

UMA CHARAN PATTNAIK ENGINEERING SCHOOL BERHAMPUR



CIRCUIT THEORY AND SIMULATION LAB MANUAL [3RD SEM ETC/EE - PR.2]

Name of the Student _____

Regd. No of the Student _____

Laboratory Section _____

[This Laboratory manual will guide you to perform the necessary experiments to meet the fulfillment of SCTE & VT Syllabus for **CIRCUIT THEORY & SIMULATION LAB** for 3rd SEMESTER Students of **ETC & EE**. This manual also consists of the supplementary materials like data sheets for some useful components to be used throughout the course.]

TABLE OF CONTENTS

Expt. No	LIST OF EXPERIMENTS	Page No.	Date	Remark
i	RULES & REGULATIONS	2		
ii	FORMAT FOR RECORD WRITING	3		
iii	MARK DISTRIBUTION	4		
iv	SCTE & VT, ODISHA SYLLABUS	5		
1.	Equivalent Resistance in Series and Parallel Circuit	06-07		
2.	a) Kirchoff's Current Law	08-09		
	b) Kirchoff's Voltage Law	10-11		
3.	Verification of Super Position Theorem	12-14		
4.	Verification of Thevenin's Theorem	15-18		
5.	Verification of Norton's Theorem	19-20		
6.	Verification of Maximum Power Transfer Theorem	21-22		
7.	Low-Pass Filter	23-24		
8.	High Pass Filter	25		
9.	R-L-C Series and Parallel Resonance Circuit	26-27		

---ALL---THE---BEST---ALL---THE---BEST---

RULES & REGULATIONS

1. **Don't be late** to the lab.
2. Put your signature on attendance sheet first, submit the record and then go to your place correspondence to experiment kit.
3. **Without fair record and lab manual** you are **not allowed** to the lab.
4. Handlings of other things, which are not related to the experiment, are strictly prohibited inside the lab.
5. You may be charged with full cost and additional penalty for destroying the equipments.
6. You should put on your own identity card.
7. Till the time you are inside the lab you should stick to your place and don't move here & there.
8. Before leaving the lab, be sure that power supply is switched off and you have returned the components proper manner otherwise all the group members will be punished.
9. Your duly filled lab manual along with stressing sheet must be verified by any of the instructors present there else your record will not be considered.
10. You yourself will be responsible for exchange or missing of any precious materials.
11. You can demand for marks on the basis of performance ONLY, but can't beg.
12. **Extra labs may be allowed** on your request if your application is forwarded through proper channel **within the time limit**. Beyond the time limit it is never accepted at any situation.

FORMAT FOR RECORD WRITING

1. Maintain the contain page with date of performing the experiment, experiment name, page number.
2. Put the page numbers on the top of the individual page, Date & experiment no.
3. Write the aim of the experiment, apparatus required short theoretical description with required circuit/block diagram, observation & conclusion.
4. Attached the tracing sheet to the left side of the paper.
5. Put your full signature at bottom right corner of the last page of that experiment.
6. Record should be very neat and clean and the entire circuits/block diagram should be drawn with **pencil only**.
7. **Bad hand** writing will **affect** the record mark.
8. **Incomplete record** will lead towards **negative** marking
9. **Data on lab manual** should match with fair record.
10. Arrange the pages properly according to the experiment list (ascending order).
11. Use the record and internal pages **without rolling (SUPPLIED format ONLY)**.
12. Don't submit loose sheets without record at any condition.

FORMAT

1. Aim of the Experiment
 2. Requirements
 3. Theory (Theoretical Circuit/block diagram must be included)
 4. Observation/Calculation (if any)
 5. Conclusion (On the basis of experiment result)
-

MARK DISTRIBUTION

Sl. No.	PARTS	MARKS (%)
1	Attendance	20
2	Record	20 + 10
3	Experiment	30
4	Viva-Voice/ Quiz	20
	TOTAL	100

I hereby Mr./Ms.....with Regd. No....., Semester & Branch have understood the above rules and regulations of the **Circuit Theory and Simulation Lab** and **will accept any kind of punishment given by the instructor for violating the rules** mentioned above.

Date:

Signature of the student

SCTE & VT SYLLABUS

(w.e.f : 2019-20 Session)

3rd Semester Electrical

Pr2. CIRCUIT AND SIMULATION LAB

Name of the Course: Diploma in Electrical Engineering			
Course code:		Semester	3 rd
Total Period:	90	Examination :	3hrs
Lab. periods:	6 P / week	Sessional:	50
Maximum marks:	100	End Semester Examination ::	50

A. Rationale:

The response of Electrical Circuit can be verified practically by applying different theorems and fundamental techniques. The students will become sure that the theoretical tricks which they have learned from books are true. The students will become competent in the field of circuit analysis

B. Objective:

On completion of the lab course the student will be able to:

1. Verify the theorems using different components.
2. Know the various types of filters.
3. Simulate different circuits using P-Spice/MATLAB software.

C. Course content in terms of specific objectives:

1. Measurement of equivalent resistance in series and parallel circuit
2. Measurement of power and power factor using series R-L-C Load.
3. Verification of KCL and KVL.
4. Verification of Super position theorem
5. Verification of Thevenin's Theorem
6. Verification of Norton's Theorem
7. Verification of Maximum power transfer Theorem
8. Determine resonant frequency of series R-L-C circuit.
9. Study of Low pass filter & determination of cut-off frequency
10. Study of High pass filter & determination of cut-off frequency
11. Analyze the charging and discharging of an R-C & R-L circuit with oscilloscope and Compute the time constant from the tabulated data and determine the rise time graphically.
12. Construct the following circuits using P-Spice/MATLAB software and compare the measurements and waveforms.
 - i. Superposition theorem
 - ii. Series Resonant Circuit
 - iii. Transient Response in R-L-C series circuit

--ALL--THE--BEST--ALL--THE--BEST--

EXPERIMENT NO – 01

Equivalent Resistance in Series and Parallel Circuit

AIM OF THE EXPERIMENT: - To measure the equivalent resistance in both series and parallel circuit.

COMPONENT REQUIRED:-

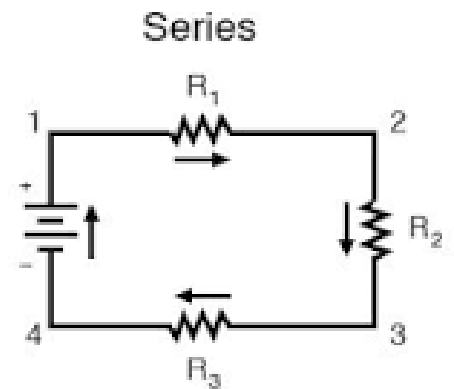
Sl. No	Name of the Components	Specification	Quantity
1	Resistors	500 Ω, 680 Ω, 330 Ω, 390 Ω, 270 Ω	Each of 2
2	Multimeter	Digital	1
3	Software	Multisim 14.1	As required

THEORY:-

SERIES CIRCUIT:-

➤ In electrical circuit is in series connected the current flowing through the conductor is constant but voltage is not constant and the resistance can be calculated by,

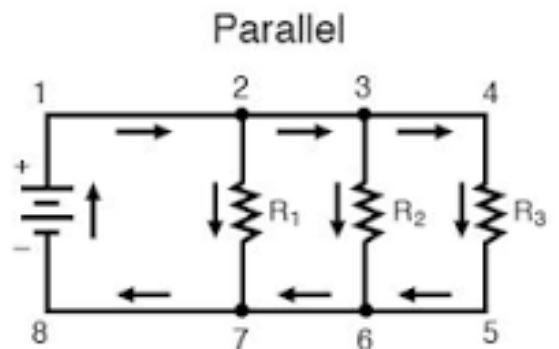
$$R_s = R_1 + R_2 + R_3 + \dots + R_n$$



PARALLEL CIRCUIT:-

➤ In electrical series is in parallel connected the current flowing through the conductor is not constant but voltage is remain constant and the resistance can be calculated by,

$$\frac{1}{R_p} = \frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3} + \frac{1}{R_4} + \dots + \frac{1}{R_n}$$



PROCEDURE:-

1. Connected the resisted the resistor as per circuit diagram.
2. Measured the individual resistance of different resistor with the help of multimeter.
3. Measured the total equivalent resistance as per circuit diagram by multimeter.
4. Compare the observed value and calculation value in both the parallel and series.

