DEPARTMENT OF ELECTRONICS & COMMUNICATION ENGINEERING

Network Analysis Laboratory – 23EC3251 Manual

PVP23

Student Name:……………………………………

Roll No:……………………………………….

Branch:……………………..Section……………

Year …………………Semester………………..

PRASAD V POTLURI SIDDHARTHA INSTITUTE OF TECHNOLOGY

(Autonomous, Accredited by NBA & NAAC, an ISO 9001:2015 certified institution) (Sponsored by Siddhartha Academy of General & Technical Education) VIJAYAWADA – 520 007

Network Analysis Laboratory

Course Code	23EC3251	Year	I	Semester	II
Course Category	Engineering Sciences	Branch	ECE	Course Type	Lab
Credits	1.5	L-T-P	0-0-3	Prerequisites	Nil
Continuous Internal Evaluation:	30	Semester End Evaluation:	70	Total Marks:	100

Course Outcomes		
	Upon successful completion of the course, the student will be able to	BL
CO1	Verify Kirchhoff's laws and network theorems and measure signal parameters.	L2
CO2	Measure time constants of RL & RC circuits	L3
CO3	Analyze behavior of RLC circuit for different cases	L4
CO4	Design resonant circuit for given specifications	L4
CO5	Characterize and model the network in terms of all network parameters	L4
CO6	Communicate concepts and technologies related to network analysis lab effectively in written reports.	L3

Mapping of course outcomes with Program outcomes (CO/PO/PSO Matrix)

Note:1-Weak correlation 2-Medium correlation 3-Strong correlation

*-Average value indicates course correlation strength with mapped PO

COs	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12	PSO1	PSO2
CO1	3				3				1				1	1
CO2		2			2				1				1	1
CO3		1			1				1				1	1
CO4		1			1				1				1	1
CO5		2			2				1				1	1
CO6										3				
Average * (Rounded to nearest integer)	3	2			2				1	3			1	1

Syllabus		
Expt. No.	**Contents**	**Mapped CO**
I	Study of components of a circuit and verification of KCL and KVL	CO1, CO6
II	Measurement of sinusoidal voltage, frequency, effective and average values using CRO	CO1, CO6
III	Verification of mesh analysis for an electrical circuit.	CO1, CO6
IV	Verification of nodal analysis for an electrical circuit.	CO1, CO6
V	Verification of superposition theorem.	CO1, CO6
VI	Verification of Thevenin's & Norton's theorems.	CO1, CO6
VII	Verification of maximum power transfer theorem.	CO1, CO6
VIII	Study of DC transients (time response) in RL, RC and RLC circuits.	CO2, CO6
IX	Study frequency response (steady state) of RL & RC networks.	CO2, CO6
X	Determine the steady state response of a 2^{nd} order circuit.	CO3, CO6
XI	Determine the Q factor and Bandwidth of a resonance circuit.	CO4, CO6
XII	Determination of open circuit (Z) and short circuit (Y) parameters	CO5, CO6
XIII	Determination of hybrid (h) and transmission (ABCD) parameters	CO5, CO6

Learning Resources
1. M.E Van Valkenburg, Network Analysis, Prentice Hall of India, Revised 3rd Ed., 2019. 2. William H. Hayt, Jack Kemmerly, Jamie Phillips,Steven M. Durbin, Engineering Circuit Analysis, Mc Graw Hill 9th Ed., 2020
Hardware Requirements
Regulated Power supplies, Analog/ Digital Function Generators, Digital Multimeters, Decade Resistance Boxes/ Rheostats, Decade Capacitance Boxes, Ammeters (Analog or Digital), Voltmeters (Analog or Digital), Active & Passive Electronic Components
Software Requirements
Multisim/ Pspice /Equivalent simulation software tool, Computer Systems with required specifications
e- Resources & other Digital Material
1. http://www.cdeep.iitb.ac.in/nptel/Electrical%20&%20Comm%20Engg/Signals%20and%20System/TOC-M1.htm 2. http://www.cdeep.iitb.ac.in/nptel/Electrical%20&%20Comm%20Engg/Signals%20and%20System/Course%20Objective.htm. 3. http://www.stanford.edu/~boyd.ee102 4. http://www.ece.gatech.edu/users/bonnie/book 5. http://ocw.mit.edu 6. https://www.youtube.com/playlist?list=PLC7D3EAEFA0CC0420&app=desktop 7. https://www.tutorialspoint.com/network_theory/network_theory_quick_guide.htm 8. https://nptel.ac.in/courses/108/105/108105159/

LABORATORY INSTRUCTIONS

1. While entering the Laboratory, the students should follow the dress code.
2. The students should bring their observation book/record, calculator, necessary stationery items and graph sheets if any for the lab classes without which the students will not be allowed for doing the experiment.
3. All the Equipment and components should be handled with utmost care.
4. If any damage or breakage is noticed, it should be reported to the concerned in charge immediately.
5. Record/observation must be submitted after completion of experiment on the same day.
6. 100% attendance should be maintained for the laboratory classes.

Precautions:

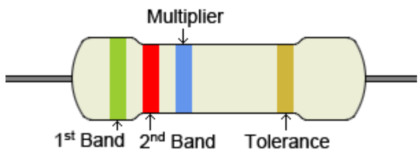
1. Check the connections before giving the supply.
2. Observations should be done carefully.

INDEX

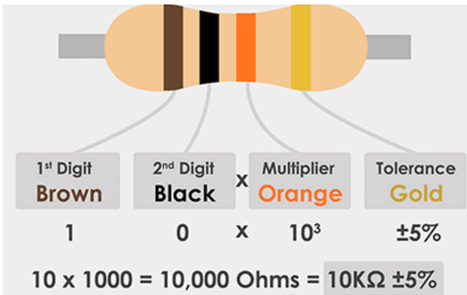
Expt. No.	Date	Contents	Marks	Sign of the faculty
1		Study of components of a circuit and verification of KCL and KVL		
2		Measurement of sinusoidal voltage, frequency, effective and average values using CRO		
3		Verification of mesh analysis for an electrical circuit.		
4		Verification of nodal analysis for an electrical circuit.		
5		Verification of superposition theorem.		
6		Verification of Thevenin's & Norton's theorems.		
7		Verification of maximum power transfer theorem.		
8		Study of DC transients (time response) in RL, RC and RLC circuits.		
9		Study frequency response (steady state) of RL & RC networks.		
10		Determine the steady state response of a 2nd order circuit.		
11		Determine the Q factor and Bandwidth of a resonance circuit.		
12		Determination of open circuit (Z) and short circuit (Y) parameters		
13		Determination of hybrid (h) and transmission (ABCD) parameters		
14		Verification of Reciprocity Theorem		
15		Verification of Millman's Theorem		

Identification: -

Color Coded Resistor (Carbon composition resistor)



Example:



By using the color chart, write the values as, $10x10^3$ = 10K ohm/10K.

The tolerance is ±5% for Gold.

±5% of 10K ohm = (10,000/100)5 = 500

Range = 10KΩ±500= (10,000-500) Ω to (10,000+500) Ω =9,500Ω to 10,5000Ω

So, the value ranges from **9.5KΩ to 10.5KΩ.**

Observations:

S.NO	1st band color	2nd band color	3rd band color	4th band color	5th band color	Actual resistance value (Ω)	Min. resistance value (Ω)	Max. resistance value (Ω)	DMM readin g (Ω)

Expt. No: 1a	**Study of components of a circuit**
Dt:	

AIM: Identification, specification, testing of R, L, C components (Color codes).

APPARATUS REQUIRED:

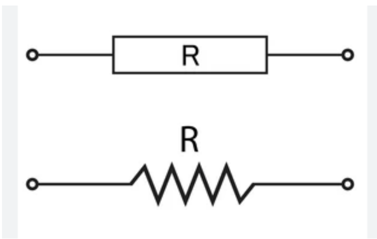
S.No	Name of the equipment	Range	Type	Quantity
1	Resistors			5
2	Capacitors			5
3	Digital Multi Meter (DMM)			1

THEORY:

1. RESISTOR:

Resistor is an electronic component whose function is to limit the flow of current in an electric circuit. It is measured in units called ohms. The symbol for ohm is **Ω** (omega). They are available in different values, shapes and sizes. Every material has some resistance. Some materials such as Rubber, Glass and air have very high opposition to current to flow. These materials are called insulators. Other materials such as Copper, Silver and Aluminum etc, has very low resistance, they are called Conductors.

Symbol:

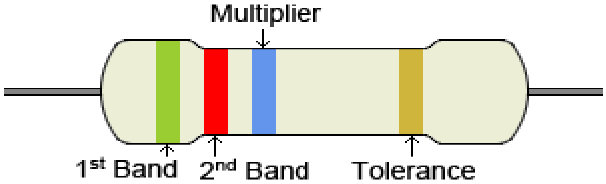


Color codes

The most common type has color bands to indicate its resistance. The code is a standard one adopted by manufacturers through their trade association, the Electronic Industries Association (EIA).

Color code and its value, multipliers, Tolerance

- **First** color band tells the first significant digit of the resistors value.
- **Second** color band indicates the second significant digit in the resistors value.
- **Third** color band indicates the no. of zeros to be appended for the first two significant numbers often called as multiplier.
- **Fourth** color band indicates the tolerance. Estimation of resistance value using color code
- Tolerance is the acceptable deviation or the actual value of the resistor may be 5% more or less than the coded value.
- If the resistor contains 5 color bands, then the first three color bands indicates the first, second and third significant digits in the resistors value, the fourth color band is the multiplier and the fifth color band indicates tolerance.



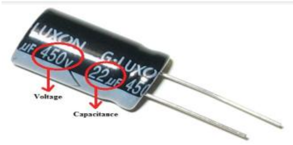
Color	1st, 2nd Band Significant Figures	Multiplier	Tolerance
Black	0	× 1	
Brown	1	× 10	±1% (F)
Red	2	× 100	±2% (G)
Orange	3	× 1K	±0.05% (W)
Yellow	4	× 10K	±0.02% (P)
Green	5	× 100K	±0.5% (D)
Blue	6	× 1M	±0.25% (C)
Violet	7	× 10M	±0.1% (B)
Grey	8	× 100M	±0.01% (L)
White	9	× 1G	
Gold		× 0.1	±5% (J)
Silver		× 0.01	±10% (K)

Specifications:

Carbon composition resistors are available from few ohms to several mega ohms. Typical resistor wattage sizes are 1/8, 1/4, 1/2, 1, 2, 5, 10 and 20 (w) units, depending on **thickness of** leads Wattage of resistors can be decided.

Reading capacitor values on Large capacitor (cylindrical capacitors)

- For large capacitors, generally capacitor value is written on the side of the capacitor.
- The above figure shows a 22 micro farad capacitor.
- Capacitance value is expressed in Farads (F or FD).
- Here are the units used for representing capacitor value.



μF = micro farads = 10^{-6}F , pF = pico farads = 10^{-12}F, nF = nano farads = 10^{-9}F

- **Voltage rating** on the capacitor indicates the maximum value voltage that capacitor can handle.

Reading the values of small capacitors (ceramic capacitors)

- Ceramic capacitors have very small area to print the value of capacitance.
- So capacitance on these capacitors is represented using short hand notation.
- Let us see how to calculate these values. Generally, capacitance of ceramic, tantalum, film capacitors is expressed in Pico Farad.

Step 1: If the capacitor has two numerical values.

- If the notation on the capacitor has 2 digits and a letter (like 22M), then it has capacitance value of 22.
- Some capacitors have letters in the second position and numerical value in first position.

Ex: 5R2 = 5.2PF.

- In place of R if the letters like p, n, u are present then they represent units of capacitance.

Ex: 4n1 = 4.1nF, p45=0.45pF

Step 2: Some of them have three numerical values.

- Capacitor shown above has notation 104 on it.
- Capacitance is calculated as 10x 10^4 = 10^5pf = 0.1uf
- If the third digit is between 0 to 7 follow the above procedure.
- If it is 8 multiply it by using 0.01. For example, 158 =15×0.01=0.15pF
- If it is 9 multiply it by using 0.1. For example, 159 =15×0.1 =1.5pF

Observations:

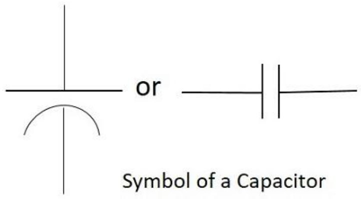
S.No	Capacitor type	Observed capacitance value	Measured capacitance value (using DMM)

2.CAPACITORS:

It is a device which stores a charge. It does not pass direct current (dc) but will effectively allow the flow of alternating current (ac). The reactance of a capacitor 'X_c' is dependent on the frequency of the ac signal and is given by $\mathbf{X_{C}=\frac{1}{2\pi fc}}$

A capacitor 'c' when charged to a voltage has a stored energy of $\frac{1}{2}CV^2$ Joules. A capacitor essentially consists of two conducting plates separated by a dielectric medium.

