1905308 ELECTRIC CIRCUITS LABORATORY

Cycle – 1

1. Simulation and experimental verification of electrical circuit problems using Kirchhoff's voltage and current laws.
2. Simulation and experimental verification of electrical circuit problems using Thevenin's theorem.
3. Simulation and experimental verification of electrical circuit problems using Norton's theorem.
4. Simulation and experimental verification of electrical circuit problems using Superposition theorem.
5. Simulation and experimental verification of Maximum Power transfer Theorem.
6. Study of Analog and digital oscilloscopes and measurement of sinusoidal voltage, frequency and power factor.

Cycle – 2

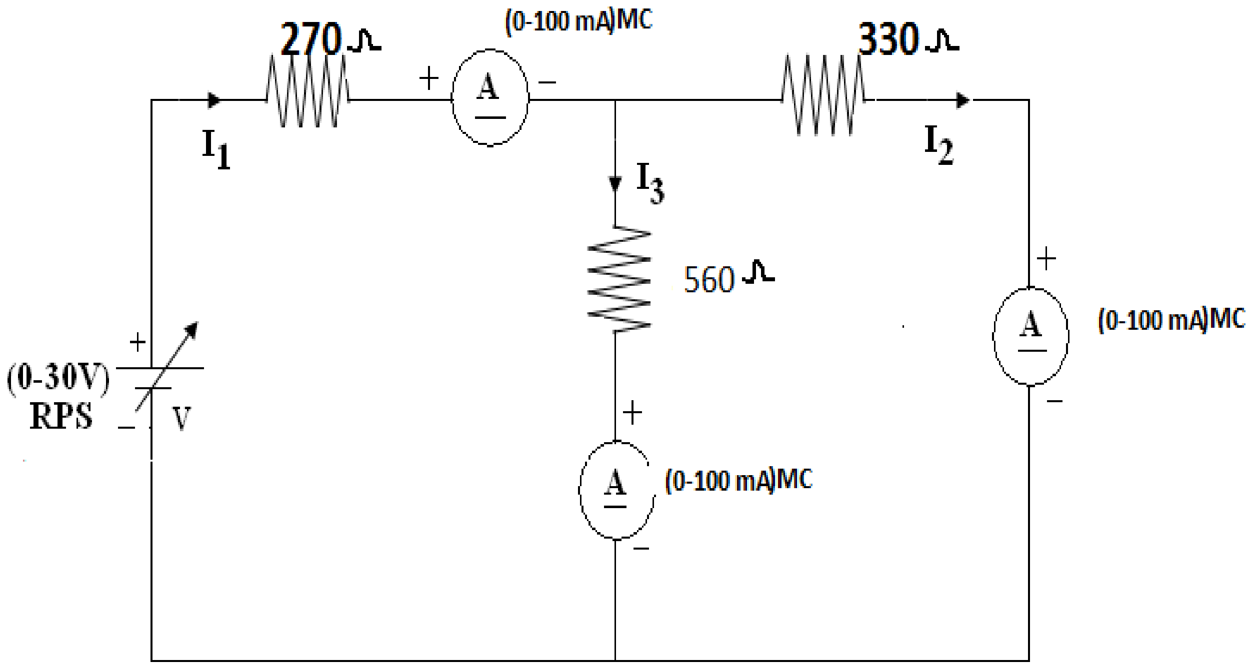
1. Simulation and Experimental validation of R-C electric circuit transients.
2. Simulation and Experimental validation of frequency response of RLC electric circuit.
3. Design and Simulation of series resonance circuit.
4. Design and Simulation of parallel resonant circuits.
5. Simulation of three phase balanced and unbalanced star, delta networks circuits.
6. Simulation and experimental verification of Millman's theorem.

ADDITIONAL EXPERIMENTS:

1. Experimental determination of power in three phase circuits by two-watt meter method
2. Determination of two port network parameters

S. No.	DATE	TITLE OF THE EXPERIMENT	MARKS	SIGN

CIRCUIT DIAGRAM FOR KIRCHHOFF'S CURRENT LAW



OBSERVATION TABLE

S.No	V (Volts)	I_1 (mA)	I_2 (mA)	I_3 (mA)	$I_1 = I_2 + I_3$ (mA)

EXP.NO:

DATE:

SIMULATION AND EXPERIMENTAL VERIFICATION OF ELECTRICAL CIRCUIT PROBLEMS USING KIRCHHOFF'S VOLTAGE AND CURRENT LAWS

AIM:

To verify (i) Kirchhoff's current law (ii) Kirchhoff's voltage law

APPARATUS REQUIRED:

S.No	Name of the apparatus	Range	Type	Quantity
1	RPS			
2	Resistor			
3	Ammeter			
4	Voltmeter			
5	Bread board			
6	Connecting wires			

SOFTWARE REQUIRED:

Matlab 7.1

KIRCHHOFF'S CURRENT LAW:

THEORY:

The law states, "The sum of the currents entering a node is equal to sum of the currents leaving the same node". Alternatively, the algebraic sum of currents at a node is equal to zero.

THEORETICAL CALCULATION

S.No.	V (Volts)	I_1 (mA)	I_2 (mA)	I_3 (mA)	$I_1 = I_2 + I_3$ (mA)

MODEL CALCULATION:

The term node means a common point where the different elements are connected. Assume negative sign for leaving current and positive sign for entering current.

PROCEDURE:

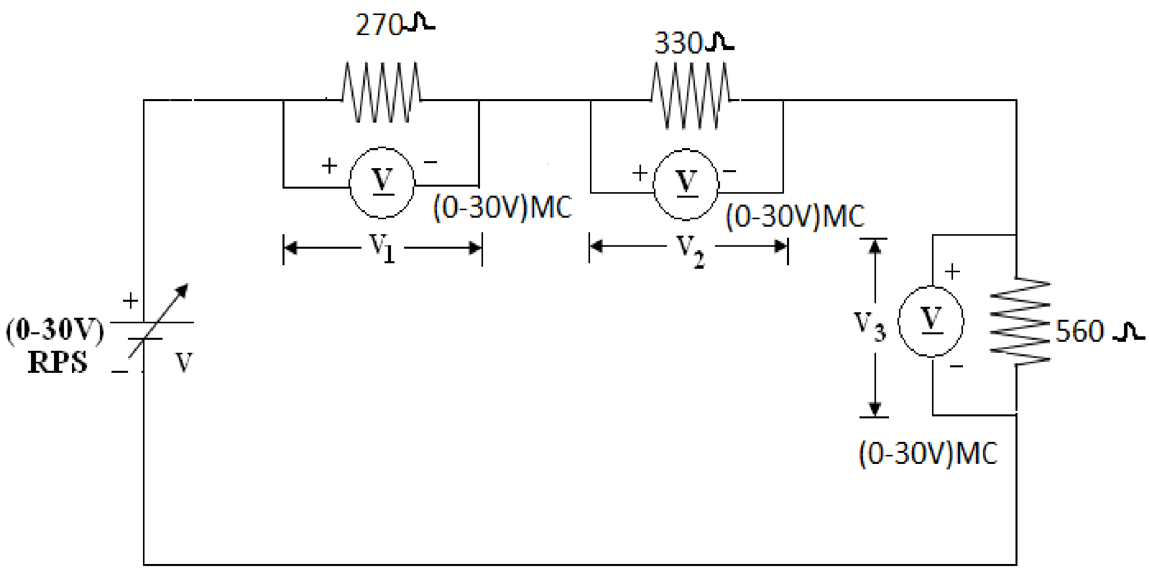
1. Connect the circuit as per the circuit diagram.
2. Switch on the supply.
3. Set different values of voltages in the RPS.
4. Measure the corresponding values of branch currents I_1, I_2 and I_3.
5. Enter the readings in the tabular column.
6. Find the theoretical values and compare with the practical values

FORMULA:

$$\sum \text{Currents entering a node} = \sum \text{Currents leaving the node}$$

$$I_1 = I_2 + I_3$$

CIRCUIT DIAGRAM FOR KIRCHHOFF'S VOLTAGE LAW:



OBSERVATION TABLE:

S.No.	V Volts	V_1 Volts	V_2 Volts	V_3 Volts	$V = V_1 + V_2 + V_3$ Volts

KIRCHHOFF'S VOLTAGE LAW:

THEORY:

The law states, "The algebraic sum of the voltages in a closed circuit/mesh is zero".

The voltage rise is taken as positive and the voltage drop is taken as negative.

PROCEDURE:

1. Connect the circuit as per the circuit diagram.
2. Switch on the supply.
3. Set different values of voltages in the RPS.
4. Measure the corresponding values of voltages (V_1, V_2 and V_3) across resistors R_1, R_2 and R_3 respectively.
5. Enter the readings in the tabular column.
6. Find the theoretical values and compare with the practical values.

FORMULA:

$$\sum \text{Voltages in a closed loop} = 0$$

$$V - V_1 - V_2 - V_3 = 0$$

THEORETICAL CALCULATION:

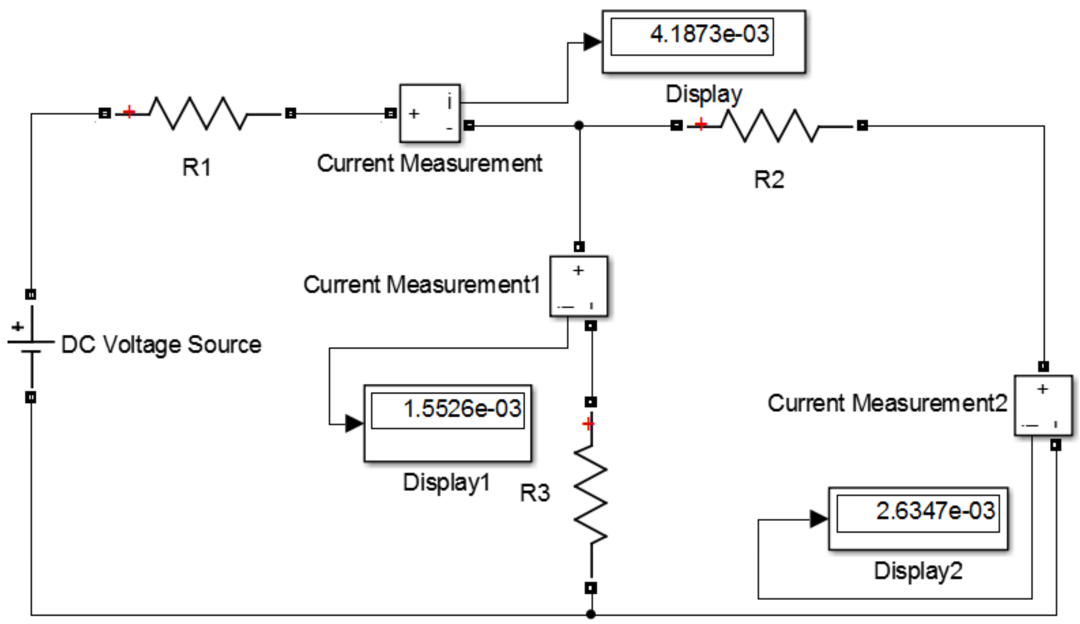
S.No.	V Volts	V_1 Volts	V_2 Volts	V_2 Volts	$V = V_1 + V_2 + V_3$ Volts

MODEL CALCULATION:

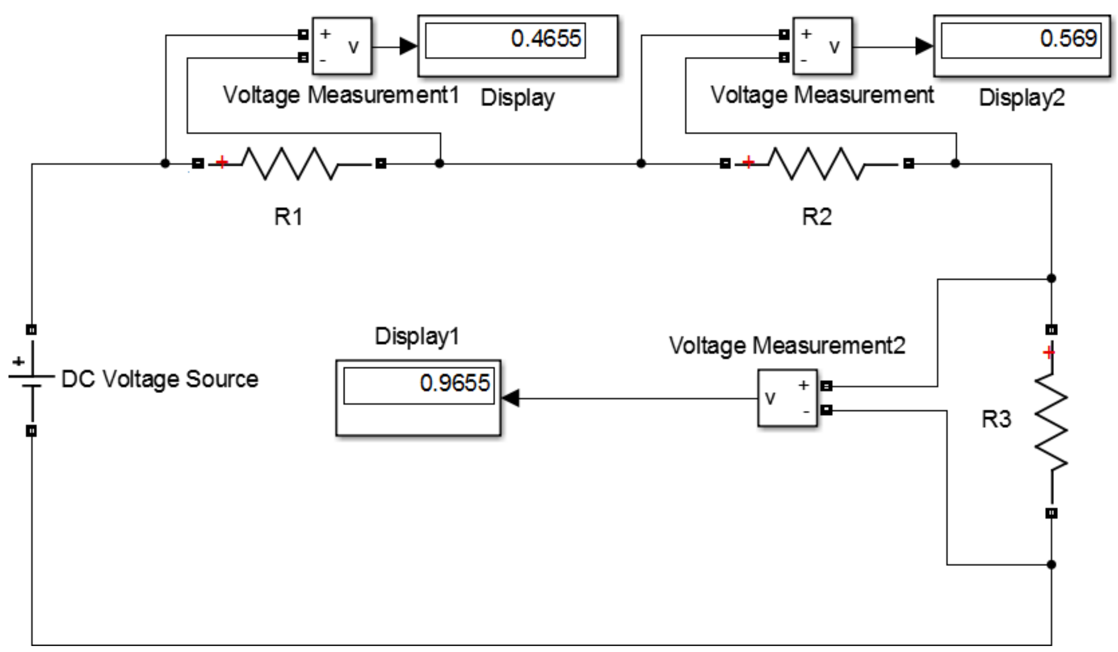
SIMULATION PROCEDURE:

1. Open a new MATLAB/SIMULINK model
2. Connect the circuit as shown in the figure
3. Debug and run the circuit
4. For different input voltages, record the current and voltages and verify with theoretical values.

SIMULATION CIRCUIT DIAGRAM FOR KIRCHHOFF'S CURRENT LAW:



SIMULATION DIAGRAM FOR KIRCHHOFF'S VOLTAGE LAW:



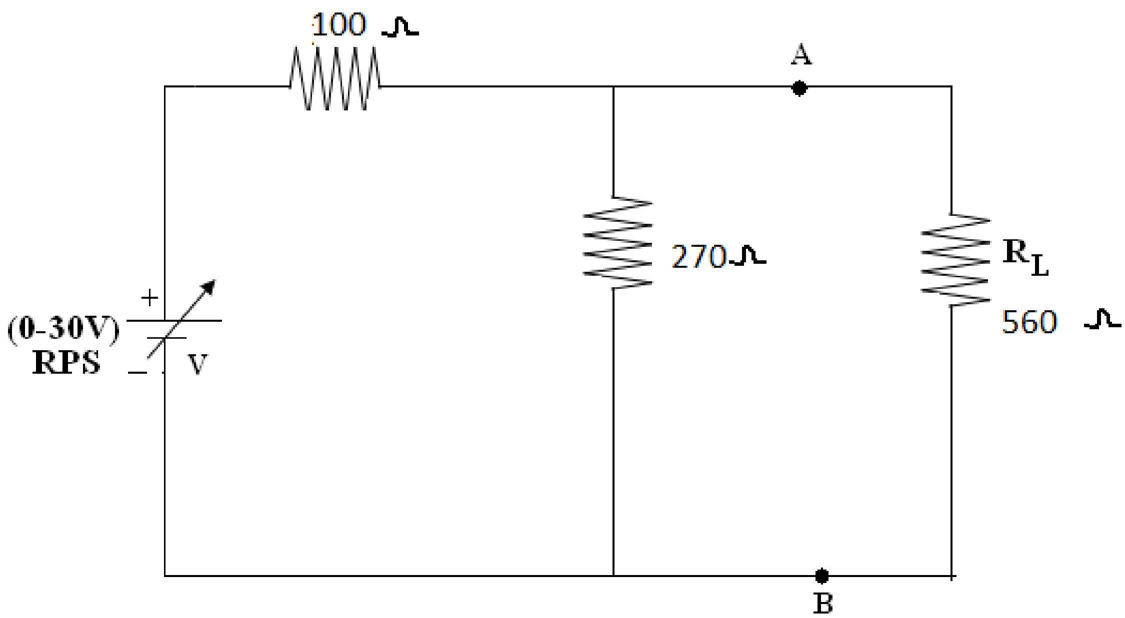
VIVA QUESTIONS:

1. State Kirchhoff's Voltage Law.
2. State Kirchhoff's Current Law.
3. What is current division rule?
4. What is voltage division rule?
5. Give the equivalent resistance when 'n' number of resistances is connected in series.
6. Give the equivalent resistance when 'n' number of resistances is connected in parallel

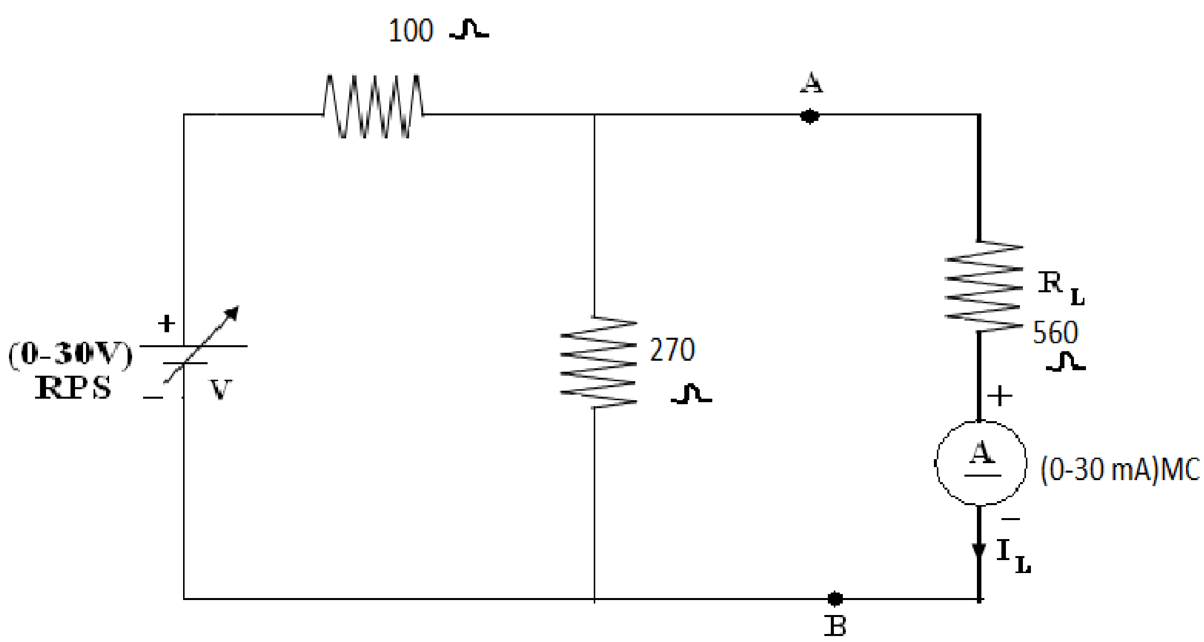
RESULT:

Thus the Kirchhoff's Current and Voltage laws are verified.

CIRCUIT DIAGRAM FOR THEVENIN'S THEOREM:



TO FIND LOAD CURRENT:



EXP.NO:

DATE:

SIMULATION AND EXPERIMENTAL VERIFICATION OF ELECTRICAL CIRCUIT PROBLEMS USING THEVENIN'S THEOREM

AIM:

To verify Thevenin's theorem.