CALCULATION:-

Let two resistor are connected in series then the total or equivalent resistor is,

$$R_1 = 330 \Omega, R_2 = 390 \Omega \rightarrow R_s = R_1 + R_2 = 330 + 390 = 720 \Omega$$

If, they are connected in parallel then the equivalent resistance is,

$$R_1 = 330 \Omega, R_2 = 390 \Omega$$

$$\frac{1}{R_p} = \frac{1}{330} + \frac{1}{390} \quad \rightarrow \quad R_p = \frac{330 \times 390}{330 + 390} = 178.8 \Omega$$

OBSERVATION TABLE:-

Sl No	R ₁ in (Ω)	R ₂ in (Ω)	RESISTANCE IN SERIES		RESISTANCE IN PARALLEL	
			Calculation	Observation	Calculation	Observation
1	500	680	1.18KΩ	1.18KΩ	293.2Ω	288.8Ω
2	270	680	950Ω	958Ω	197.7Ω	193.26Ω
3	330	390	721Ω	720Ω	118.5Ω	178.8Ω

CONCLUSION:-

From the above experiment it we have studied and verified that the observation value is approximately same to the calculation value in both parallel and series circuit.

DATE:

Signature of the Student

NAME:

ROLL NO.:

SEMESTER:

GROUP NO.

DATE:

Instructor's Signature

---ALL---THE---BEST---ALL---THE---BEST---

EXPERIMENT NO – 2 (A)

KIRCHHOFF'S CURRENT LAW

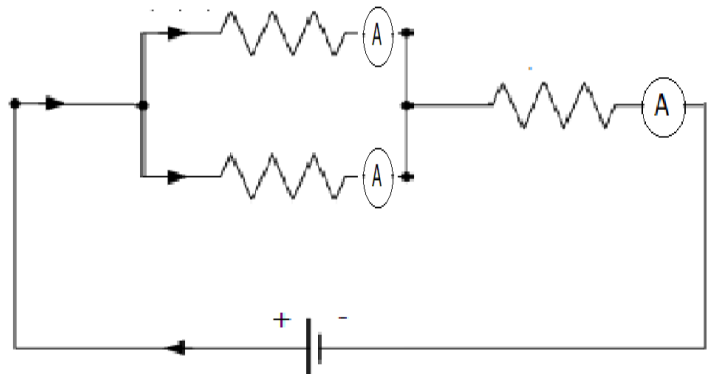
AIM OF THE EXPERIMENT: - To study & verify Kirchoff's current law (KCL).

EQUIPMENT REQUIRED: -

SI No	Name of the Components	Specification	Quantity
1	Verification Kit	(OMEGA-ETB-201)	1
2	Patch Cord	-----	As per required
3	Power Supply	0-12 Volt	-----
4	Multimeter	Digital meter	1

THEORY: -

- KCL states that the algebraic sum of all the current meeting at a point or junction is equal to zero.
- It can be stated that total incoming current at a point will be equal to the total out going current.
- For verification of kcl we consider the given circuit.



PROCEDURE: -

1. Connect the circuit as per the circuit diagram.
2. Vary the voltage to take 5 different reading.
3. Observe different ammeter reading for each input voltage.
4. Compare the reading with the total current following the ckt.

CALCULATION: -

$$R_1 = 270\Omega \quad R_2 = 330\Omega$$

$$R_3 = 500\Omega \quad V = 9V$$

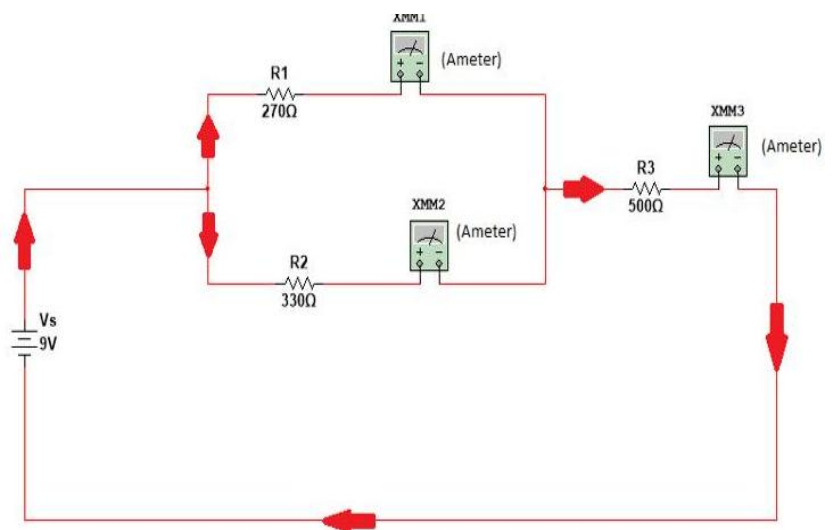
$$R_{eq} = (270 \parallel 330) + 500$$

$$= 148.5 + 500 = 648.5\Omega$$

$$I_3 = \frac{V}{R_{eq}} = \frac{9}{648.5} = 0.013A = 13mA$$

$$I_1 = \frac{(0.013) \times 330}{270 + 330} = 7.15mA$$

$$I_2 = \frac{(0.013) \times 270}{270 + 330} = 5.85mA$$



$$\rightarrow I_1 + I_2 = 7.15mA + 5.85mA = 13mA$$

OBSERBATION TABLE: -

SI No	Input voltage	I_1 (270 Ω) in mA	I_2 (330 Ω) in mA	I_3 (500 Ω) in mA	$I_1 + I_2$ in mA
1.	10V	8.53	6.95	15.49	15.48
2.	9V	7.60	6.2	13.9	13.8
3.	8V	6.84	5.58	12.44	12.42
4.	7V	6	4.89	10.91	10.89
5.	5V	4.30	3.51	7.82	7.81

CONCLUSION: -

From the above experiment sum of all input current at a junction is equal to the sum of outgoing current from the junction.

DATE:

Signature of the Student

NAME:

ROLL NO.:

SEMESTER:

GROUP NO.

DATE:

Instructor's Signature

---ALL---THE---BEST---

EXPERIMENT NO – 2 (B)

KIRCHHOFF’S VOLTAGE LAW

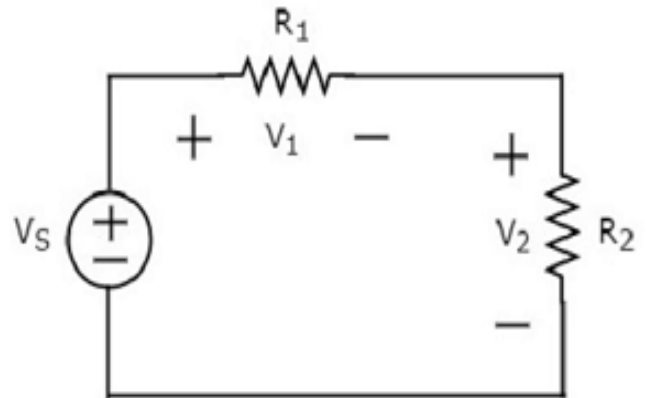
AIM OF THE EXPERIMENT: - To study & verify Kirchoff’s voltage law (KVL).

EQUIPMENT REQUIRED: -

SI No	Name of the Components	Specification	Quantity
1	Verification Kit	(OMEGA-ETB-201)	1
2	Patch Cord	-----	As per required
3	Power Supply	0-12 Volt	-----
4	Multimeter	Digital meter	1

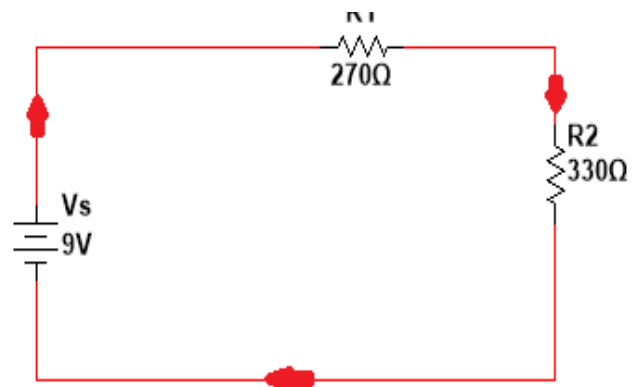
THEORY: -

- KVL states that the algebraic sum of ‘EMF’ and product of current and resistance in a closed loop is equal to zero.
- For the verification of this theorem we have taken a circuit as shown in the figure.
- In the given circuit we have one ‘EMF’ and two resistance value 270 Ω and 330 Ω.
- The voltage across 270 Ω resistor is taken ‘V₁’ and across 330 Ω resistor is taken ‘V₂’.



