Department of Electrical Engineering
Govt. Polytechnic Mayurbhanj, Tikarpada,
Podaastia, Baripada, Odisha - 757049

LABORATORY MANUAL

Circuit & Simulation Lab

(3rd Semester)

Prepared By: **Hemata Patra**

Department of Electrical Engineering
Govt. Polytechnic Mayurbhanj, Tikarpada,
PodaAstia, Baripada, Odisha - 757049

TABLE OF CONTENTS

EXPERIMENT NO - 01

AIM OF THE EXPERIMENT: - Measurement of equivalent resistance in series and parallel circuit

COMPONENT REQUIRED:-

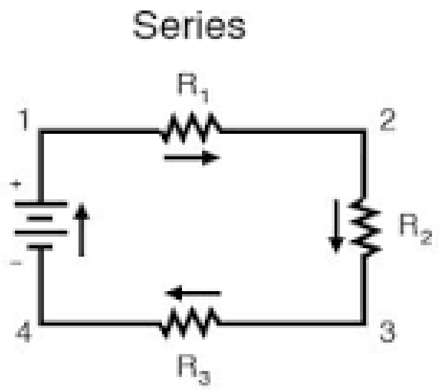
Sl. No	Name of the Components	Specification	Quantity
1	Resistors	500 Ω, 680 Ω, 330 Ω,390 Ω, 270 Ω	Each of 2
2	Multimeter	Digital	1
3	Software	Multisim 14.1	As required

THEORY:-

SERIES CIRCUIT:-

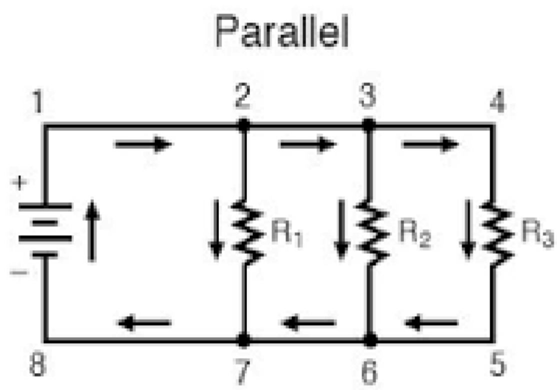
- In electrical circuit is in series connected the current flowing through the conductor is constant but voltage is not constant and the resistance can be calculated by,

$$\mathbf{R_s = R_1 + R_2 + R_3 + \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots + R_n}$$



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,



$$\frac{1}{R_P} = \frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3} + \frac{1}{R_4} + \ldots\ldots\ldots\ldots\ldots\ldots + \frac{1}{R_n}$$

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,

$R_1 = 330\ \Omega$, $R_2 = 390\ \Omega$ → $R_S = R_1 + R_2 = 330 + 390 = 720\ \Omega$

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.

Experiment - 02

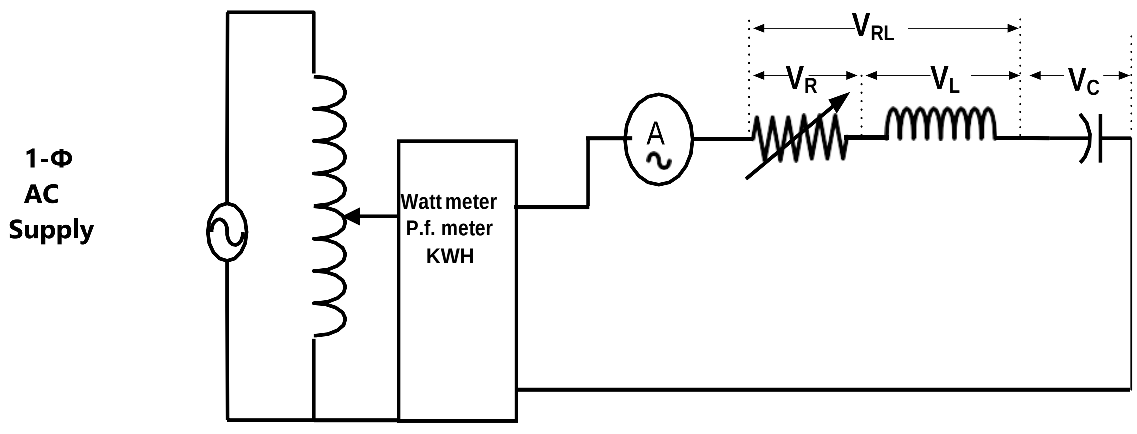
Aim- Measurement of power and power factor using series R-L-C Load.

APPARATUS REQUIRED:-

Sl. No	Name of the Equipment	Specification	Quantity
1	Variable Resistor	0-100 Ω	1 no
2	Inductor	40W, 250V	1 no
3	Capacitor	2.5 μF	1 no
4	1-Φ Dimmer Set	0-250 v	1 no
5	Voltmeter	0-300 v	3 nos
6	Ammeter	0-5 A	1 no
7	1-Φ Wattmeter	250V, 1 kW	1 no
8	Power factor Meter	250v, 5A	1 no
9	Connecting Wires	-	As per required

Theory:- A series RLC circuit is one the resistor, inductor and capacitor are connected in series across a voltage supply. The resulting circuit is called **series RLC circuit.**

Circuit Diagram:-



Observation Table:-

Sl .no	Type of Load	Reading of Wattmeter	Reading of PF Meter
1	R		
2	L		
3	C		
4	R-L		
5	R-C		
6	L-C		
7	R-L-C		

Procedure:-

1- We should take all the tools & instrument for this experiment.
2- Connect as per Circuit diagram.
3- Then switch ON the supply.
4- Take reading of wattmeter and PF meter.

Conclusion:- From the above experiment, we learnt about the measurement of power and power factor using series R-L-C Load.

EXPERIMENT NO – 03

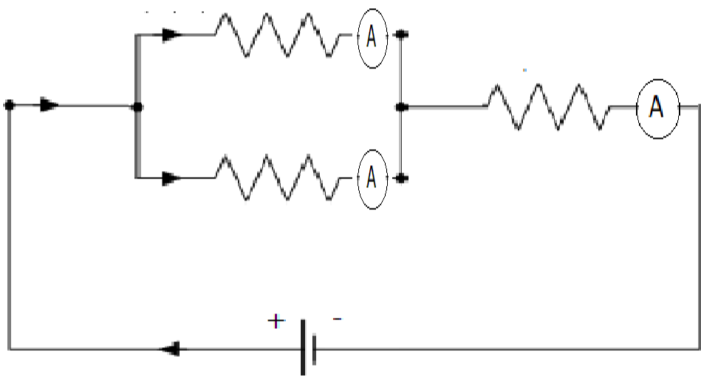
AIM OF THE EXPERIMENT: -Verification of KCL & KVL.

EQUIPMENT REQUIRED: -

Sl 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: -

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

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

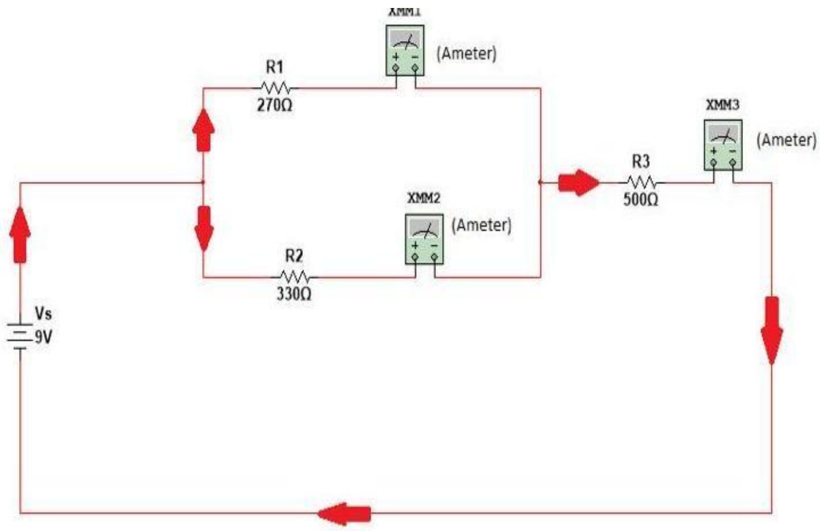
$R_{eq} = (270 \| 330) + 500$

$= 148.5 + 500 = 648.5\Omega$

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

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

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



➔ $I_1 + I_2 = 7.15mA + 5.85mA = 13mA$

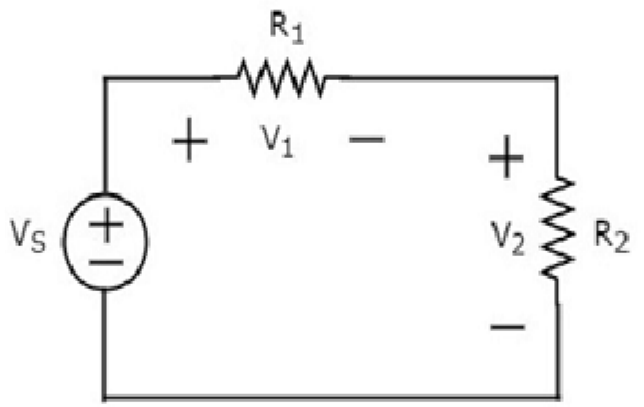
OBSERBATION TABLE: -

Sl 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

KIRCHHOFF'S VOLTAGE LAW

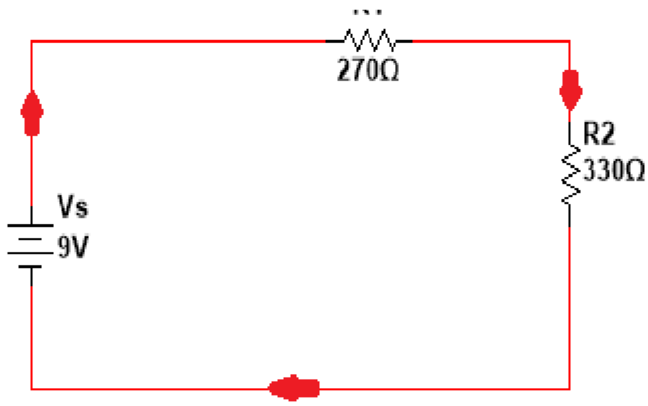
THEORY: -

- 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 'V_1' and across 330 Ω resistor is taken 'V_2'.



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, $V - IR_1 - IR_2 = 0$

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

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

$\Rightarrow 9 = I \times (270 + 330)$ ➔ $I = \frac{9}{270 + 330} = 0.015A$ ➔ $\boxed{I = 15\ mA}$

OBSERBATION TABLE: -

Sl 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.

EXPERIMENT NO – 04

SUPER POSITION THEOREM

AIM OF THE EXPERIMENT: - Verification of 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	**-**	**-**

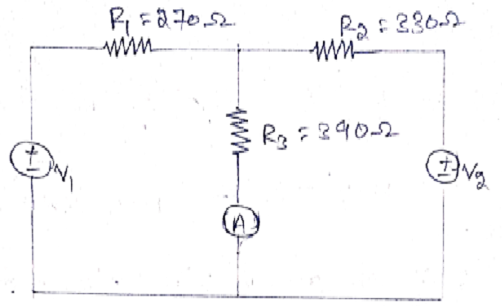
THEORY: -

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.