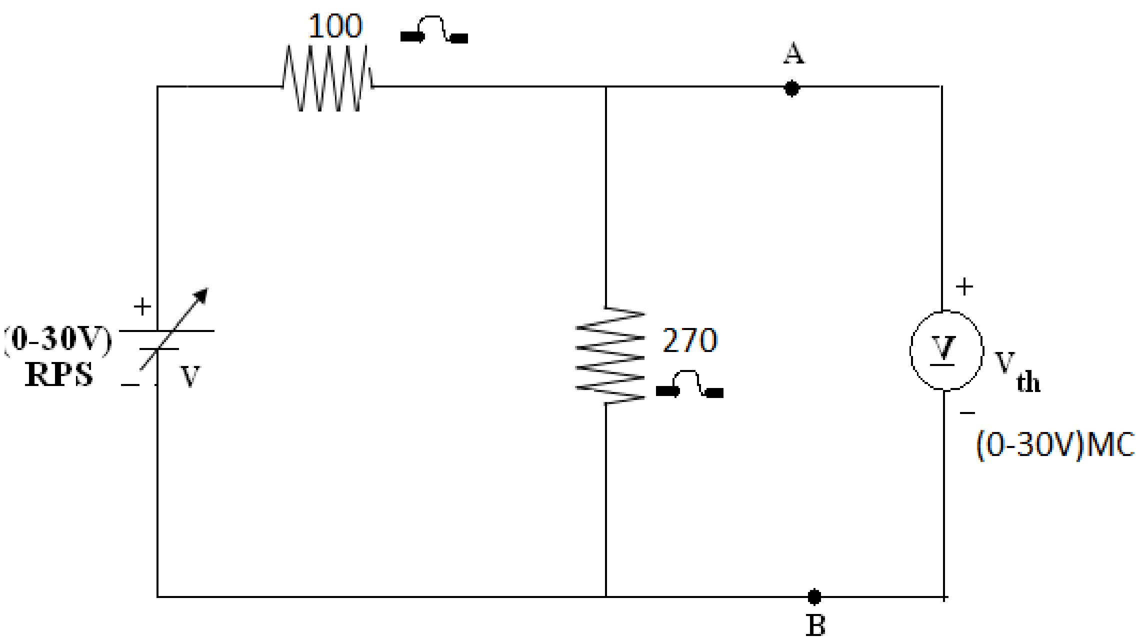
APPARATUS REQUIRED:

S.No no	Name of the Components / Equipment	Type/Range	Quantity required
1	Resistor		
2	Dc power supply		
3	Voltmeter		
4	Ammeter		
5	Wires		
6	Bread board		

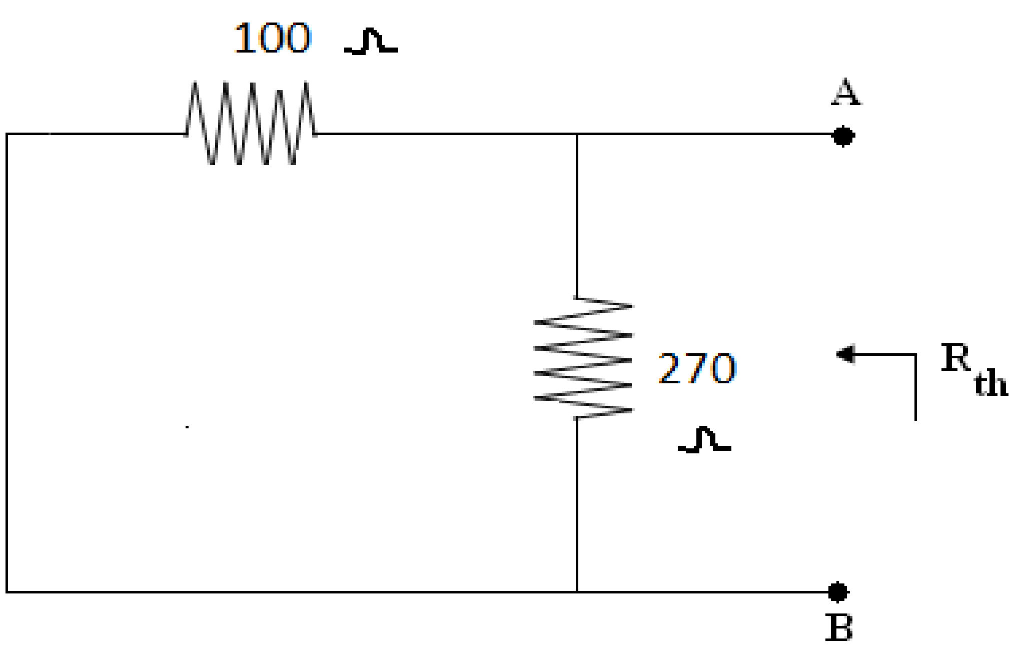
SOFTWARE REQUIRED:

Matlab 7.1

TO FIND V_{th}:



TO FIND R_{th}:

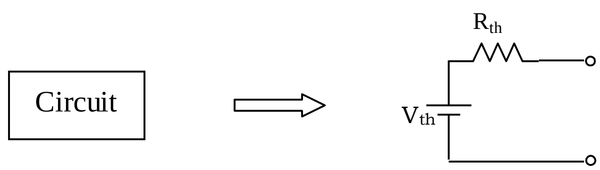


THEVENIN'S THEOREM:

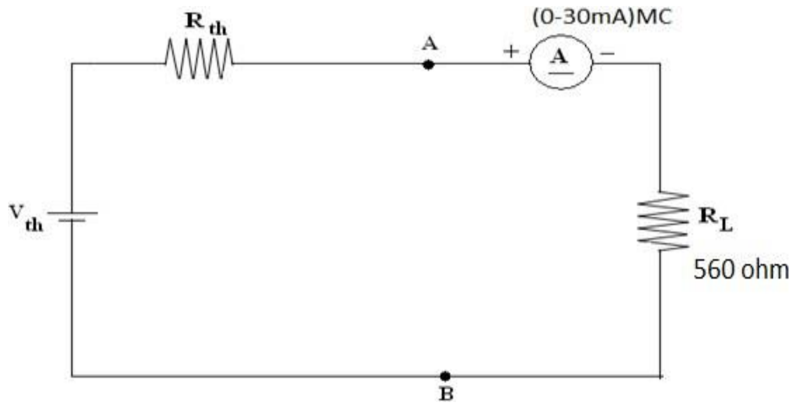
STATEMENT:

Any two-terminal linear network, composed of voltage sources, current sources, and resistors,

can be replaced by an equivalent two-terminal network consisting of an independent voltage source in series with a resistor. The value of voltage source is equivalent to the open circuit voltage (V_{th}) across two terminals of the network and the resistance is equal to the equivalent resistance (R_{th}) measured between the terminals with all energy sources replaced by their internal resistances.



THEVENIN'S EQUIVALENT CIRCUIT



OBSERVATION TABLE

S. No	V_{dc}	V_{th} (Volts)		R_{th} (Ω)		Current through Load Resistance I_L(mA)	
		Practical Value	Theoretical Value	Practical Value	Theoretical Value	Practical Value	Theoretical Value

PROCEDURE:

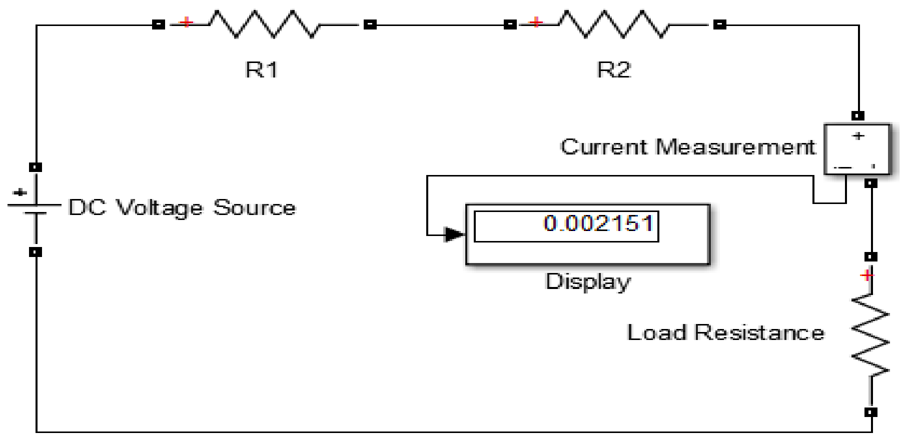
1. Give connections as per the circuit diagram.
2. Measure the current through R_L in the ammeter.
3. Open circuit the output terminals by disconnecting load resistance $R_{L.}$
4. Connect a voltmeter across AB and measure the open circuit voltage V_{th}.
5. To find R_{th}, replace the voltage source by short circuit.
6. Give connections as per the Thevenin's Equivalent circuit.
7. Measure the current through load resistance in Thevenin's Equivalent circuit.
8. Verify Thevenin's theorem by comparing the measured currents in Thevenin's Equivalent circuit with the values calculated theoretically.

SIMULATION PROCEDURE:

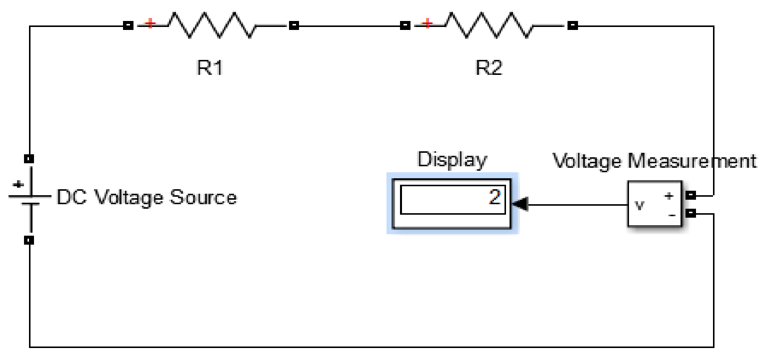
1. Open a new MATLAB/SIMULINK model
2. Connect the circuit as shown in the figure
3. Debug and run the circuit
4. For different input voltages, record the current and voltages and verify with theoretical values.

SIMULATION:

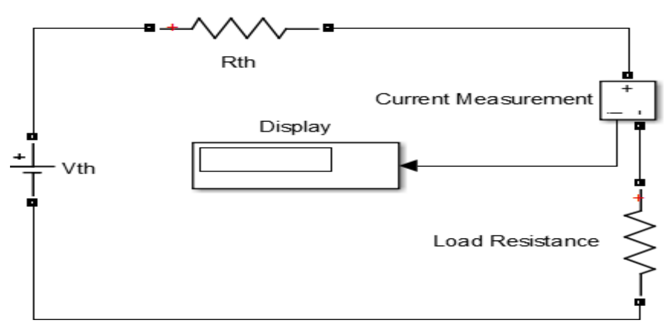
TO FIND LOAD CURRENT:



TO FIND V_{th}:



THEVENIN'S EQUIVALENT CIRCUIT:



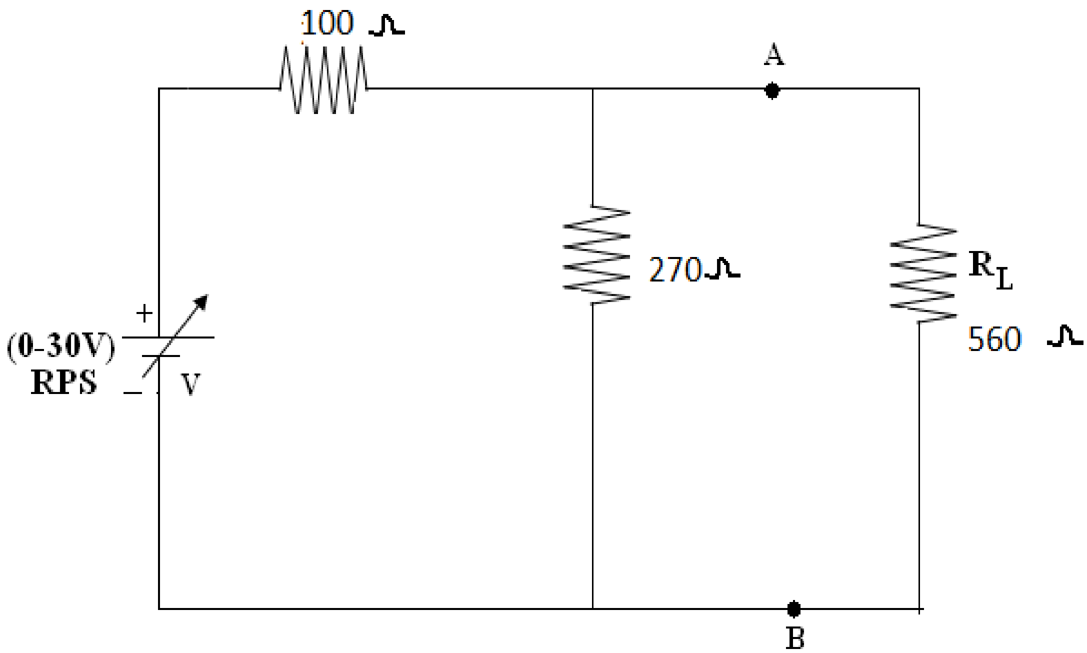
VIVA QUESTIONS:

1. What is meant by a linear network?
2. State Thevenin's Theorem.
3. How do you calculate thevenin's resistance?

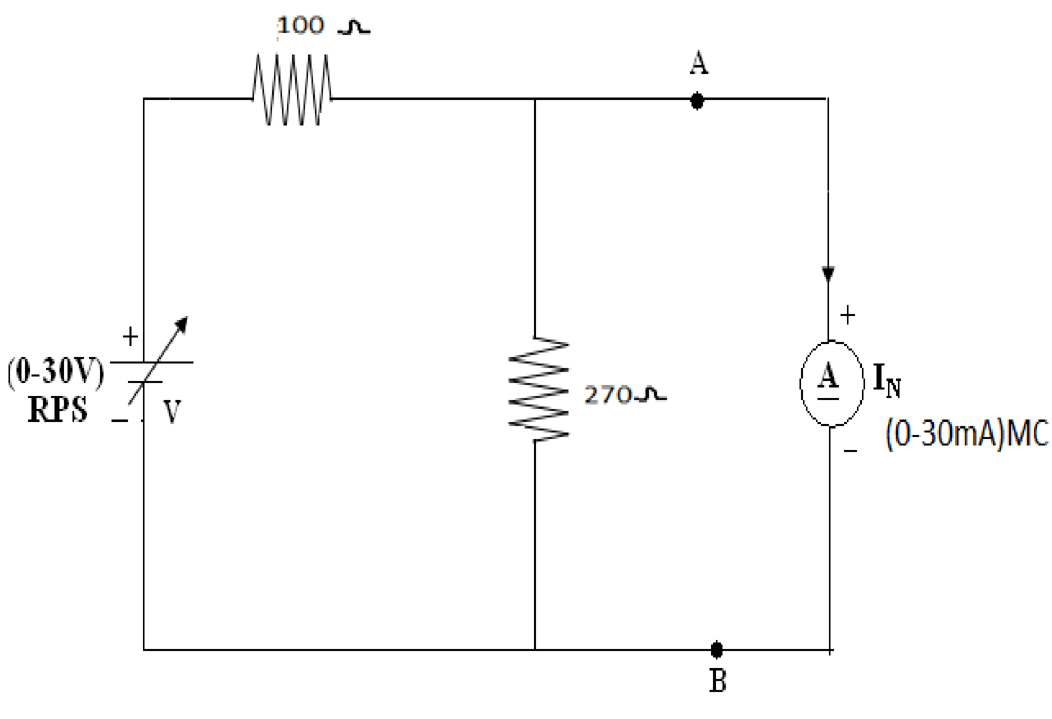
RESULT:

Thus the Thevenin's theorem was verified.

CIRCUIT DIAGRAM FOR NORTON'S THEOREM:



TO FIND NORTON'S CURRENT:



EXP.NO:

DATE:

SIMULATION AND EXPERIMENTAL VERIFICATION OF ELECTRICAL CIRCUIT PROBLEMS USING NORTON'S THEOREM

AIM:

To verify Norton's theorem.

APPARATUS REQUIRED:

S.No no	Name of the Components / Equipment	Type/Range	Quantity required
1	Resistor		
2	Dc power supply		
3	Voltmeter		
4	Ammeter		
5	Wires		
6	Bread board		

SOFTWARE REQUIRED:

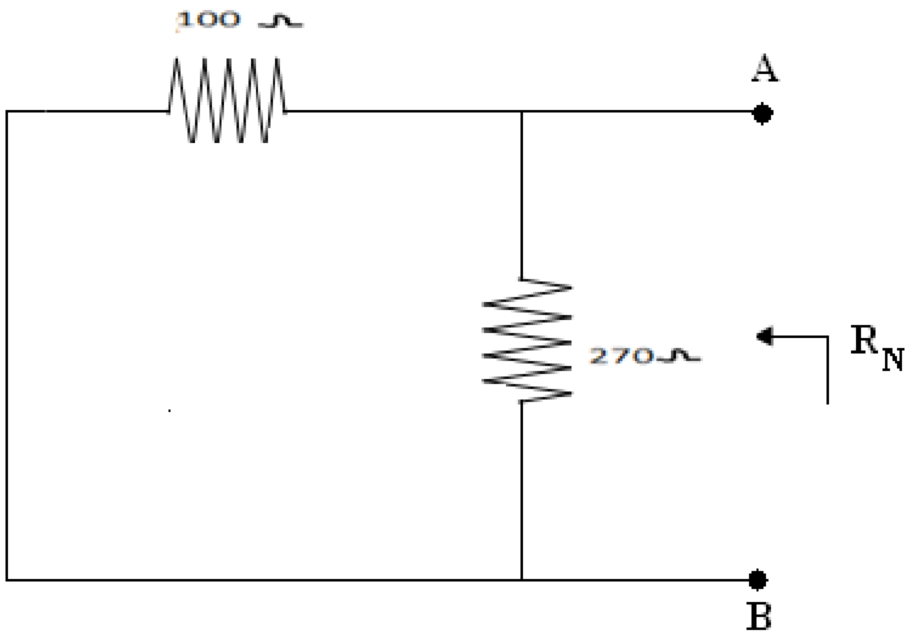
Matlab 7.1

NORTON'S THEOREM

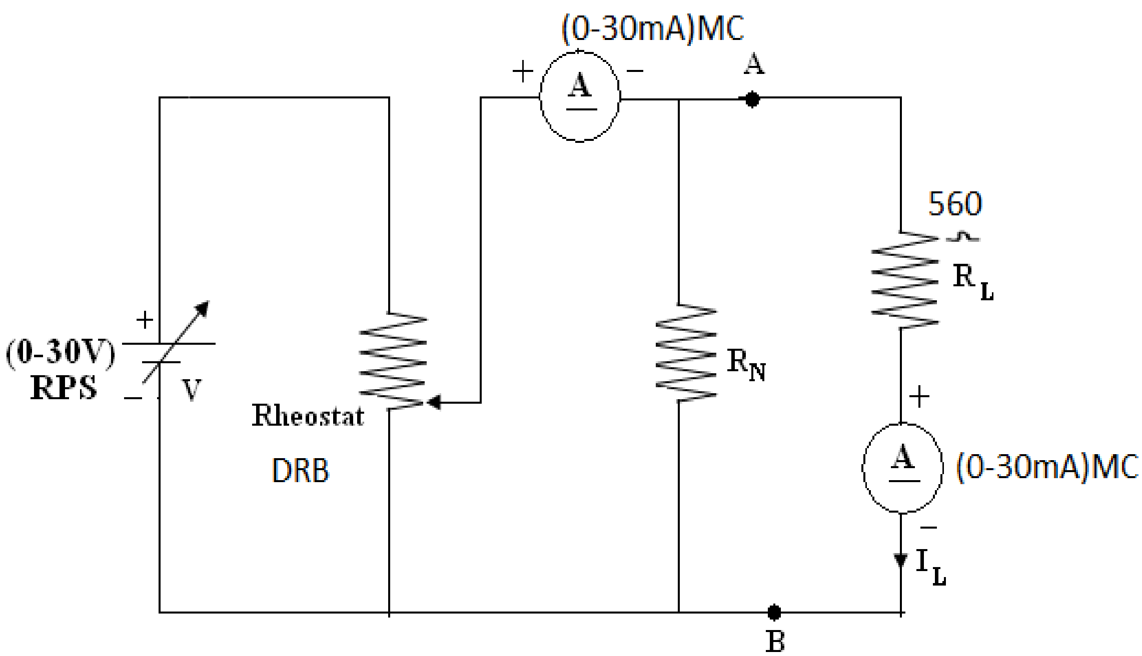
STATEMENT:

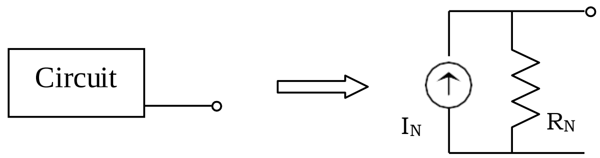
Any two-terminal linear network, composed of voltage sources, current sources, and resistors, can be replaced by an equivalent two-terminal network consisting of an independent current source in parallel with a resistor. The value of the current source is the short circuit current (I_N) between the two terminals of the network and the resistance is equal to the equivalent resistance (R_N) measured between the terminals with all energy sources replaced by their internal resistances.