PROCEDURE: -

1. Connect circuit as per the circuit diagram.
2. Give the power apply to the circuit.
3. Now the measure the voltage across each resistor using Multimeter and note down the observe value in the observation table.
4. Now add all the three values of voltage obtain and compare it with the emf value.
5. This procedure may be respected for variable voltage values.



CALCULATION: -

Theoretically applying KVL to the given circuit, $V - IR_1 - IR_2 = 0$

$$\Rightarrow 9 - I \times 270\Omega - I \times 330\Omega = 0$$

$$\Rightarrow 9 - I \times (270 + 330) = 0$$

$$\Rightarrow 9 = I \times (270 + 330) \quad \rightarrow I = \frac{9}{270 + 330} = 0.015A \quad \rightarrow$$

I = 15 mA

OBSERBATION TABLE:-

SI No	V ₁ (270 Ω)	V ₂ (330 Ω)	V _t (V ₁ +V ₂)	Total EMF Applied
1	4.57	5.70	10.27	10V
2	3.98	4.88	8.86	9V
3	3.22	3.91	7.13	7V
4	2.30	2.80	5.10	5V
5	0.90	1.10	2.00	2V

CONCLUSION:-

From the above experiment we observe that sum of emf and voltage drop is equal to zero.

DATE:

Signature of the Student

NAME:

ROLL NO.:

SEMESTER:

GROUP NO.

DATE:

Instructor's Signature

---ALL---THE---BEST---ALL---THE---BEST---

EXPERIMENT NO – 3

SUPER POSITION THEOREM

AIM OF THE EXPERIMENT: -To verify and verify the super position theorem.

EQUIPMENT REQUIRED: -

SL. NO	NAME OF THE COMPONENT	SPECIFICATION	QUANTITY
01	VERIFICATION KIT	OMEGA TYPE - ETB 201	01
02	POWER SUPPLY	0-12 V	-
03	PATCH CORDS	-	As required
04	MULTIMETER	-	-

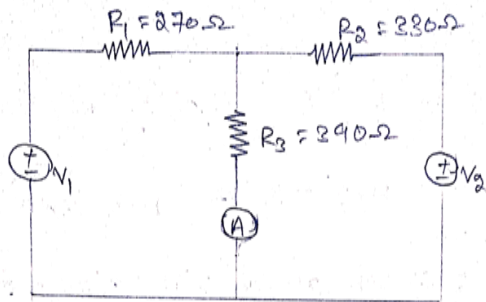
THEORY: -

In any linear bilateral network containing two or more independent sources (voltage or current sources or combination of voltage and current sources), the resultant current / voltage in any branch is the algebraic sum of currents / voltages caused by each independent source acting along, with all other independent sources being replaced meanwhile by their respective internal resistances.

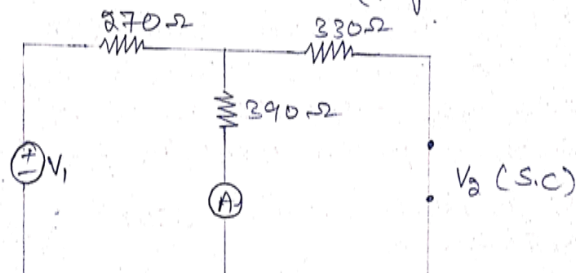
The voltage source replaced by short circuit and the current circuit replaced by open circuit. The voltage source replaced by short the resistances of the source are replaced at the time of source elimination.

If the current produced by one source is in one direction while that produced by the other is in the opposite direction through the same resistor, the resulting current is the difference of the two and has the direction of the larger current. If the individual currents are in the same direction, the resulting current is the sum of two and has the direction of either current..

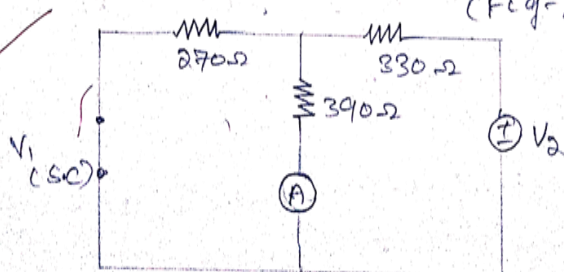
The total power delivered to a resistive element must be determined using the total current through or the total voltage across the element and cannot be determined by a simple sum of the power levels established by each source.



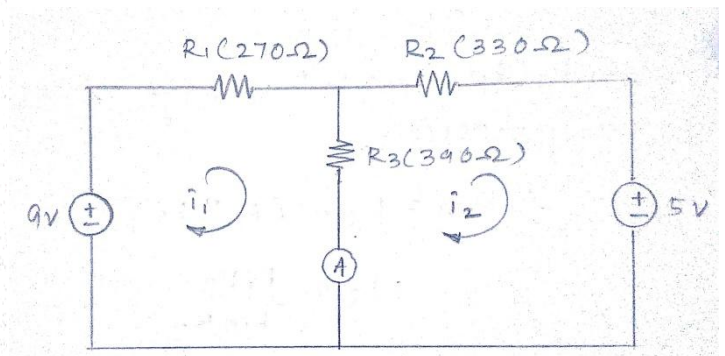
(Fig-1)



(Fig-2)



(Fig-3)



PROCEDURE: -

- Connect the power supply to the verification kit.
- Make the connection as per the circuit diagram.
- Remove V_2 and close the circuit through a patch cord.
- Measure I_{31} in the ammeter.
- Now put V_2 in the circuit and remove V_1 from the circuit. Close the circuit through a patch cord in place of V_1 .
- Now replace and measure I_{32} in the ammeter.
- Now replace V_1 and switch on both the sources.

CALCULATION: -

Now eliminate the V_2 voltage from the circuit , $R_{eq} = R_1 + \frac{R_2 R_3}{R_2 + R_3}$

Due to voltage source V_1 , $I_{31} = I_1 \times \frac{R_2}{R_2 + R_3} = \frac{V_1}{R_{eq}} \times \frac{R_2}{R_2 + R_3}$

Now eliminate the V_1 voltage source from the circuit, $R_{eq} = R_2 + \frac{R_1 R_3}{R_1 + R_3}$

Due to voltage source V_2 , $I_{32} = I_2 \times \frac{R_1}{R_1 + R_3} = \frac{V_2}{R_{eq}} \times \frac{R_1}{R_1 + R_3}$

So, $V_1 = 9\text{ V}$, $V_2 = 5\text{ V}$, $R_1 = 270\ \Omega$, $R_2 = 330\ \Omega$, $R_3 = 390\ \Omega$,

$$R'_{eq} = 448.75\ \Omega \text{ , } R''_{eq} = 489.54$$

$$\text{So, } I_{31} = \frac{9}{448.75} \times \frac{330}{330+390} = 9.19\text{ mA}$$

$$\text{Similarly, } I_{32} = \frac{5}{489.54} \times \frac{270}{270+390}$$

$$I_3 = I_{31} + I_{32} = 9.19 + 4.17 = \mathbf{13.36\text{ mA}}$$

Loop-1 , $9 - 270I_1 - (I_1 - I_2) 390 = 0$

$$9 - 270I_1 - 390I_1 + 390I_2 = 0$$

$$\Rightarrow 660I_1 - 390I_2 = 9 \text{ (i)}$$

Loop-2 , $- 330I_2 - 5 - (I_2 - I_1)390 = 0$

$$\Rightarrow -330I_2 - 5 - 390I_2 + 390I_1 = 0$$

$$\Rightarrow 390I_1 - 720I_2 = 5 \text{ (ii)}$$

Now solving the equations (i) and (ii) we get ,

$$I_1 = 0.014 = 14\text{ mA} \text{ , } I_2 = 6.5 \times 10^{-4} = 0.65\text{ mA}$$

$$\text{So , } I = I_1 - I_2 = 14 - 0.65 = 13.35\text{ mA}$$

TABULATION: -

Sl No.	V ₁ (Volt)	V ₂ (Volt)	Total I ₃ where both (V ₁) and (V ₂) active	Current where (V ₁) active I ₃₁	Current where (V ₂) active I ₃₂	Total (I = I ₃₁ + I ₃₂)
01	9	5	13.04	8.97	4.09	13.06
02	9	7	14.65	8.97	5.65	14.26
03	9	9	16.29	8.97	7.30	16.27
04	9	10	17.28	8.97	8.12	17.09
05	9	11	17.98	8.97	8.97	17.64

CONCLUSION: -

From the above experiment we studied and observed that different branch current of the circuit using Super position theorem.