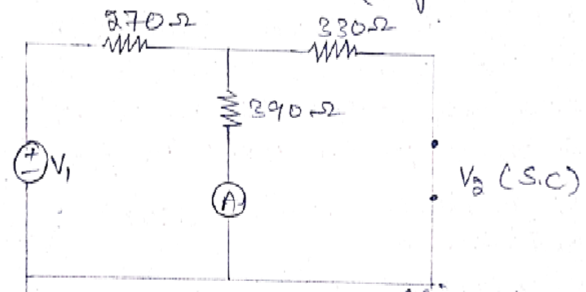
The voltage source replaced by short circuit and the current circuit replaced by open circuit. The voltage source replaced by short the resistances of the source are replaced at the time of source elimination.

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..

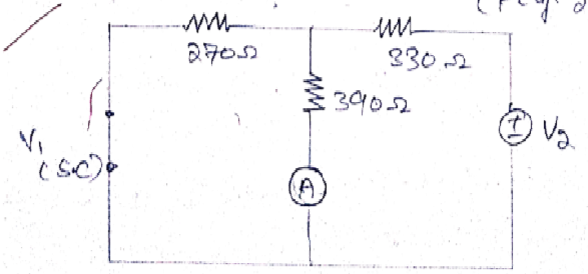
The total power delivered to a resistive element must be determined using the total current through or the total voltage across the element and cannot be determined by a simple sum of the power levels established by each source.



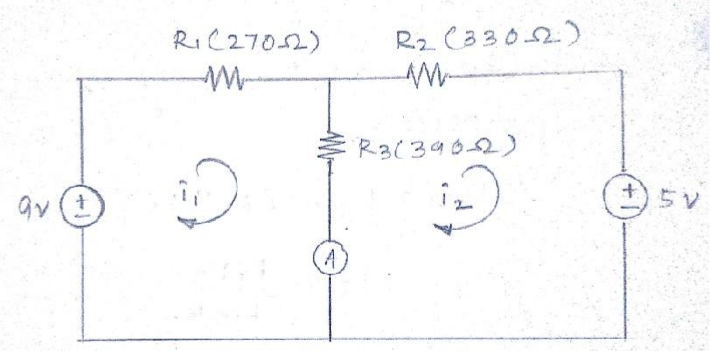
(Fig-1)



(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 , $R_{eq} = R_1 + \frac{R2R3}{R2+R3}$

Due to voltage source V_1 , $I_{31} = I_1 \times \frac{R2}{R2+R3} = \frac{V1}{Req} \times \frac{R2}{R2+R3}$

Now eliminate the V1 voltage source from the circuit, $R_{eq} = R_2 + \frac{R1R3}{R1+R3}$

Due to voltage source V_2 , $I_{32} = I_2 \times \frac{R1}{R1+R3} = \frac{V2}{Req} \times \frac{R1}{R1+R3}$

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

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

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

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

$I_3 = I_{31} + I_{32} = 9.19 + 4.17 =$ **13.36 mA**

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

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

⇨ $660I_1 - 390I_2 = 9$ .... (i)

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

⇨ $-330I_2 - 5 - 390I_2 + 390\,I_1 = 0$

⇨ $390I_1 - 720I_2 = 5$ .... (ii)

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

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

So , $I = I_1 - I_2 = 14 - 0.65 = 13.35$ 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.

EXPERIMENT NO – 05

VERIFICATION OF THEVENIN'S THEORM

AIM OF THE EXPERIMENT:- Verification of Thevenin's Theorem.

EQUIPMENTS REQUIRED

Sl 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 VAB 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 Ohm, R_2= 500 Ohm, R_3= 680 Ohm, R_L= 9v

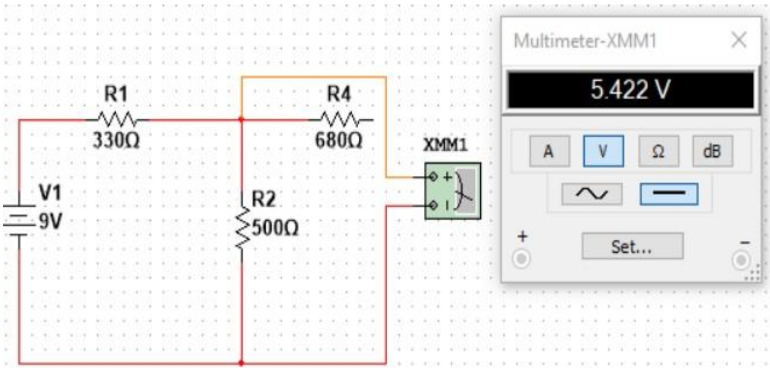
STEP- 1

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

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

$I = V_s/R_1+R_2 = 9/330+500 = 1.08$ mA

$V_{th} = V_{AB=} I\times500 = 1.08\times500 = 5.42V$



STEP-2

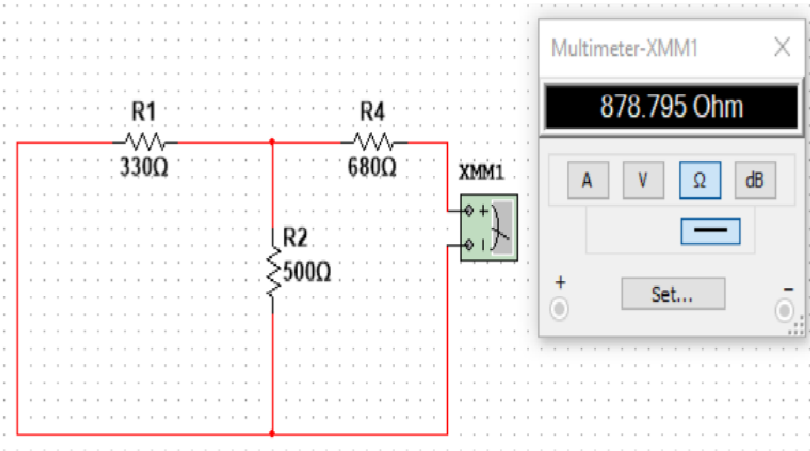
Calculate the R_{th} across AB terminal by short circuit the voltage source.

$R_{ab} = (330 \,||\, 500) + 680 = \frac{330\times500}{330+500} + 680 = 878.795$Ohm

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

$I_{th} = \frac{Vth}{Rth + Rl} = \frac{5.422}{878.795+270} = 4.71$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.

EXPERIMENT NO – 06

VERIFICATION OF NORTON’S THEOREM

AIM OF THE EXPERIMENT: - Verification of Norton’s Theorem.

COMPONENT REQUIRED:-

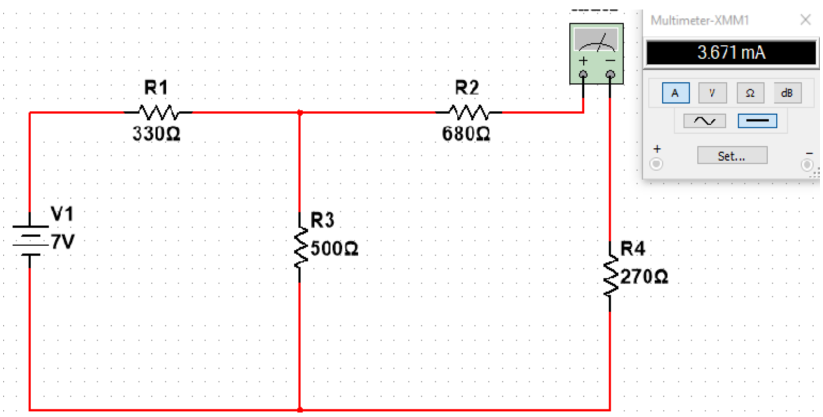
Sl No	Name of the Components	Specification
1	Software	Multisim-14.1
2	Resistors	500 Ω, 680 Ω, 330 Ω, 270 Ω
3	DC Power source	7 Volt
4	Multimeter	As required

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, ➔ $7 - 330I_1 - 500I_1 + 500I_2 = 0$

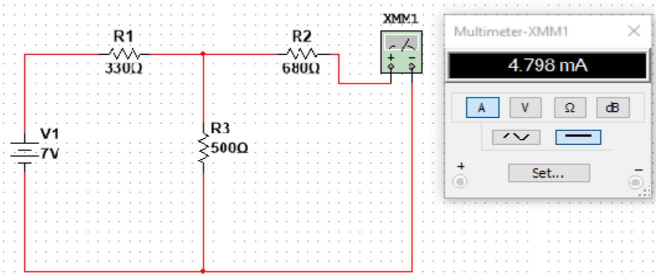
➔ $7 - 830I_1 + 500I_2 = 0$ ➔ $830I_1 - 500I_2 = 7$ --------------- (1)

Apply mesh analysis, in loop 2 we get, ➔ $- 500I2 + 500\ I1 - 680I2 = 0$

➔ $-1180I2 + 500\ I_1 = 0$ ➔ $500\ I1-1180I2= 0$ --------------- (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] = **4.79mA**

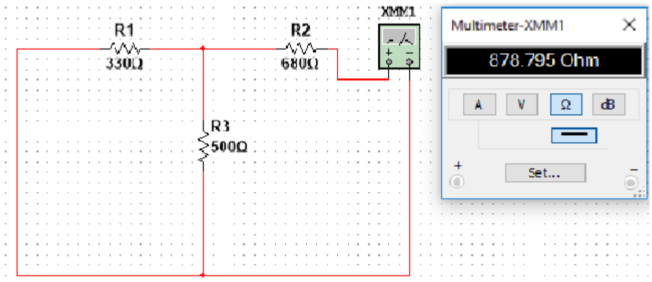


STEP - 3

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

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

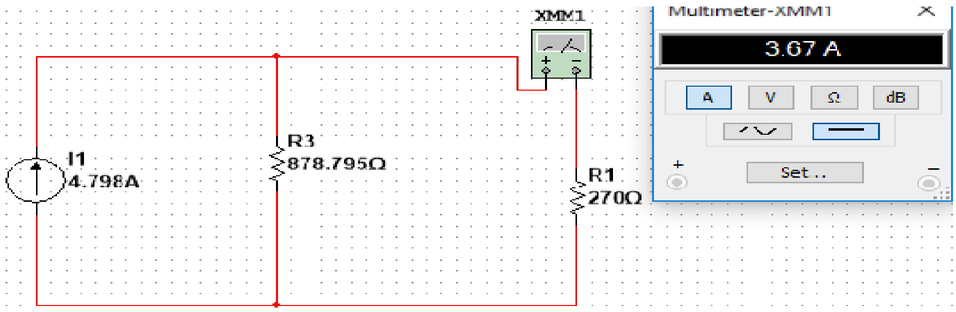
$$=198.795 + 680 = \mathbf{878.795\ \Omega}$$