TO FIND NORTON'S RESISTANCE:



NORTON'S EQUIVALENT CIRCUIT:





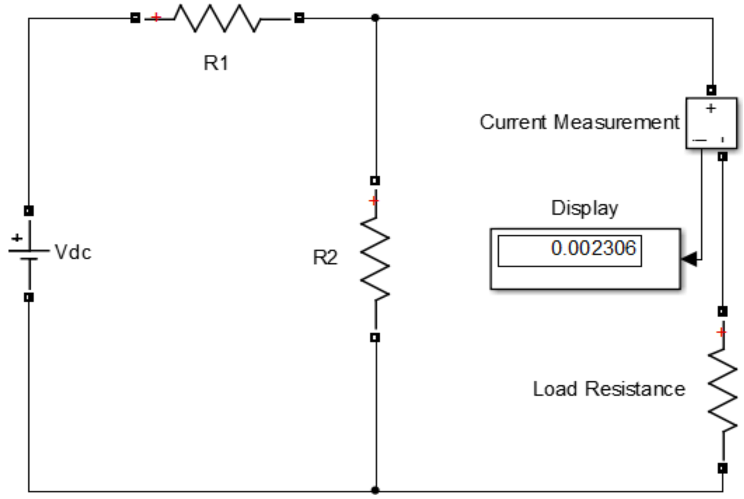
PROCEDURE:

1. Give connections as per the circuit diagram.
2. Measure the current through R_L in ammeter.
3. Short circuit A and B through an ammeter.
4. Measure the Norton current in the ammeter.
5. Find out the Norton's Resistance viewed from the output terminals.
6. Give connections as per the Norton's Equivalent circuit.
7. Measure the current through $R_{L.}$
8. Verify Norton's theorem by comparing currents in R_L directly and that obtained with the equivalent circuit.

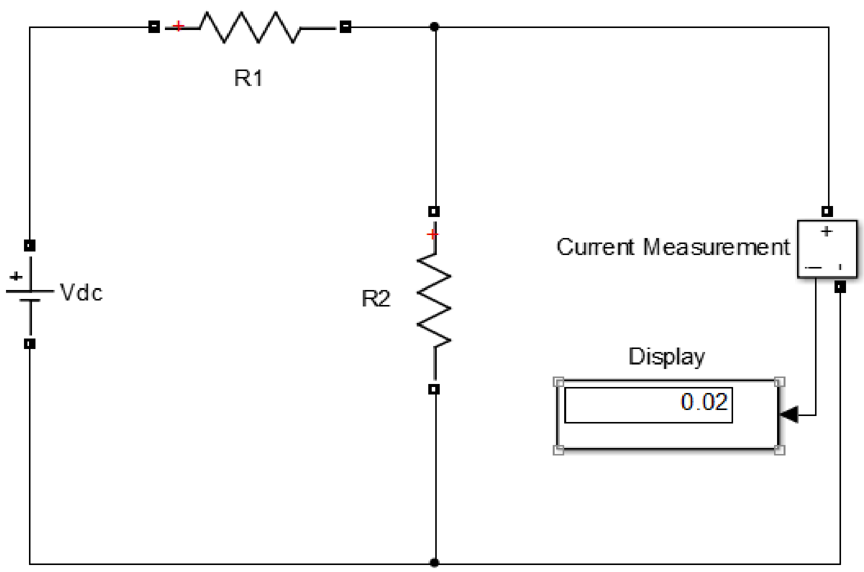
SIMULATION PROCEDURE:

1. Open a new MATLAB/SIMULINK model
2. Connect the circuit as shown in the figure
3. Debug and run the circuit
4. For different input voltages, record the current and voltages and verify with theoretical values.

TO FIND LOAD CURRENT:



TO FIND NORTON'S CURRENT:



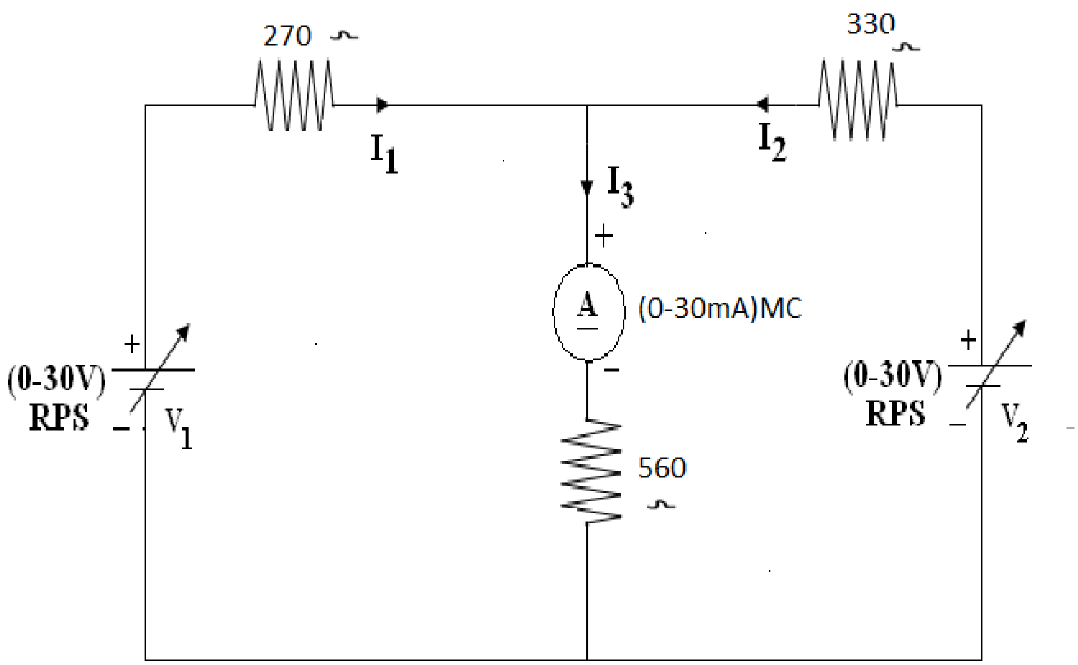
VIVA QUESTIONS:

1. How do you calculate Norton's resistance?
2. State Norton's Theorem.
3. Give the usefulness of Norton's theorems.

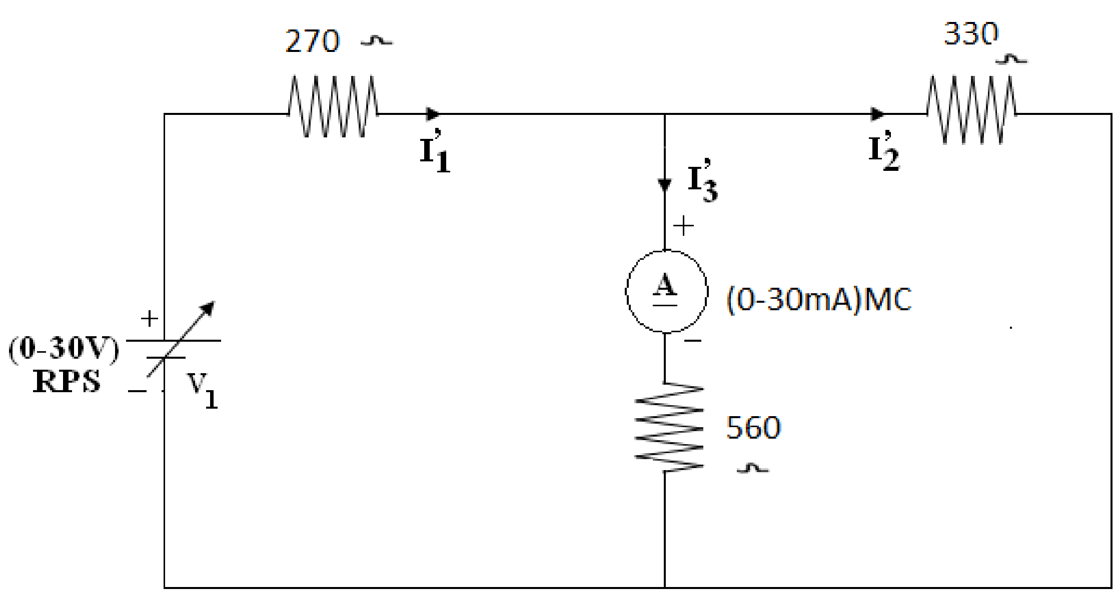
RESULT:

Thus the Norton's theorem was verified.

CIRCUIT DIAGRAM FOR SUPERPOSITION THEOREM:



CIRCUIT DIAGRAM WITH V_1 ACTING INDEPENDENTLY:



EXP.NO:

DATE:

SIMULATION AND EXPERIMENTAL VERIFICATION OF ELECTRICAL CIRCUIT PROBLEMS USING SUPERPOSITION THEOREM

AIM:

To verify superposition theorem.

APPARATUS REQUIRED:

S.No	Name of the Components / Equipment	Type/Range	Quantity required
1	Resistor		
2	Dc power supply		
3	Voltmeter		
4	Ammeter		
5	Wires		
6	Bread board		

SOFTWARE REQUIRED:

Matlab 7.1

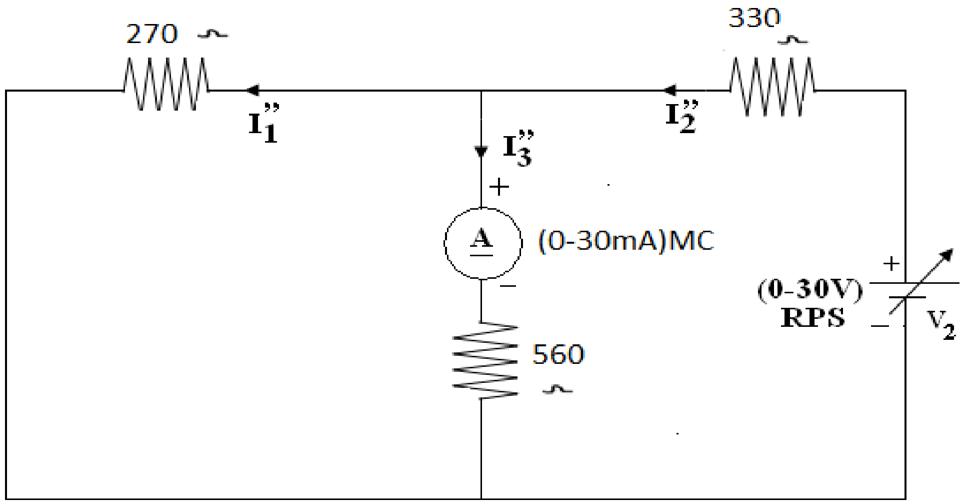
SUPERPOSITION THEOREM:

STATEMENT:

In any linear, bilateral network energized by two or more sources, the total response is equal to the algebraic sum of the responses caused by individual sources acting alone while the other sources are replaced by their internal resistances.

To replace the other sources by their internal resistances, the voltage sources are short- circuited and the current sources open- circuited.

CIRCUIT DIAGRAM WITH V_2 ACTING INDEPENDENTLY:



OBSERVATION TABLE:

Experimental Values:

V_1 (Volts)	V_2 (Volts)	I_3 (mA)

Theoretical Values:

V_1 (Volts)	V_2 (Volts)	I_3 (mA)

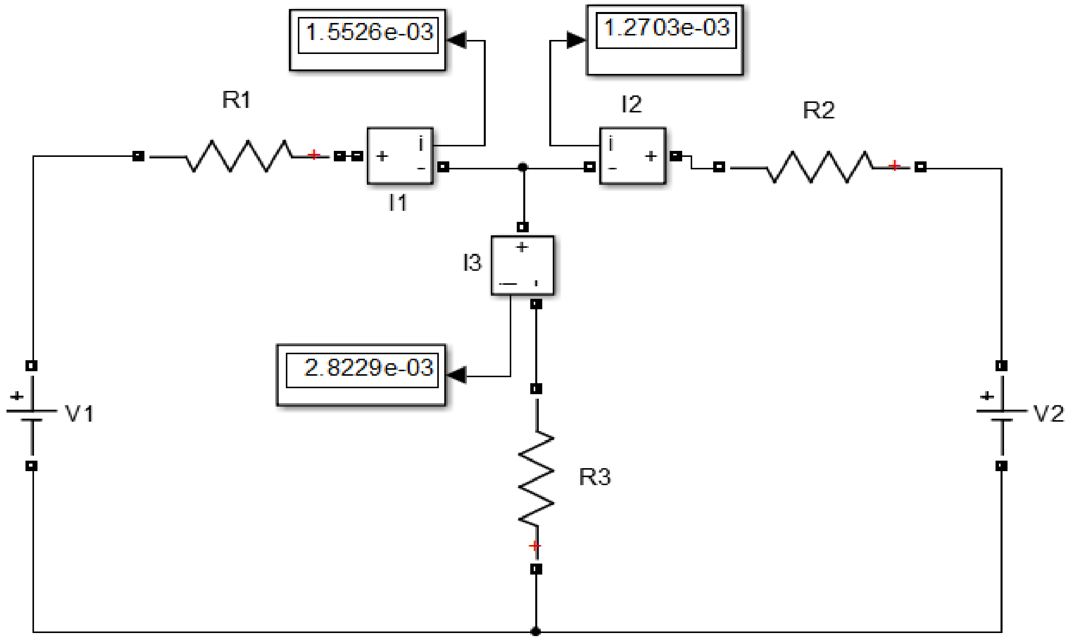
FORMULAE :

$$I_3' + I_3'' = I_3$$

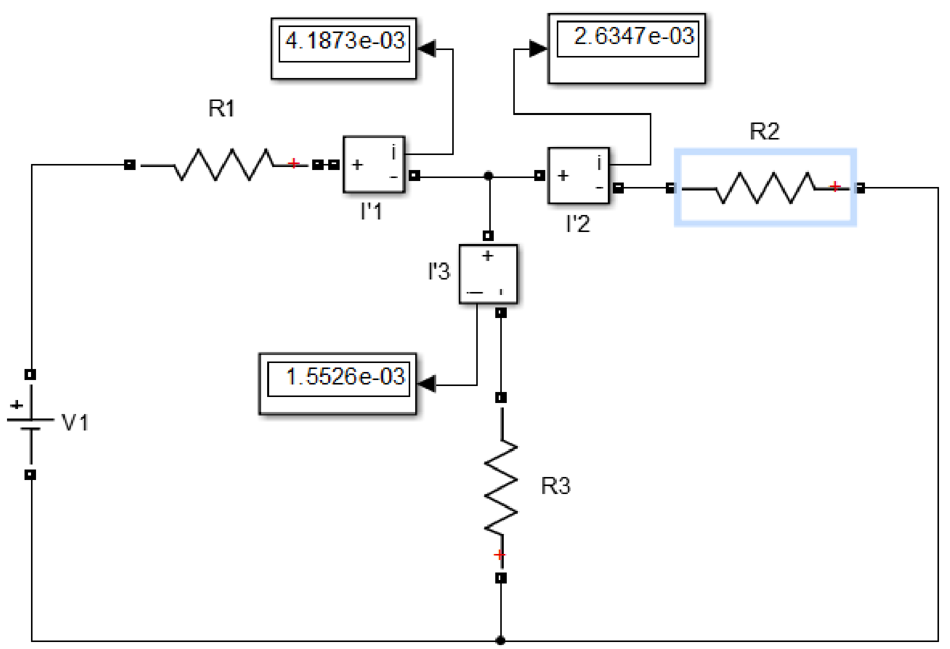
PROCEDURE :

1. Connections are made as per the circuit diagram given in Fig. 1.
2. Switch on the supply.
3. Note the readings of three Ammeters.
4. One of the voltage source V_1 is connected and the other voltage source V_2 is short circuited as given in Fig.2.
5. Note the three ammeter readings.
6. Now short circuit the voltage source V_1 and connect the voltage source V_2 as given in the circuit diagram of Fig. 3.
7. Note the three ammeter readings.
8. Algebraically add the currents in steps (5) and (7) above to compare with the current in step (3) to verify the theorem.
9. Verify with theoretical values.