---ALL---THE---BEST---ALL---THE---BEST---

EXPERIMENT NO – 4

VERIFICATION OF THEVENIN’S THEORM

AIM OF THE EXPERIMENT:-To verify Thevenin’s theorem and calculate Thevenin’s equivalent of given circuit.

EQUIPMENTS REQUIRED

SI No	Name of the Components	Specification	Quantity
1	Resistors	500 Ω, 680 Ω, 330 Ω, 270 Ω	As Required
2	Multimeter	Digital	1
3	Connecting wire	-----	-----
4	DC power supply	-----	-----

THEORY

Any linear active 2 terminal n/w consisting in of voltage and current source with some resistance. It can be replaced by an equivalent Thevenin’s voltage source or voltage source having its value equal to the Thevenin’s equivalent voltage with a series resistor which is known as Thevenin’s resistance. The equivalent voltage source is represented by ‘ V_{th} ’ and equivalent resistance is represented by ‘ R_{th} ’. To find the Thevenin’s equivalent voltage first we have to open circuit the load terminals. The open circuited voltage V_{AB} is the required Thevenin’s voltage. We have again equal to the voltage across the point ‘P’ and ‘Q’ so $V_{PQ} = V_{AB} = V_{TH}$.

PROCEDURE

1. Start – Electronics workbench – Multisim 14.1
2. Select the component from place Component library according to given circuit diagram.
3. Connect the multimeter.
4. Make connection as per a circuit diagram.
5. Simulate Run.
6. Double click to the multimeter.
7. See the output result.

CALCULATION

$$R_1 = 300 \text{ Ohm}, R_2 = 500 \text{ Ohm}, R_3 = 680 \text{ Ohm}, R_L = 9v$$

STEP- 1

Calculate the V_{th} across load AB terminal and open the 270 ohm resistor.

$$V_s = IR_1 + IR_2 = I(R_1 + R_2)$$

$$I = V_s / R_1 + R_2 = 9 / 330 + 500 = 1.08 \text{ mA}$$

$$V_{th} = V_{AB} = I \times 500 = 1.08 \times 500 = 5.42 \text{ V}$$

STEP-2

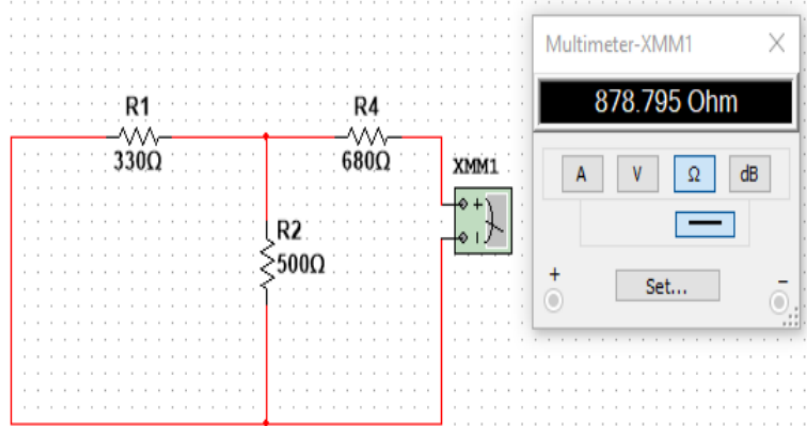
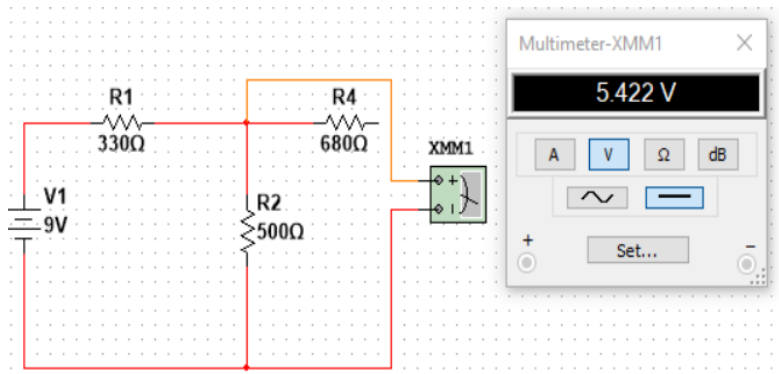
Calculate the R_{th} across AB terminal by short circuit the voltage source.

$$R_{ab} = (330 \parallel 500) + 680 = \frac{330 \times 500}{330 + 500} + 680 = 878.795 \text{ Ohm}$$

Then find I_{th} in the Thevenin's equivalent circuit,

$$I_{th} = \frac{V_{th}}{R_{th} + R_l} = \frac{5.422}{878.795 + 270} = 4.71 \text{ mA}$$

I_l in find I will be equal to the I in thevenin's equivalent circuit.



TABULATION

CALCULATED TABLE: -

Sl No.	Applied voltage in V	V_{th} In Volt	R_{th} in ohm	I_L in mA
01	09	5.42	878.795	4.72

OBSERVATION TABLE: -

Sl. No	Applied voltage in volt	V_{th} in volt	R_{th} in ohm	I_L in mA
01	07	4.217	878.795	3.671
02	08	4.819	878.795	4.195
03	09	5.422	878.795	4.72
04	10	6.020	878.795	5.244
05	11	6.627	878.795	5.768
06	12	7.229	878.795	6.293

CONCLUSION

From the above experiment we know that how to verify Thevenin's theorem by using multisim14.1.

---ALL---THE---BEST---

EXPERIMENT NO – 4

THEVININ'S THEOREM

AIM OF THE EXPERIMENT: - To verify Thevenin's theorem and calculate Thevenin's equivalent of the given circuit.

EQUIPMENT REQUIRED: -

- I. Software Multisim – 14.1
- II. Components: -
 - a) Resistors – 330 Ω, 500Ω, 680Ω, 270Ω
 - b) DC power source – 10 V
- III. Instruments – Multimeter

THEORY: -

Any linear active two terminal networks consisting of voltage and current source with some resistances. It can be replaced by an equivalent Thevenin's voltage source or voltage source having its value is equal to the Thevenin's equivalent voltage with a series resistor which is known as Thevenin's resistance.

The equivalent voltage source is represented by V_{th} and equivalent resistance is represented by R_{th} . To find the Thevenin's equivalent voltage, first we have to open the circuit load terminals. The open circuited voltage V_{AB} is the required. Thevenin's voltage we charge again equal to the voltage across the point P and Q.

$$\text{So, } V_{PQ} = V_{AB} = V_{th}$$

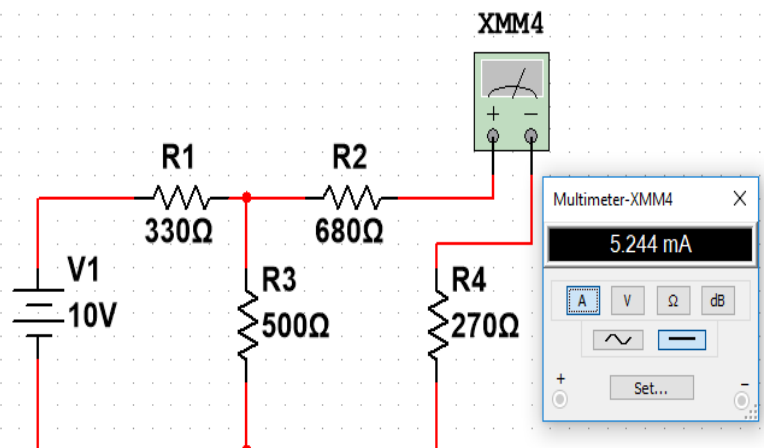
PROCEDURE: -

- i. Start → Electronics workbench → Multisim-14.1
- ii. Select the component from place → Component library according to given circuit.
- iii. Connect the multimeter.
- iv. Make the connections as per circuit diagram.
- v. Stimulate → Run.
- vi. Double click on the multimeter.
- vii. See the output results.