STEP - 4

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

$$I_L = \frac{4.79\times10^{-3}\times878.795}{878.795+270} = \mathbf{3.66mA}$$



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 - 07

VERIFICATION OF MAXIMUM POWER TRANSFER THEOREM

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

COMPONENT REQUIRED:-

Sl 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 'V_0' 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, $\mathbf{I_O} = \frac{V_O}{R_{TH}+R_L}$

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

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

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

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

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

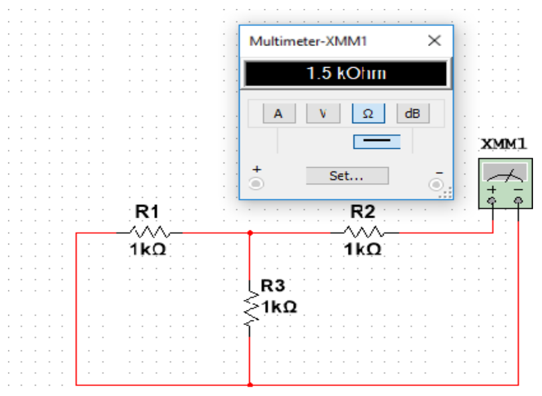
By replacing $R_{TH}=R_L$ in this equitation, $\mathbf{P_L} = \frac{V_O{}^2 R_L}{(2R_L)^2} \Rightarrow \mathbf{P_L} = \frac{V_O{}^2 R_L}{4R_L{}^2} \Rightarrow \mathbf{P_L} = \frac{V_O{}^2}{4R_L}$

$$\boxed{\mathbf{P_{max}} = \frac{V_O{}^2}{4R_L}}$$

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.

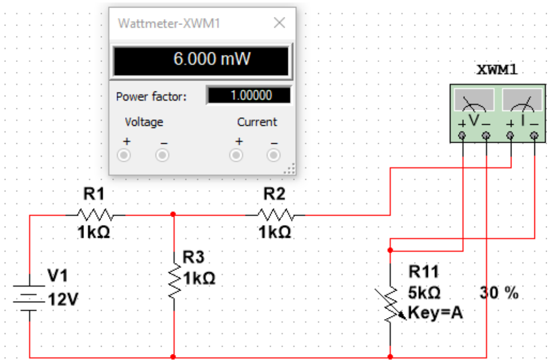
STEP - 2

Assume load resistance as open circuited & calculate Thevenin's equivalent voltage as short circuited path Apply mesh analysis, in loop we get,

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

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

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



STEP - 3

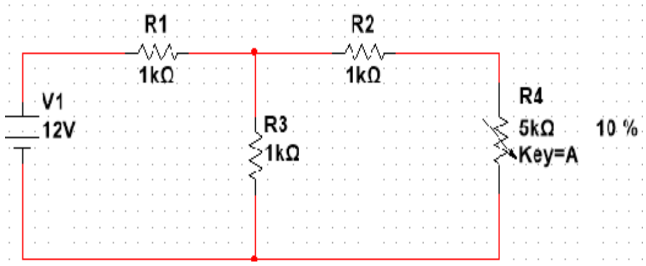
Assume load resistance as open circuit and find Thevenin's equivalent resistance,

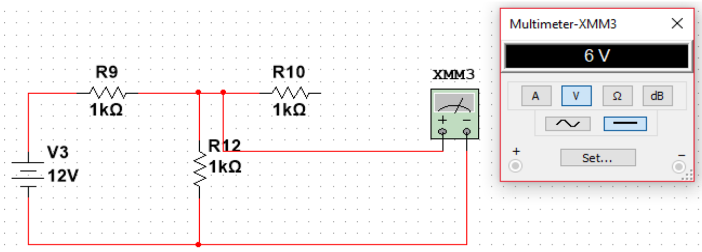
$R_{TH} = (1000 \| 1000) + 1000 = 500 + 1000 = 1500\ \Omega \rightarrow \mathbf{R_{TH} = 1.5K\Omega}$

STEP - 4

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

$\mathbf{P_{max}} = \frac{V_O^2}{4R_L} = \frac{6^2}{4\times1500}$ $\mathbf{P_{max} = 6mW}$

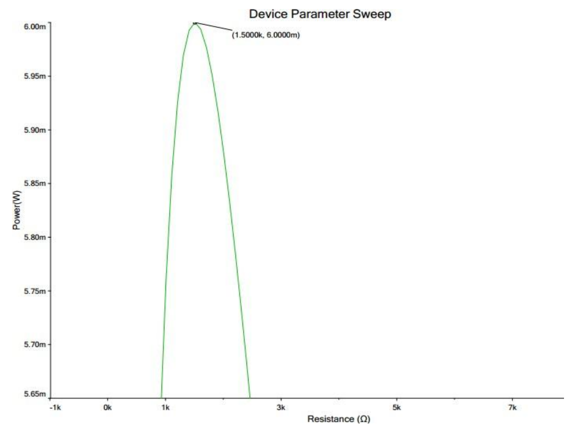




TABULATION

Sl No	Load resistance (R_L) (5KΩ)	V_{RL} in (V)	Po (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

GRAPH



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 – 08

RESONANCE CIRCUIT

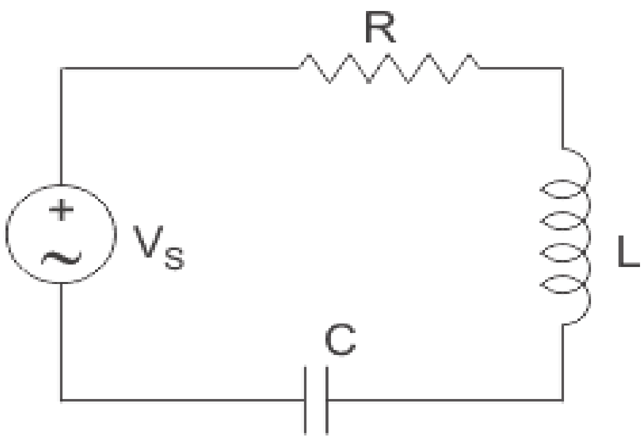
AIM OF THE EXPERIMENT: - Determine resonant frequency of series R-L-C circuit

COMPONENTREQUIRED:-

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:-

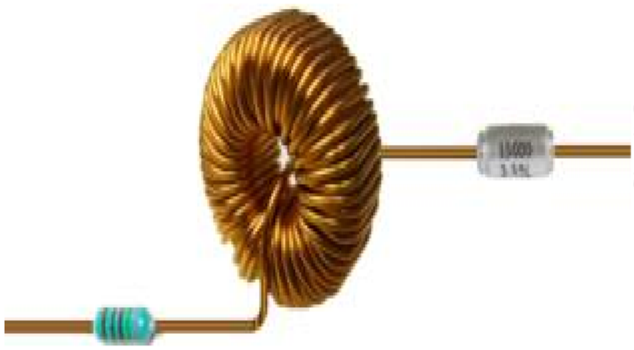
SERIES RESONANCE:-

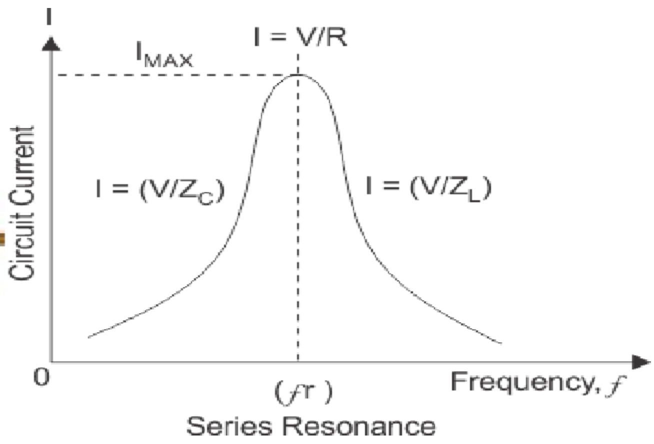


When an inductor and capacitor are connected in series the output current or voltage are maximum at a particular frequency depending on the values of inductor and capacitor.

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, $F_O = \frac{1}{2\pi\sqrt{LC-\frac{R^2}{L^2}}}$





]

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.

EXPERIMENT NO - 09

LOW-PASS FILTER

AIM OF THE EXPERIMENT: - Study of Low pass filter & determination of cut-off frequency.

COMPONENT REQUIRED:-

Sl 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.

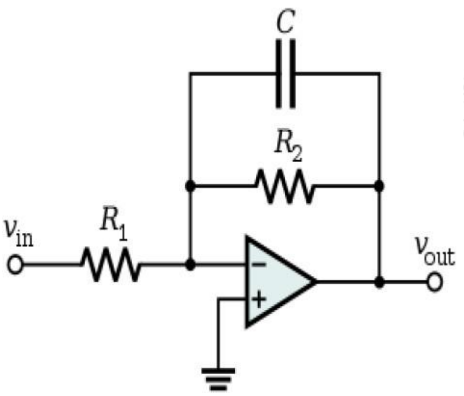
A low-pass filter is the opposite of a high-pass filter, and a band-pass filter is a combination of a low-pass and a high-pass.

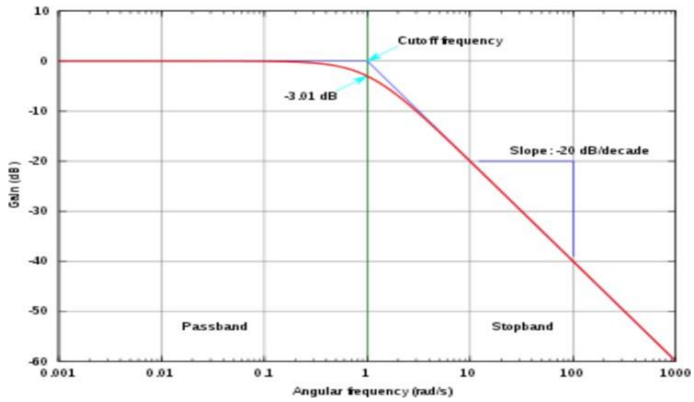
Low-pass filters exist in many different forms, including electronic circuits (such as a hiss filter used in audio), digital filters for smoothing sets of data, acoustic barriers, blurring of images, and so on.

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.

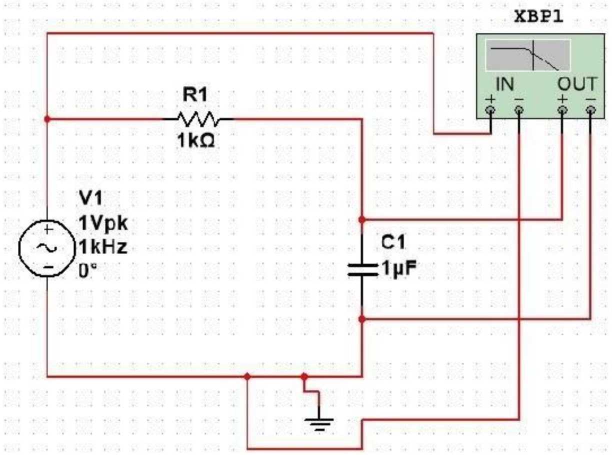
In an electronic low-pass RC filter for voltage signals, high frequencies contained in the input signal are attenuated but the filter has little attenuation below its cutoff frequency which is $\mathbf{F_c = \frac{1}{2\pi R_c}}$ determined by its RC time constant.