Symbol:



Symbol of a Capacitor

Capacitance of a capacitor 'c' is given by $C = \frac{\epsilon A}{d}$ F

The SI unit of capacitance is **Farad (F)**

The Farad is the capacitance of a capacitor that contains a charge of 1 Coulomb when the potential difference between its terminals is 1Volt and it stores energy, capacitor does not pass Direct current but allows the flow of alternating current.

Identification:

Capacitors dielectric is largely responsible for determining its most important characteristics. Hence capacitor is usually identified by the type of dielectric used.

Ex: Air capacitors, mica capacitor, ceramic capacitor, plastic film capacitor, electrolytic capacitor, and tantalum capacitors.

There are two types of capacitors

1) Electrolytic 2) non-electrolytic

The electrolytic capacitors use insulation (dielectric) which is chemically active. The capacitor is marked with a +ve & -ve lead polarity.

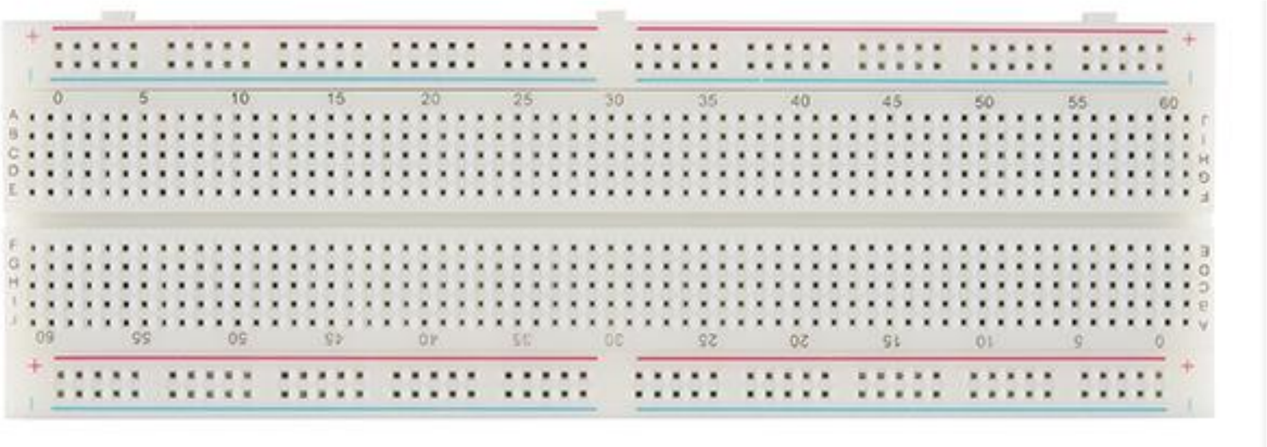
Non electrolytic capacitors can be connected to the circuit with any polarity.

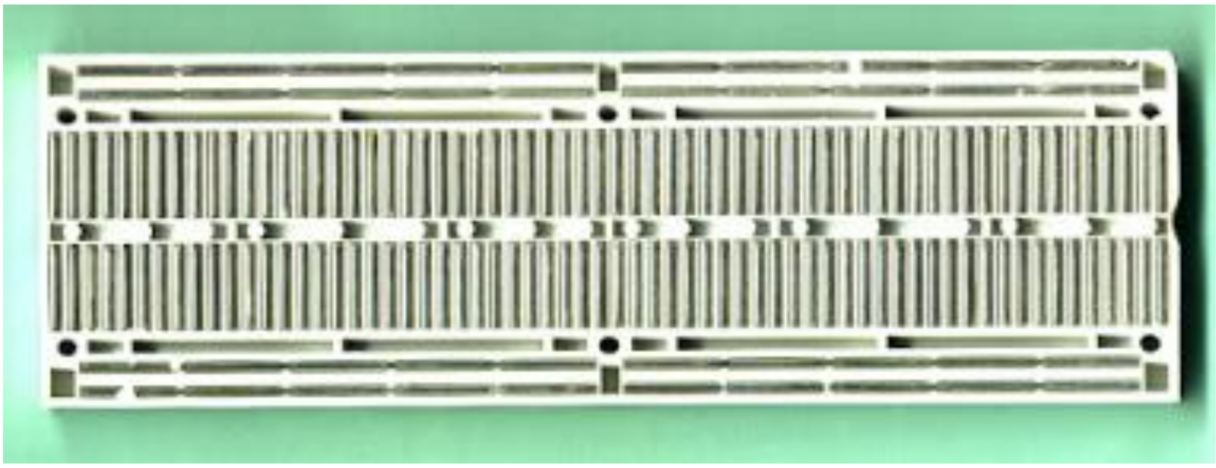
Note:

1) Charging of capacitor: When a capacitor is connected to a power source it is charged, and maintains the charge even after the power source is disconnected.

2) Discharging of capacitor: When capacitor leads are connected to a resistor or short circuited the stored charge results in current flow and when stored charge is removed the capacitor is said to be discharged.

Bread Board:





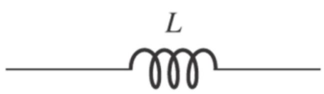
3. INDUCTOR:

An **inductor**, also called a **coil**, **choke**, or **reactor**, is a passive two-terminal electrical component that stores energy in a magnetic field when electric current flows through it. An inductor typically consists of an insulated wire wound into a coil.

When the current flowing through the coil changes, the time-varying magnetic field induces an **electromotive force (*emf*)** (voltage) in the conductor, described by Faraday's law of induction. According to Lenz's law, the induced voltage has a polarity (direction) which opposes the change in current that created it. As a result, inductors oppose any changes in current through them.

An inductor is characterized by its inductance, which is the ratio of the voltage to the rate of change of current. In the International System of Units (SI), the unit of inductance is the **henry (H)** named for 19th century American scientist Joseph Henry. In the measurement of magnetic circuits, it is equivalent to weber/ampere. Inductors have values that typically range from 1µH (10^{-6} H) to 20H. Many inductors have a magnetic core made of iron or ferrite inside the coil, which serves to increase the magnetic field and thus the inductance.

Symbol:



PROCEDURE:

Testing for Resistor:

1. Determine the resistance value of various resistors using color code.
2. Measure the resistance of each resistor using DMM and complete the table.

Testing for capacitance:

1. Determine the value and type of each capacitor from its value printed on it.
2. Measure the value of capacitance using DMM and complete the table.

4. Bread board

- A breadboard allows for easy and quick creation of temporary electronic circuits or to carry out experiments with circuit design.
- Breadboards enable developers to easily connect components or wires thanks to the rows and columns of internally connected spring clips underneath the perforated plastic enclosure.

RESULT:

Hence, different electronic circuit components have been identified, observed and measured their values.

CIRCUIT DIAGRAM:

1. KVL:

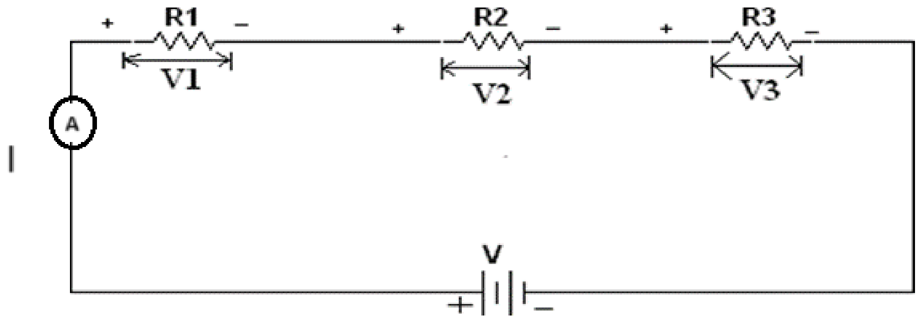


Fig (1a)

CALCULATIONS:

OBSERVATION TABLE:

S.No	Voltage Across	Theoretical (V)	Practical (V)
1	R_1		
2	R_2		
3	R_3		
4	Total Voltage		

Expt. No: 1b	Verification of KVL and KCL
Dt:	

AIM: To verify the Kirchhoff's voltage law and Kirchhoff's current law for the given circuit.

APPARATUS REQUIRED:

S.No	Name of the equipment	Range	Type	Quantity
1	RPS	0-30V	-	1
2	Voltmeter	0-20 V	Digital	4
3	Ammeter	0-20mA	Digital	4
4	Bread board	-	-	1
5	Resistors	1kΩ	-	1
		2.2kΩ	-	1
		3.3kΩ	-	1
6	Connecting wires	-	-	Required

THEORY:

a) **Kirchhoff's Voltage law** states that the algebraic sum of the voltage around any closed path in a given circuit is always zero. In any circuit, voltage drops across the resistors always have polarities opposite to the source polarity. When the current passes through the resistor, there is a loss in energy and therefore a voltage drop. In any element, the current flows from a higher potential to lower potential. Consider the fig (1a) shown above in which there are 3 resistors are in series. According to kickoff's voltage law….

$$\mathbf{V = V_1 + V_2 + V_3}$$

Where

$V_1 = \frac{R_1}{R_1+R_2+R_3} V \qquad V_2 = \frac{R_2}{R_1+R_2+R_3} V \qquad V_2 = \frac{R_3}{R_1+R_2+R_3} V$

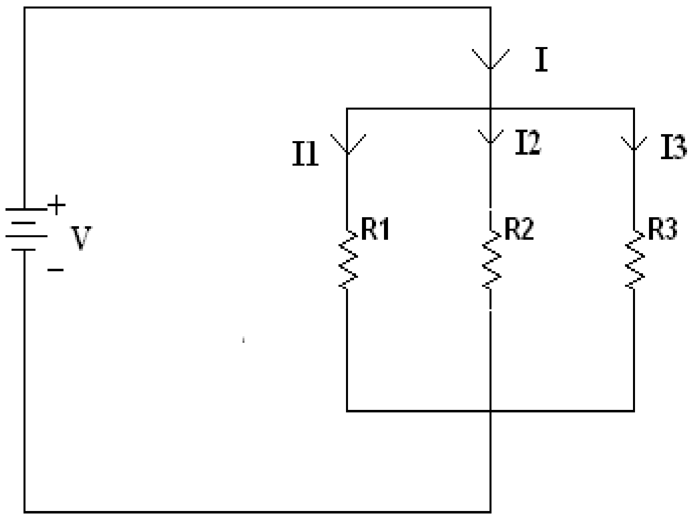
b) **Kirchhoff's current law** states that the sum of the currents entering a node equal to the sum of the currents leaving the same node. Consider the fig(1b) shown above in which there are 3 parallel paths. According to Kirchhoff's current law...

$$\mathbf{I = I_1 + I_2 + I_3}$$

Where

$\boldsymbol{I_1 = \frac{V}{R_1}} \qquad \boldsymbol{I_2 = \frac{V}{R_2}} \qquad \boldsymbol{I_3 = \frac{V}{R_3}}$

2. KCL:



Fig(1b)

CALCULATIONS:

OBSERVATION TABLE:

S.No	Current Through	Theoretical (mA)	Practical (mA)
1	R_1		
2	R_2		
3	R_3		
4	Total Current		

PROCEDURE:

a) Kirchhoff's Voltage law:

- Connect the circuit as shown in fig (2a).
- Measure the voltages across the resistors.
- Observe that the algebraic sum of voltages in a closed loop is zero.

b) Kirchhoff's current law:

- Connect the circuit as shown in fig (2b).
- Measure the currents through the resistors.
- Observe that the algebraic sum of the currents at a node is zero.