CALCULATION: -

STEP-1

First draw the given circuit and measure the flow of current through it.



STEP – 2

Let load terminal be the 270Ω. Remove the load terminal and apply KVL to loop in figure- 2

$$10 - 330I_1 - 500I_1 = 0$$

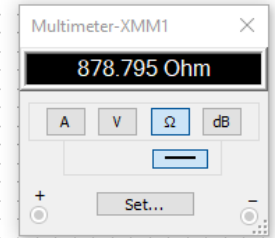
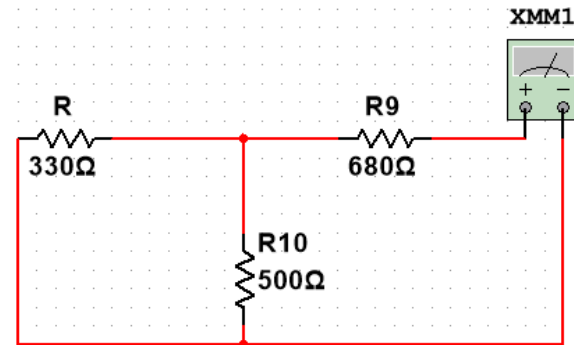
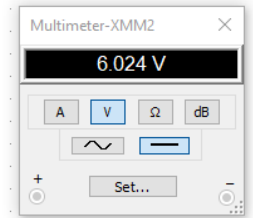
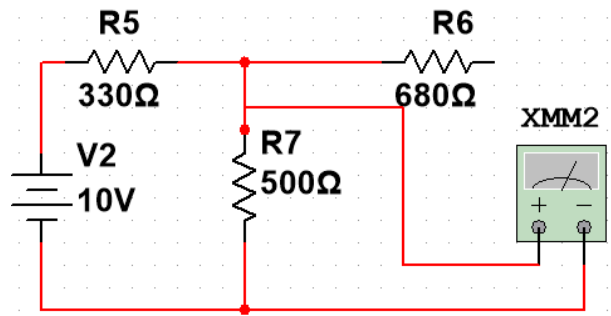
$$\Rightarrow 830 I_1 = 10$$

$$\Rightarrow I_1 = \frac{10}{830}$$

$$\Rightarrow I_1 = 0.012 \text{ A}$$

$$\Rightarrow I_1 = 12 \text{ mA}$$

$$\begin{aligned} \text{So, } V_{th} &= 500I_1 \\ &= 500 \times 0.012 = 6.02 \text{ V} \end{aligned}$$



STEP – 3

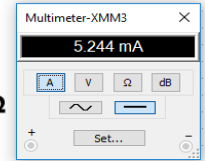
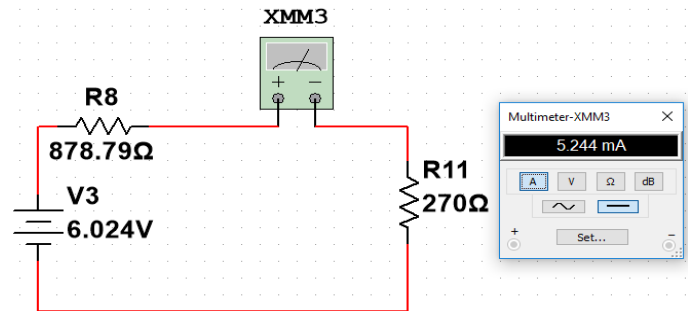
Short the voltage source and also find equivalent resistance as shown in figure-3,

$$\text{So, } R_{th} = \frac{R1R2}{R1+R2} + R_3 = \frac{330 \times 500}{330+500} + 680 = 878.795 \Omega$$

STEP – 4

Now draw the Thevenin's equivalent circuit and find the current across the circuit.

$$I_L = \frac{V_{th}}{R_{th} + R_L} = \frac{6.02}{878.795 + 270} = 5.24 \text{ mA}$$



TABULATION: -

CALCULATED TABLE: -

Sl No.	Applied voltage in V	V _{th} In Volt	R _{th} in ohm	I _L in mA
01	10	6.02	878.795	5.24

OBSERVATION TABLE: -

Sl No	Applied voltage in volt	V _{th} in volt	R _{th} in ohm	I _L in mA
01	07	4.217	878.795	3.671
02	08	4.819	878.795	4.195
03	09	5.422	878.795	4.72
04	10	6.020	878.795	5.244
05	11	6.627	878.795	5.768
06	12	7.229	878.795	6.293

CONCLUSION: -

From the above experiment we verified and studied Thevenin's Theorem by using Multisim 14.1 software.

EXPERIMENT NO – 5

VERIFICATION OF NORTON’S THEOREM

AIM OF THE EXPERIMENT: - To study and verify Norton’s theorem.

COMPONENT REQUIRED:-

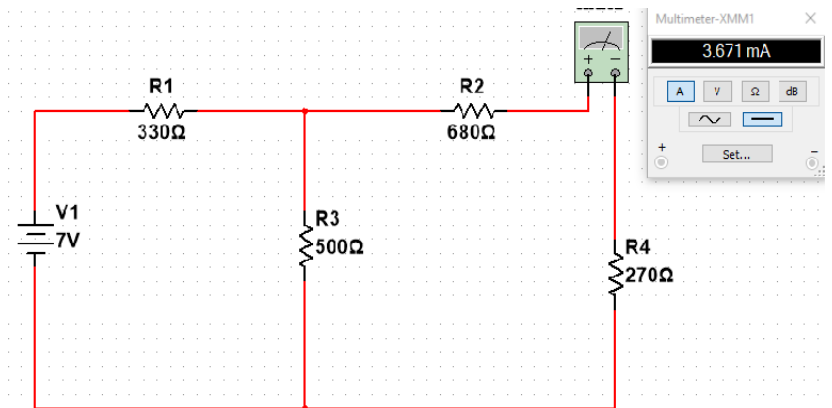
SI No	Name of the Components	Specification
1	Software	Multisim-14.1
2	Resistors	500 Ω, 680 Ω, 330 Ω, 270 Ω
3	DC Power source	7 Volt
4	Multimeter	As required

THEORY:-

- In any linear bilateral network containing one or more voltage source can be replaced by an equivalent circuit.
- Consisting of current [I_N] in parallel with the equivalent resistance.
- In is the short circuited current following through the load terminals.

PROCEDURE:-

1. Start → Electronics work bench → Multisim 14.1.
2. Select component from place →Component library according to following circuit.
3. Connect the multimeter.
4. Make connection according.
5. Simulate →Run
6. Double click on the multimeter.
7. See the output result.



CALCULATION:-

STEP - 1

First draws the given original circuit.

STEP - 2

Assume load resistance as short circuited and calculate Norton’s equivalent current as short circuited path.

Apply mesh analysis, in loop 1 we get, $\rightarrow 7 - 330I_1 - 500I_1 + 500I_2 = 0$

$\rightarrow 7 - 830I_1 + 500I_2 = 0 \rightarrow 830I_1 - 500I_2 = 7$ ----- (1)

Apply mesh analysis, in loop 2 we get, $\rightarrow - 500I_2 + 500 I_1 - 680I_2 = 0$

$\rightarrow -1180I_2 + 500 I_1 = 0 \rightarrow 500 I_1 - 1180I_2 = 0$ ----- (2)

By comparing or calculating Eq.1 & Eq.2 we get, $I_1 = 0.011A = 11mA$ & $I_2 = 4.79mA$

So, current across short circuit path or Norton's equivalent current $[I_N] = 4.79\text{mA}$

STEP - 3

Assume load resistance as open circuit and find equivalent resistance or Norton's equivalent resistance,