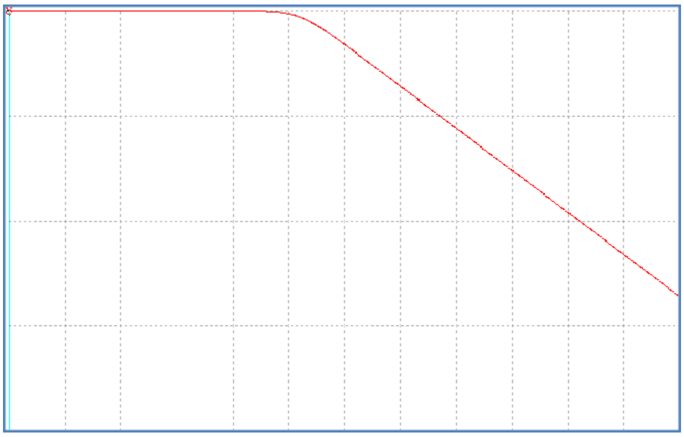




PROCEDURE:-

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.

EXPERIMENT NO – 10

AIM OF THE EXPERIMENT

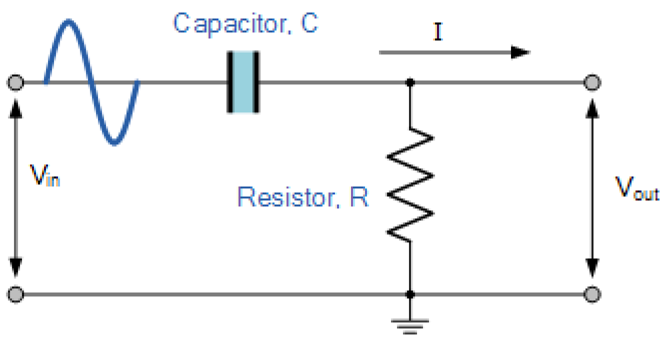
Study of High pass filter & determination of cut-off frequency

EQUIPMENT REQUIRED

Sl 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

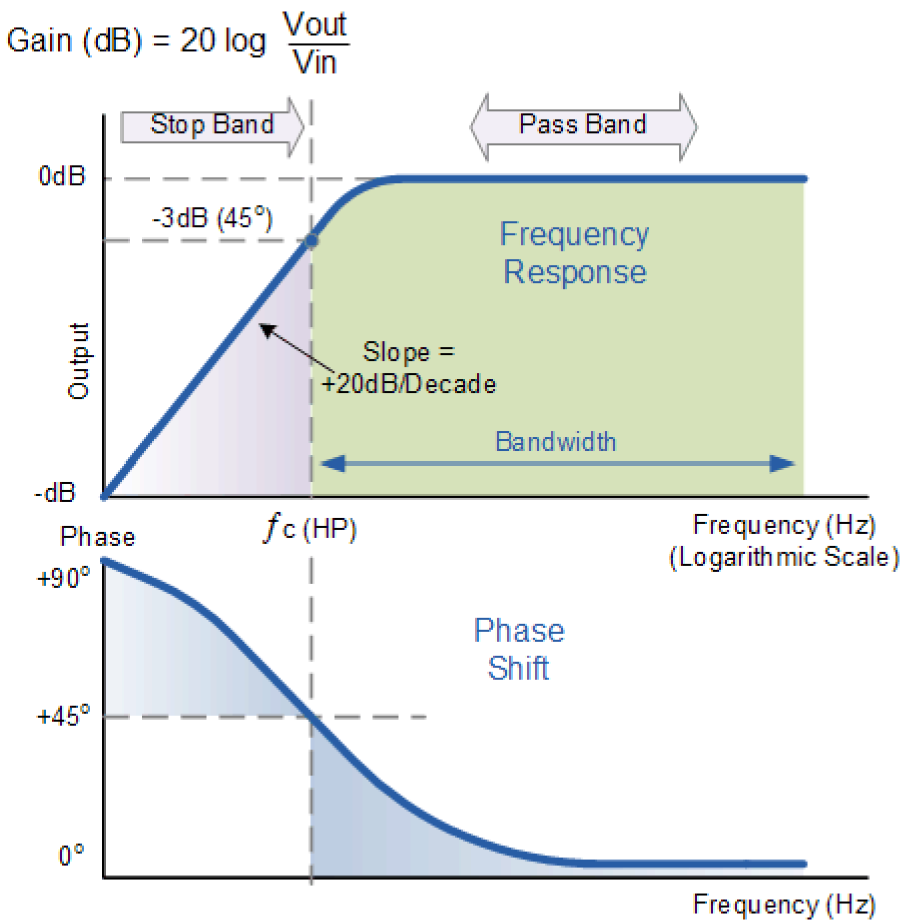
THEORY

The High Pass Filter Circuit



In this circuit arrangement, the reactance of the capacitor is very high at low frequencies so the capacitor acts like an open circuit and blocks any input signals at V_{IN} until the cut-off frequency point (f_C) is reached. Above this cut-off frequency point the reactance of the capacitor has reduced sufficiently as to now act more like a short circuit allowing all of the input signal to pass directly to the output as shown below in the filters response curve.

Frequency Response of a 1st Order High Pass Filter



The **Bode Plot** or Frequency Response Curve above for a passive high pass filter is the exact opposite to that of a low pass filter. Here the signal is attenuated or damped at low frequencies with the output increasing at +20dB/Decade (6dB/Octave) until the frequency reaches the cut-off point (ƒc) where again R = Xc. It has a response curve that extends down from infinity to the cut-off frequency, where the output voltage amplitude is 1/√2 = 70.7% of the input signal value or -3dB (20 log (Vout/Vin)) of the input value.

Also we can see that the phase angle (Φ) of the output signal **LEADS** that of the input and is equal to **+45°** at frequency ƒc. The frequency response curve for this filter implies that the filter can pass all signals out to infinity. However in practice, the filter response does not extend to infinity but is limited by the electrical characteristics of the components used.

The cut-off frequency point for a first order high pass filter can be found using the same equation as that of the low pass filter, but the equation for the phase shift is modified slightly to account for the positive phase angle as shown below.

Cut-off Frequency and Phase Shift

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

$$Phase\ Shift\ \phi = arctan\ \frac{1}{2\pi fRC}$$

The circuit gain, Av which is given as Vout/Vin (magnitude) and is calculated as:

$$A_V = \frac{V_{OUT}}{V_{IN}} = \frac{R}{\sqrt{R^2 + Xc^2}} = \frac{R}{Z}$$

$$\text{at low } f: \ Xc \to \infty, \ \ Vout = 0$$
$$\text{at high } f: \ Xc \to 0, \ \ Vout = Vin$$

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 - 11

Aim- 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.

APPARATUS REQUIRED:-

Sl. No	Name of the Equipment	Specification	Quantity
1	RLC charging & discharging Kit	-	1 no
2	Oscilloscope	2 MHZ	1 no
3	Connecting probes	-	As per required

Theory:-

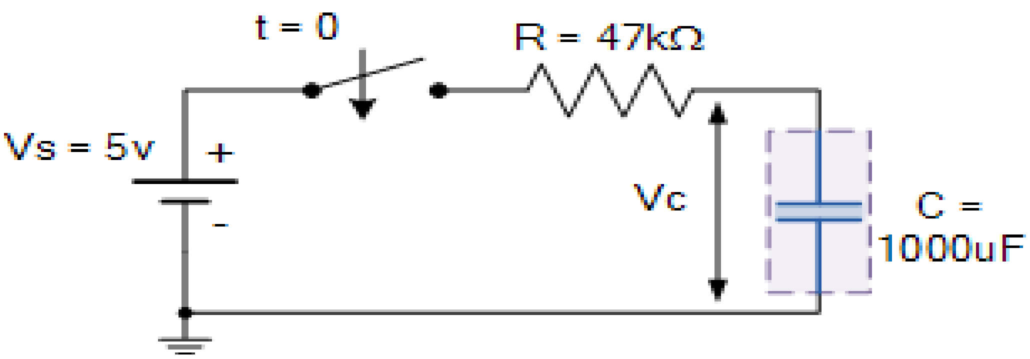
R-C charging circuit:-

When a voltage source is applied to an RC circuit, the capacitor, C charges up through the resistance, R
A capacitor, (C) in series with a resistor, (R) forming a **RC Charging Circuit** connected across a DC battery supply (Vs) via a mechanical switch. at time zero, when the switch is first closed, the capacitor gradually charges up through the resistor until the voltage across it reaches the supply voltage of the battery.
Let us assume above, that the capacitor, C is fully "discharged" and the switch (S) is fully open. These are the initial conditions of the circuit, then t = 0, i = 0 and q = 0. When the switch is closed the time begins at t = 0 and current begins to flow into the capacitor via the resistor.