SIMULATION DIAGRAM FOR SUPERPOSITION THEOREM:



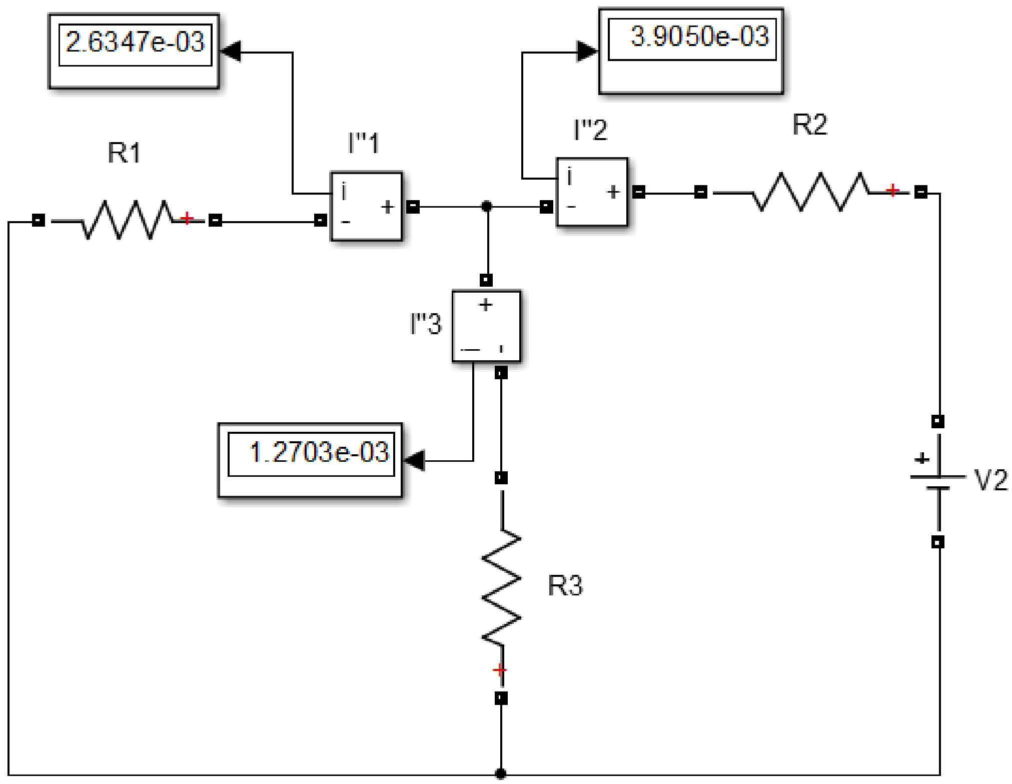
CIRCUIT DIAGRAM WITH V_1 ACTING INDEPENDENTLY:



SIMULATION PROCEDURE:

1. Open a new MATLAB/SIMULINK model
2. Connect the circuit as shown in the figure
3. Debug and run the circuit
4. For different input voltages, record the current and voltages and verify with theoretical values.

CIRCUIT DIAGRAM WITH V_2 ACTING INDEPENDENTLY:



VERIFICATION OF SUPERPOSITION THEOREM:

Practical:

S.No.	I_3 (mA)	I_3’ (mA)	I_3’’ (mA)	I_3= I_3’ +I_3’’ (mA)

Theoretical:

S.No.	I_3 (mA)	I_3' (mA)	I_3'' (mA)	$I_3 = I_3' + I_3''$ (mA)

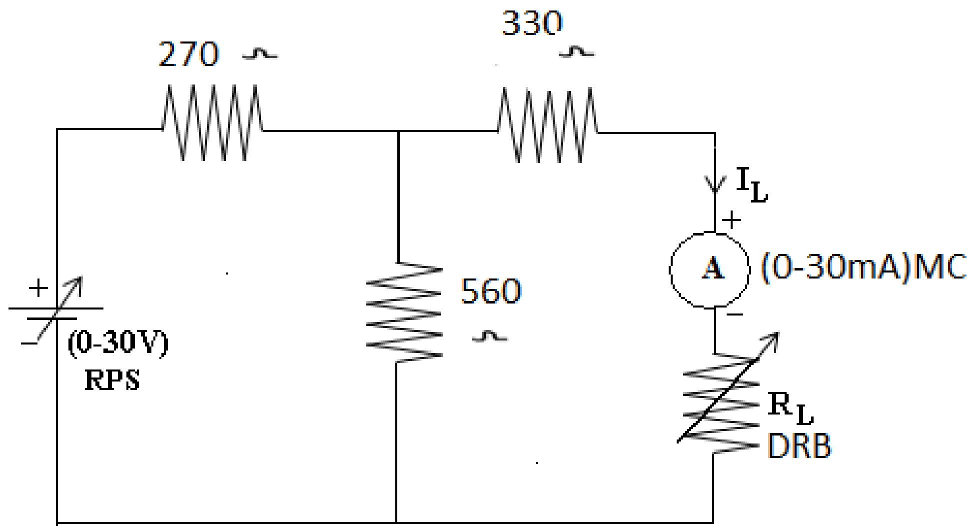
VIVA QUESTIONS:

1. State Superposition Theorem.
2. What is meant by a linear system?
3. Give the usefulness of Superposition Theorem.
4. How will you apply Superposition Theorem to a linear circuit containing both dependent and independent sources?
5. State the limitations of Superposition theorem.

RESULT:

Thus the Superposition theorem was verified.

CIRCUIT DIAGRAM FOR MAXIMUM POWER TRANSFER THEOREM:



OBSERVATION TABLE:

S.No.	R_L (kΩ)	I_L (mA)		P = I^2R_L (mW)	
		Practical Value	Theoretical Value	Practical Value	Theoretical Value

EXP.NO:

DATE:

SIMULATION AND EXPERIMENTAL VERIFICATION OF ELECTRICAL CIRCUIT PROBLEMS USING MAXIMUM POWER TRANSFER THEOREM

AIM:

To verify maximum power transfer theorem.

APPARATUS REQUIRED:

S.No	Name of the Components / Equipment	Type/Range	Quantity required
1	Resistor		
2	Dc power supply		
3	Voltmeter		
4	Ammeter		
5	Wires		
6	Bread board		

SOFTWARE REQUIRED:

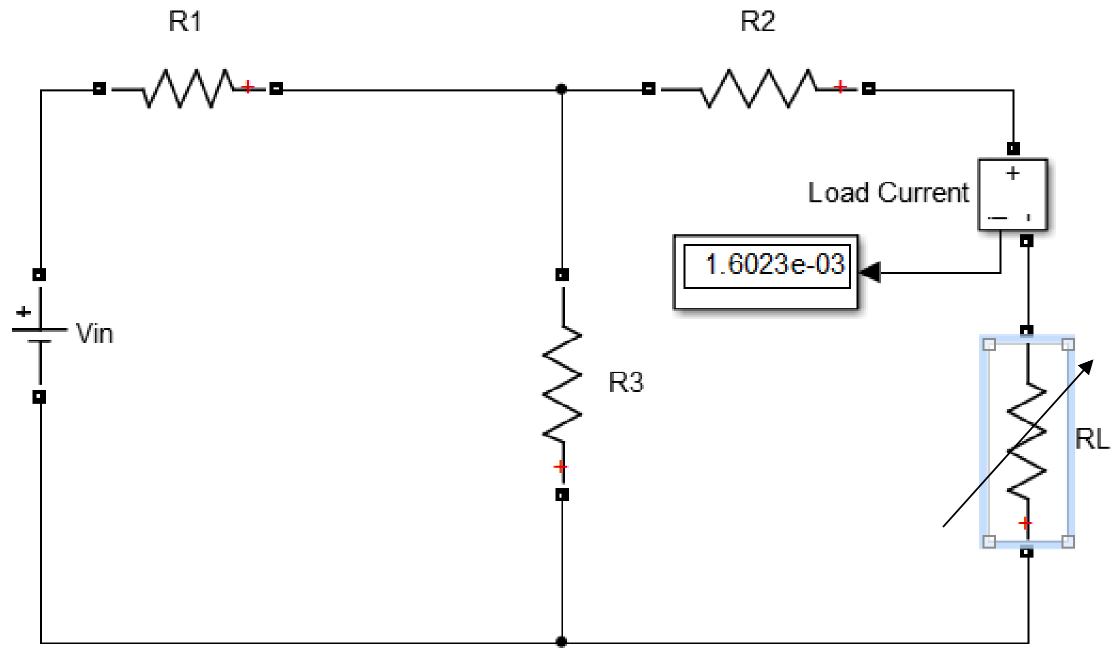
Matlab 7.1

MAXIMUM POWER TRANSFER THEOREM:

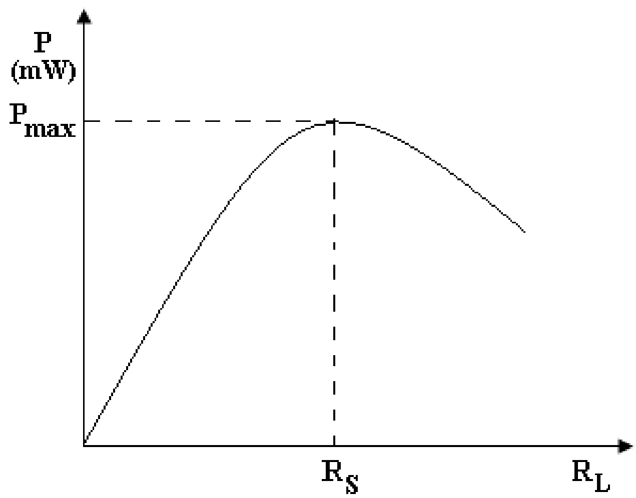
THEORY:

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.

SIMULATION DIAGRAM FOR MAXIMUM POWER TRANSFER THEOREM:



MODEL GRAPH:



MODEL CALCULATION:

PROCEDURE:

1. Find the Load current for the minimum position of the Rheostat theoretically.
2. Select the ammeter Range.
3. Give connections as per the circuit diagram.
4. Measure the load current by gradually increasing R_L .
5. Enter the readings in the tabular column.
6. Calculate the power delivered in R_L.
7. Plot the curve between R_L and power.
8. Check whether the power is maximum at a value of load resistance that equals source resistance.
9. Verify the maximum power transfer theorem.

SIMULATION PROCEDURE:

1. Open a new MATLAB/SIMULINK model
2. Connect the circuit as shown in the figure
3. Debug and run the circuit
4. For different input voltages, record the current and voltages and verify with theoretical values.

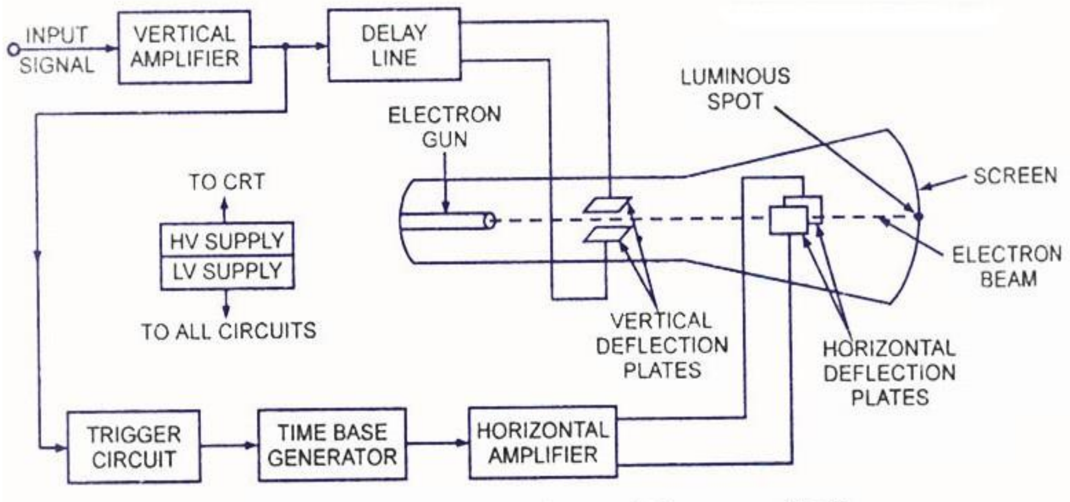
VIVA QUESTIONS:

1. Define Power. What is the unit of Power?
2. State Maximum Power Transfer Theorem

RESULT:

Thus the Maximum power transfer theorem was verified.

BLOCK DIAGRAM OF GENERAL PURPOSE CRO:



Block Diagram of a General Purpose CRO

EXP NO.:

DATE:

STUDY OF ANALOG AND DIGITAL OSCILLOSCOPES AND MEASUREMENT OF SINUSOIDAL VOLTAGE, FREQUENCY AND POWER FACTOR

AIM:

The aim of the experiment is to understand the operation of cathode ray oscilloscope (CRO) and to become familiar with its usage, also to perform an experiment using function generator to measure amplitude, time period, frequency & power factor of the time varying signals using a calibrated cathode ray oscilloscope.

APPARATUS REQUIRED:

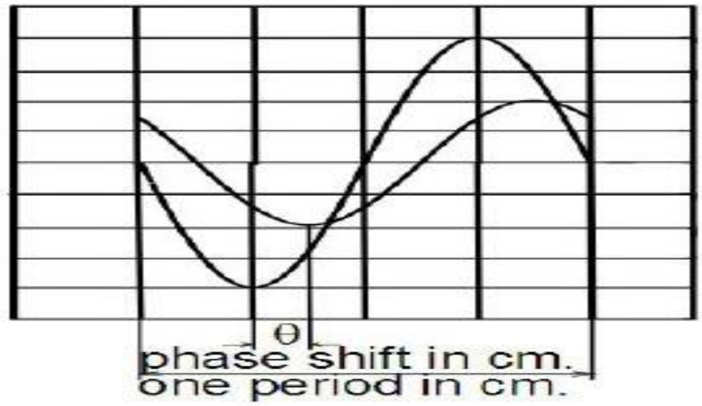
S.No	Name of the Components/Equipment	Qty
1.	CRO	1
2.	Function generator	2
3.	Probes	2

THEORY:

The cathode ray oscilloscope (CRO) provides a visual presentation of any waveform applied to the input terminal. The oscilloscope consists of the following major subsystems.

- Cathode ray tube (CRT)
- Vertical amplifier
- Horizontal amplifier
- Sweep Generator
- Trigger circuit
- Associated power supply

It can be employed to measure quantities such as peak voltage, frequency, phase difference, pulse width, delay time, rise time, and fall time.



S.No	Type of wave	Time period (T)	Amplitude	Theoretical Frequency	Practical Frequency
1.					
2.					
3.					

CATHODE RAY TUBE:

The CRT is the heart of the CRO providing visual display of an input signal waveform. A CRT contains four basic parts:

- An electron gun to provide a stream of electrons.
- Focusing and accelerating elements to produce a well define beam of electrons.
- Horizontal and vertical deflecting plates to control the path of the electron beam.
- An evacuated glass envelope with a phosphorescent which glows visibly when struck by electron beam.

A Cathode containing an oxide coating is heated indirectly by a filament resulting in the release of electrons from the cathode surface. The control grid which has a negative potential, controls the electron flow from the cathode and thus control the number of electron directed to the screen. Once the electron passes the control grid, they are focused into a tight beam and accelerated to a higher velocity by focusing and accelerating anodes. The high velocity and well defined electron beam then passed through two sets of deflection plates.

The First set of plates is oriented to deflect the electron beam vertically. The angle of the vertical deflection is determined by the voltage polarity applied to the deflection plates. The electron beam is also being deflected horizontally by a voltage applied to the horizontal deflection plates. The tube sensitivity to deflecting voltages can be expressed in two ways that are deflection factor and deflection sensitivity.

The deflected beam is then further accelerated by very high voltages applied to the tube with the beam finally striking a phosphorescent material on the inside face of the tube. The phosphor glows when struck by the energetic electrons.

CONTROL GRID:

Regulates the number of electrons that reach the anode and hence the brightness of the spot on the screen.

FOCUSING ANODE:

Ensures that electrons leaving the cathode in slightly different directions are focused down to a narrow beam and all arrive at the same spot on the screen.

ELECTOR GUN:

Cathode, control grid, focusing anode, and accelerating anode.

DEFLECTING PLATES:

Electric fields between the first pair of plates deflect the electrons horizontally and an electric field between the second pair deflects them vertically. If no deflecting fields are present, the electrons travel in a straight line from the hole in the accelerating anode to the center of the screen, where they produce a bright spot. In general purpose oscilloscope, amplifier circuits are needed to increase the input signal to the voltage level required to operate the tube because the signals measured using CRO are typically small. There are amplifier sections for both vertical and horizontal deflection of the beam.

VERTICAL AMPLIFIER:

Amplify the signal at its input prior to the signal being applied to the vertical deflection plates.

HORIZONTAL AMPLIFIER:

Amplify the signal at its input prior to the signal being applied to the horizontal deflection plates.

SWEEP GENERATOR:

Develop a voltage at the horizontal deflection plate that increases linearly with time.

OPERATION:

The four main parts of the oscilloscope CRT are designed to create and direct an electron beam to a screen to form an image. The oscilloscope links to a circuit that directly connects to the vertical deflection plates while the horizontal plates have linearly increasing charge to form a plot of the circuit voltage over time. In an operating cycle, the heater gives electrons in the cathode enough energy to escape. The electrons are attracted to the accelerating anode and pulled through a control grid that regulates the number of electrons in the beam, a focusing anode that controls the width of the beam, and the accelerating anode itself. The vertical and horizontal deflection plates create electric field that bend the beam of electrons. The electrons finally hit the fluorescent screen which absorbs the energy from the electron beam and emits it in the form of light to display an image at the end of the glass tube.

PRECAUTIONS:

1. Do not leave a 'bright spot' on the screen for any length of time.
2. Do not apply signals that exceed the scopes voltage rating.
3. Do not try make accurate measurements on signals whose frequency is outside the scope's frequency specifications.
4. Be aware that the scope's input circuitry can cause loading effects on the circuitry under test-use correct probe for the work.

PRODEDURE:

1. Measurement of Voltage Using CRO : A voltage can be measured by noting the Y deflection produced by the voltage; using this deflection in conjunction with the Y-gain setting, the voltage can be calculated as follows : V = (no. of boxes in cm.) x (selected Volts/cm scale)

2 .Measurement of Current and Resistance Using a CRO: Using the general method, a correctly calibrated CRO can be used in conjunction with a known value of resistance R to determine the current I flowing through the resistor.

3 Measurement of Frequency Using a CRO: A simple method of determining the frequency of a signal is to estimate its periodic time from the trace on the screen of a CRT. However this method has limited accuracy, and should only be used where other methods are not available. To calculate the frequency of the observed signal, one has to measure the period, i.e. the time taken for 1 complete cycle, using the calibrated sweep scale. The period could be calculated by T = (no. of squares in cm) x (selected Time/cm scale) Once the period T is known, the frequency is given by f (Hz)= 1/T(sec)

4. Measurement of Phase: The calibrated time scales can be used to calculate the phase shift between two sinusoidal signals of the same frequency. If a dual trace or beam CRO is available to display the two signals simultaneously (one of the signals is used for synchronization), both of the signals will appear in proper time perspective and the amount of time difference between the waveforms can be measured. This, in turn can be utilized to calculate the phase angle θ, between the two signals.

Referring to the fig below the phase shift can be calculated by the formula;

$$\theta^\circ = \frac{phase\ shift\ in\ cm}{one\ period\ in\ cm}$$

MEASUREMENT OF PF:

The power factor is calculated by the formula

pf=VICOS θ.

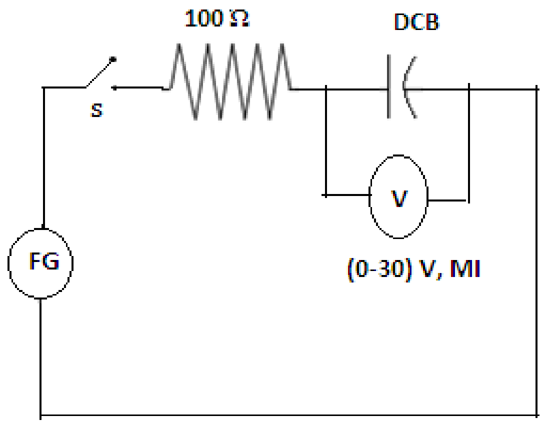
VIVA QUESTIONS:

1. What is a CRO?
2. How can we measure the voltage using a CRO?
3. Explain the different parts of the CRO
4. Explain the operation of a CRO.