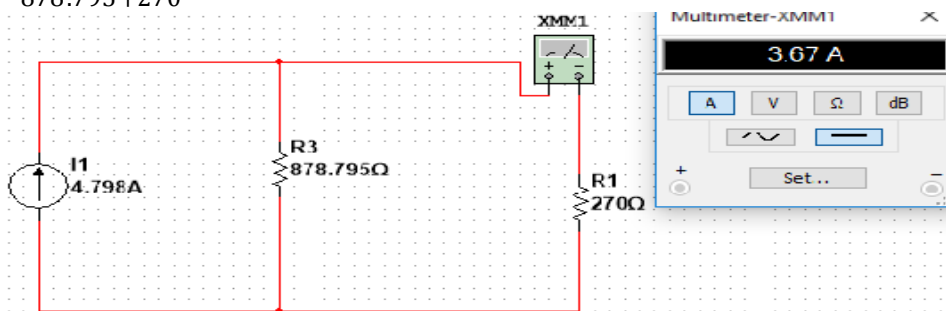
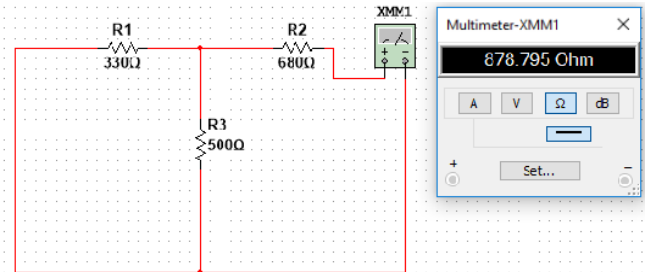
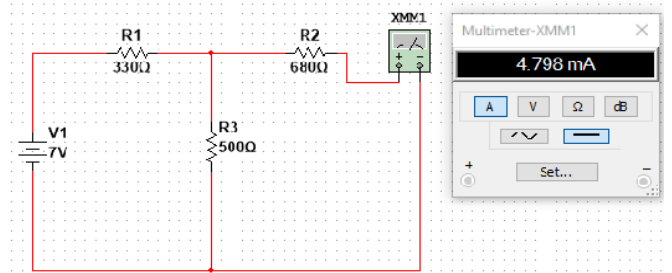
$$R_N = (330 \parallel 500) + 680$$

$$= 198.795 + 680 = 878.795 \Omega$$

STEP - 4

Now draw Norton's equivalent circuit and find current across load resistance.

$$I_L = \frac{4.79 \times 10^{-3} \times 878.795}{878.795 + 270} = 3.66\text{mA}$$



TABULATION: -

Calculated Tabulation: -

Sl No	Applied voltage in (V)	I_N in (mA)	R_N in (mA)	I_L in (mA)
1	7V	4.798	878.795	3.67

Observation Tabulation:-

Sl No	Applied voltage in (V)	I_N in (mA)	R_N in (mA)	I_L in (mA)
1	7	4.798	878.795	3.670
2	8	5.484	878.795	4.195
3	9	6.169	878.795	4.719
4	10	6.855	878.795	5.244
5	11	7.540	878.795	5.768
6	12	8.226	878.795	6.293

CONCLUSION:-

From the above experiment we know that how to verify the Norton's theorem by using software Multisim 14.1.

EXPERIMENT NO – 06

VERIFICATION OF MAXIMUM POWER TRANSFER THEOREM

AIM OF THE EXPERIMENT:- To study & verify Maximum power transfer theorem.

COMPONENT REQUIRED:-

SI No	Name of the Components	Specification
1	Software	Multisim-14.1
2	Resistors	1KΩ
3	Variable Resistor	10 KΩ
4	DC Power source	12 Volt
5	Multimeter	As required
6	Voltmeter	As required

THEORY:-

A resistive load being connected to a DC network receives maximum power when the load resistance is equal to the internal resistance of the source network as seen from load end.

EXPLANATION:-

A variable resistance 'R_L' is connected to a dc source network where 'V₀' represent the Thevenin's Voltage and 'R_{th}' represent the Thevenin's resistance of the source network. We have to find out the value of 'R_L' such that it receives the maximum from the dc source with reference to the fig the following can be written.

The current through the network 'I₀' will be equal to mean, $I_0 = \frac{V_0}{R_{TH} + R_L}$

The power delivered to the resistive load, $P_L = [I_0]^2 R_L = \frac{V_0^2 R_L}{(R_{TH} + R_L)^2}$

P_L can be maximized by varying the R_L & hence maximum power (P_{max}) can be delivered when, $\frac{d}{dR_L} P_L = 0$

$$\Rightarrow \frac{d}{dR_L} \frac{V_0^2 R_L}{(R_{TH} + R_L)^2} = 0 \Rightarrow V_0^2 \frac{d}{dR_L} \frac{R_L}{(R_{TH} + R_L)^2} = 0 \Rightarrow V_0^2 \frac{(R_{TH} + R_L)^2 \frac{d}{dR_L} R_L - R_L \frac{d}{dR_L} (R_{TH} + R_L)^2}{(R_{TH} + R_L)^4} = 0$$

$$\Rightarrow \frac{V_0^2}{(R_{TH} + R_L)^4} (R_{TH} + R_L)^2 - R_L 2(R_{TH} + R_L) = 0 \Rightarrow (R_{TH} + R_L) [(R_{TH} + R_L) - 2R_L] = 0$$

$$\Rightarrow R_{TH} + R_L - 2R_L = 0 \Rightarrow R_{TH} - R_L = 0 \Rightarrow R_{TH} = R_L \Rightarrow \text{Thus } P_L = \frac{V_0^2 R_L}{(R_{TH} + R_L)^2}$$

By replacing R_{TH}=R_L in this equitation, $P_L = \frac{V_0^2 R_L}{(2R_L)^2} \Rightarrow P_L = \frac{V_0^2 R_L}{4R_L^2} \Rightarrow P_L = \frac{V_0^2}{4R_L}$

$$P_{max} = \frac{V_0^2}{4R_L}$$

PROCEDURE:-

- 1) Start → Electronics work bench → Multisim 14.1.
- 2) Select component from place → Component library according to following circuit.
- 3) Connect the multimeter.
- 4) Make connection according.

- 5) Simulate → Run
- 6) Double click on the multimeter.
- 7) See the output result

CALCULATION:-

STEP - 1

First draws the given original circuit.

STEP - 2

Assume load resistance as open circuited & calculate Thevenin’s equivalent voltage as short circuited path

Apply mesh analysis, in loop we get,

$$12 - 1000I_1 - 1000I_1 = 0 \rightarrow 12 - 2000I_1 = 0$$

$$2000I_1 = 12 \rightarrow I_1 = 6\text{mA}$$

So, voltage across open circuit path or Thevenin’s equivalent voltage $[V_{TH}] = 6\text{V}$

STEP - 3

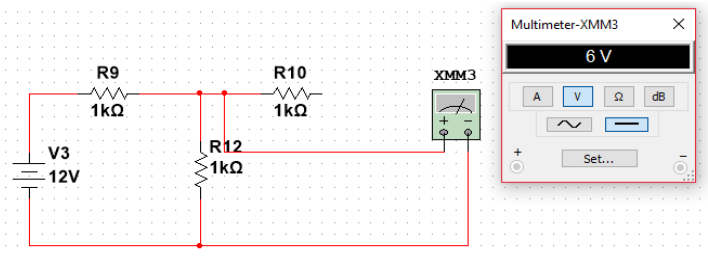
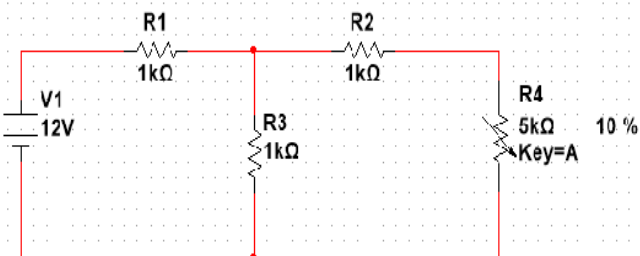
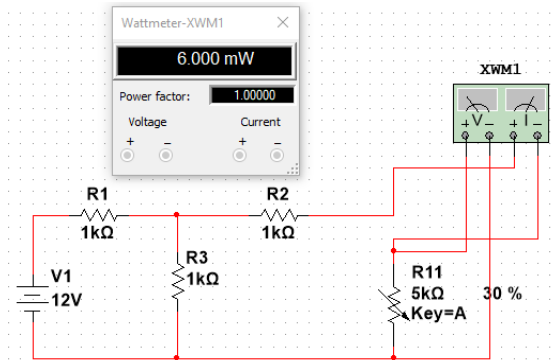
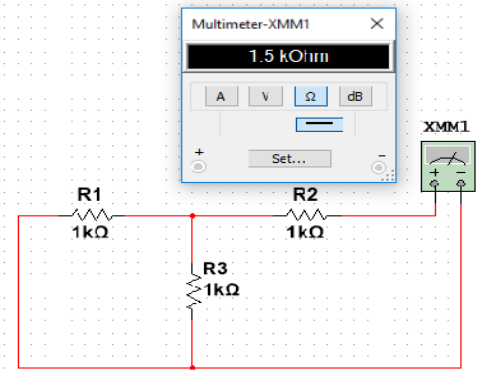
Assume load resistance as open circuit and find Thevenin’s equivalent resistance,

$$R_{TH} = (1000 \parallel 1000) + 1000 = 500 + 1000 = 1500 \Omega \rightarrow R_{TH} = 1.5\text{K}\Omega$$