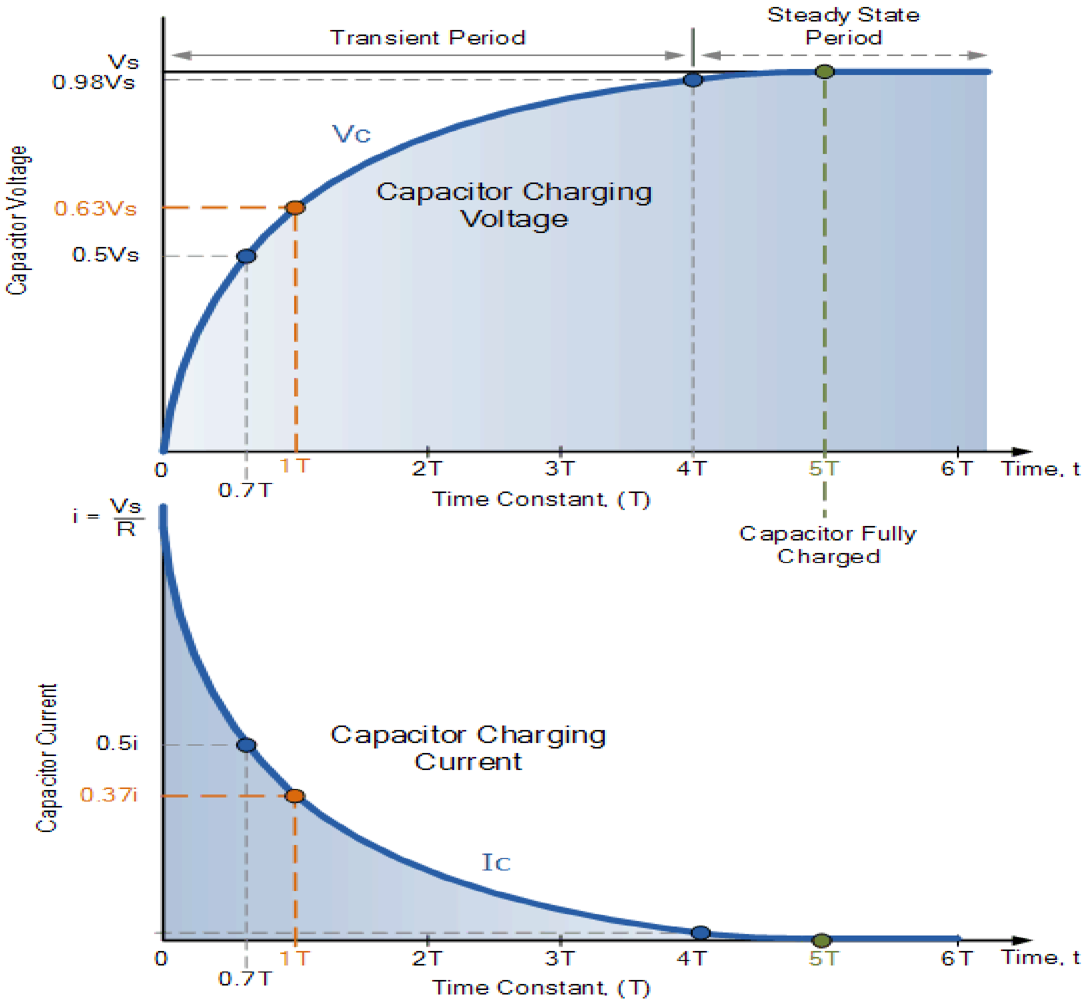
Since the initial voltage across the capacitor is zero, (Vc = 0) at t = 0 the capacitor appears to be a short circuit to the external circuit and the maximum current flows through the circuit restricted only by the resistor R. Then by using Kirchhoff's voltage law (KVL), the voltage drops around the circuit are given as:

$$V_S - R \times i(t) - V_C(t) = 0$$

The current now flowing around the circuit is called the **Charging Current** and is found by using Ohms law as: i = Vs/R.



[R-C charging circuit]



[RC Charging Circuit Curves]

RC Discharging Circuit:-

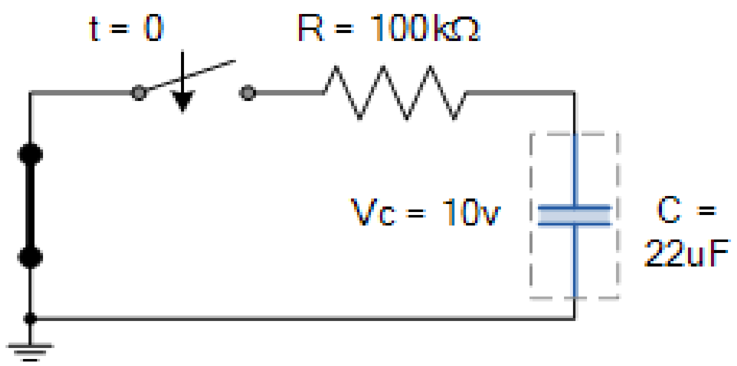
A Capacitor, C charges up through the resistor until it reaches an amount of time equal to 5 time constants known as 5T, and then remains fully charged as long as a constant supply is applied to it.
If this fully charged capacitor is now disconnected from its DC battery supply voltage, the stored energy built up during the charging process would stay indefinitely on its plates, (assuming an ideal capacitor and ignoring any internal losses), keeping the voltage stored across its connecting terminals at a constant value.

If the battery was replaced by a short circuit, when the switch is closed the capacitor would discharge itself back through the resistor, R as we now have a **RC discharging circuit**. As the capacitor discharges its current through the series resistor the stored energy inside the capacitor is extracted with the voltage Vc across the capacitor decaying to zero

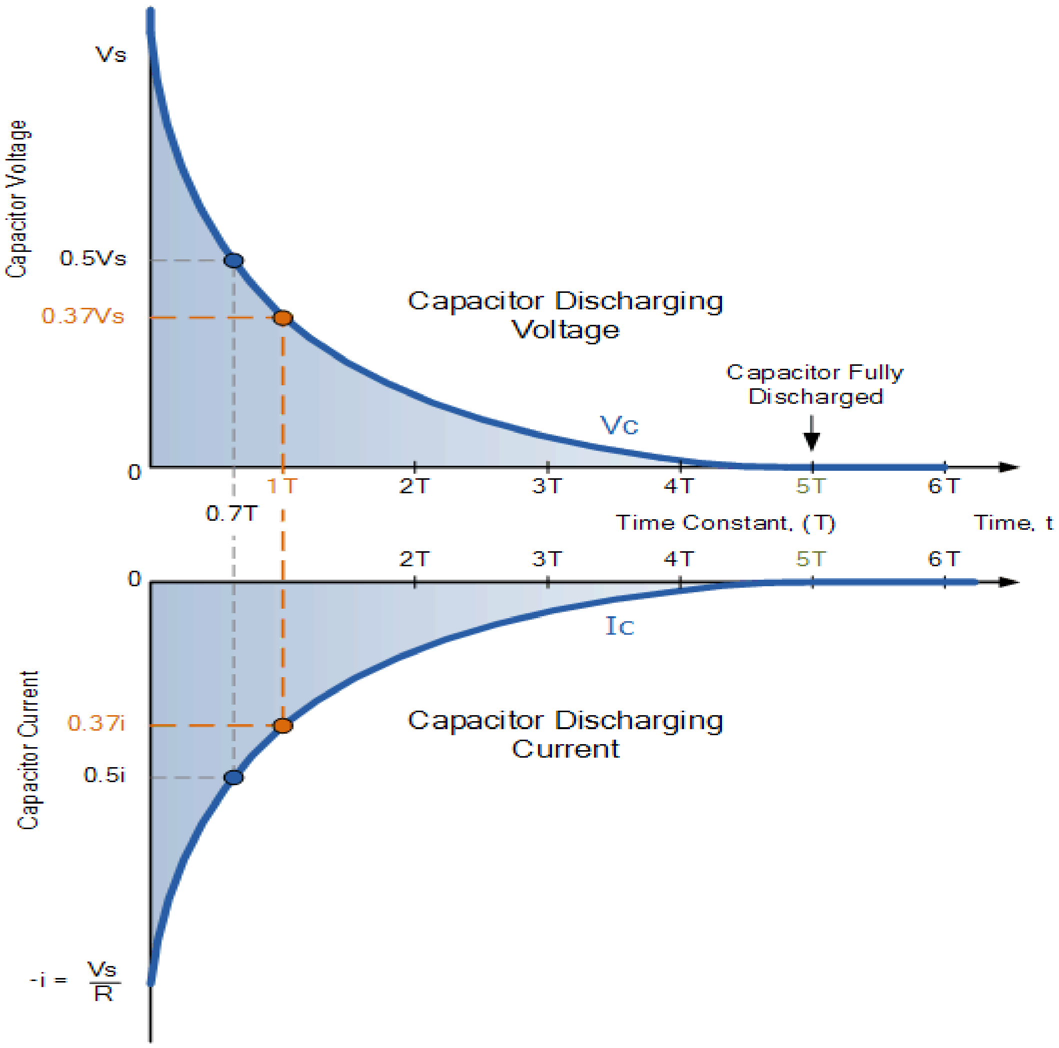
In a **RC Discharging Circuit** the time constant (τ) is still equal to the value of 63%. Then for a RC discharging circuit that is initially fully charged, the voltage across the capacitor after one time constant, 1T, has dropped by 63% of its initial value which is 1 – 0.63 = 0.37 or 37% of its final value.

Thus the time constant of the circuit is given as the time taken for the capacitor to discharge down to within 63% of its fully charged value. So one time constant for an RC discharge circuit is given as the voltage across the plates representing 37% of its final value, with its final value being zero volts (fully discharged), and in our curve this is given as 0.37Vs.

As the capacitor discharges, it does not lose its charge at a constant rate. At the start of the discharging process, the initial conditions of the circuit are: t = 0, i = 0 and q = Q. The voltage across the capacitors plates is equal to the supply voltage and $V_C = V_S$. As the voltage at t = 0 across the capacitors plates is at its highest value, maximum discharge current therefore flows around the RC circuit.



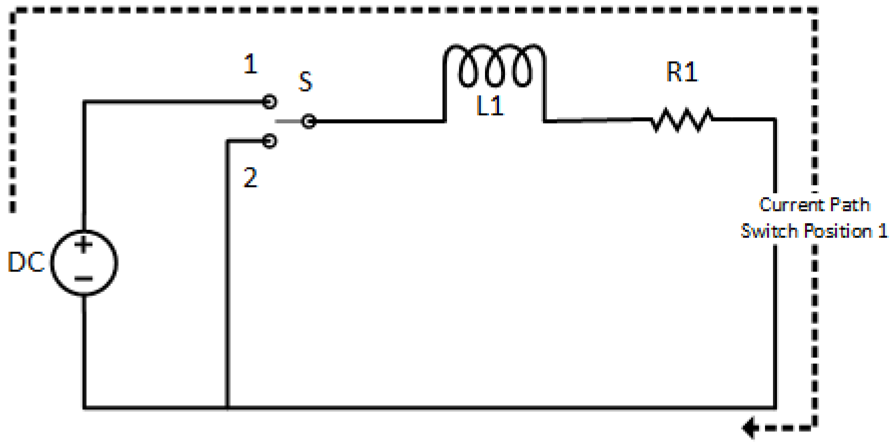
[R-C discharging circuit]



[RC Discharging Circuit Curves]

R-L charging circuit:-

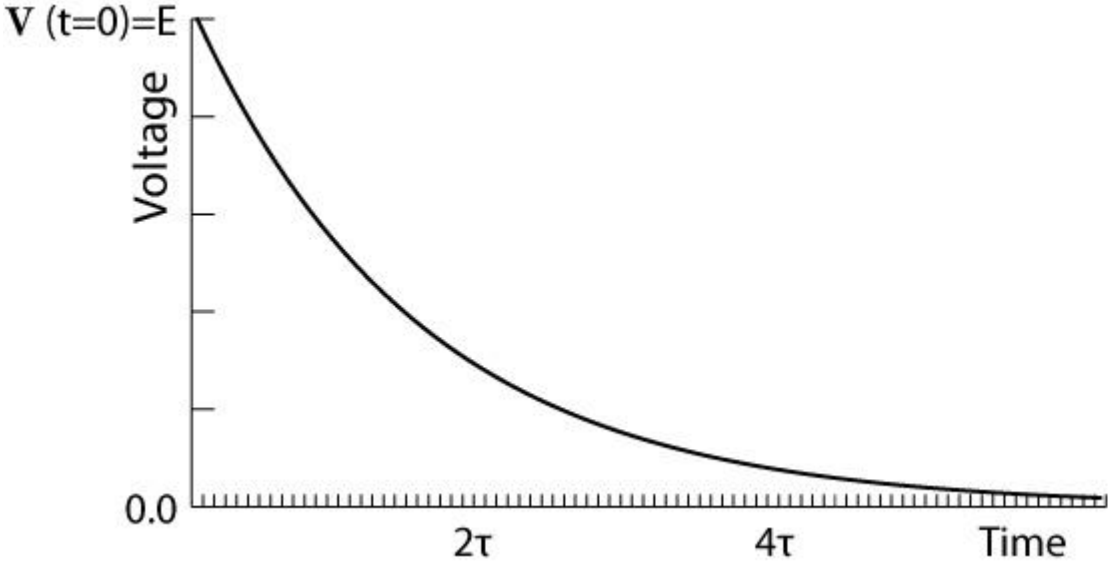
Suppose the inductor has no energy stored initially. At some point in time, the switch is moved to position 1, the moment is called time t=0. As the switch closes the source voltage will appear across the inductor and will try to pass current (I=V/R) abruptly through the inductor. But according to the Lenz Law, the inductor will oppose the change in current. The current will gradually increase unless it reaches its final value of current (I=V/R). At the same time, the voltage across the inductor will decrease unless it reaches zero.