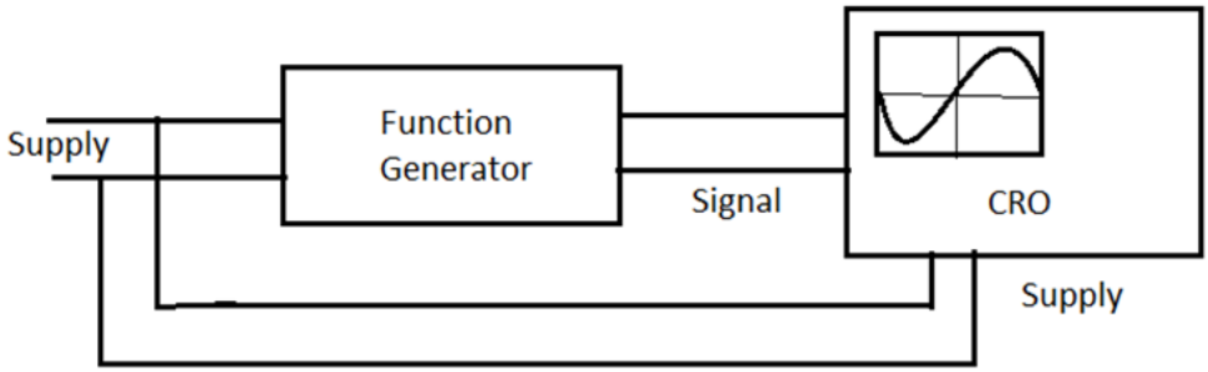
PRECAUTIONS:

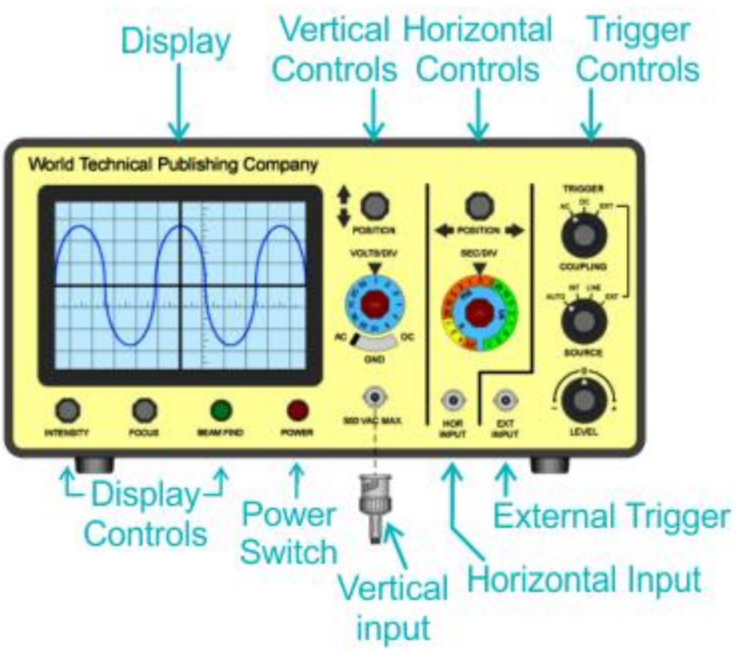
1. Avoid loose connections.
2. Keep all the knobs in minimum position while switch on and off of the supply.

RESULT:

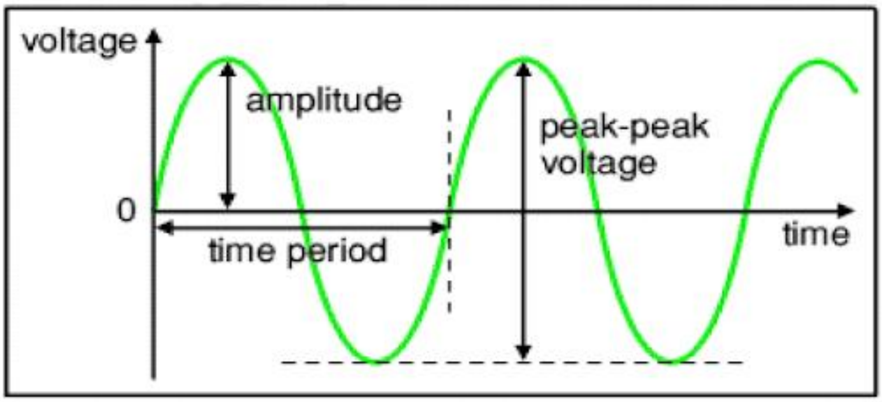
Hence, the Kirchhoff's voltage law and Kirchhoff's current law have been verified for the given circuit.

CIRCUIT DIAGRAM:





MODEL WAVEFORM:



Expt. No: 2	Measurement of sinusoidal voltage, frequency, effective and average values using CRO
Dt:	

AIM:

To familiarize with the measurement of sinusoidal voltage, frequency, effective (RMS), and average values using a Cathode Ray Oscilloscope (CRO).

APPARATUS REQUIRED:

S.	Name of the equipment	Range	Type	Quantity
1	Function generator		-	1
2	Cathode Ray Oscilloscope		Digital	1
3	Connecting cables		Digital	2
4	Multimeter	-	-	1

THEORY:

Cathode Ray Oscilloscope (CRO) controls are essential for adjusting the display, capturing signals, and making accurate measurements. Here's an overview of the typical controls of a CRO:

1. Vertical Controls:

•**Volts/Division (V/Div):** Adjusts the vertical scale to display the voltage per division on the screen.

•**Position:** Moves the trace up or down on the screen.

•**Invert:** Inverts the polarity of the displayed waveform.

•**AC/DC/Ground:** Selects the coupling mode for the vertical input (AC, DC, or ground reference).

2. Horizontal Controls:

•**Time/Division (Time/Div):** Adjusts the horizontal scale to display time per division on the screen.

•**Position:** Moves the trace left or right on the screen.

•**Trigger Level:** Sets the voltage level at which the trigger circuit initiates a sweep.

3. Trigger Controls:

•**Trigger Source:** Selects the channel that will trigger the sweep (Channel 1, Channel 2, External, etc.).

•**Trigger Mode:** Selects the trigger mode (Auto, Normal, Single).

•**Trigger Slope:** Selects the trigger slope (Rising, Falling, or both).

Measurement of Voltage:

Measure the signal voltage of the signal by counting the no.of vertical divisions between +ve peak and -ve peak.

$$\mathbf{V_{PP} = N_V \times (Volts/Division)}$$

Where N_V is vertical divisions on the scope.

$$\mathbf{V_{rms} = V_{PP}/\sqrt{2}}$$

Measure the signal with an AC milli voltmeter as well. It gives the rms value of the signal.

Measurement of Time period:

Measure the signal time period T of the signal by counting the no.of horizontal divisions.

$T = N_H \times$ (Time/Division)

Where N_H is horizontal divisions on the scope covering the span of one cycle.

Calculate frequency as f = 1/T.

OBSERVATIONS:

1. **Display Controls:**

- **Intensity/Brightness:** Adjusts the brightness of the trace.
- **Focus:** Adjusts the sharpness of the trace.

2. **Additional Controls:**

- **XY Mode:** Enables XY mode for displaying Lissajous figures or other two-dimensional plots.
- **Beam Finder:** Helps locate the trace if it's not visible on the screen.
- **Calibration Controls:** Allows for fine-tuning and calibration of the oscilloscope.

PROCEDURE:

1. Connect the function generator to the CRO using appropriate cables.
2. Generate a sinusoidal waveform using the function generator.
3. Connect the CRO probes to measure the sinusoidal voltage.
4. Adjust the frequency of the function generator and observe the changes in the waveform on the CRO.
5. Measure the peak-to-peak voltage and time period on the CRO and calculate the average voltage, rms voltage and frequency.
6. Compare the calculated values with theoretical values obtained from the function generator.

RESULT:

Hence, familiarized with the functions of CRO and measured different parameters of a signal like voltage, frequency, effective (RMS), and average values using a Cathode Ray Oscilloscope (CRO).

29

CIRCUIT DIAGRAM:

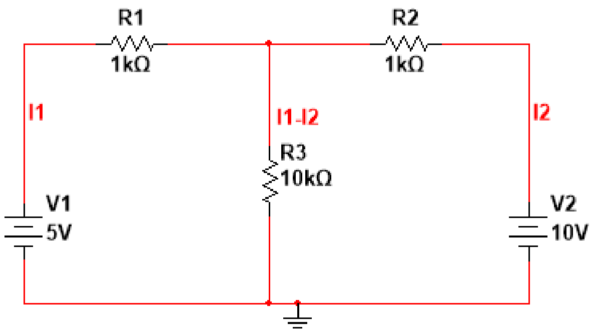


figure 1

CALCULATIONS:

OBSERVATIONS:

	Theoretical value I (mA)	Practical Value I (mA)
I_1		
I_2		
I_1-I_2		

Expt. No: 3	Verification of Mesh analysis for an Electrical Circuit
Dt:	

AIM:

To find the branch currents of a given circuit using mesh analysis.

APPARATUS REQUIRED:

S.	Name of the equipment	Range	Type	Quantity
1	RPS	0-30V	-	2
2	Voltmeter	0-20 V	Digital	4
3	Ammeter	0-20mA	Digital	4
4	Bread board	-	-	1
5	Resistors	1kΩ	-	2
		10kΩ	-	1
6	Connecting wires	-	-	Required

THEORY:

Mesh and nodal analysis are two basic important techniques which are useful to find solutions in a network. The suitability of either mesh or nodal analysis to a particular problem mainly depends on the number of voltage sources or current sources. If a network has a large number of voltage sources, it is useful to use mesh analysis; if, on the other hand, the network has more current sources, nodal analysis is the useful method. Mesh analysis is applied to most of the networks. Unfortunately, it is applicable only for planar networks. For non-planar circuits mesh analysis is not applicable. A circuit is said to be planar, if it can be drawn on a plane surface without crossovers. A non-planar circuit cannot be drawn on a plane surface without a crossover.

PROCEDURE:

1. Construct the circuit as shown in figure 1.
2. By using mesh method, the circuit is solved and the loop currents are determined.
3. Connect ammeters in series with resistances in all branches.
4. Tabulate the readings and compare the practical values with theoretical values.

RESULTS:

Thus the mesh analysis for the given electrical circuit is verified.

CIRCUIT DIAGRAM:

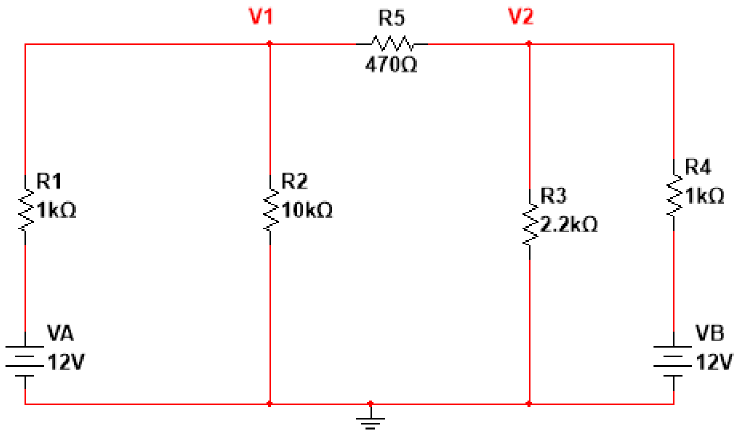


Figure 1

CALCULATIONS:

OBSERVATIONS:

	Theoretical value V(volts)	Practical Value V(volts)
V_1		
V_2		

Expt. No: 4	**Verification of Node analysis for an Electrical Circuit**
Dt:	

AIM: To find the node voltages of a given circuit using nodal analysis.

APPARATUS REQUIRED:

S.No	Name of the equipment	Range	Type	Quantity
1	RPS	0-30V	-	2
2	Voltmeter	0-20 V	Digital	4
3	Ammeter	0-20mA	Digital	4
4	Bread board	-	-	1
5	Resistors	1k	-	2
		10kΩ, 2.2kΩ, 470Ω	-	Each 1
6	Connecting wires	-	-	Required

THEORY:

Nodal analysis is a mathematical method for calculating the distribution of voltages between nodes in a circuit. Also referred to as the node-voltage method, this strategy uses Ohm's law, Kirchhoff's voltage law, and Kirchhoff's current law to define an equation relating the voltage measured between each circuit node and some reference (usually ground). The voltage drops measured between neighboring nodes are taken as the variables in a set of linear equations and the system can be solved using a standard algorithm

PROCEDURE:

1. Construct the circuit as shown in figure 1.
2. Set V_A & V_B to defined voltages.
3. By using multimeter measure node voltages atV_1 and V_2.
4. Tabulate all results.

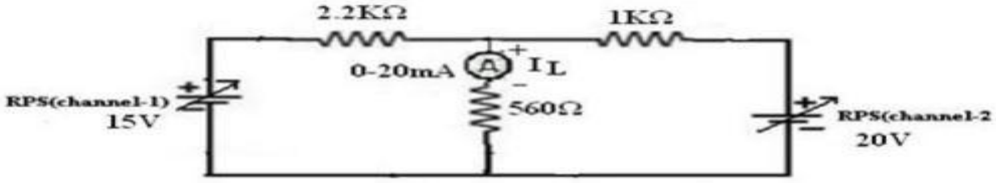
RESULT:

Thus the nodal analysis for the given electrical circuit has been verified.