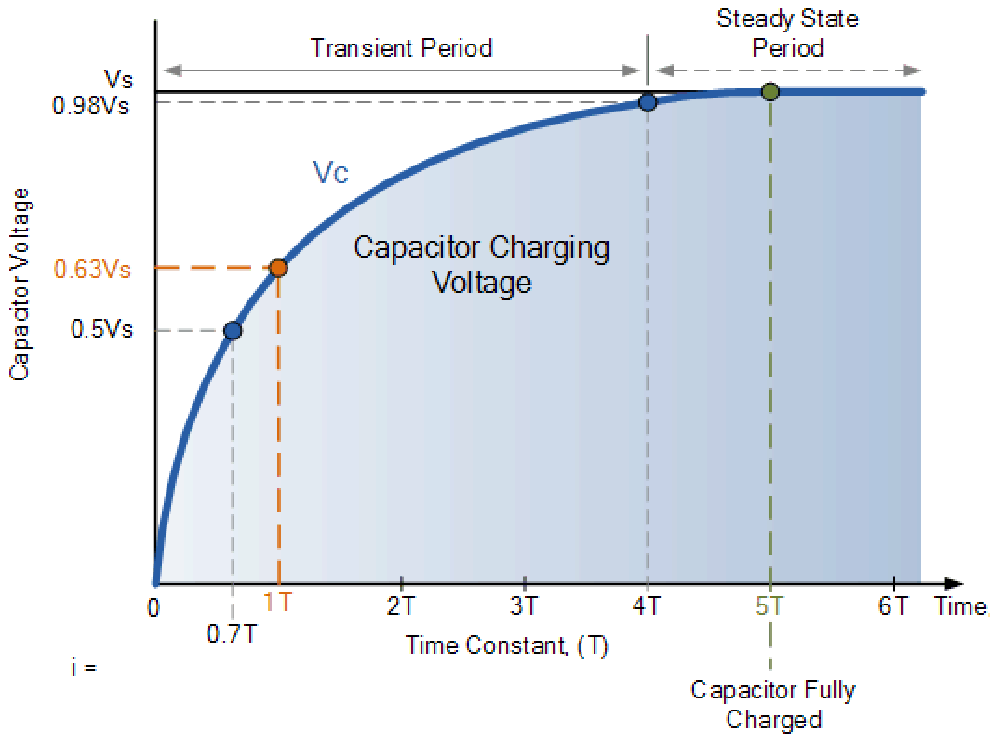
RESULT:

Thus the Analog and digital oscilloscopes were studied and measurement of sinusoidal voltage, frequency and power factor was done.

CIRCUIT DIAGRAM FOR RC TRANSIENT:



MODEL GRAPH:



EXP NO. :

DATE :

SIMULATION AND EXPERIMENTAL VALIDATION OF R-C ELECTRIC CIRCUIT TRANSIENTS

AIM:

To find the time constant of series R-C electric circuits

SOFTWARE REQUIRED:

PSpice Lite

APPARATUS REQUIRED:

S.No.	Name of the Components/Equipment	Range/Type	Quantity required
1	Resistor	100 Ω	1
2	Function generator	-	1
3	Voltmeter	(0-30)V MI	1
4	Decade capacitance box	-	1
5	Wires	Single strand	Few nos
6	Bread board		1

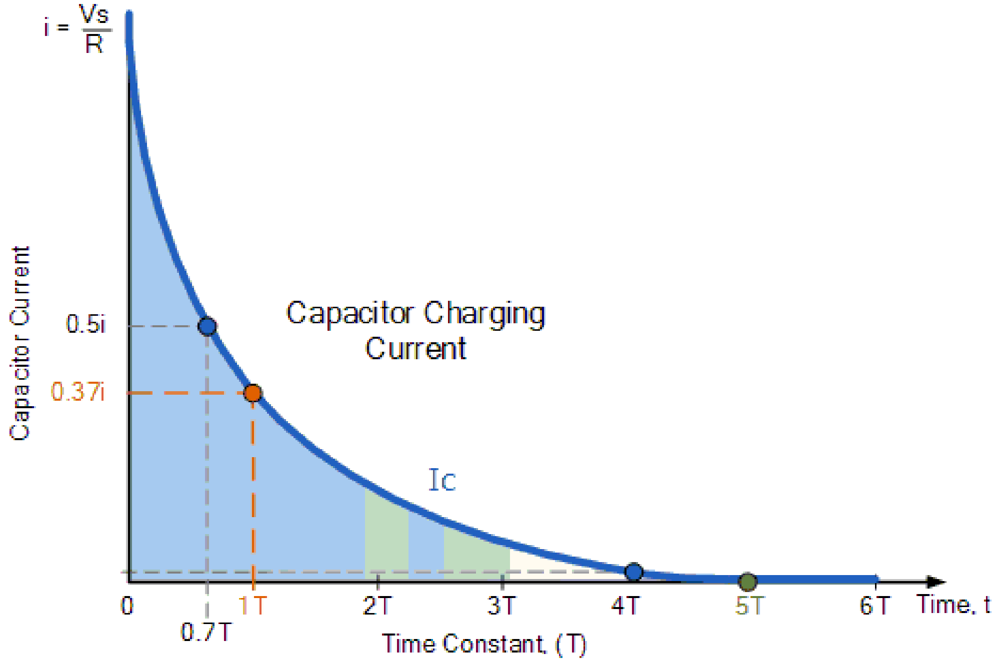
THEORY:

RC CIRCUIT:

Consider a series RC circuit as shown. The switch is in open state initially. There is no charge on condenser and no voltage across it. At instant t=0, switch is closed.

Immediately after closing a switch, the capacitor acts as a short circuit, so current at the time of switching is high. The voltage across capacitor is zero at $t= 0^+$ as capacitor acts as a short circuit, and the current is maximum given by,

$i = V/R$ Amps



OBSERVATION TABLE:

S.No.	Frequency (Hz)	Time (s)	Voltage across the capacitor V_C (v)

MODEL CALCULATION:

This current is maximum at t=0+ which is charging current. As the capacitor starts charging, the voltage across capacitor V_C starts increasing and charging current starts decreasing. After some time, when the capacitor charges to V volts, it achieves steady state. In steady state it acts as an open circuit and current will be zero finally.

Charging current and voltage in capacitor are given as below,

$$I_C = \frac{V_{in}}{R} e^{\frac{-t}{RC}} \qquad V_C = V_{in}(1 - e^{\frac{-t}{RC}})$$

$$V_C = V_{in}(1 - e^{-1})$$

The term RC in equation of V_C or I_C is called Time constant and denoted by τ, measured in seconds.

When, $t = RC = \tau$ then,

$$V_C = 0.632V_{in}$$

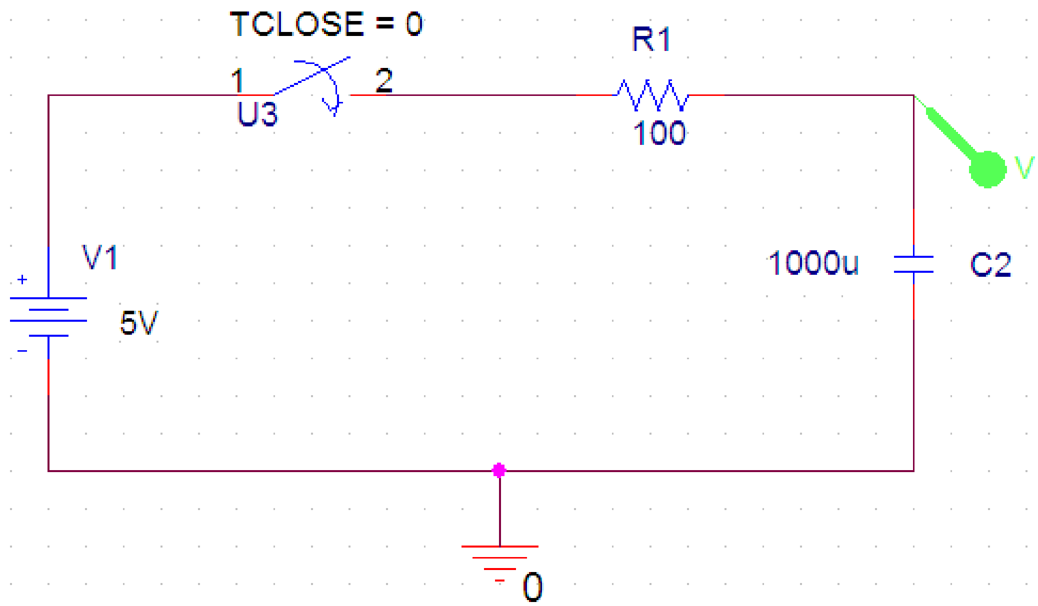
So time constant of series RC circuit is defined as time required by the capacitor voltage to rise from zero to 0.632 of its final steady state value during charging.

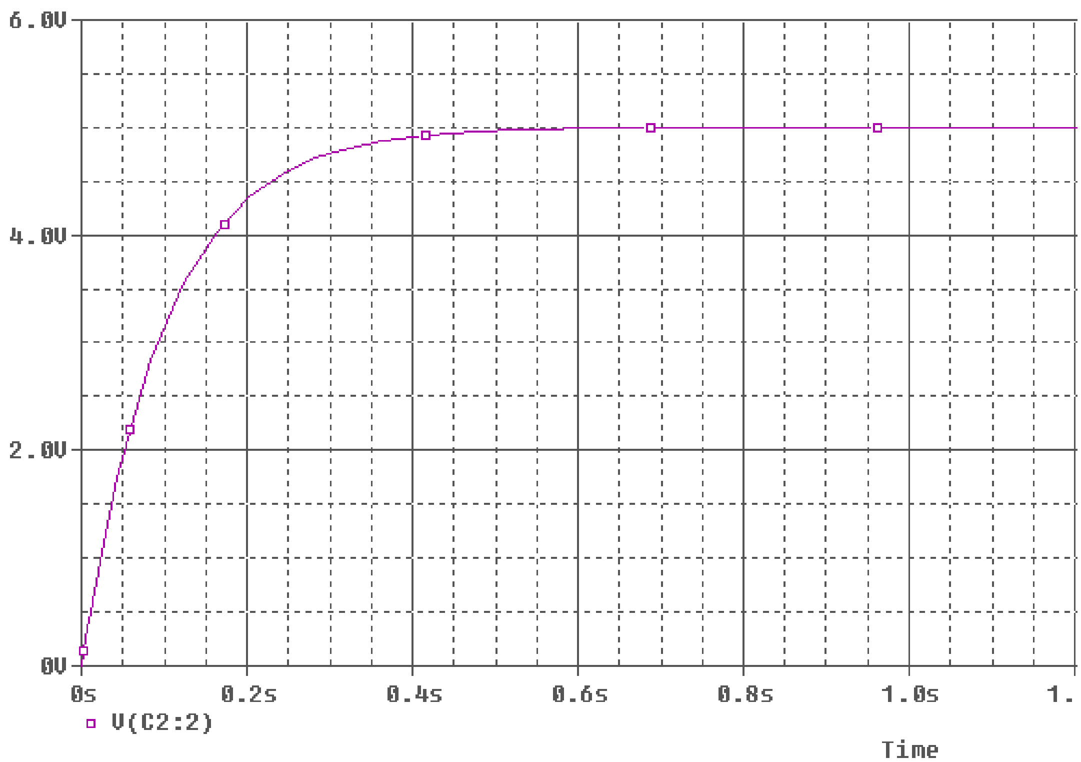
Thus, time constant of RC circuit can be defined as time seconds, during which voltage across capacitor (stating from zero) would reach its final steady state value if its rate of change was maintained constant at its initial value throughout charging period.

PROCEDURE:

1. Make the connections as per the circuit diagram.
2. Vary the frequency by using function generator.
3. For different frequencies tabulate the value of voltage across the capacitor .
4. Calculate the time period.
5. Plot the graph for time period Vs voltage across the capacitor.

SIMULATION DIAGRAM:





SIMULATION PROCEDURE:

1. Open a new PSpice CAPTURE project.
2. Connect the circuit as shown in the figure.
3. Create simulation profile and run the model

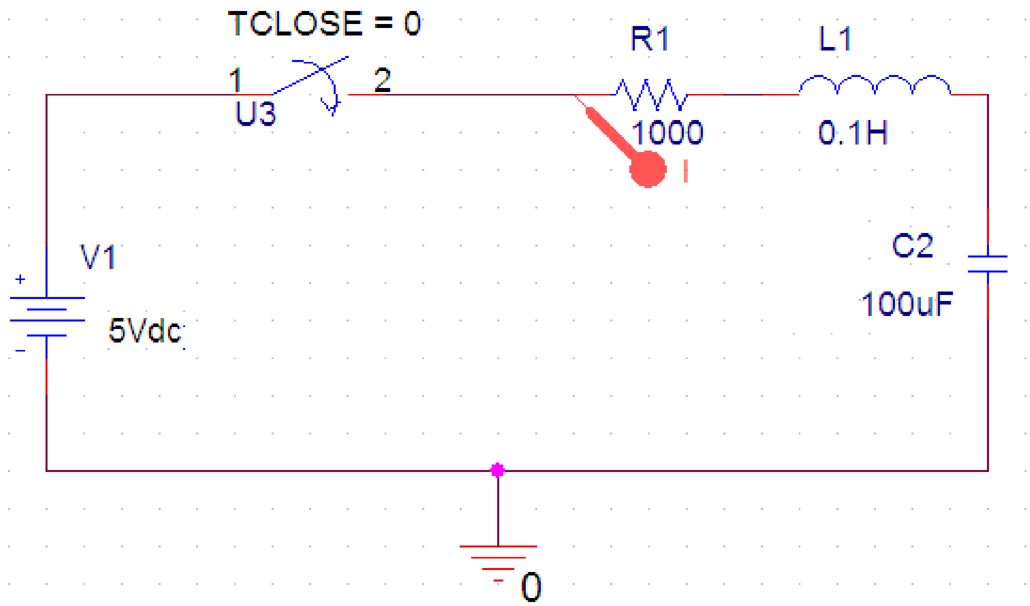
VIVA QUESTIONS:

1. Differentiate steady state and transient state.
2. What is meant by transient response?
3. Define the time constant of a RL Circuit.
4. Define the time constant of a RC Circuit.
5. What is meant by forced response?

RESULT:

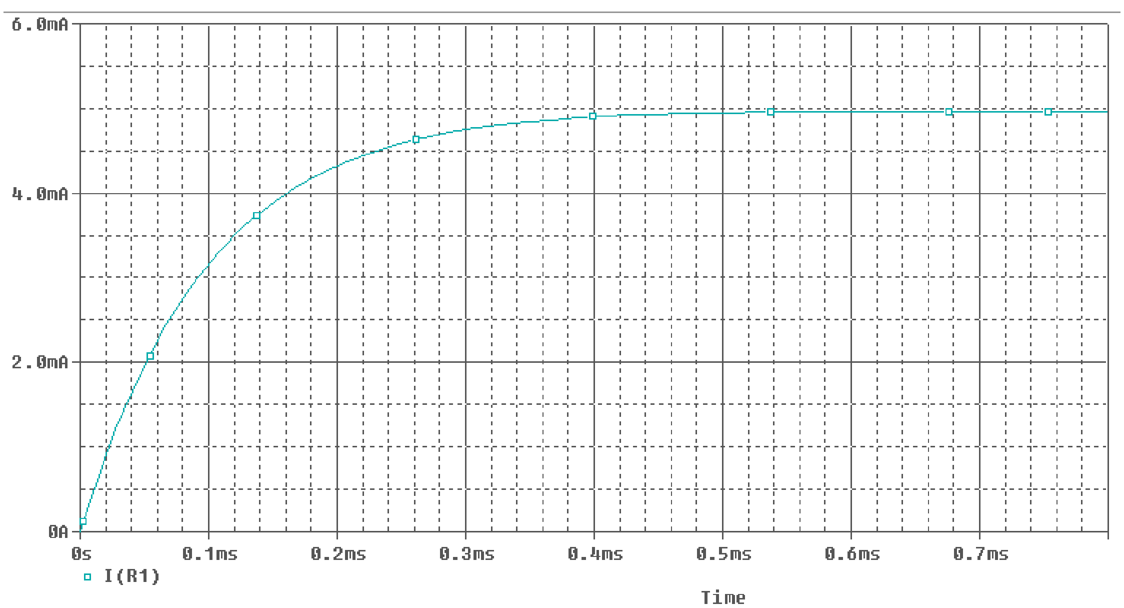
Thus the transient responses of RC circuit are found practically.

SIMULATION CIRCUIT DIAGRAM:



OUTPUT WAVEFORM:

Case (i):



EXP NO.:

DATE :

SIMULATION AND EXPERIMENTAL VALIDATION OF FREQUENCY RESPONSE OF RLC ELECTRIC CIRCUIT

AIM:

To simulate and find the frequency response of RLC electric circuits.

SOFTWARE REQUIRED:

PSpice Lite

APPARATUS REQUIRED:

S.No.	Name of the Components/Equipment	Range/Type	Quantity required
1	Resistor	1000 Ω	1
2	Function generator	-	1
3	Voltmeter	(0-30)V MI	1
4	Decade capacitance box	-	1
5	Decade Inductance box	-	1
6	Wires	Single strand	Few nos
7	Bread board		1

THEORY:

RLC CIRCUIT:

Consider a series RLC circuit as shown. The switch is in open state initially. There is no charge on condenser and no voltage across it. At instant t=0, switch is closed.

Immediately after closing a switch, the capacitor acts as a short circuit, so current at the time of switching is high. The voltage across capacitor is zero at $t = 0^+$ as capacitor acts as a short circuit, and the current is maximum given by,

$$i = V/R \text{ Amps}$$

OBSERVATION TABLE:

S.No.	Frequency (Hz)	Time (s)	Voltage across the capacitor V_C (v)

MODEL CALCULATION:

This current is maximum at t=0+ which is charging current. As the capacitor starts charging, the voltage across capacitor V_C starts increasing and charging current starts decreasing. After some time, when the capacitor charges to V volts, it achieves steady state. In steady state it acts as an open circuit and current will be zero finally.

Laplace transform of current flowing through the circuit is,

$$I(s)=\frac{V/L}{s^2+\frac{R}{L}s+\frac{1}{LC}}$$

Case (i):

If $\left[\frac{R}{2L}\right]^2 > \frac{1}{LC}$

The roots are real and distinct. The current is over damped.

Case (ii):

If $\left[\frac{R}{2L}\right]^2 = \frac{1}{LC}$

The roots are equal. The current is critically damped.