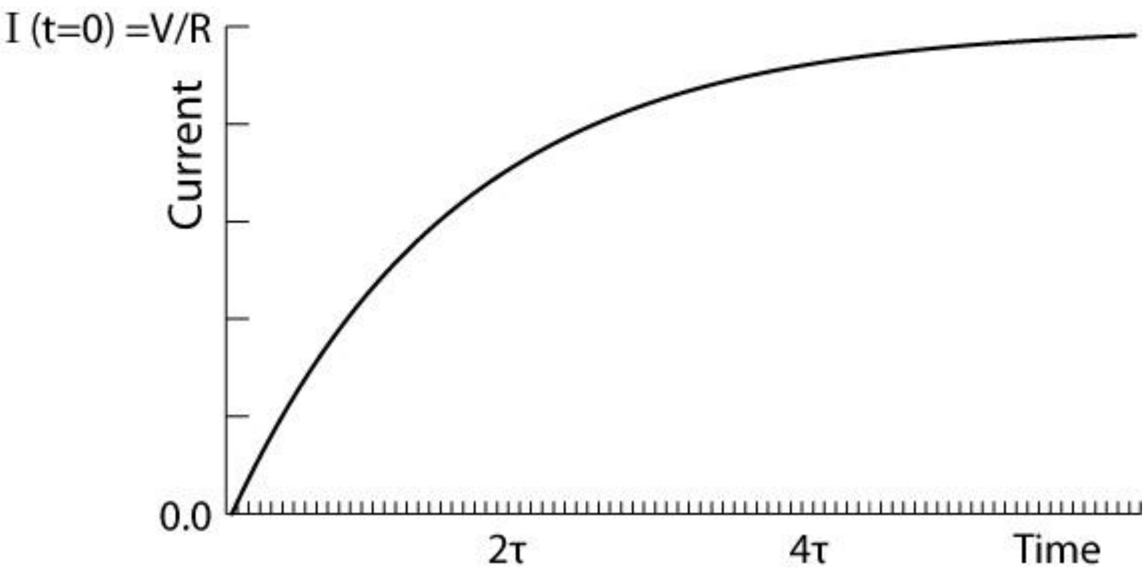
[RL series circuit for inductor charging]

$$i_L = \frac{E}{R}(1 - e^{-\frac{t}{\tau}})$$

where $\tau = L/R$



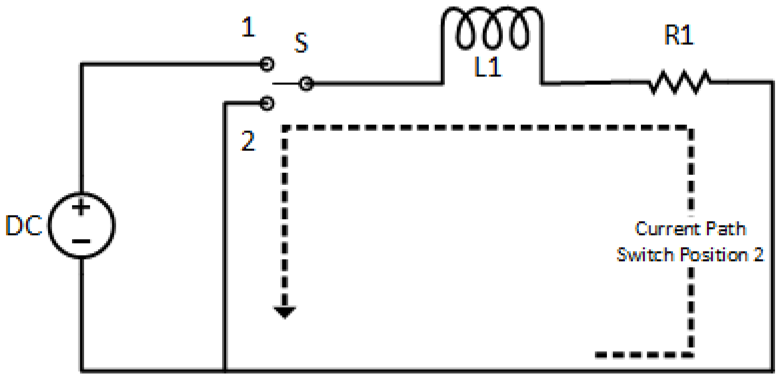
[Inductor voltage during the charging phase]



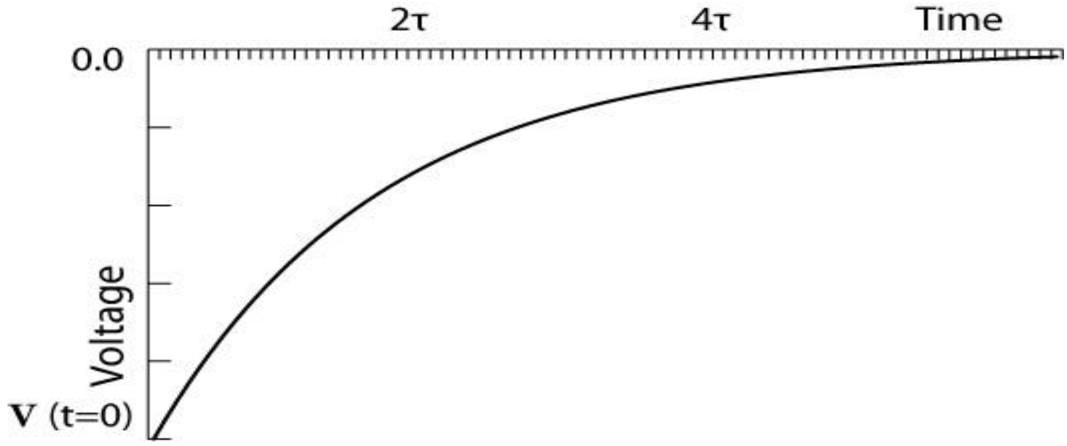
[Inductor current during the charging phase]

R-L discharging circuit:-

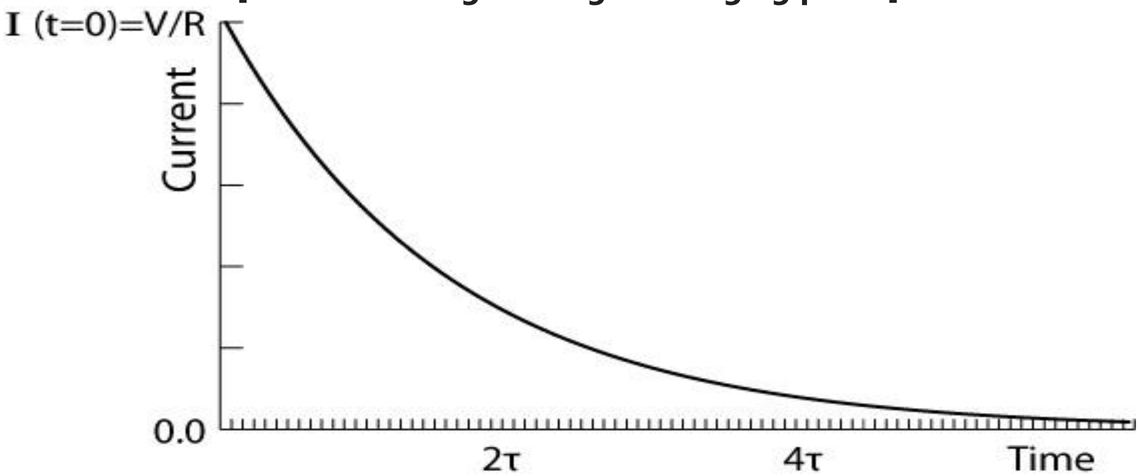
Suppose the above inductor is charged (has stored energy in the magnetic field around it) and has been disconnected from the voltage source. Now connected to the resistive load i.e. the switch is moved to position 2 at the time t=0. The energy stored will be discharged to a resistive load and will be dissipated in the resistor. The current will continue to flow in the same direction and will gradually decrease to zero as well as the voltage across the inductor. But if the inductor is disconnected and not connected to any load, so current will stop abruptly because of no closed path. According to the equations above, it will cause a huge voltage across the inductor and you will observe in the form of spark at switch terminals. The same phenomenon is used for car engine ignition.



[RL series circuit during the decay phase]



[Inductor voltage during discharging phase]



[Inductor current during discharging phase]

Procedure:-

1- We should take all the tools & instrument for this experiment.
2- Connect as per Circuit diagram.
3- Connect the oscilloscope at the output of circuits.
4- Then switch ON the supply.
5- Then observe the wave forms.

Conclusion:- From the above experiment, we learnt about the analyze the charging and discharging of an R-C & R-L circuit with oscilloscope.

Experiment - 12

Aim- Construct the following circuits using P-Spice/MATLAB software and compares the measurements and waveforms.

i. Superposition theorem
ii. Series Resonant Circuit
iii. Transient Response in R-L-C series circuit

SOFTWARE USED : MULTISIM / MATLAB Simulink

(I) SUPERPOSITION THEOREM:

In a linear network with several independent sources which include equivalent sources due to initial conditions, and linear dependent sources, the overall response in any part of the network is equal to the sum of individual responses due to each independent source, considered separately, with all other independent sources reduced to zero".

Procedure:

Step 1:

1. Make the connections as shown in the circuit diagram by using MULTISIM/MATLAB Simulink.
2. Measure the response 'I' in the load resistor by considering all the sources 10V, 15V and 8V in the network.

Step 2:

1. Replace the sources 15V and 8V with their internal impedances (short circuited).
2. Measure the response 'I1' in the load resistor by considering 10V source in the network.

Step 3:

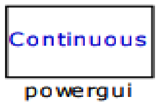
1. Replace the sources 10V and 8V with their internal impedances (short circuited).
2. Measure the response 'I2' in the load resistor by considering 15V source in the network.

Step 4:

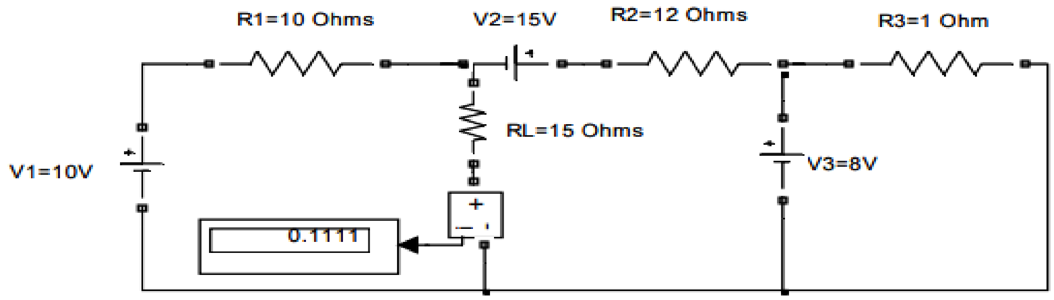
1. Replace the sources 10V and 15V with their internal impedances (short circuited).
2. Measure the response 'I3' in the load resistor by considering 8V source in the network. The responses obtained in step 1 should be equal to the sum of the responses obtained in step 2, 3 and 4.