37

CIRCUIT DIAGRAM:

When V_1 & V_2 source acting (To find I_L):-



Fig(1)

When V_1 Source Acting (To Find I_L^I)

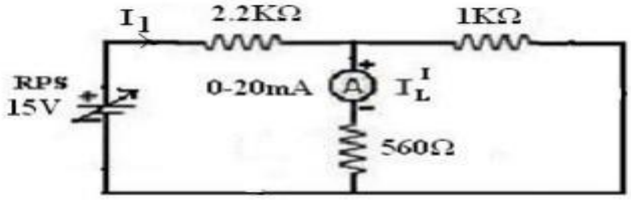


Fig (2)

When V_2 source acting (To find I_L^{II}):

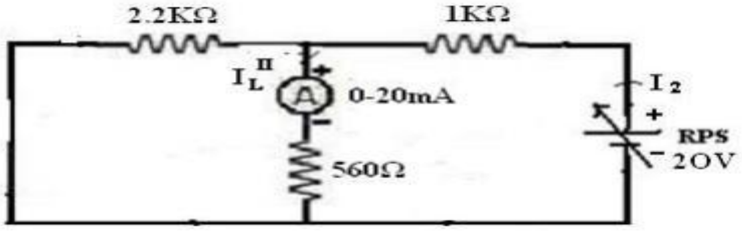


Fig (3)

CALCULATIONS:

Expt. No: 5	Verification of Superposition theorem
Dt:	

AIM: To verify the superposition theorem for the given circuits.

APPARATUS REQUIRED:

S.No	Apparatus	Range	Quantity
1.	DC voltage source	0-20V	1 No.
2.	Resistor	100 ohm	1 No.
3.	Ammeter	0 - 200mA	1 No.
4.	Ammeter	0-200µA	1 No.
5.	Voltmeter	0 - 20V	2 Nos.
6.	Connecting wires		Required.

THEORY:

SUPERPOSITION THEOREM:

Superposition theorem states that in a lumped, linear, bilateral network consisting more number of sources each branch current (voltage) is the algebraic sum all currents (branch voltages), each of which is determined by considering one source at a time and removing all other sources. In removing the sources, voltage and current sources are replaced by internal resistances.

OBSERVATIONS:

Source	Theoretical	Practical
Source 1	I_L^{I} =	I_L^{I} =
Source 2	I_L^{II} =	I_L^{II} =
All Sources	I_L =	I_L =

PROCEDURE:

1. Connect the circuit as per the fig (1).
2. Adjust the sources X and Y to appropriate values (Say 15V and 20V respectively).
3. Note down the current (I_L) through the 560 0hm resistor by using the ammeter.
4. Connect the circuit as per fig (2) and set the source Y (20V) to 0V.
5. Note down the current (I_L^l) through 560ohm resistor by using ammeter.
6. Connect the circuit as per fig(3) and set the source X (15V) to 0V and source Y to 20V.
7. Note down the current (I_L^{ll}) through the 560ohm resistor branch by using ammeter.

PRECAUTIONS:

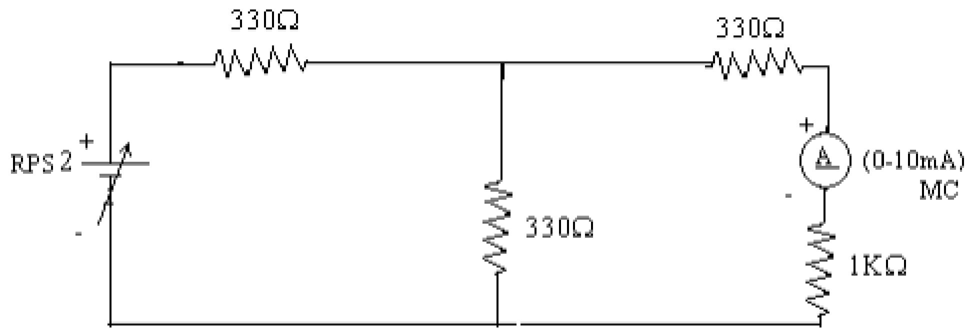
1. Check the connections before giving the supply.
2. Observations should be done carefully.

RESULT:

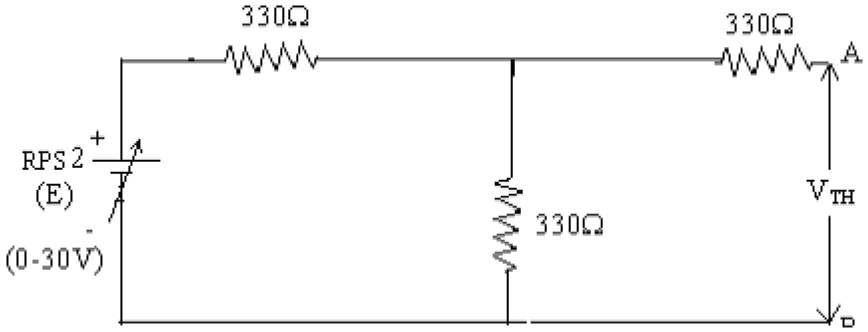
Thus the superposition theorem has been verified for a given circuit.

CIRCUIT DIAGRAM:

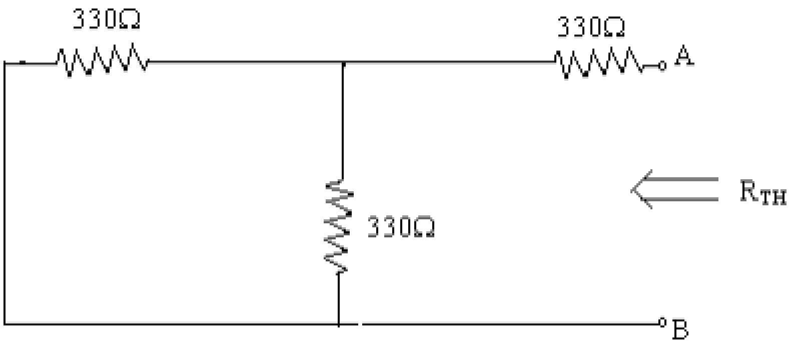
Circuit - 1 : To find load current



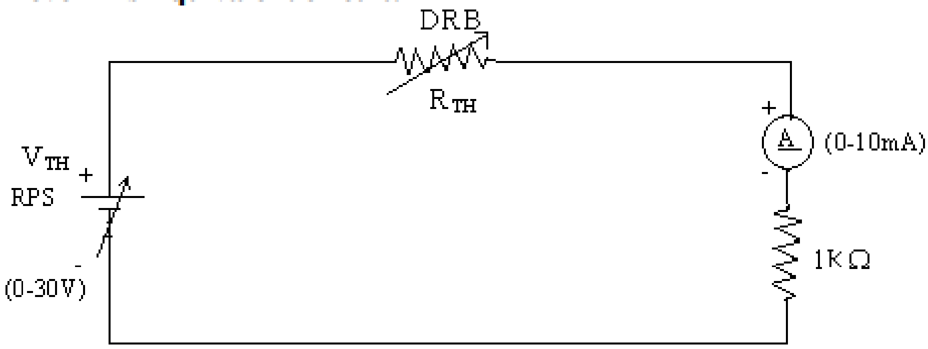
To find V_{TH}



To find R_{TH}



Thevenin's Equivalent circuit:



Expt. No: 6	Verification of Thevenin’s & Norton’s theorems
Dt:	

AIM: To verify Thevenin’s and Norton’s theorems and to find the full load current for the given circuit.

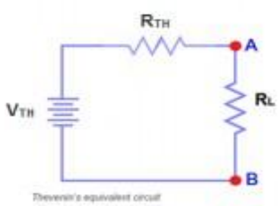
APPARATUS REQUIRED:

S.No	Apparatus	Range	Quantity
1.	DC voltage source	0-20V	1 No.
2.	Resistor	100 ohm	1 No.
3.	Ammeter	0 - 200mA	1 No.
4.	Ammeter	0-200µA	1 No.
5.	Voltmeter	0 - 20V	2 Nos.
6.	Connecting wires		Required.

THEORY:

Thevenin’s theorem:

Thevenin’s theorem states that any two output terminals (A & B) of an active linear network containing independent sources (it includes voltage and current sources) can be replaced by a simple voltage source of magnitude V_{th} in series with a single resistor R_{th} where R_{th} is the equivalent resistance of the network when looking from the output terminals A & B with all sources (voltage and current) removed and replaced by their internal resistances and the magnitude of V_{th} is equal to the open circuit voltage across the A & B terminals.



Thevenin’s Equivalent Circuit

CALCULATIONS:

OBSERVATIONS:

	V(V)	V_{TH}(V)	R_{TH}(Ω)	I_L (mA)	
				Circuit - I	Equivalent Circuit
Theoretical					
Practical					

Thevenin's Theorem:

Procedure:

1. Connections are given as per the circuit diagram.
2. Set a particular value of voltage using RPS and note down the corresponding ammeter readings.

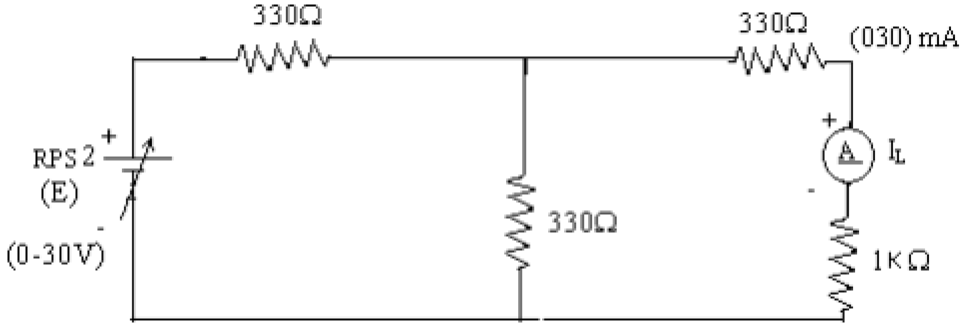
To find V_{TH}

3. Remove the load resistance and measure the open circuit voltage using multimeter (V_{TH}).

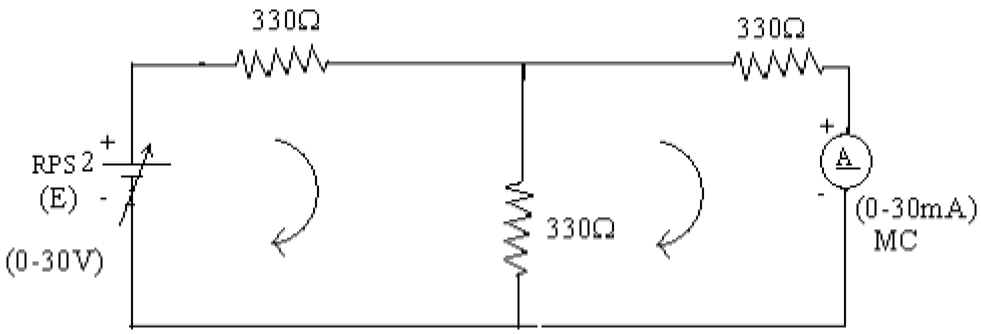
To find R_{TH}

4. To find the Thevenin's resistance, remove the RPS and short circuit it and find the R_{TH} using multimeter.
5. Give the connections for equivalent circuit and set V_{TH} and R_{TH} and note the corresponding ammeter reading.
6. Verify Thevenins theorem.