Case (iii):

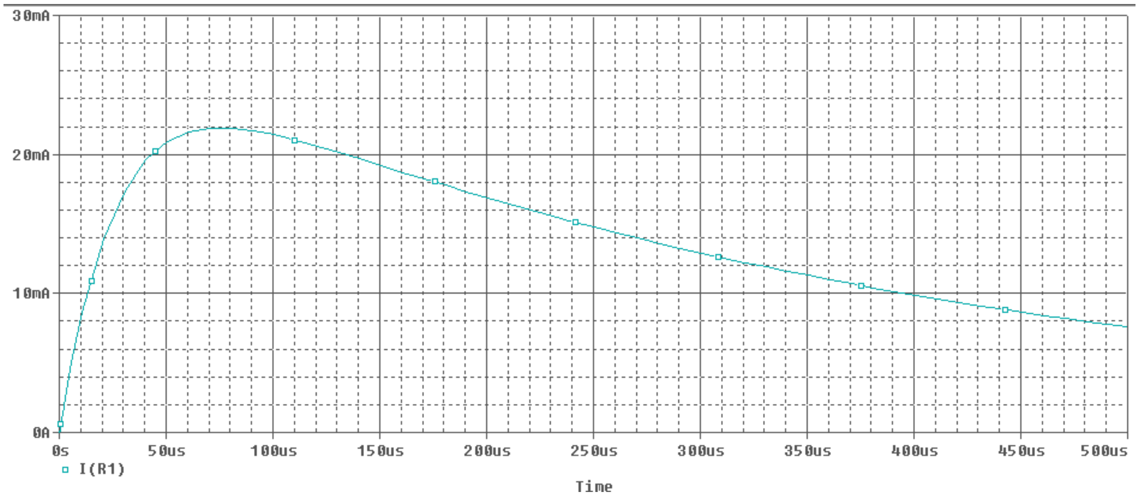
If $\left[\frac{R}{2L}\right]^2 < \frac{1}{LC}$

The roots become complex conjugate. The current is oscillatory in nature.

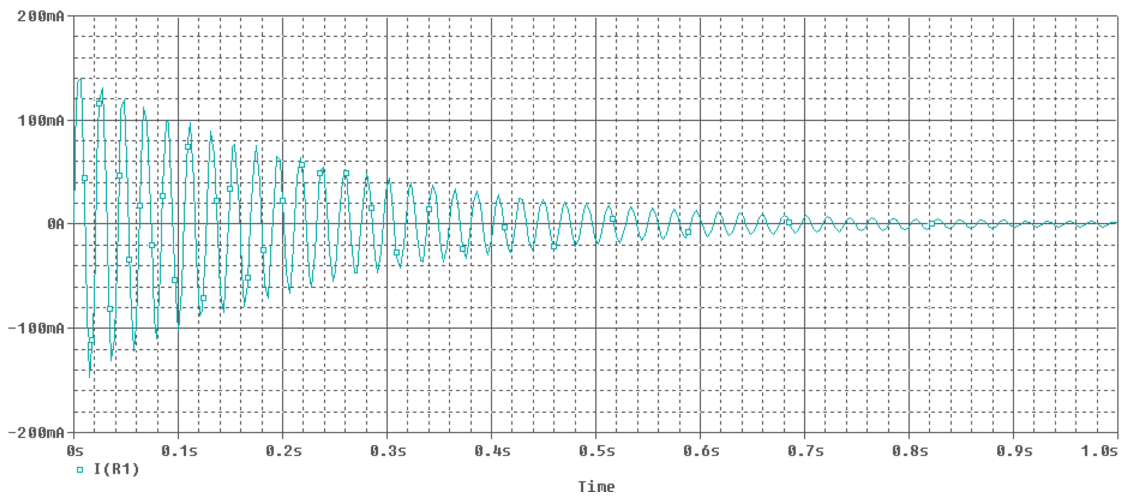
PROCEDURE:

1. Make the connections as per the circuit diagram
2. Vary the frequency by using function generator
3. For different frequencies tabulate the value of voltage across the capacitor
4. Calculate the time period
5. Plot the graph for time period Vs voltage across the capacitor.

Case (ii):



Case (iii):



SIMULATION PROCEDURE:

4. Open a new PSpice CAPTURE project.
5. Connect the circuit as shown in the figure.
6. Create simulation profile and run the model

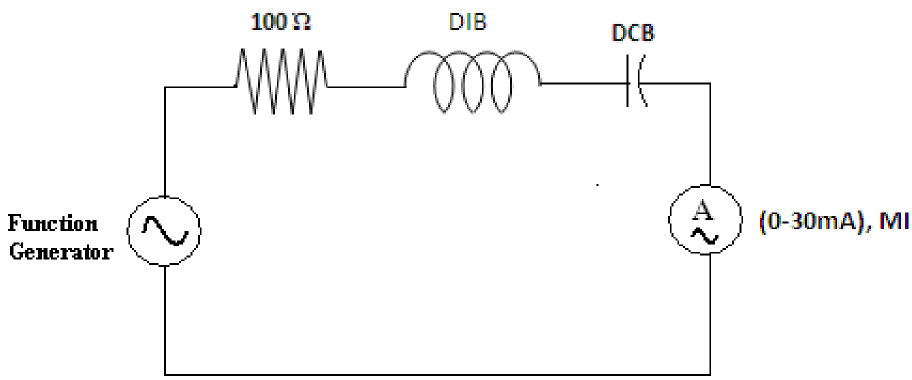
VIVA QUESTIONS

1. What is meant by transient response?
2. Define the time constant of a RL Circuit.
3. Define the time constant of a RC Circuit.
4. What is meant by forced response?

RESULT:

Thus the transient responses of RLC circuit are found practically.

CIRCUIT DIAGRAM FOR SERIES RESONANCE:



OBSERVATION TABLE:

S.No.	Frequency in Hz	Output Current in mA

MODEL CALCULATION:

EXP NO.:

DATE :

DESIGN AND SIMULATION OF SERIES RESONANCE CIRCUIT

AIM:

To plot the current Vs frequencies graph of series resonant circuits and hence measure their bandwidth, resonant frequency and Q factor.

SOFTWARE REQUIRED:

PSpice 9.1 Lite

APPARATUS REQUIRED:

S.No.	Name of the Components/Equipment	Type	Range	Quantity required
1	Function Generator	-	-	1
2	Resistor	-	100 Ω	1
3	Decade Inductance Box	-	-	1
4	Decade Capacitance Box	-	-	1
5	Ammeter	MI	(0-30) mA	1
6	Connecting Wires	Single strand	-	Few nos

THEORY:

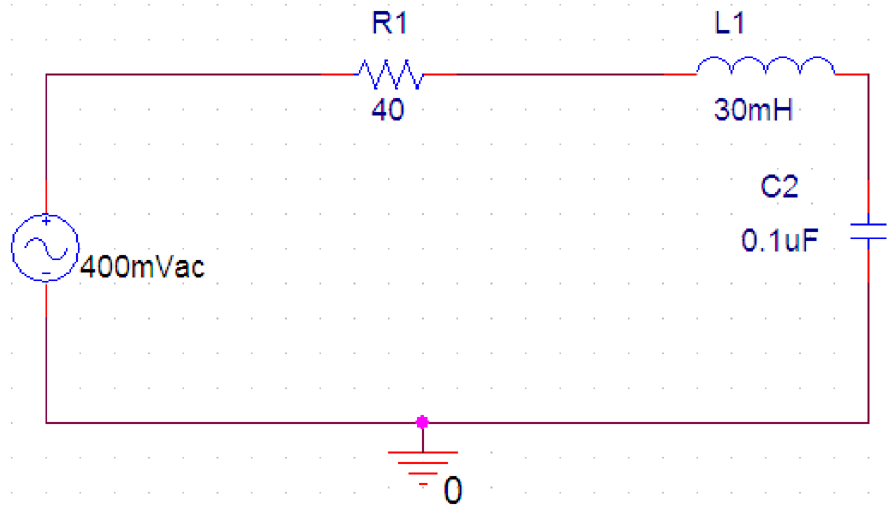
A circuit is said to be in resonance when applied voltage V and current I are in phase with each other. Thus at resonance condition, the equivalent complex impedance of the circuit consists of only resistance (R) and hence current is maximum. Since V and I are in phase, the power factor is unity.

The complex impedance

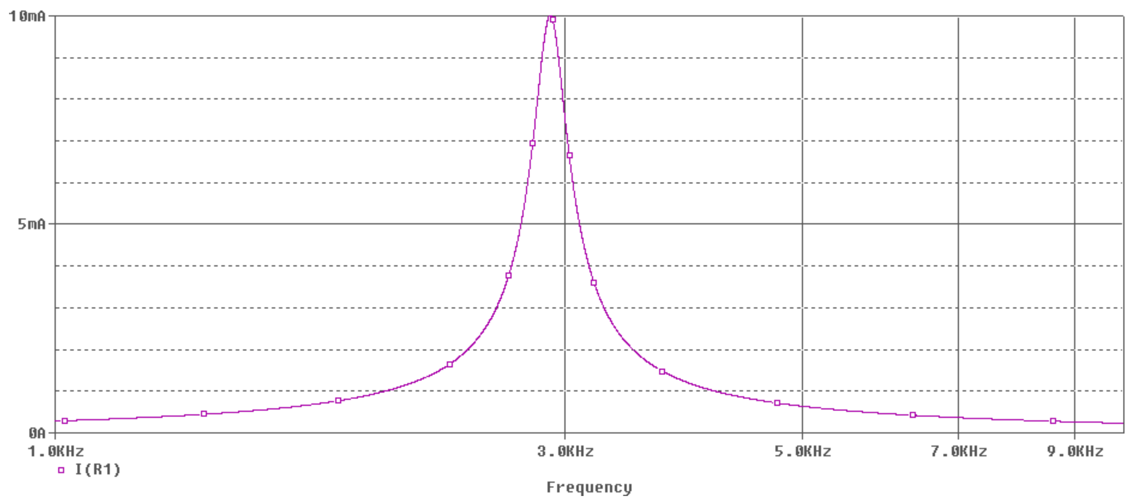
$$Z = R + j (X_L – X_C)$$

Where $X_L = \omega L$

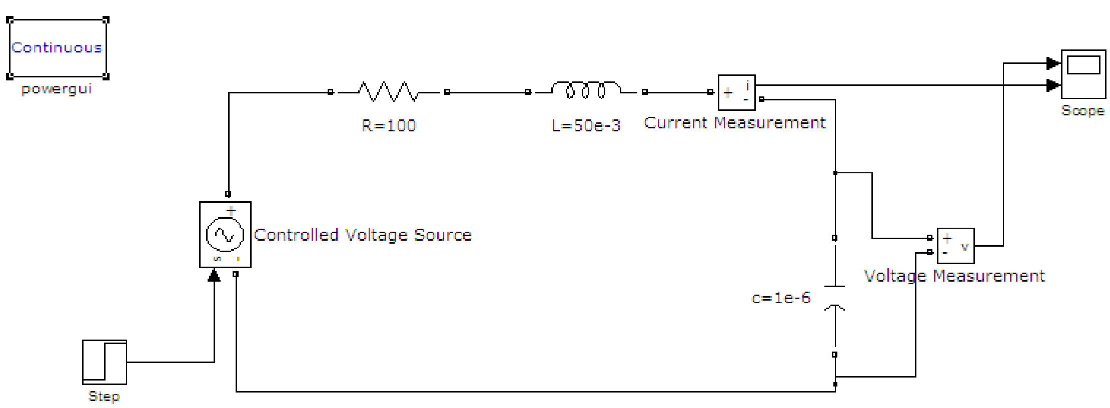
PSpice SIMULATION:



OUTPUT WAVWFORM:



MATLAB SIMULATION:



$X_C = 1/\omega C$

At resonance, $X_L = X_C$ 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 $f_2 - f_1$. f_1 is the lower cut off frequency and f_2 is the upper cut off frequency.

Q - FACTOR:

In the case of a RLC series circuit, Q-factor is defined as the voltage magnification in the circuit at resonance. At resonance, current is maximum. $I_o = V/R$.

The applied voltage $V = I_o R$

Voltage magnification = $V_L/V = I_o X_L$

In the case of resonance, high Q factor means not only high voltage, but also higher sensitivity of tuning circuit. Q factor can be increased by having a coil of large inductance, not of smaller ohmic resistance.

$Q = \omega L / R$

FORMULAE USED:

Resonant frequency $f_r = \frac{1}{2\pi\sqrt{LC}}$ Hz

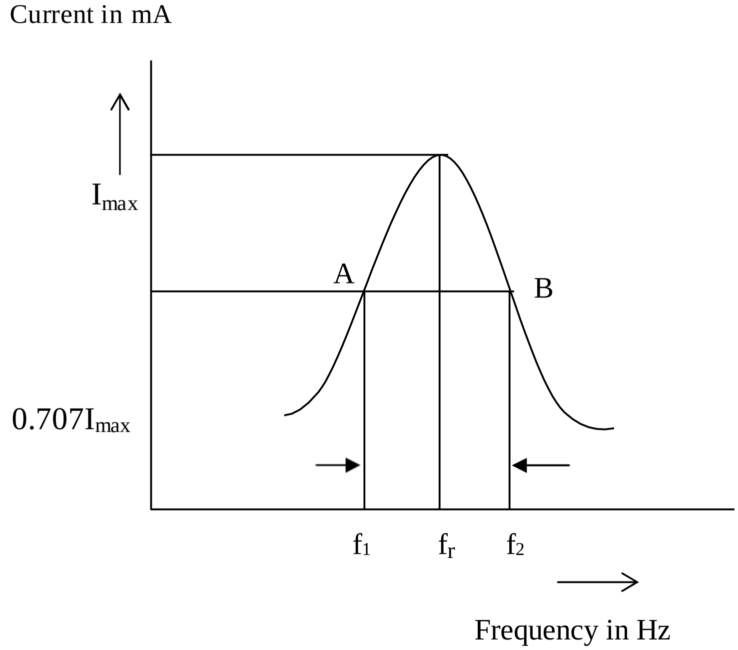
Bandwidth $BW = f_2 - f_1$

Quality Factor = $\frac{f_r}{BW}$

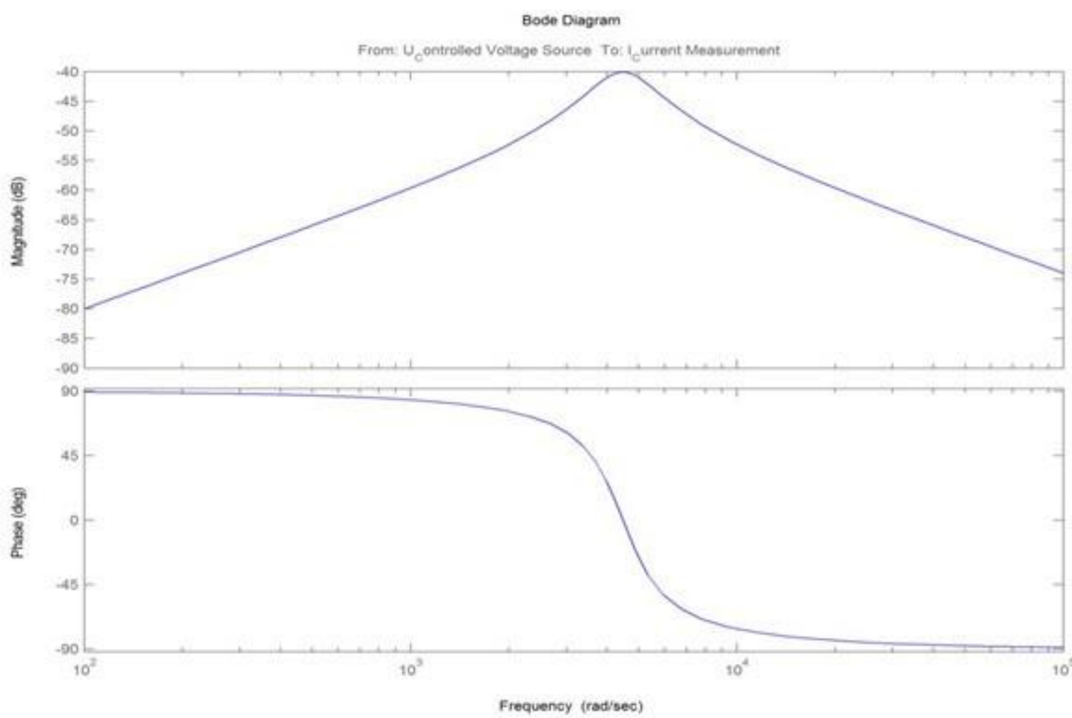
PROCEDURE:

1. Connect the circuit as per the circuit diagram.
2. Vary the frequency and note down the corresponding meter reading.
3. Draw the current Vs frequency curve and measure the bandwidth, resonant frequency and Q factor.

MODEL GRAPH FOR SERIES RESONANCE



PLOT OF MAGNITUDE & PHASE ANGLE OF CURRENT FOR VARIOUS FREQUENCIES:



SIMULATION PROCEDURE:

1. Open a new MATLAB/SIMULINK model or PSpice CAPTURE project.
2. Connect the circuit as shown in the figure.
3. Debug and run the circuit.
4. By double clicking the power gui plot the value of current for the different values of frequencies (for MATLAB Simulink).
5. For PSpice CAPTURE run the model create simulation profile and run the model.

VIVA QUESTIONS:

1. Define Bandwidth.
2. Define Quality factor.
3. What is meant by selectivity?
4. Give the significance of Q- factor.
5. What is meant by resonance?
6. What are the characteristics of a series resonant circuit?
7. What will be the power factor of the circuit at resonance?