STEP - 4

After Thevenin’s equivalent resistance found the P_{max} or maximum power of the circuit,

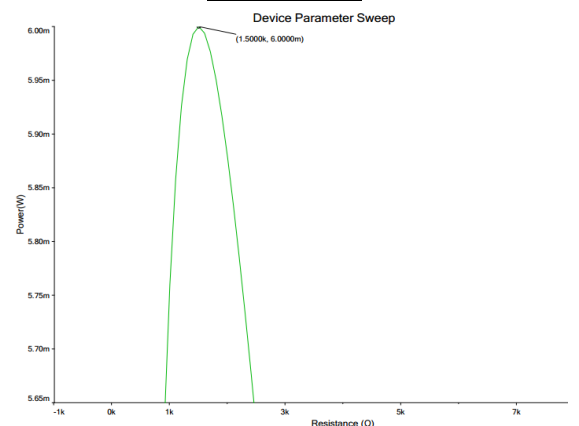
$$P_{max} = \frac{V_0^2}{4R_L} = \frac{6^2}{4 \times 1500} \quad P_{max} = 6\text{mW}$$



TABULATION

GRAPH

Sl No	Load resistance (RL) (5KΩ)	V _{RL} in (V)	P _o (V _{RL} /R _L) in mW
1	500 (10%)	1.5	4.500
2	1000 (20%)	2.4	5.760
3	1500 (30%)	3.0	6.000
4	2000 (40%)	3.4	5.870
5	2500 (50%)	3.7	5.625



CONCLUSION:-

From the above experiment we concluded that for $R_L = R_{th}$ we get the maximum power transferred to the load end by using Multisim 14.1.

EXPERIMENT NO – 07

LOW-PASS FILTER

AIM OF THE EXPERIMENT: - To study the characteristic of low pass filter.

COMPONENT REQUIRED:-

SI No	Name of the Components	Specification
1	Software	Multisim-14.1
2	Resistors	1kΩ
3	Capacitor	1μF
3	Sine wave function generator	1kHz,1Vpk
4	Bode Plotter	As required

THEORY:-

A low-pass filter is a filter that passes low-frequency signals but attenuates (reduces the amplitude of) signals with frequencies higher than the cutoff frequency.

The actual amount of attenuation for each frequency varies from filter to filter.

It is sometimes called a high-cut filter, or treble cut filter when used in audio applications.

A low-pass filter is the opposite of a high-pass filter, and a band-pass filter is a combination of a low-pass and a high-pass.

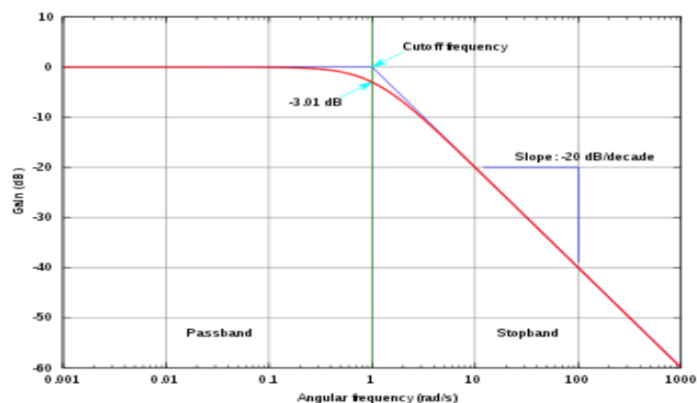
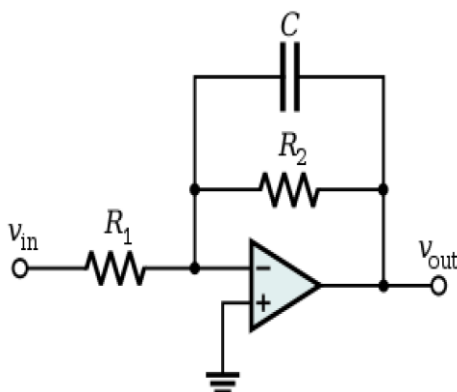
Low-pass filters exist in many different forms, including electronic circuits (such as a hiss filter used in audio), digital filters for smoothing sets of data, acoustic barriers, blurring of images, and so on.

The moving average operation used in fields such as finance is a particular kind of low-pass filter, and can be analyzed with the same signal processing techniques as are used for other low-pass filters.

Low-pass filters provide a smoother form of a signal, removing the short-term fluctuations, and leaving the longer-term trend.

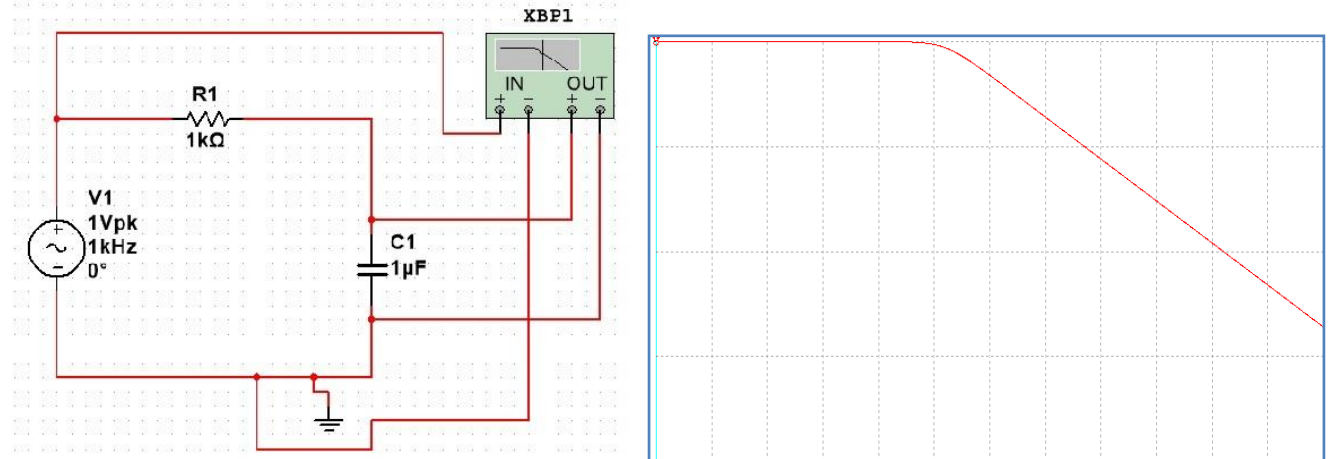
In an electronic low-pass RC filter for voltage signals, high frequencies contained in the input signal are attenuated but the filter has little attenuation below its cutoff frequency

which is $F_c = \frac{1}{2\pi R_c}$ determined by its RC time constant.



PROCEDURE:-

1. Start – Electronics Workbench – Multisim14.1.
2. Select the components from place – Components library according to following circuit.
3. Connect the Bode Plotter [Simulate – Instrument- Bode Plotter].
4. Simulate- Run.
5. Double click on Bode Plotter.



OBSERVATION:-

In the above experiment we observe that the output of CRO is much difference when the theoretical characteristic curve and the practical and theoretical curve are different.

CONCLUSION:-

From the above experiment we studied those characteristics of Low pass Filter by using software Multisim14.1.

DATE:

Signature of the Student

NAME:

ROLL NO.:

SEMESTER:

GROUP NO.