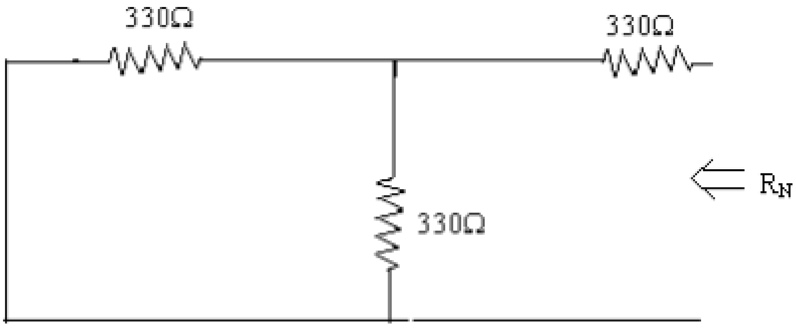
To find load current in circuit 1:



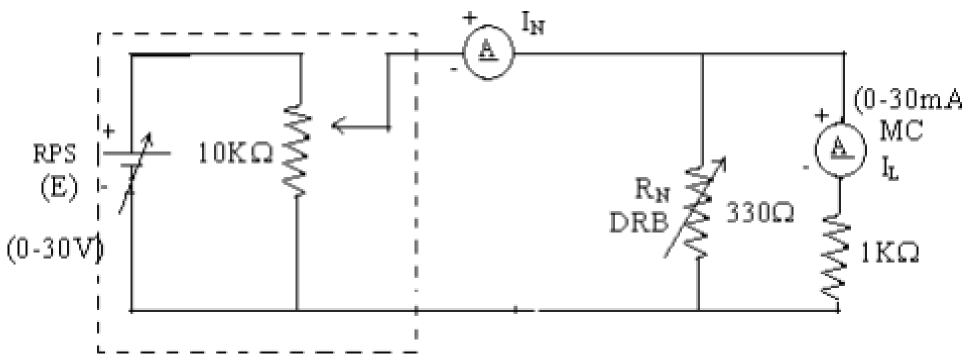
To find I_N



To find R_N



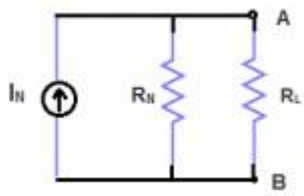
Norton's equivalent circuit



Constant current source

Norton's Theorem

Norton's Theorem states that Any Linear Electric Network or complex circuit with Current and Voltage sources can be replaced by an equivalent circuit containing of a single independent Current Source I_N and a Parallel Resistance R_N.



Norton's Equivalent Circuit

PROCEDURE:

1. Connections are given as per circuit diagram.
2. Set a particular value in RPS and note down the ammeter readings in the original circuit.

To Find I_N:

3. Remove the load resistance and short circuit the terminals.
4. For the same RPS voltage note down the ammeter readings.

To Find R_N:

5. Remove RPS and short circuit the terminal and remove the load and note down the resistance across the two terminals. Equivalent Circuit:
6. Set I_N and R_N and note down the ammeter readings.
7. Verify Norton's theorem.

CALCULATIONS:

OBSERVATIONS:

	V(V)	I_N (A)	$R_N(\Omega)$	I_L (mA)	
				Circuit - I	Equivalent Circuit
Theoretical					
Practical					

PRECAUTIONS:

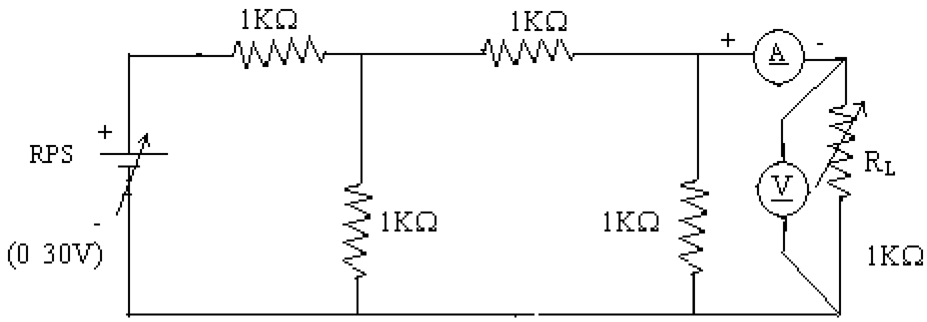
1. Check the connections before giving the supply.
2. Observations should be done carefully.

RESULT:

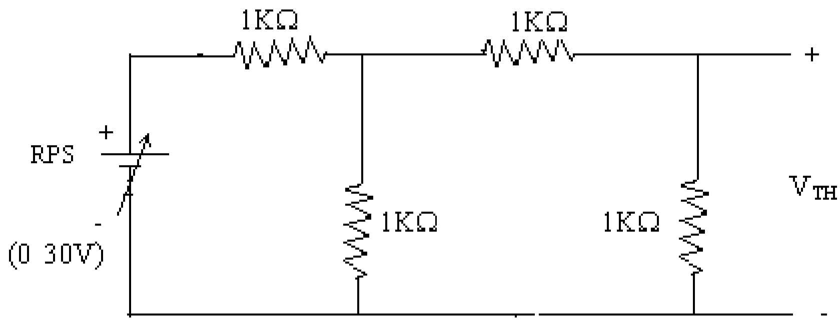
Thus the Thevinen's and Norton's theorems has been verified for a given circuit.

CIRCUIT DIAGRAM:

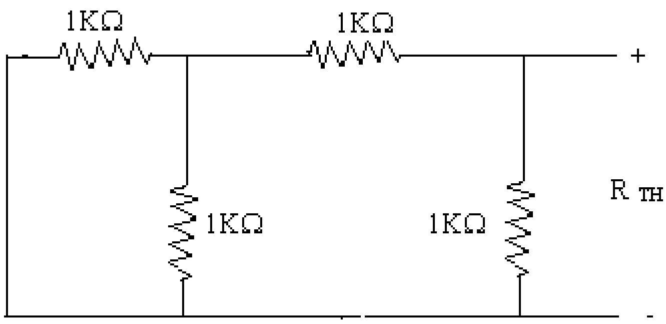
Circuit - 1



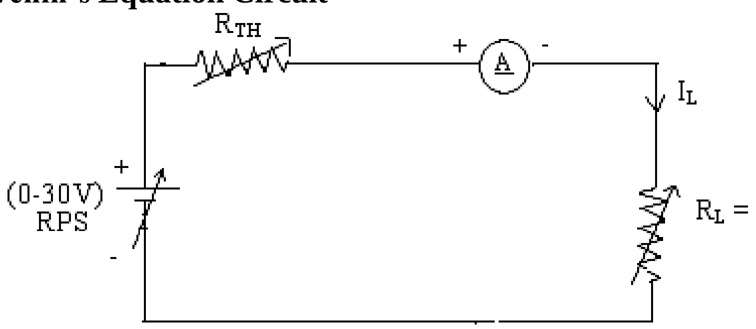
To find V_{TH}



To find R_{TH}



Thevenin's Equation Circuit



Expt. No: 7	Verification of Maximum Power Transfer Theorem
Dt:	

AIM:

To verify maximum power transfer theorem for the given circuit

APPARATUS REQUIRED:

Sl.No.	Apparatus	Range	Quantity
1	RPS	(0-30V)	1
2	Voltmeter	(0-10V) MC	1
3	Resistor	1KΩ, 1.3KΩ, 3Ω	3
4	DRB	--	1
5	Bread Board & wires	--	Required

THEORY:

Statement:

In a linear, bilateral circuit the maximum power will be transferred to the load when load resistance is equal to source resistance.

PROCEDURE:

Circuit – I

1. Connections are given as per the diagram and set a particular voltage in RPS.
2. Vary R_L and note down the corresponding ammeter and voltmeter reading.
3. Repeat the procedure for different values of R_L & Tabulate it.
4. Calculate the power for each value of R_L.

To find V_{TH}:

5. Remove the load, and determine the open circuit voltage using multimeter (V_{TH})

To find R_{TH}:

6. Remove the load and short circuit the voltage source (RPS).
7. Find the looking back resistance (R_{TH}) using multimeter.

OBSERVATIONS:
Circuit – I

Sl.No.	R_L (Ω)	I (mA)	V(V)	P=VI (watts)

To find Thevenin's equivalent circuit

	V_{TH} (V)	R_{TH} (Ω)	I_L (mA)	P (milli watts)
Theoretical				
Practical				

Equivalent Circuit:

8. Set V_{TH} using RPS and R_{TH} using DRB and note down the ammeter reading.
9. Calculate the power delivered to the load ($R_L = R_{TH}$)
10. Verify maximum transfer theorem.

PRECAUTIONS:

1. **PRECAUTIONS:**

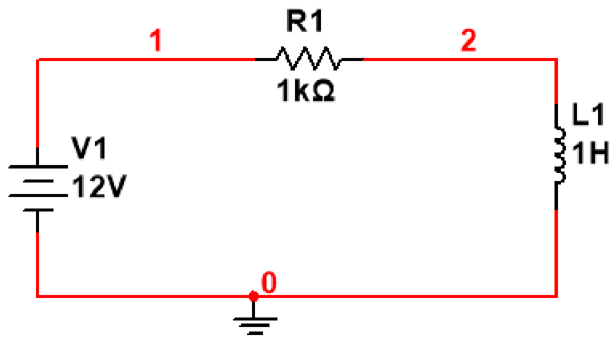
1. Voltage control knob of RPS should be kept at minimum position.
2. Check the connections before giving the supply.
3. Observations should be done carefully.

RESULT:

Thus maximum power theorem was verified both practically and theoretically

RL Circuit:

CIRCUIT DIAGRAM:



CALCULATIONS:

$$i = \frac{V}{R}\left(1 - e^{-\left(\frac{R}{L}\right)t}\right)$$

$$\Rightarrow i = \frac{V}{R}\left(1 - e^{-\left(\frac{t}{\tau}\right)}\right)$$

Where, **τ is** the **time constant** and its value is equal to L/R

i(T) =

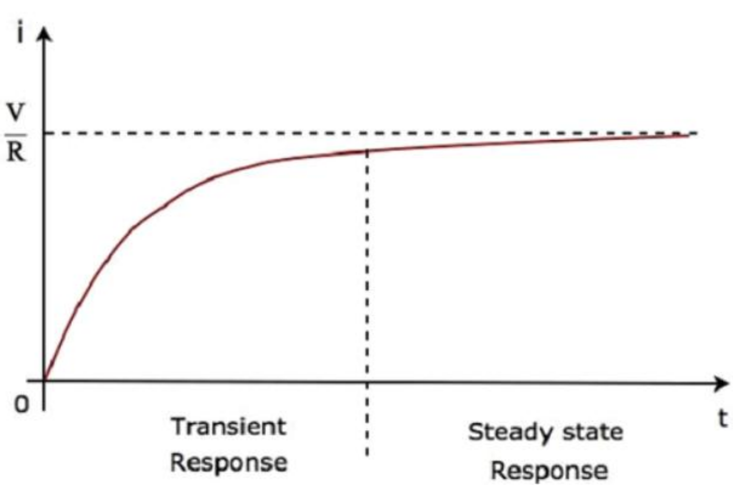
i(2T) =

i(3T) =

i(4T) =

i(5T) =

MODEL GRAPH



OBSERVATIONS:

i(t)	Theoritical (A)	Practical (A)
i(T)		
i(2T)		
i(3T)		
i(4T)		
i(5T)		

Expt. No: 8 Dt:	**Study of DC transients (time response) in RL, RC and RLC circuits**

AIM: To study the transient response of a series RL, RC and RLC circuit and understand the time constant concept with DC Power Supply

APPARATUS REQUIRED:

S.No	Name of the equipment	Range	Type	Quantity
1	PC loaded with multisim software	-----	------	1

THEORY:

When a circuit is switched from one condition to another either by a change in the applied voltage or a change in one of the circuit elements, there is a transitional period during which the branch currents and voltage drops change from their former values to new ones. After this transition interval called the transient, the circuit is said to be in the steady state.

Time Constant (T): It is a measure of time required for certain changes in voltages and currents in RC and RL circuits. Generally, when the elapsed time exceeds five time constants (5T) after switching has occurred, the currents and voltages have reached their final value, which is also called steady-state response. The time constant of an RL circuit is the equivalent inductance divided by the Thévenin resistance as viewed from the terminals of the equivalent inductor.

$$T = L / R$$

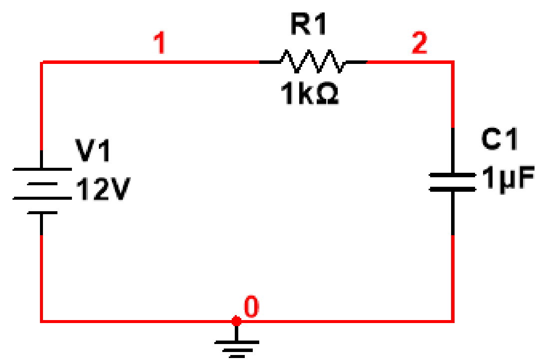
The time constant of an RC circuit is the product of equivalent capacitance and the Thévenin resistance as viewed from the terminals of the equivalent capacitor.

$$T = R*C$$

When a circuit is switched from one condition to another either by a change in the applied voltage or a change in one of the circuit elements, there is a transitional period during which the branch currents and voltage drops change from their former values to new ones. After this transition interval called the transient, the circuit is said to be in the steady state. Let us consider the R-L-C circuit as shown.