RESULT:

Thus the current Vs frequency graphs of series and parallel resonant circuits were plotted and the bandwidth, resonant frequency and Q factor were measured. They were found to be

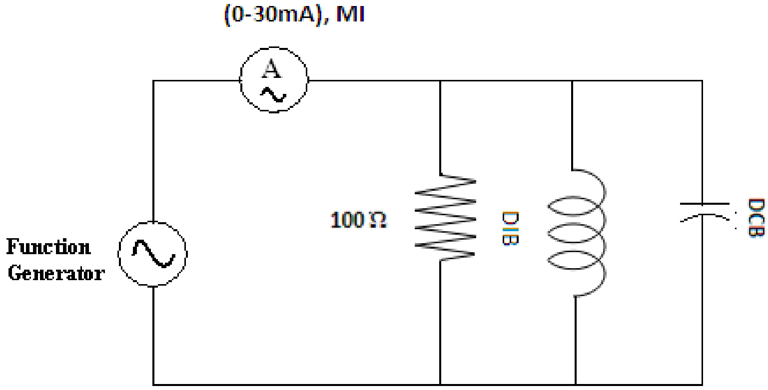
(a) Series resonance

Resonant frequency = ____________

Bandwidth = ____________

Q- Factor = ____________

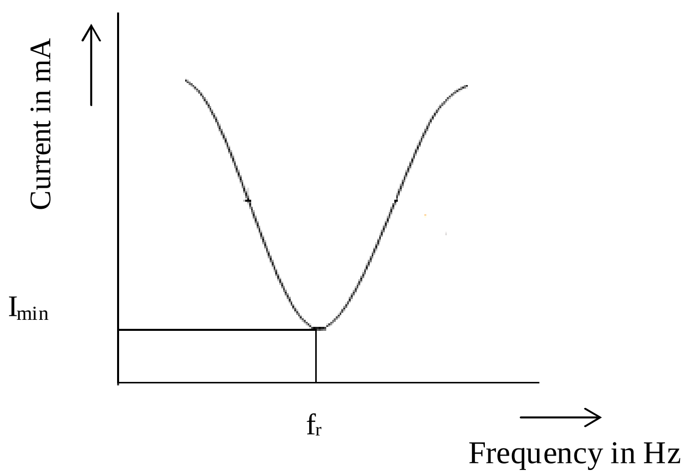
CIRCUIT DIAGRAM FOR PARALLEL RESONANCE:



OBSERVATION TABLE:

S.No.	Frequency in	Output

MODEL GRAPH FOR PARALLEL RESONANCE:



EXP NO. :

DATE :

DESIGN AND SIMULATION OF PARALLEL RESONANT CIRCUITS

AIM:

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

SOFTWARE REQUIRED:

Matlab 7.1 or PSpice 9.1 Lite

APPARATUS REQUIRED:

S.No.	Name of the Components/Equipment	Type	Range	Quantity required
1	Function Generator	-	-	1
2	Resistor	-	100 Ω	1
3	Decade Inductance Box	-	-	1
4	Decade Capacitance Box	-	-	1
5	Ammeter	MI	(0-30) mA	1
6	Connecting Wires	Single strand	-	Few nos

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

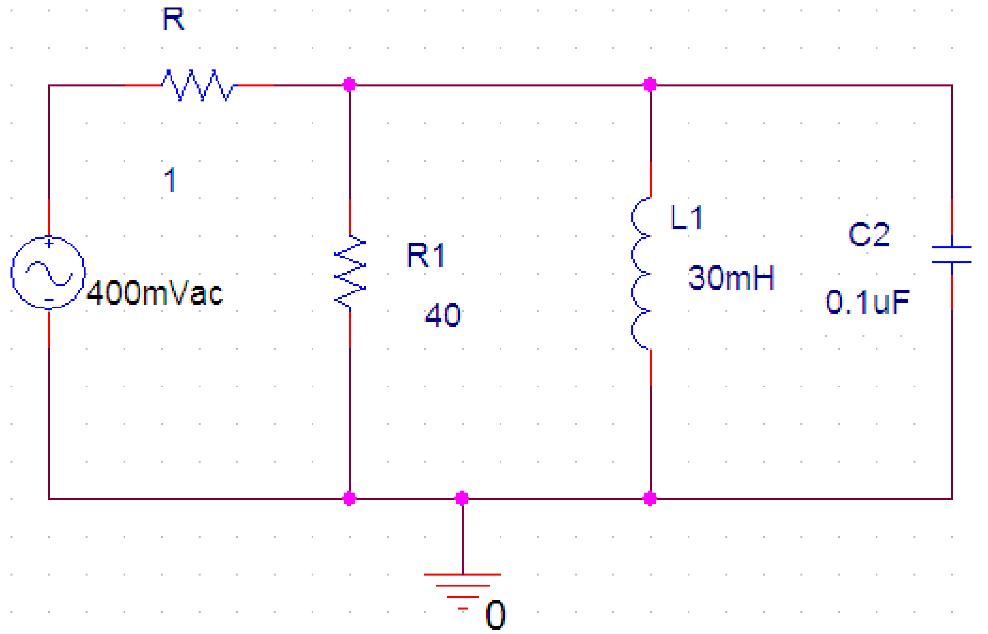
$Z = R + j (X_L – X_C)$

Where $X_L = \omega L$

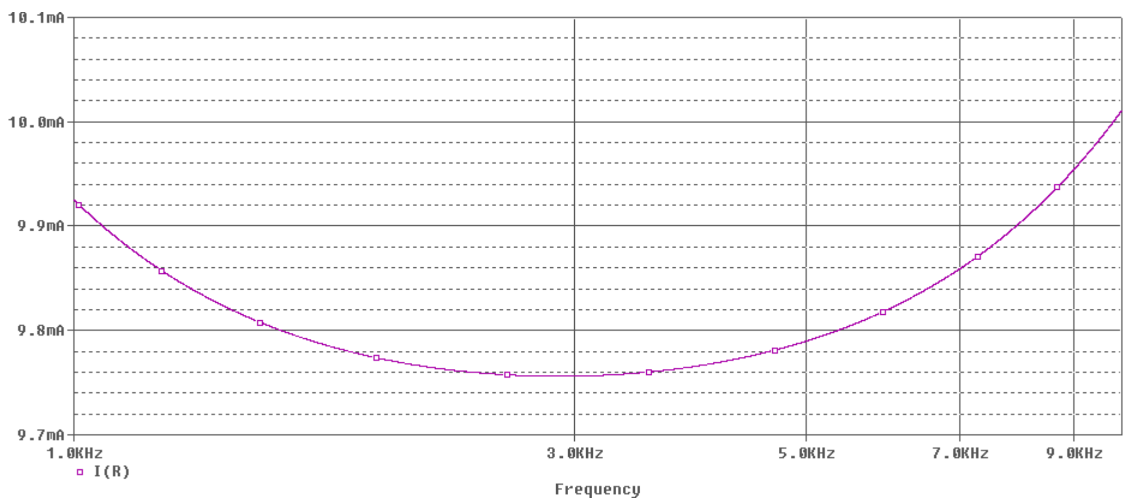
$X_C = 1/\omega C$

At resonance, $X_L = X_C$ and hence $Z = R$

PSpice SIMULATION:



OUTPUT WAVEFORM:



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 $f_2 - f_1$. f_1 is the lower cut off frequency and f_2 is the upper cut off frequency.

Q - FACTOR:

In the case of a RLC series circuit, Q-factor is defined as the voltage magnification in the circuit at resonance. At resonance, current is maximum. $I_o = V/R$.

The applied voltage $V = I_oR$

Voltage magnification = $V_L/V = I_oX_L$

In the case of resonance, high Q factor means not only high voltage, but also higher sensitivity of tuning circuit. Q factor can be increased by having a coil of large inductance, not of smaller ohmic resistance.

$Q = \omega L / R$

FORMULAE USED:

Resonant frequency $f_r = \dfrac{1}{2\pi\sqrt{LC}}$ Hz

Bandwidth BW = $f_2 - f_1$

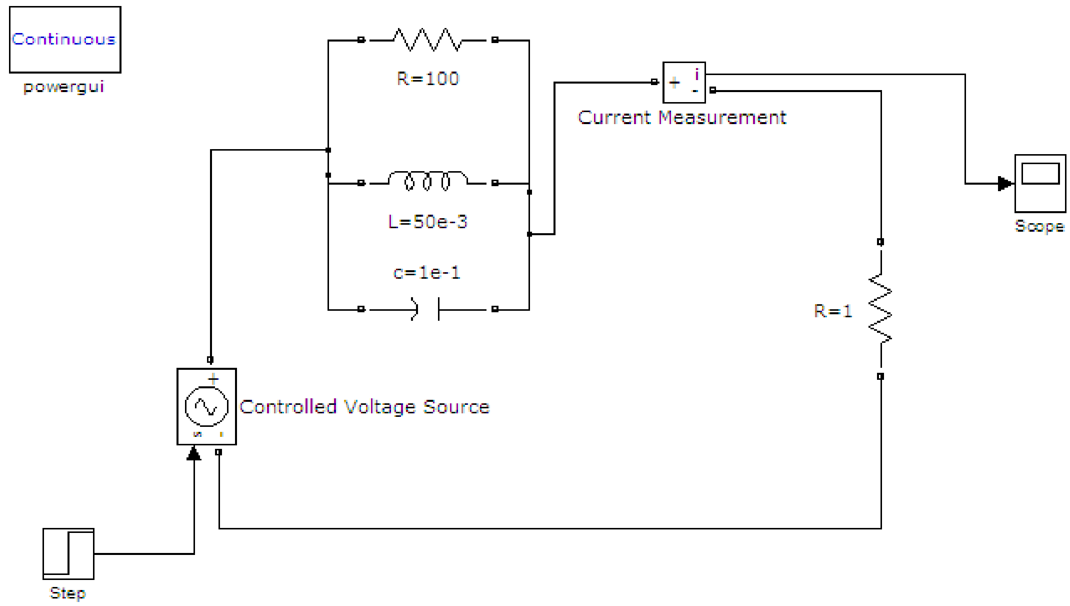
Quality Factor = $\dfrac{f_r}{BW}$

PROCEDURE:

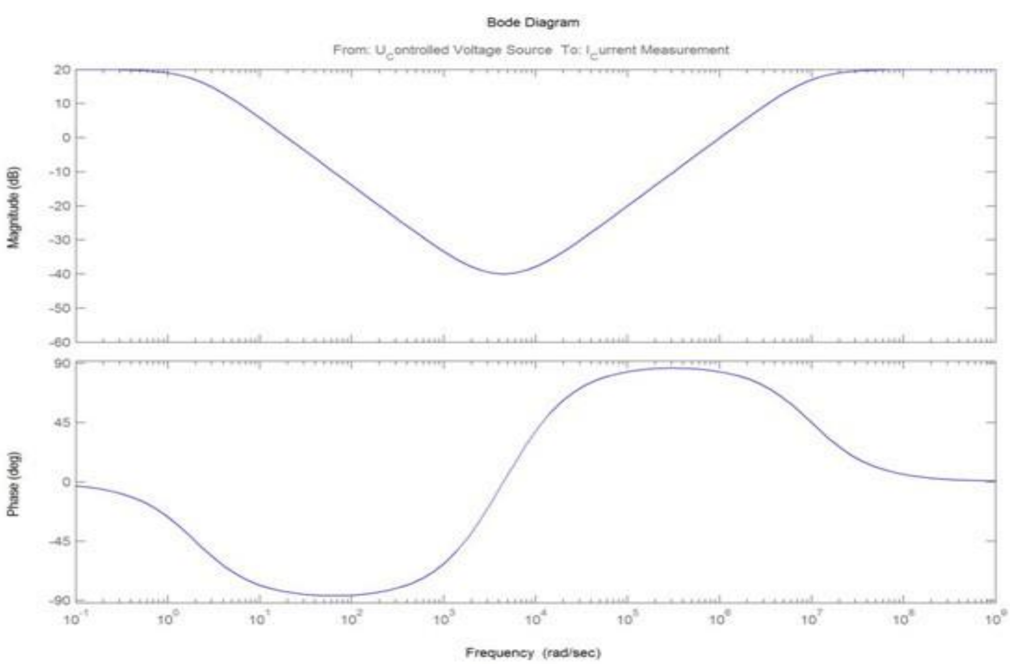
1. Connect the circuit as per the circuit diagram.
2. Vary the frequency and note down the corresponding meter reading.
3. Draw the current Vs frequency curve and measure the bandwidth, resonant frequency and Q factor.

.

MATLAB SIMULATION:



PLOT OF MAGNITUDE & PHASE ANGLE OF CURRENT FOR VARIOUS FREQUENCIES:



SIMULATION PROCEDURE:

1. Open a new MATLAB/SIMULINK model or PSpice CAPTURE project.
2. Connect the circuit as shown in the figure.
3. Debug and run the circuit.
4. By double clicking the power gui plot the value of current for the different values of frequencies (for MATLAB Simulink).
5. For PSpice CAPTURE run the model create simulation profile and run the model

VIVA QUESTIONS:

1. Define Bandwidth.
2. Define Quality factor.
3. What is meant by selectivity?
4. Give the significance of Q- factor.
5. What is meant by resonance?
6. What are the characteristics of a parallel resonant circuit?
7. What will be the power factor of the circuit at resonance?

RESULT:

Thus the current Vs frequency graphs of series and parallel resonant circuits were plotted and the bandwidth, resonant frequency and Q factor were measured. They were found to be

(a) Parallel resonance

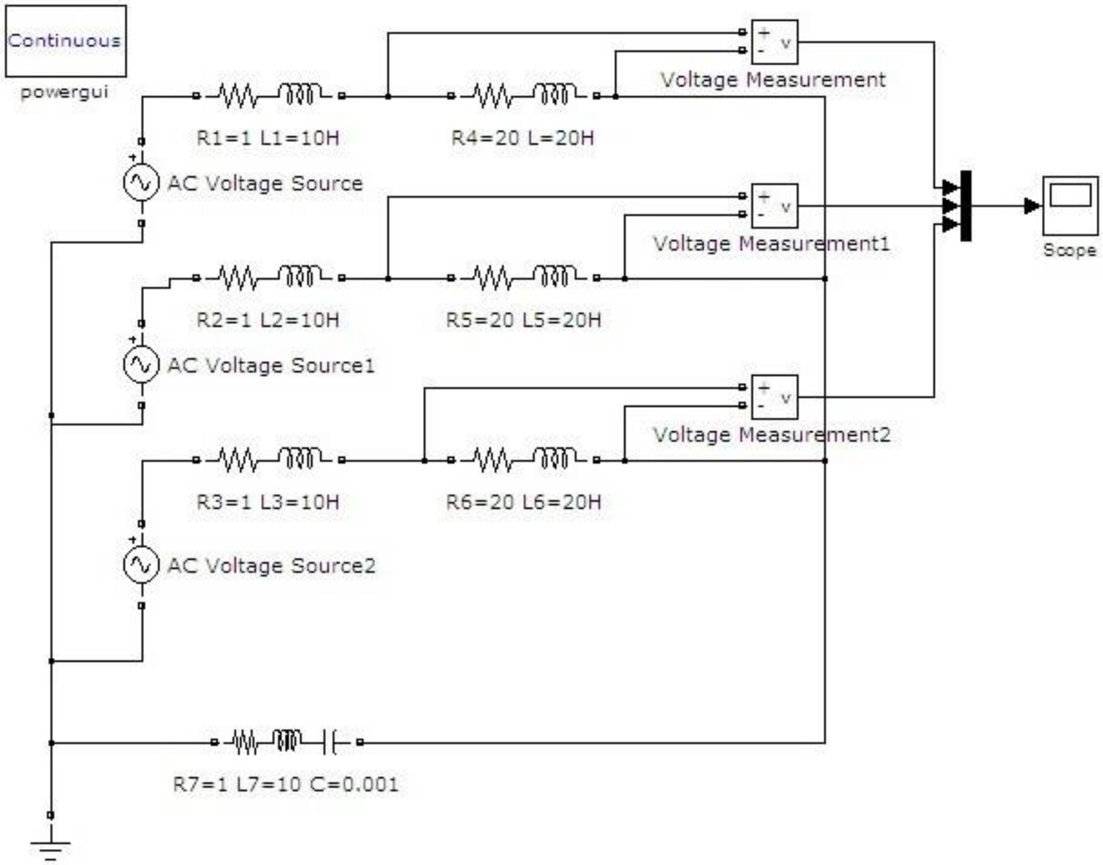
Resonant frequency = _____________

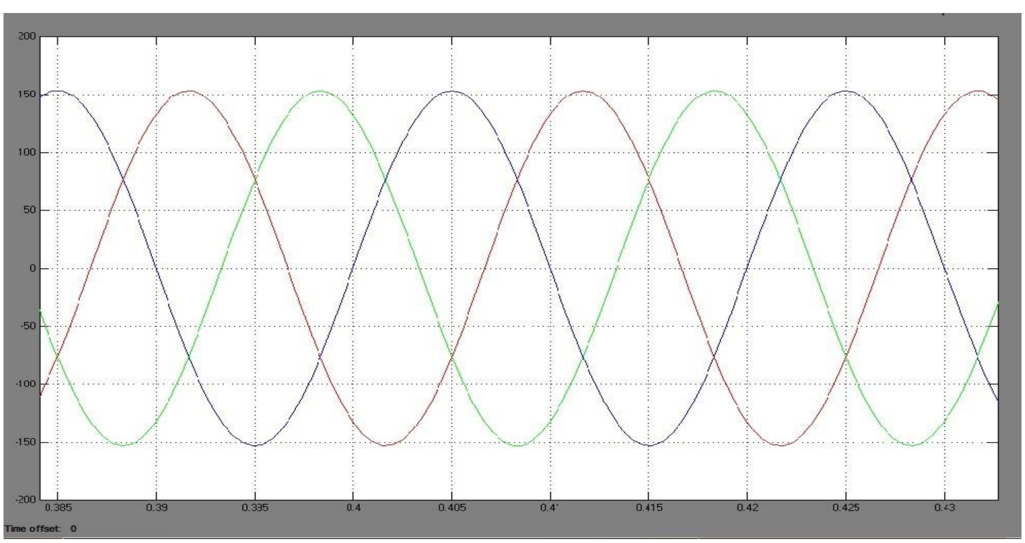
Bandwidth = _____________

Q- Factor = _____________

SIMULATTION DIAGRAM:

3 Φ BALANCED STAR CONNECTED NETWORK:





EXP NO.:

DATE :

SIMULATION OF THREE PHASE BALANCED AND UNBALANCED STAR, DELTA NETWORKS CIRCUITS

AIM:

To simulate three phase balanced and unbalanced star, delta networks circuits.

SOFTWARE REQUIRED:

Matlab 7.1

THEORY:

BALANCED THREE- PHASE CIRCUIT:

Balanced phase voltages are equal in magnitude and are out of phase with each other by 120°.The phase sequence is the time order in which the voltages pass through their respective maximum values. A balanced load is one in which the phase impedances are equal in magnitude and in phase.