I=I1+I2+I3

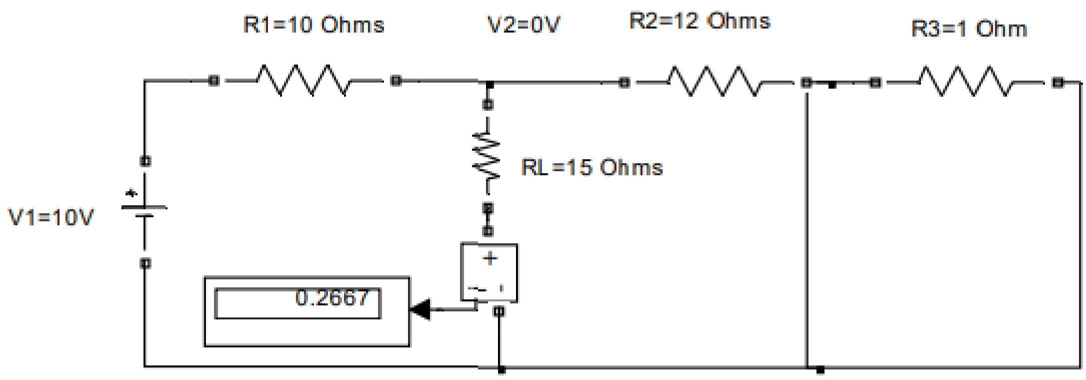
Hence Superposition Theorem is verified.



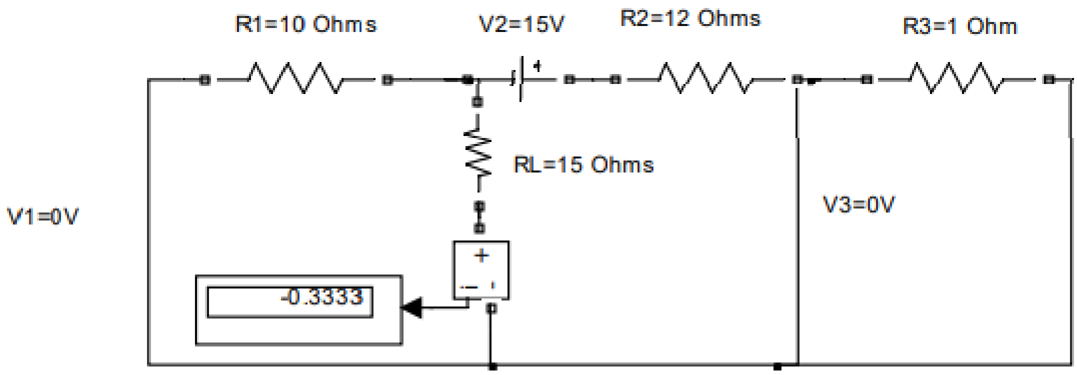
Step 1 : By Considering All Sources In The Network



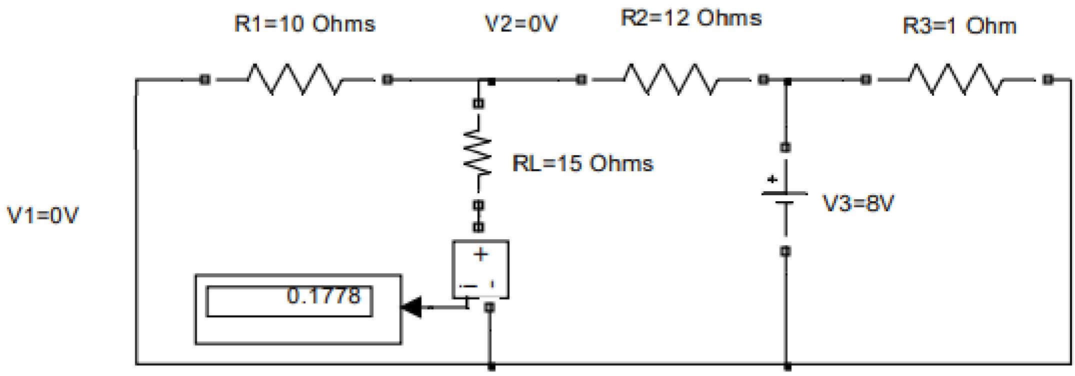
Step 2 : By Considering 10 V Sources In The Network



Step 3 : By Considering 15 V Sources In The Network



Step 4 : By Considering 8V Sources In The Network



.

Current through Load Resistor 15 Ohms :

Considering 10V Source I1= 0.2667A

Considering 15V Source I2 = 0.3333A

Considering 8V Source I3 = 0.1778

With all the sources in the network I = 0.1111A I=I1+I2+I3

Total Current : I1+I2+I3=0.2667-0.3333+0.1778

=0.1112A

Hence SuperPosition Theorem is Verified.

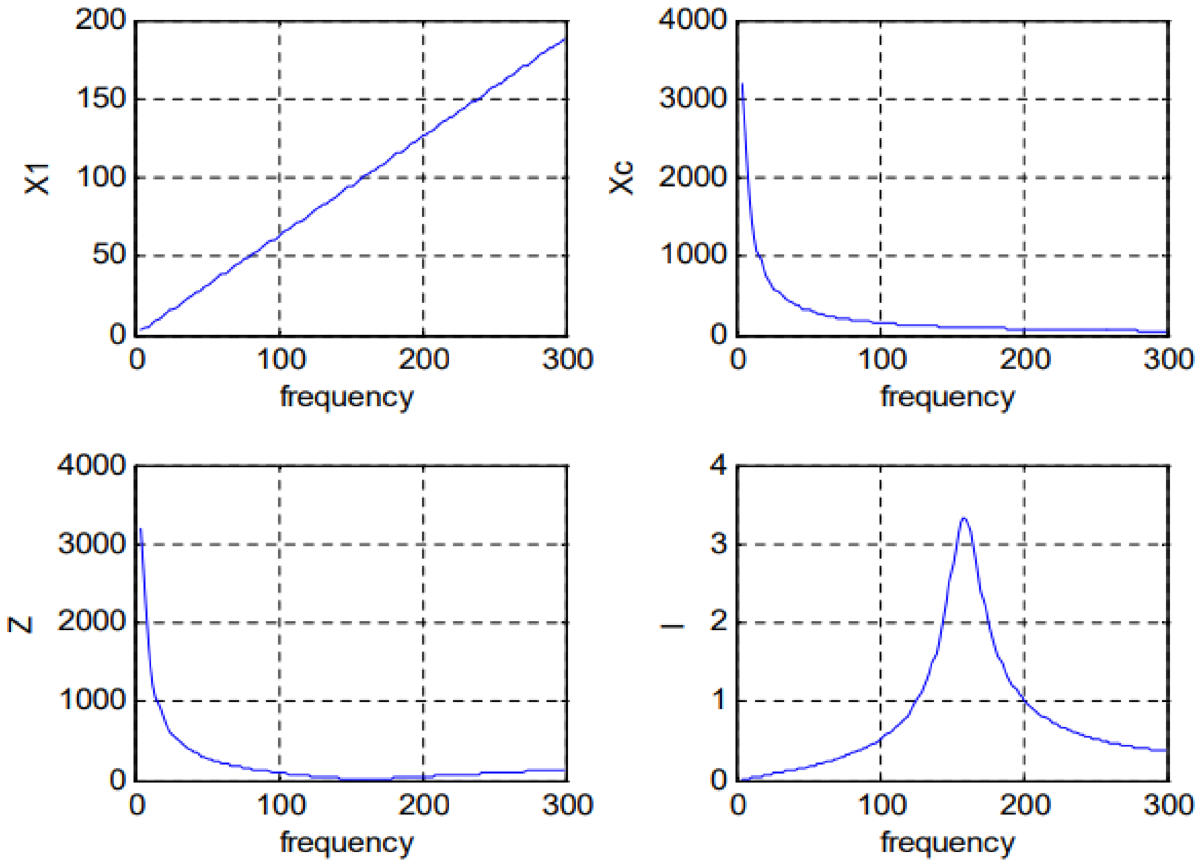
(ii) Series Resonant Circuit:-

Aim: - To obtain the plot of of frequency vs. XL, frequency vs. XC , frequency vs. impedance and frequency vs. current for the given series RLC circuit and determine the resonant frequency and check by theoretical calculations. R = 15Ω , C = 10 μ F, L = 0.1 H, V = 50V vary frequency in steps of 1 Hz using Matlab.

```
%Program to find the Parallel Resonance
 clc;
clear all;
close all;
 r=input('enter the resistance value   >');
l=input('enter the inductance value>');
c=input('enter the capacitance value    >');
 v=input('enter the input voltage->');
 f=5:2:300;
 xl=2*pi*f*;
 xc=(1./(2*pi*f*c)
 );
 x=xl-xc;
z=sqrt((r^2)+(x.^2));
i=v./z;
%plotting the graph
subplot(2,2,1);
plot(f,xl);
grid;
xlabel('frequency');
ylabel('X1');
subplot(2,2,2);
plot(f,xc);
grid;
xlabel('frequency');
ylabel('Xc');
subplot(2,2,3);
plot(f,z);
grid;
xlabel('frequency');
ylabel('Z');
subplot(2,2,4);
plot(f,i);
grid;
xlabel('frequency');
ylabel('I');
```

PROGRAM RESULT:

```
enter the resistance value >15
enter the inductance value->0.1
enter the capacitance value----->10*10^-6
enter the input voltage------->50
```



(iii) Transient Response in R-L-C series circuit:-

The transient response is the fluctuation in current and voltage in a circuit (after the application of a step voltage or current) before it settles down to its steady state. This lab will focus on simulation of series RLC (resistor inductor-capacitor) circuits to demonstrate transient analysis.

Transient Response of Circuit Elements:

A. Resistors: As has been studied before, the application of a voltage V to a resistor (with resistance R ohms), results in a current I, according to the formula:

$I = V/R$

The current response to voltage change is instantaneous; a resistor has no transient response.