RC Circuit:

CIRCUIT DIAGRAM:



CALCULATIONS:

$$i = \frac{V}{R} e^{-\frac{1}{RC}t} \quad \text{for } t > 0$$

i(T) =

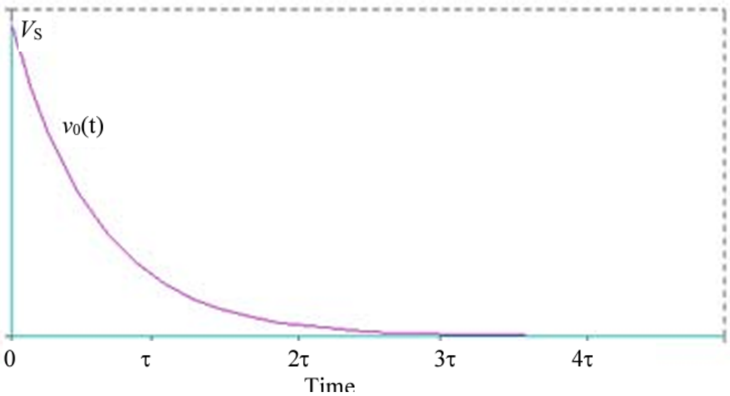
i(2T) =

i(3T) =

i(4T) =

i(5T) =

MODEL GRAPH



OBSERVATIONS:

i(t)	Theoritical (A)	Practical (A)
i(T)		
i(2T)		
i(3T)		
i(4T)		
i(5T)		

Obtain transient response of a series RLC circuit, excited by a unit step input, where L=10mH and C=1μF and for the following conditions:

Case 1. $R < 2\sqrt{\frac{L}{C}}$, under damped case where R=50

Case 2. $R = 2\sqrt{\frac{L}{C}}$, critically damped case where R=200

Case 3. $R > 2\sqrt{\frac{L}{C}}$, over damped case where R=300

PROCEDURE:

RL Circuit:

1. Construct RL circuit as shown in fig1 using Multisim simulation software.
2. Observe the transient response.

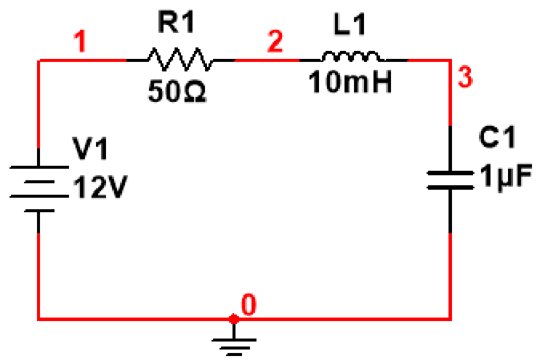
RC Circuit:

1. Construct RC circuit as shown in fig 2 using Multisim simulation software.
2. Observe the transient response.

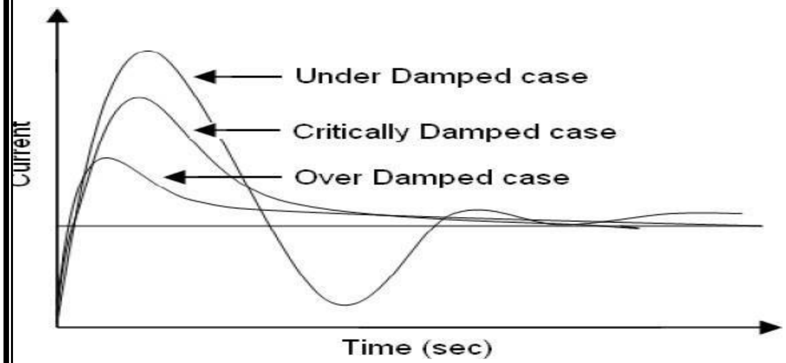
RLC Circuit:

1. Construct RLC circuit as shown in fig 3 using Multisim simulation software.
2. Observe the transient response for three cases.

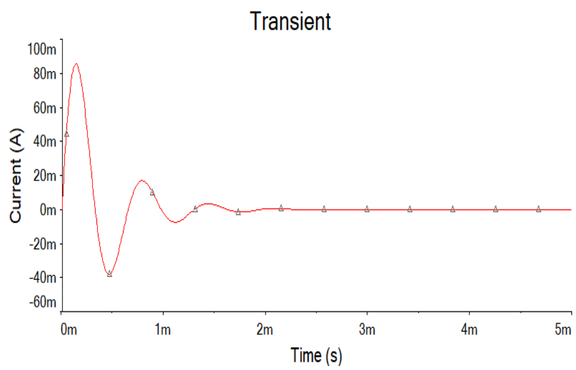
RLC CIRCUIT DIAGRAM:



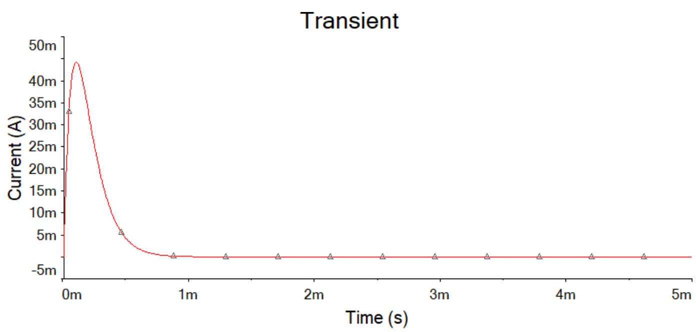
MODEL GRAPHS:



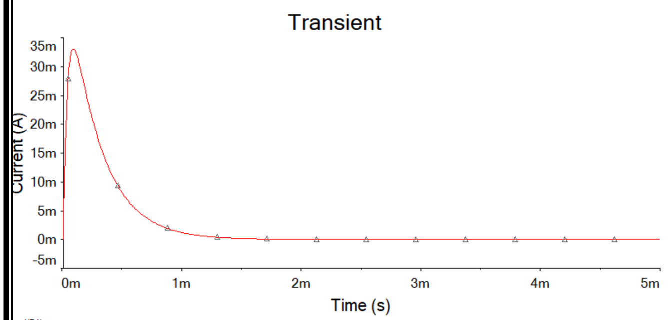
Case1:



Case2:



Case3:



PRECAUTIONS:

1. Choose components properly from the built-in library to get accurate simulations.
2. Ensure proper grounding of circuits to avoid inaccurate results or simulation errors.

RESULT:

Hence, Transient analysis of Series RL, RC and RLC circuits were analyzed using Multisim. The Theoritical and simulated graphs are verified.

RL CIRCUIT

CIRCUIT DIAGRAM:

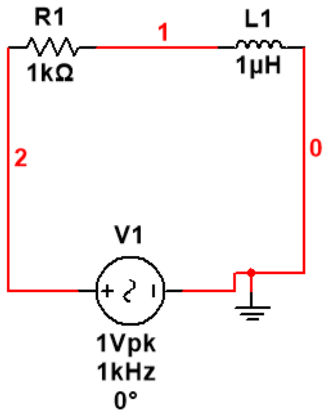


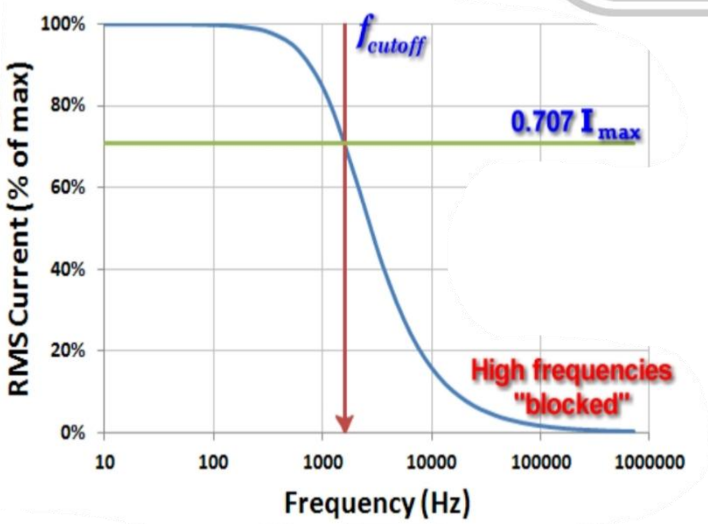
Fig1: RL Circuit

CALCULATIONS:

The cutoff frequency (f_c) of an RL circuit is

$$f_c = \frac{R}{2\pi L}$$

MODEL GRAPH:



OBSERVATIONS:

	Theoretical	Practical
Cutoff frequency f_C (Hz)		

Expt. No: 9	**Study of frequency response (steady state) of RL & RC networks**
Dt:	

AIM: To Study frequency response (steady state) of RL & RC networks.

APPARATUS REQUIRED:

S.No	Name of the equipment	Range	Type	Quantity
1	PC loaded with multisim software	-----	------	1

THEORY:

The frequency response of RL (resistor-inductor) and RC (resistor-capacitor) networks refers to how these circuits behave in response to sinusoidal input signals across a range of frequencies in a steady state. Understanding the frequency response is crucial for various applications, including signal processing, filtering, and impedance matching.

RL Network:

In an RL network, which consists of a resistor (R) and an inductor (L), the behavior at different frequencies is influenced by the inductive reactance (X_L), given by:

$$X_L = 2\pi fL$$

where:

- f is the frequency of the input signal.
- L is the inductance of the inductor.

At low frequencies (compared to the cutoff frequency), the inductive reactance dominates, and the impedance of the circuit is mainly determined by the inductor. As the frequency increases, the inductive reactance decreases, allowing more current to flow through the circuit. However, the impedance of the inductor limits the current flow, resulting in a phase lag between voltage and current.

The cutoff frequency (f_c) of an RL circuit is given by:

$$f_c = \frac{R}{2\pi L}$$

Above the cutoff frequency, the inductive reactance becomes negligible compared to the resistance, and the impedance of the circuit is mainly determined by the resistor. Consequently, the phase shift approaches zero, and the voltage across the resistor and inductor becomes nearly in-phase.

RC Circuit:

CIRCUIT DIAGRAM:

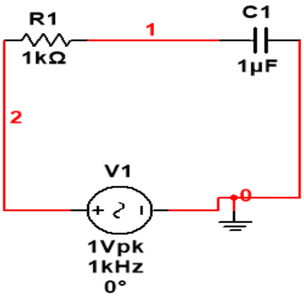


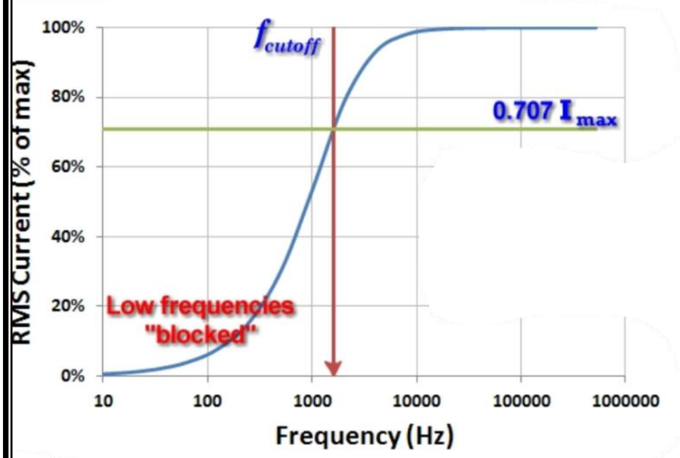
Fig2: RC Circuit

CALCULATIONS:

The cutoff frequency (f_c) of an RC circuit is

$$f_c = \frac{1}{2\pi RC}$$

MODEL GRAPHS:



OBSERVATIONS:

	Theoretical	Practical
Cutoff frequency f_C (Hz)		

RC Network:

In an RC network, which consists of a resistor (R) and a capacitor (C), the behavior at different frequencies is influenced by the capacitive reactance (XC), given by:

$$X_C = \frac{1}{2\pi f C}$$

At low frequencies, the capacitive reactance is high, and the impedance of the circuit is mainly determined by the capacitor. As the frequency increases, the capacitive reactance decreases, allowing more current to flow through the circuit. However, the impedance of the capacitor limits the current flow, resulting in a phase lead between voltage and current.

The cutoff frequency (f_c) of an RC circuit is given by:

$$f_c = \frac{1}{2\pi RC}$$

Above the cutoff frequency, the capacitive reactance becomes negligible compared to the resistance, and the impedance of the circuit is mainly determined by the resistor. Consequently, the phase shift approaches zero, and the voltage across the resistor and capacitor becomes nearly in-phase.