POSSIBLE LOAD CONFIGURATIONS:

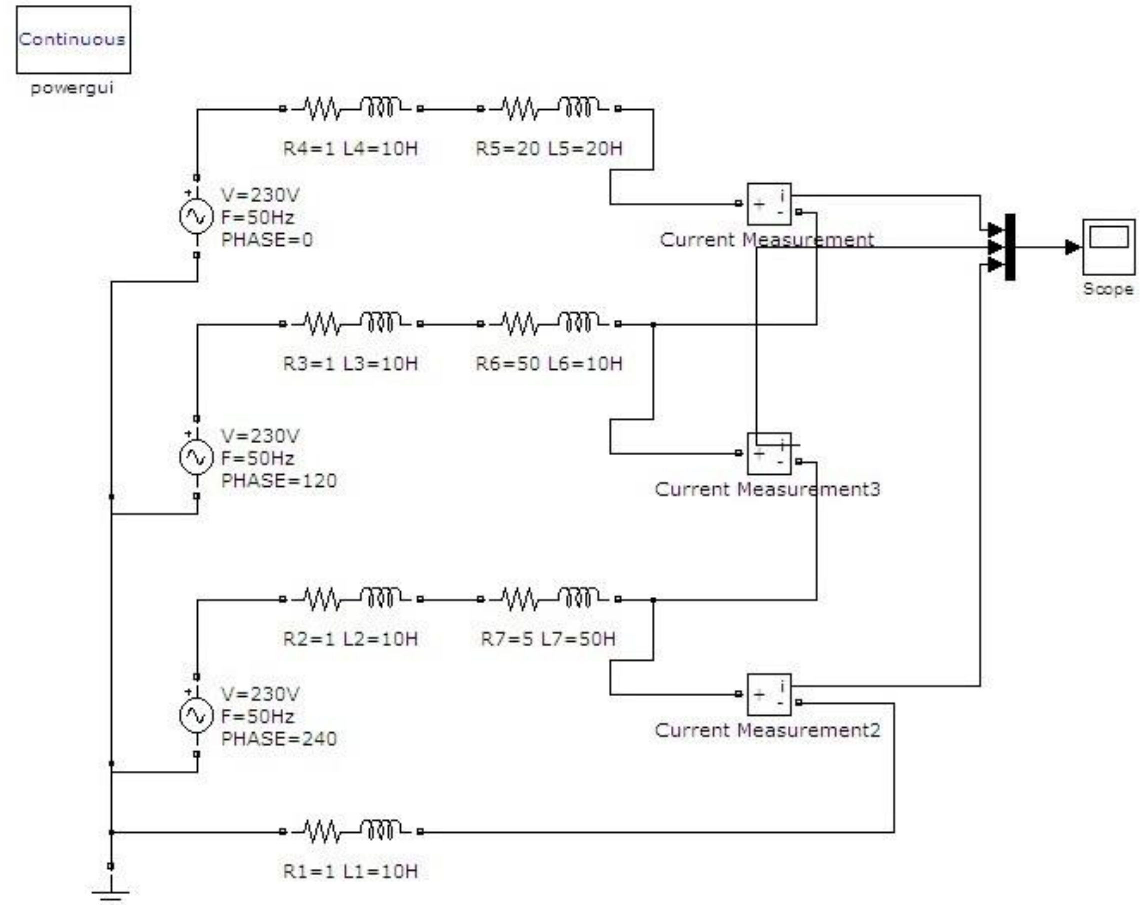
Four possible connections between source and load:

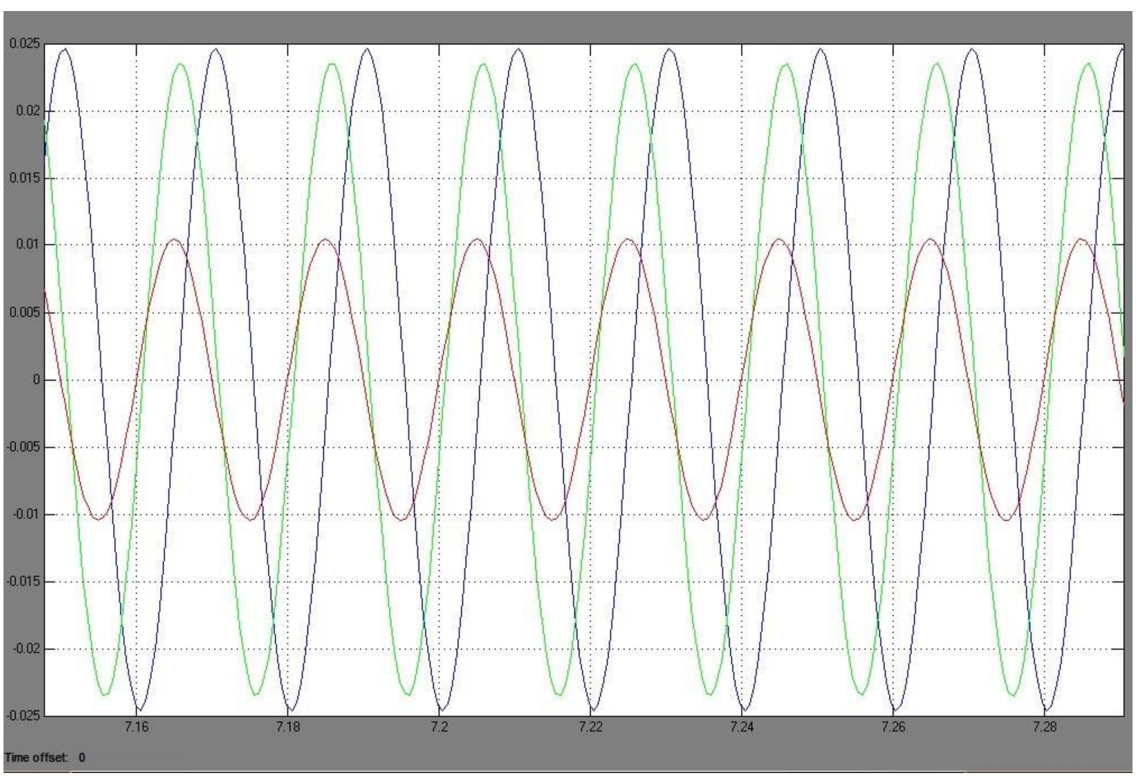
1. Y-Y connection (Y-connected source with a Y-connected load)
2. Y-Δ connection (Y-connected source with a Δ-connected load)
3. Δ-Δ connection
4. Δ-Y connection

UNBALANCED THREE- PHASE CIRCUIT:

An unbalanced system is due to unbalanced voltage sources or an unbalanced load. To calculate power in an unbalanced three-phase system requires that we find the power in each phase. The total power is not simply three times the power in one phase but the sum of the powers in the three phases.

3 Φ UNBALANCED DELTA CONNECTED NETWORK:





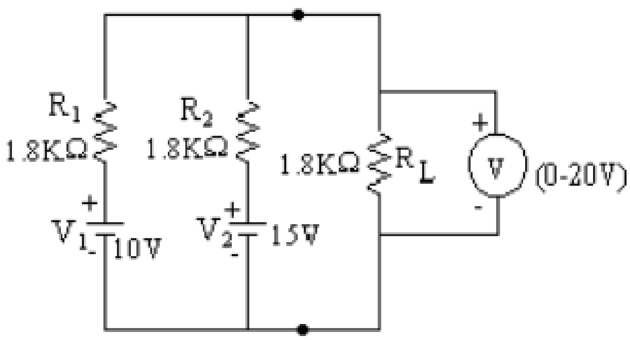
VIVA QUESTIONS:

1. What do you meant by balanced circuit?
2. List the possible load configuration?
3. What is mean by unbalanced circuit?

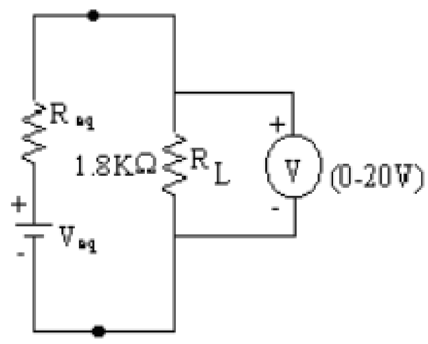
RESULT:

Thus the three phase balanced and unbalanced star, delta network circuits were simulated and verified.

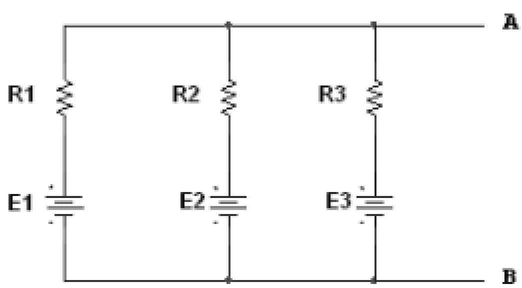
CIRCUIT DIAGRAM:

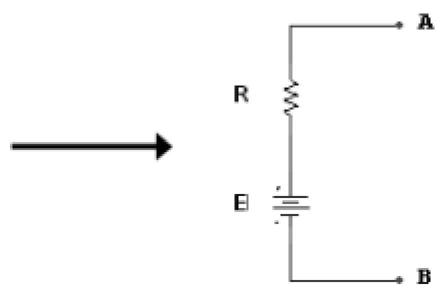


CIRCUIT-1



CIRCUIT-2





EXP NO.:

DATE :

SIMULATION AND EXPERIMENTAL VERIFICATION OF MILLMAN'S THEOREM

AIM:

To verify millman's theorem.

APPARATUS REQUIRED:

SLNO	NAME OF ITEM	SPECIFICATION	QUANTITY
1.	Resistor		
2.	Dc power supply		
3.	Voltmeter		
4.	Ammeter		
5.	Wires		
6	Bread board		

SOFTWARE REQUIRED:

Matlab 7.1

MILLMAN'S THEOREM:

STATEMENT:

The utility of this theorem is that, any number of parallel voltage sources can be reduced to one equivalent one. When a number of voltage sources (V_1,V_2,........V_n) are in parallel having internal resistances(R_1,R_2,.........R_n) respectively, the arrangement can be replaced by a single equivalent voltage source V in series with an equivalent series resistance R.

$$V = \frac{\pm V_1 G_1 \pm V_2 G_2 \pm \ldots\ldots\ldots \pm V_n G_n}{G_1 + G_2 + \ldots\ldots . + G_n} \quad \text{and}$$

$$R = \frac{1}{G} = \frac{1}{G_1 + G_2 + ,,,,,,, + G_n}$$

OBSERVATION TABLE:

S.NO	V_1	V_2	PRACTICAL	THEORETICAL
			V_{L1}	V_{L1}

MODEL CALCULATION:

PROCEDURE:

1. Connect the circuit as shown Fig-1 and Note down the reading of voltmeter as V_{L1} .
2. Connect the equivalent circuit as shown in Fig-2 , by calculating
3. Veq=(V1G1 +V2G2)/ (G1 + G2) and R eq= 1/ (G1+G2)
4. Note down the reading of the voltmeter as $V_{L2.}$
5. If V L1 = V L2, the Milliman's Theorem is verified.
6. Verify with theoretical values.

SIMULATION PROCEDURE:

1. Open a new MATLAB/SIMULINK model
2. Connect the circuit as shown in the figure
3. Debug and run the circuit
4. For different input voltages, record the voltage and verify with theoretical values.

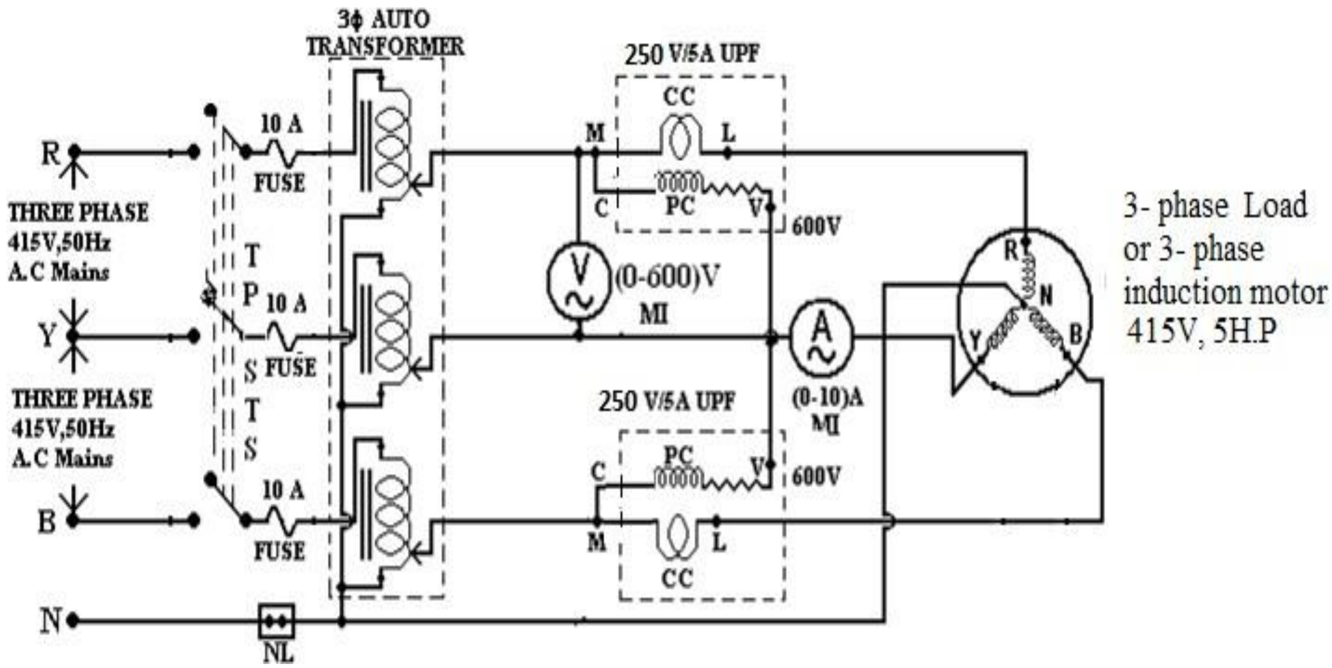
VIVA QUESTIONS:

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?
4. State application of Millman's theorem

RESULT:

Thus the Millman's theorem was verified.

CIRCUIT DIAGRAM:



EXP NO.:

DATE :

EXPERIMENTAL DETERMINATION OF POWER IN THREE PHASE CIRCUITS BY TWO-WATT METER METHOD

AIM:

To determine the power in three-phase balanced and unbalanced circuit using two-watt meter method.

APPARATUS REQUIRED:

SLNO	NAME OF ITEM	SPECIFICATION	QUANTITY
1.	3-phase Auto transformer	20 Amp. 440v 50 Hz	1
2.	Ammeter	MI(0-10A)	1
3.	Voltmeter	MI(0-600V)	1
4.	Wattmeter	250v, 5A	2
5.	3- phase Load or 3- phase induction motor	415V, 5H.P	1
6	Connecting wires	-	Few

THEORY:

Two wattmeter method can be employed to measure power in a 3- phase,3 wire star or delta connected balance or unbalanced load. In this method, the current coils of the watt meters are connected in any two lines say R and Y and potential coil of each watt meters is joined across the same line and third line i.e. B. Then the sum of the power measured by two watt meters W1 and W2 is equal to the power absorbed By the 3- phase load

PROCEDURE:

1. Connect the Voltmeter, Ammeter and Watt meters to the load through 3ϕ Auto transformer as shown fig and set up the Autotransformer to Zero position.
2. Switch on the 3ϕ A.C. supply and adjust the autotransformer till a suitable voltage.

OBSERVATION TABLE:

S. No	Voltmeter reading V_L	Ammeter reading I_L	Wattmeter reading (watts)				Total power P	Reactive power Q	Power factor
	(V)	(A)	W_1 observed	W_1 Actual	W_2 Observed	W_2 actual	(watts)	(watts)	

MODEL CALCULATION:

3. Note down the readings of watt meters, voltmeter& ammeter
4. Vary the voltage by Autotransformer and note down the Various readings.
5. Now after the observation switch off and disconnect all the Equipment or remove the lead wire.

FORMULAE USED:

1. Total power or Real power P = $\sqrt{3}V_LI_LCOS\phi$ =$W_{1actual}+W_{2actual}$
2. Reactive power of load= Q=$\sqrt{3}(W_{1actual}-W_{2actual})$
3. tan ϕ= $[\sqrt{3}(W_{1actual}-W_{2actual})]/[W_{1actual}+W_{2actual}]$
4. Power factor=cos ϕ

PRECAUTION & SOURCES OF ERROR:

1. Proper currents and voltage range must be selected before putting the instruments in the circuit.
2. If any Wattmeter reads backward, reverse its pressure coil connection and the reading as negative.
3. As the supply voltage Fluctuates it is not possible to observe the readings correctly.

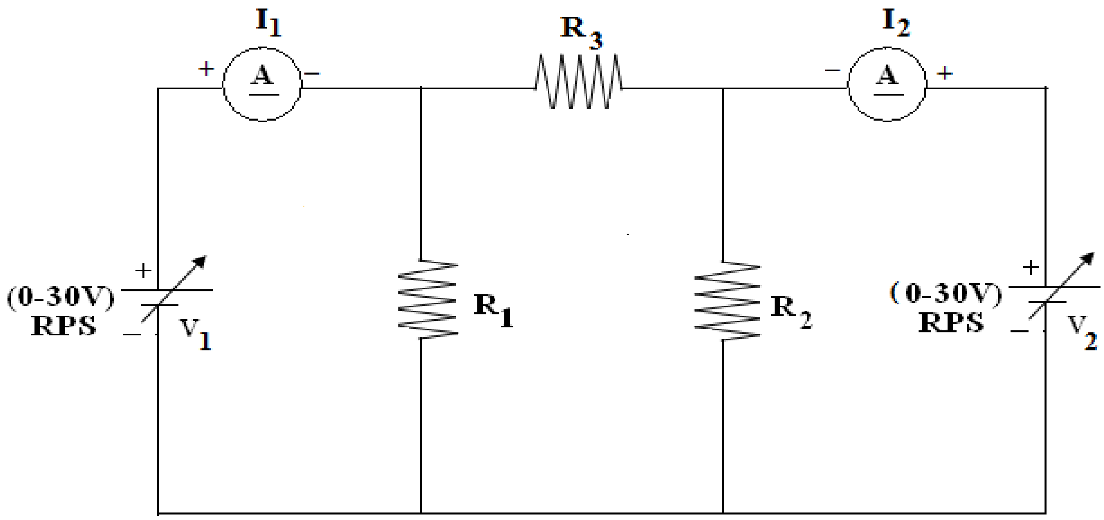
VIVA QUESTIONS:

1. What are the various types of wattmeter?
2. How many coils are there in wattmeter?
3. What is meant by real power?
4. What is meant by apparent power?

RESULT:

The power measured in the 3-phase circuit and there corresponding power factors are in observation table.

CIRCUIT DIAGRAM FOR TWO- PORT NETWORK



OBSERVATION TABLE:

S.No	V_1 (Volts)	V_2 (Volts)	I_1 (Amps)	I_2 (Amps)	Z_{11} (Ohms)	Z_{12} (Ohms)	Z_{22} (Ohms)	Z_{21} (Ohms)

EXP NO.:

DATE :

DETERMINATION OF TWO PORT NETWORK PARAMETERS

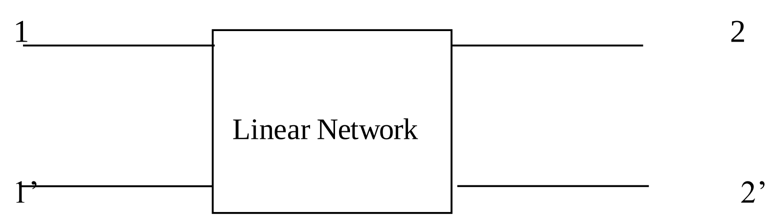
AIM:

To determine the two port parameters for the given electric circuit.

APPARATUS REQUIRED:

S.NO	Name of the Apparatus/Component	Range	Type	Quantity
1	Ammeter		MC	
2	Power Supply			
3	Voltmeter		MC	
4	Connecting wires			
5	Resistors			
6	Breadboard			

THEORY:



The terminal pair where the signal enters the network is called as the INPUT PORT and the terminal pair where it leaves the network is called as the OUTPUT PORT. V_1& I_1 are measured at the Input terminals and V_2 & I_2 are measured at the Output terminals. The two port network parameters express the inter relationship between V_1, I_1,V_2 and I_2 They are Z- parameters, Y- parameters, H-parameters, ABCD parameters and image parameters.

MODEL CALCULATION:

The Impedance parameters are also called as Z parameters.

$$V_1 = Z_{11}I_1 + Z_{12}I_2 \qquad \text{(i)}$$

$$V_2 = Z_{21}I_1 + Z_{22}I_2 \qquad \text{(ii)}$$

where, Z_{11}, Z_{12}, Z_{22} and Z_{21} are constants of the network called Z parameters.

When $I_2=0$, (Open circuit the output terminal)

$$Z_{11}=V_1/I_1 \qquad \text{(iii)}$$

$$Z_{21}=V_2/I_1 \qquad \text{(iv)}$$

When $I_1=0$, (Open circuit the Input terminal)

$$Z_{12}=V_1/I_2 \qquad \text{(v)}$$

$$Z_{22}=V_2/I_2 \qquad \text{(vi)}$$

PROCEDURE:

1. Connect the circuit as per the circuit diagram.
2. Open circuit the output terminal (2,2’).
3. Vary the power supply to a fixed value and note down the ammeter and voltmeter readings.
4. Open circuit the Input terminal (1,1’).
5. Vary the power supply to a fixed value and note down the ammeter and voltmeter readings.
6. Tabulate the readings and calculate the Z parameters.

VIVA QUESTIONS:

1. What is meant by a two-port network?
2. Give the use of two-port network model.
3. What are impedance parameters?
4. What are admittance parameters?
5. What are hybrid parameters?
6. What are ABCD parameters? Mention their significance.

RESULT:

Thus the two port parameters are measured for the given electric circuit.