DATE:

Instructor's Signature

---ALL---THE---BEST---

EXPERIMENT NO – 08

HIGH PASS FILTER

AIM OF THE EXPERIMENT

To study the characteristics of high pass filter.

EQUIPMENT REQUIRED

Software- Multisim 14.1
Components- Resistance-1k ohm
Capacitor- 1uf.
1khz, 1vpk, sine wave function generator.
Bode plotter

THEORY

The filter is a circuit that respond to the frequency of the I/p signal.

The o/p of the filter depends on the frequency of the I/P signal.

The filter don't contain very active component's are called passive filter.

According to the connecting wire R and C filter act as high pass and low pass filter.

The cut off frequency for a filter is $F_c = 1/2\pi R_c$.

The HPF the o/p is low for the frequency is low then the cut off frequency and o/p is high for higher frequency.

This is because at the capacitor is series with the I/p signal acts as a open circuit for lower frequency and acts as a short circuit for higher frequency.

The filter is contain active component are called active filter.

High pass filter are obtained by connecting R and C combination to open.

The operation in filter is same as that basis filter

PROCEDURE

1. Start- Electronic workbench- Multisim 14.1.
2. Select the components from place- components library according to the following circuit.
3. Connect the oscilloscope [simulate-instrument-oscilloscope].
4. Simulate- Run.
5. Double click on the Bode plotter.

OBSERVATION

In the above experiment we observe that the o/p of CRO is much difference when the theoretical characteristics curve and the practical and theoretical curve are different.

CONCLUSION

From the above experiment we observe that the characteristics of high pass filter.

EXPERIMENT NO – 09

RESONANCE CIRCUIT

AIM OF THE EXPERIMENT: - To study the resonance circuit when is in both series and parallel.

COMPONENT REQUIRED:-

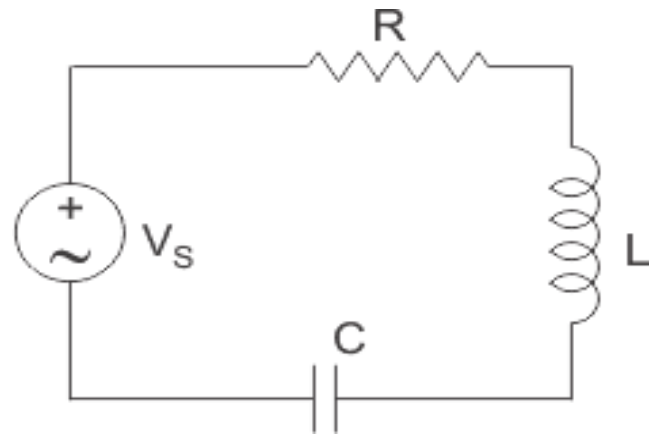
Sl No	Name of the Components	Specification
1	Software	Multisim-14.1
2	Resistors	40kΩ
3	Capacitor	30mF
3	Inductor	0.1mH
4	Sine wave function generator	1kHz,1Vpk

THEORY:-

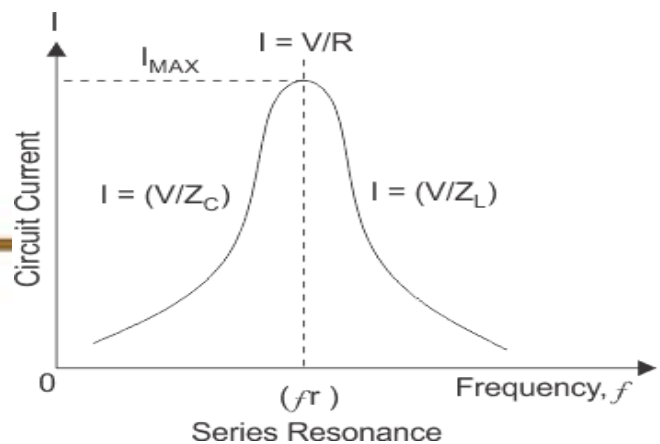
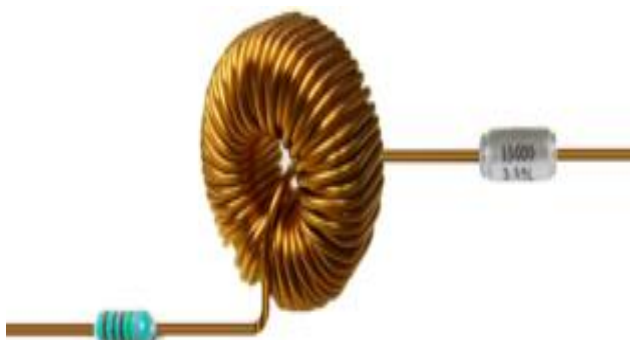
SERIES RESONANCE:-

When an inductor and capacitor are connected in series the output current or voltage are maximum at a particular frequency depending on the values of inductor and capacitor.

This is called as resonance condition and the frequency is called resonating frequency at which the circuit attains resonance.



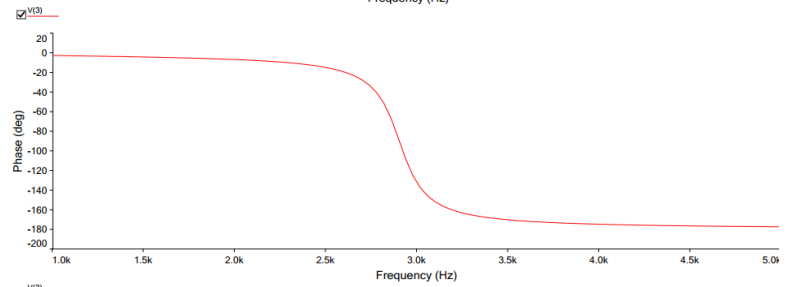
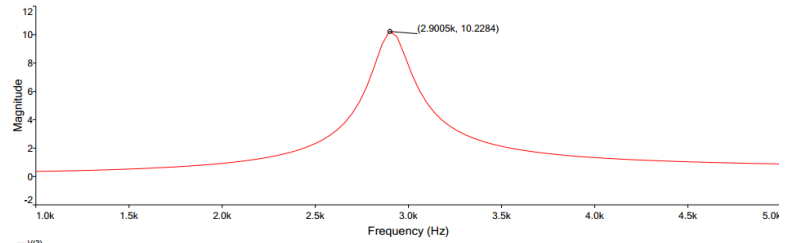
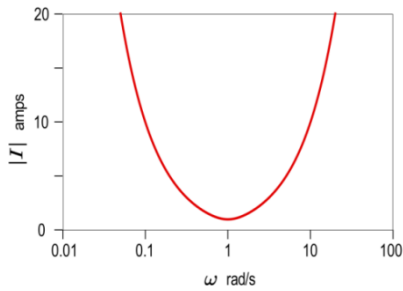
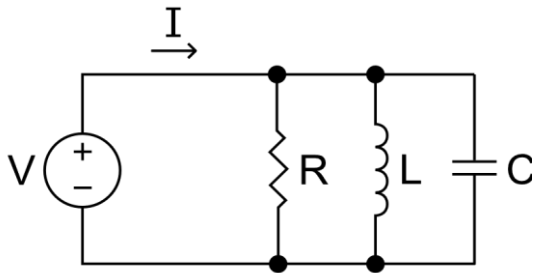
For a series L-C Resonant is given by,
$$F_0 = \frac{1}{2\pi \sqrt{LC - \frac{R^2}{L^2}}}$$



PARALLEL RESONANCE:-

When an inductor and capacitor are connected in parallel the output current or voltage are minimum at a particular frequency depending on the values of inductor and capacitor.

This is called as resonance condition and the frequency is called resonating frequency at which the circuit attains resonance. For a parallel L-C Resonant is given by, $F_0 = \frac{1}{2\pi\sqrt{LC}}$



PROCEDURE:-

- 1) Start – Electronics Workbench – Multisim14.1.
- 2) Select the components from place – Components library according to the following circuit
- 3) Connect the Power source [simulate – instrument-Power source (A.C Battery)]
- 4) Simulate- Run

CONCLUSION:-

The circuit at resonance at particular frequency the frequency at which the amplitude get increased.

DATE:

Signature of the Student

NAME:

ROLL NO.:

SEMESTER:

GROUP NO.

DATE:

Instructor’s Signature

---ALL---THE---BEST---ALL---THE---BEST---