PROCEDURE:

RL circuit

3. Construct RL circuit as shown in fig1 using Multisim simulation software.
4. Plot graphs of amplitude versus frequency and phase shift versus frequency for RL circuit.
5. Analyze the frequency response characteristics such as cutoff frequency, and phase shift.

RC circuit

1. Construct RC circuit as shown in fig2 using Multisim simulation software.
2. Plot graphs of amplitude versus frequency and phase shift versus frequency for RC circuit.
3. Analyze the frequency response characteristics such as cutoff frequency, and phase shift.

PRECAUTIONS:

1. Choose components properly from the built-in library to get accurate simulations.
2. Ensure proper grounding of circuits to avoid inaccurate results or simulation errors.

RESULT:

Hence, frequency response (steady state) of RL & RC networks has been analyzed.

CIRCUIT DIAGRAM:

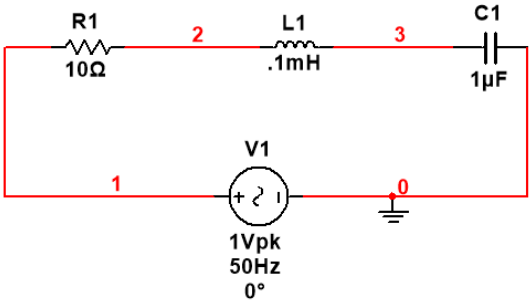


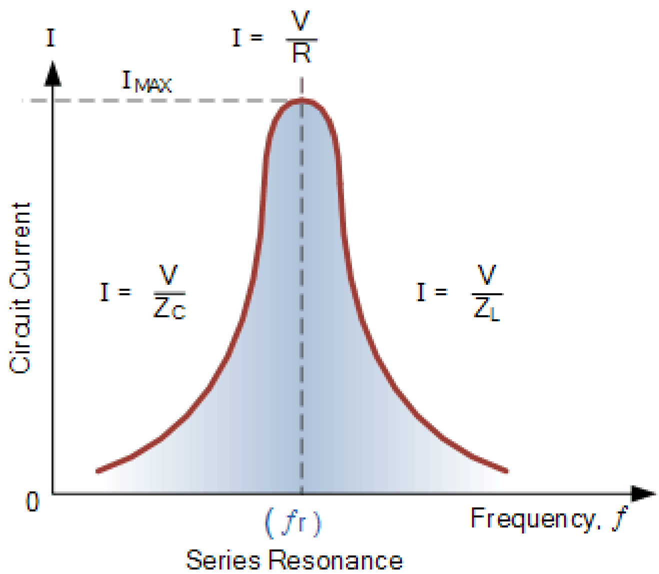
Fig: RLC Circuit

CALCULATIONS:

The cutoff frequency (f_c) of an RLC circuit is

$$f_0 = \frac{1}{2\pi\sqrt{LC}}$$

MODEL GRAPH:



Expt. No: 10	Study state response of 2^{nd} order circuit
Dt:	

AIM: To Study frequency response of 2^{nd} order circuit i.e RLC circuit.

APPARATUS REQUIRED:

S.No	Name of the equipment	Range	Type	Quantity
1	PC loaded with multisim software	-----	------	1

THEORY:

In a series **RLC circuit** Resistor (R), Inductor (L), and Capacitor (C) are all connected in series with the AC voltage source. In this circuit, the current flow through all the network elements is the same, but only the supply voltage (AC) will get divided among the passive elements.

RLC circuits have many applications as oscillator circuits. Radio receivers and television sets use them for tuning to select a narrow frequency range from ambient radio waves. In this role, the circuit is often referred to as a tuned circuit. An RLC circuit can be used as a band-pass filter, band-stop filter, low-pass filter or high-pass filter.

$$\mathbf{Z} = R + jX_L - jX_C = R + j(X_L - X_C)$$

At resonance, the circuit is purely resistive.

$$X_L - X_C = 0$$

$$X_L = X_C$$

$$\omega_0 L = \frac{1}{\omega_0 C}$$

$$\omega_0^2 = \frac{1}{LC}$$

$$\omega_0 = \frac{1}{\sqrt{LC}}$$

$$f_0 = \frac{1}{2\pi\sqrt{LC}}$$

where f_0 is called the **resonant frequency** of the circuit.

Current I_0: $I_0 = \frac{V}{R}$

OBSERVATIONS:

	Theoretical	**Practical**
Resonant frequency f_0 (Hz)		

PROCEDURE:

1. Construct RLC circuit as shown in fig using Multisim simulation software.
2. Plot graphs of amplitude versus frequency and phase shift versus frequency for RLC circuits.
3. Analyze the frequency response characteristics such as cutoff frequency, and phase shift.

PRECAUTIONS:

1. Choose components properly from the built-in library to get accurate simulations.
2. Ensure proper grounding of circuits to avoid inaccurate results or simulation errors.

RESULT:

Hence, frequency response of 2^{nd} order circuit has been analyzed.

SERIES RESONANCE

CIRCUIT DIAGRAM:

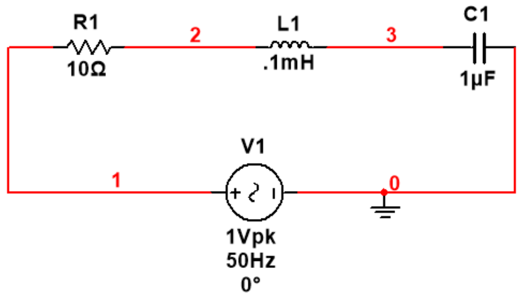


Fig1:

CALCULATIONS:

1. The resonance frequency (f_o) of an RLC circuit is

$$f_0 = \frac{1}{2\pi\sqrt{LC}}$$

2. Lower Cut off Frequency $f_L = f_0 - (R/4\pi L)$

3. Upper Cut off Frequency $f_H = f_0 - (R/4\pi L)$

4. Bandwidth = $f_H - f_L$

5. Quality Factor $Q_0 = \frac{1}{R}\sqrt{\frac{L}{C}} = \frac{V_L \, or \, V_C}{V}$

Expt. No: 11	Determination of The Q factor and Bandwidth of a resonance circuit
Dt:	

AIM: To find the resonant frequency, quality factor and band width of a given series resonant circuits.

APPARATUS REQUIRED:

S.No	Name of the equipment	Range	Type	Quantity
1	PC loaded with multisim software	-----	------	1

THEORY:

Resonance is a particular type of phenomenon inherently found normally in every kind of system, electrical, mechanical, optical, Acoustical and even atomic. The definition of resonance in electrical system is studied state operation of a circuit or system at that frequency for which the resultant response is in time phase with the forcing function.

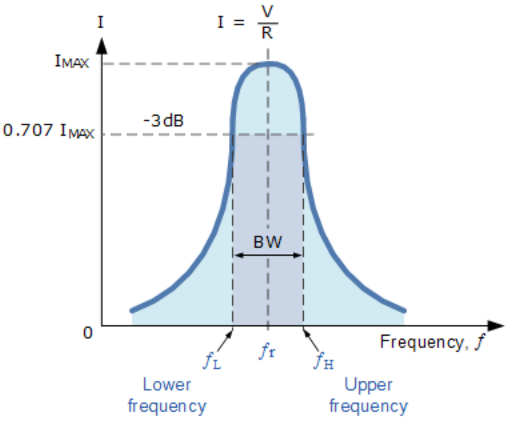
Series Resonance: A circuit is said to be under resonance, when the applied voltage 'V' and current are in phase. Thus a series RLC circuit, under resonance behaves like a pure resistance network and the reactance of the circuit should be zero. Since V & I are in phase, the power factor is unity at resonance. The frequency at which the resonance will occur is known as resonant frequency.

Resonant frequency,

$$f_0 = \frac{1}{2\pi\sqrt{LC}}$$

Thus at resonance the impedance Z is minimum. Since I = V/Z. The current is maximum So that current amplification takes place. Quality factor is the ratio of reactance power inductor (or) capacitor to its resistance.

MODEL GRAPHS:



f_L= lower cutoff frequency

f_H = upper cutoff frequency

f_r= Resonant Frequency

OBSERVATIONS:

	Theoretical	**Practical**
The resonance frequency (f_0)		
Lower Cut off Frequency f_L		
Upper Cut off Frequency f_H		
Bandwidth		
Quality Factor Q_0		

PROCEDURE:

1. Construct Series Resonance circuit as shown in fig1 using Multisim simulation software.
2. Plot graphs of amplitude versus frequency and phase shift versus frequency for RLC circuits.
3. Analyze the frequency response characteristics such as quality factor, cutoff frequency and bandwidth.

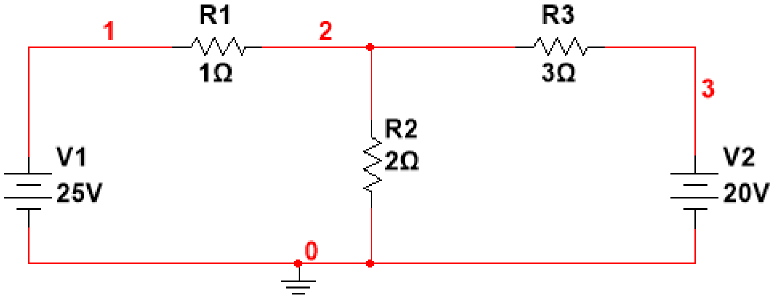
PRECAUTIONS:

1. Choose components properly from the built-in library to get accurate simulations.
2. Ensure proper grounding of circuits to avoid inaccurate results or simulation errors.

RESULT:

Hence, the resonant frequency, quality factor and band width of a given series and resonant circuits have been analyzed.

CIRCUIT DIAGRAM:



Fig

CALCULATIONS:

Expt. No: 12 Dt:	Determination of open circuit (Z) and short circuit (Y) parameters

AIM: To determine the Impedance (Z) and admittance (Y) parameters of a two port network.

APPARATUS REQUIRED:

S.No	Name of the equipment	Range	Type	Quantity
1	PC loaded with multisim software	-----	------	1

THEORY:

A pair of terminals between which a signal may enter or leave the network is known as port. If a network has one such type pair of terminals it is known as One-Port Network and that have two such type of ports is known as Two-Port Network.

If we relate the voltage of one port to the current of the same port, we get driving point admittance. On the other hand, if we relate the voltage of one port to the current at another port, we get transfer admittance. Admittance is a general term used to represent either the impedance or the admittance of a network. We will consider a general two-port network composed of linear, bilateral elements and no independent sources. The voltage and current at port -1 are V_1 and I_1 and at port -2 are V_2 and I_2. The position of V_1 and V_2 and the directions of I_1 and I_2 are customarily selected. Out of four variables only two are independent. The other two are expressed in terms of the independent variable of network parameters. The relation between the voltages and currents in terms of Z and Y parameters are as follows.

Z Parameters:

$V_1=Z_{11} (I_1) +Z_{12} (I_2)$

$V_2=Z_{21} (I_1) +Z_{22} (I_2)$

Y Parameters:

$I_1=Y_{11} (V_1) +Y_{12} (V_2)$

$I_2=Y_{21} (V_1) +Y_{22} (V_2)$

OBSERVATIONS:

Z-PARAMETERS:

Z-parameters	**Theoretical**	**Practical**
Z_{11}		
Z_{12}		
Z_{21}		
Z_{22}		

Y-PARAMETERS:

Y-Parameters	**Theoretical**	**Practical**
Y_{11}		
Y_{12}		
Y_{21}		
Y_{22}		

PROCEDURE:

1. Connections are made as per the circuit diagram.
2. Open circuit the port – 1 i.e., I_1=0, find the values ofV_1, I_2 and V_2.
3. Short circuit the port-1 i.e. V_1 =0, find the values of V_2, $I_1$and I_2.
4. Open circuit the port – 2 i.e., I_2=0, find the values of V_1, I_1 and V_2.
5. Short circuit the port-2 i.e. V_2 =0, find the values of V_1, $I_1$and I_2.
6. Find the Z and Y parameters of the given two port network.

PRECAUTIONS:

1. Choose components properly from the built-in library to get accurate simulations.
2. Ensure proper grounding of circuits to avoid inaccurate results or simulation errors.

RESULT:

Hence, the Impedance (Z) and admittance (Y) parameters of a two port network has been determined.

CIRCUIT DIAGRAM:

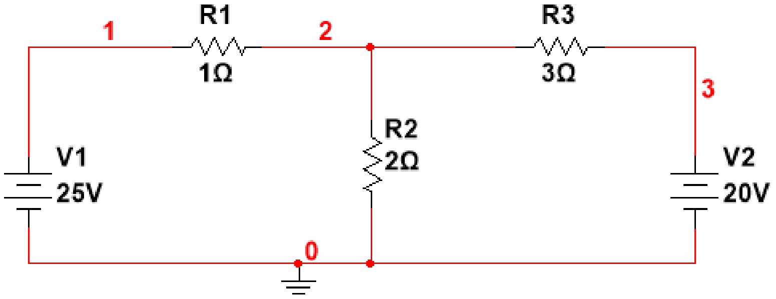


Fig:

CALCULATIONS:

Expt. No: 13	Determination of hybrid (h) and transmission (ABCD) parameters
Dt:	

AIM: To determine the Transmission and Hybrid parameters of a two port network.

APPARATUS REQUIRED:

S.No	Name of the equipment	Range	Type	Quantity
1	PC loaded with multisim software	-----	------	1

THEORY:

The relation between the voltages and currents of a two port network in terms of ABCD and h–parameters is given as follows.

ABCD PARAMETERS:

$V_1 = AV_2 - BI_2$

$I_1 = CV_2 - DI_2$

H-PARAMETERS

$V1 = h_{11}I1 + h_{12}V2$

$I2 = h_{11}I1 + h_{22}V2$

ABCD PARAMETERS:

$A = \frac{V1}{V2}$ / I2 =0

$B = \frac{-V1}{I2}$ / $V2$ =0

$C = \frac{I1}{V2}$ / I2 =0

$D = \frac{-I1}{I2}$ / V2 =0

H-PARAMETERS:

$h11 = \frac{V1}{I1}$ / V2 =0

$h12 = \frac{V1}{V2}$ / I1 =0

$h21 = \frac{I2}{I1}$ / V2 =0

$h22 = \frac{I2}{V2}$ / I1 =0

OBSERVATIONS:

ABCD -PARAMETERS:

ABCD-parameters	Theoretical	Practical
A		
B		
C		
D		

h-PARAMETERS:

h-Parameters	Theoretical	Practical
h_{11}		
h_{12}		
h_{21}		
h_{22}		

PROCEDURE:

1. Connections are made as per the circuit diagram.
2. Open circuit the port - 1 i.e., I_1=0 find the values ofV_1, I_2 and V_2.
3. Short circuit the port-1 V_1 =0 find the values of V_2, $I_1$and I_2.
4. Open circuit the port -2 i.e., I_2=0 find the values of V_1, I_1 and V_2.
5. Short circuit the port-2 i.e. V_2 =0 find the values of V_1, $I_1$and I_2.
6. Find the ABCD and h-parameters of the given two port network from the above data.

PRECAUTIONS:

1. Choose components properly from the built-in library to get accurate simulations.
2. Ensure proper grounding of circuits to avoid inaccurate results or simulation errors.

RESULT:

Hence, the ABCD and h-parameters of a two port network has been determined.

CIRCUIT DIAGRAM:

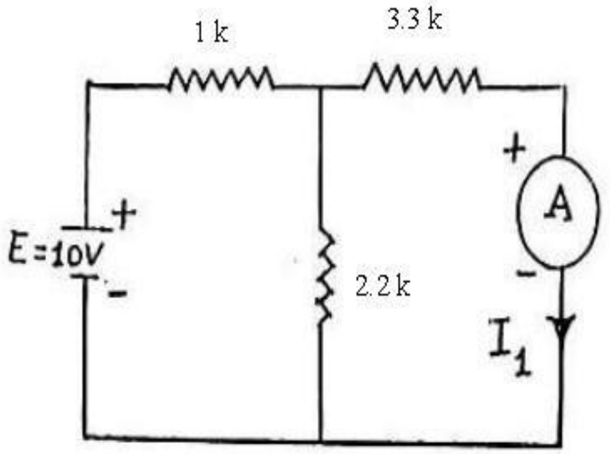


Figure.1

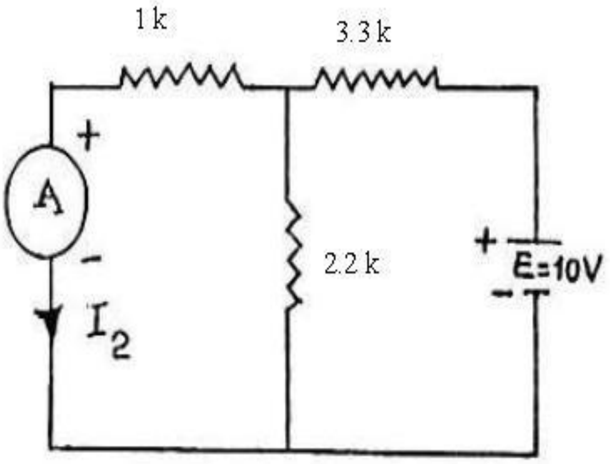


Figure.2

CALCULATIONS:

OBSERVATIONS:

Parameter	Theoritical	Practical
V(Volts)		
I_1		
I_2		
$V/I_1 = V/I_2$		

Expt. No: 14 Dt:	**Verification of Reciprocity Theorem**

AIM: To verify reciprocity theorem by observing source to excitation ratio.

APPARATUS REQUIRED:

S.No	Name of the equipment	Range	Type	Quantity
1	PC loaded with multisim software	-----	------	1

THEORY:

"In a linear bi-lateral single source network, the ratio of excitation to the response is constant when the position of excitation and response are interchanged".

PROCEDURE:

1. Connect the circuit as per circuit diagram.
2. Adjust the supply voltage to 10 V in Figure 1
3. Note down the response of I_1 through 3.3 kΩ resistor and tabulate the values.
4. The circuit connections are changed as per the Figure 2, change the source to load end and replace the source with internal resistance.
5. Note down the response of I_2 through 1 kΩ resistor and tabulate the values.
6. Compare the theoretical & practical values and prove reciprocity theorem

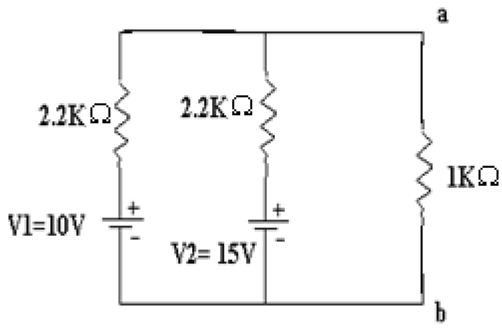
PRECAUTIONS:

1. Choose components properly from the built-in library to get accurate simulations.
2. Ensure proper grounding of circuits to avoid inaccurate results or simulation errors.

RESULT:

Hence, the reciprocity theorem has been verified for the given circuit.

CIRCUIT DIAGRAM:



PRACTICAL CIRCUITS:

CIRCUIT-1

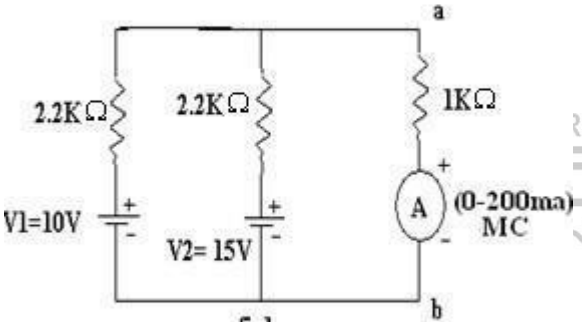


fig1

CIRCUIT-2:

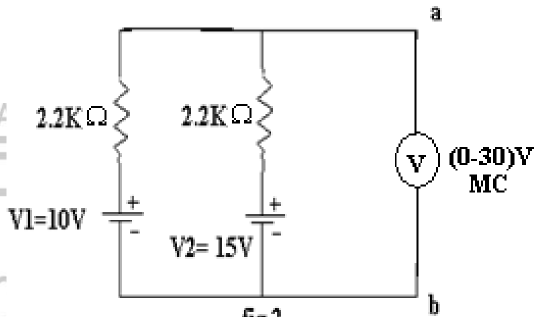


fig 2

CIRCUIT-3:

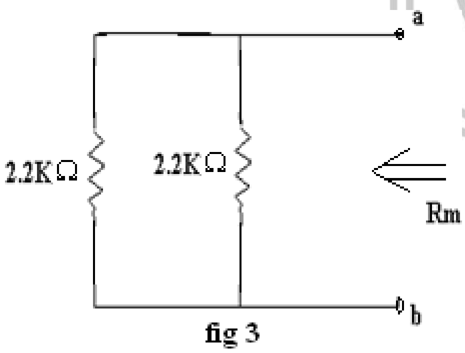


fig 3

CIRCUIT-4:

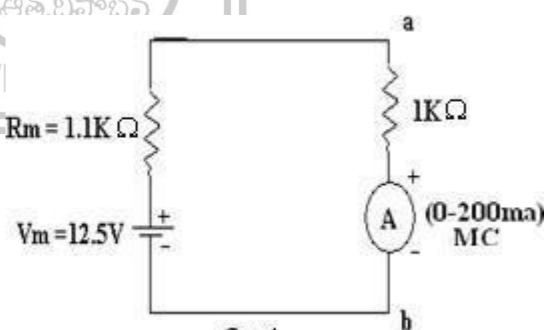


fig 4

Expt. No: 15	Verification of Millman's Theorem
Dt:	

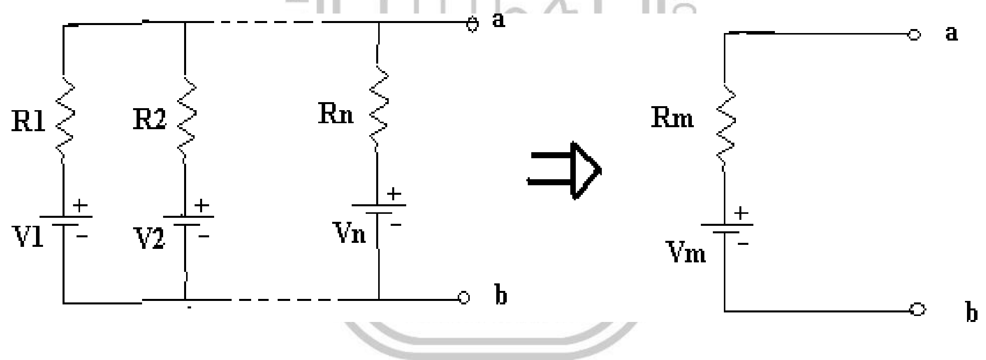
AIM: To verify Millman's theorem for the given circuit.

APPARATUS REQUIRED:

S.No	Name of the equipment	Range	Type	Quantity
1	PC loaded with multisim software	-----	------	1

THEORY:

Millman's theorem states that in any network, if the voltage sources V1, V2,……..Vn in series with internal resistances R1,R2,…….Rn, respectively are in parallel then these sources may be replaced by a single voltage source "Vm" in series with "Rm" as shown.



According to Millman's theorem,

$$V_m = \frac{V_1 G_1 + V_2 G_2 + \cdots + V_n G_n}{G_1 + G_2 + \cdots + G_n}$$

$$R_m = \frac{1}{G_1 + G_2 + \cdots + G_n}$$

CALCULATIONS:

OBSERVATION TABLE:

S.No	V_1 (volts)	V_2 (volts)	I_L (mA)

Table 1

S.No	V_1 (volts)	V_2 (volts)	V_m (volts)	I_L (mA)

Table 2

PROCEDURE:

1. Make the connections as per the circuit diagram shown in fig 1.
2. Apply the source voltages V_1=10V & V_2=15V.
3. Note down the readings of ammeter and tabulate in table1.
4. Make the connections as per the circuit diagram shown in fig 2.
5. Switch on the supply, apply the source voltages V_1=10V & V_2=15V.
6. Note down the readings of ammeter and tabulate in table2
7. Make the connections as per the circuit diagram shown in fig 3 and determine R_M using Multimeter
8. Make the connections of equivalent Millman's circuit as shown in fig 4.
9. Switch on the supply, apply the millman voltage Vm (calculated) and note down the readings of ammeter and tabulate in table2.
10. Repeat the experiment at different source voltages and compare the readings.

PRECAUTIONS:

1. Choose components properly from the built-in library to get accurate simulations.
2. Ensure proper grounding of circuits to avoid inaccurate results or simulation errors.

RESULT:

Hence, the Millman's theorem has been verified for the given circuit.