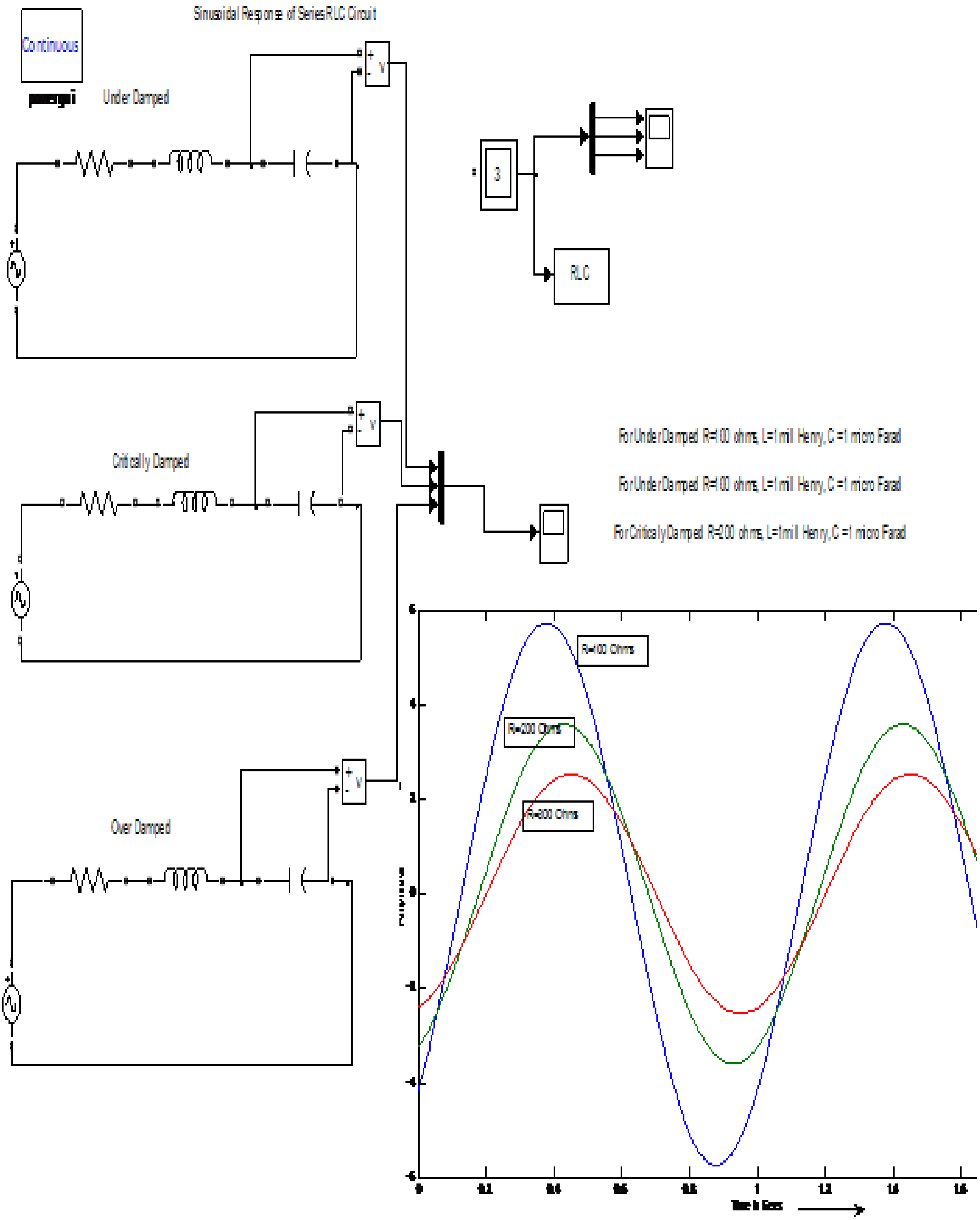
B. Inductors: A change in voltage across an inductor (with inductance L Henrys) does not result in an instantaneous change in the current through it. The i-v relationship is described with the equation: $v=L\, di/ dt$

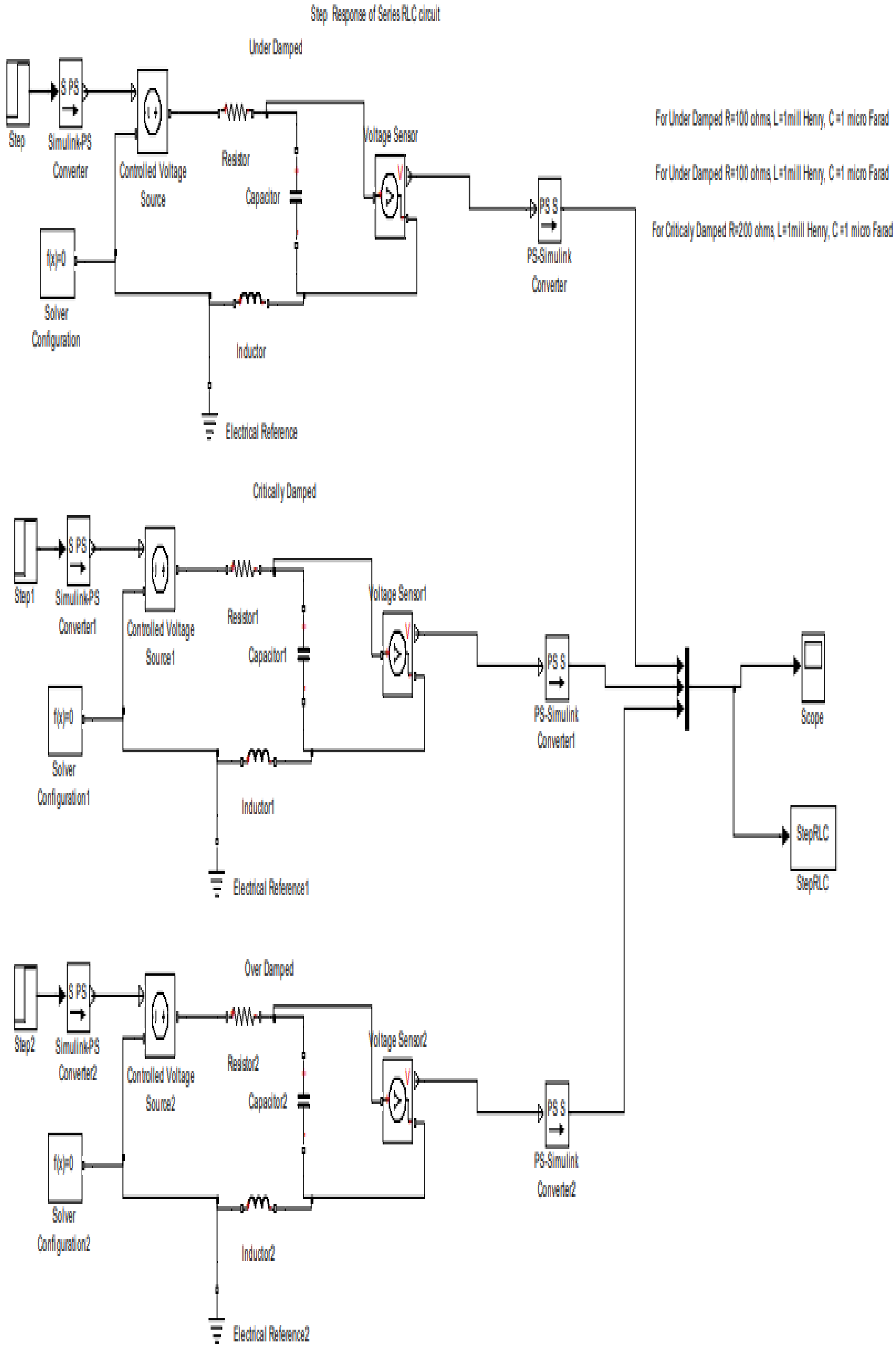
This relationship implies that the voltage across an inductor approaches zero as the current in the circuit reaches a steady value. This means that in a DC circuit, an inductor will eventually act like a short circuit.

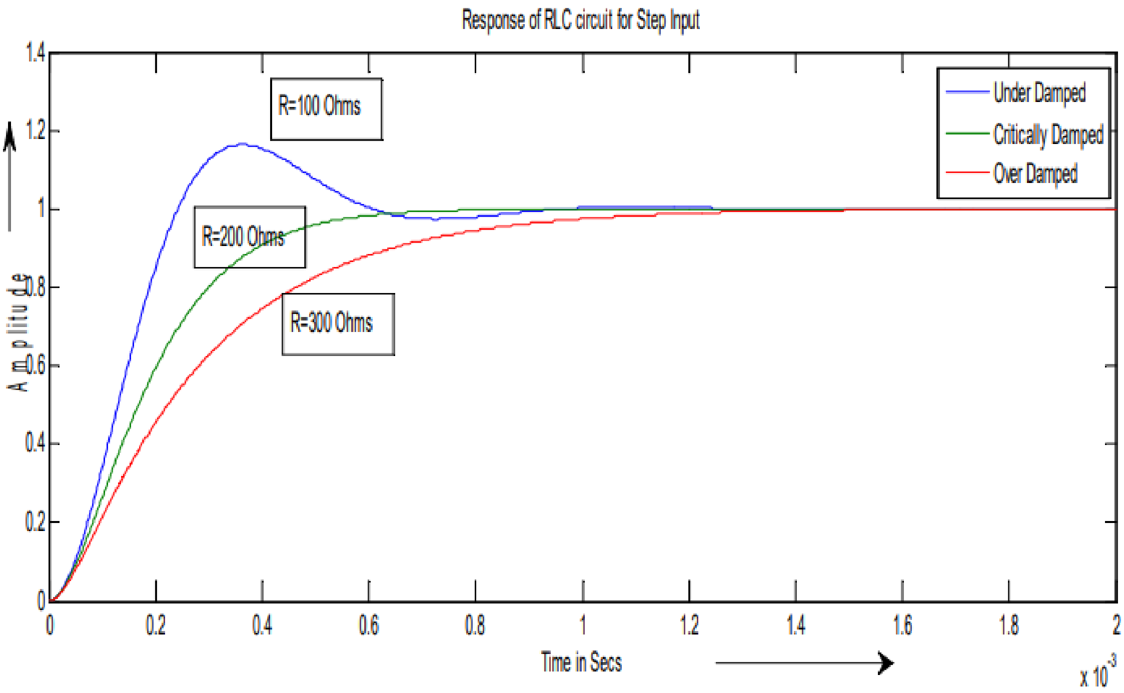
C. Capacitors: The transient response of a capacitor is such that it resists instantaneous change in the voltage across it. Its i-v relationship is described by: $i=C\, dv /dt$

This implies that as the voltage across the capacitor reaches a steady value, the current through it approaches zero. In other words, a capacitor eventually acts like an open circuit in a DC circuit Series Combinations of Circuit Elements: Solving the circuits involves the solution of first and second order differential equations.

[Sinusoidal response of series RLC Circuit]


Sinusoidal Response of Series RLC Circuit
Continuous
powergui
Under Damped
Critically Damped
Over Damped
3
RLC
For Under Damped R=100 ohms, L=1 mili Henry, C =1 micro Farad
For Under Damped R=100 ohms, L=1 mili Henry, C =1 micro Farad
For Critically Damped R=200 ohms, L=1mili Henry, C =1 micro Farad
R=100 Ohms
R=200 Ohms
R=300 Ohms
Time in Secs


Step Response of Series RLC circuit
Under Damped
For Under Damped R=100 ohms, L=1mill Henry, C =1 micro Farad
For Under Damped R=100 ohms, L=1mill Henry, C =1 micro Farad
For Criticaly Damped R=200 ohms, L=1mill Henry, C =1 micro Farad
Step
Simulink-PS Converter
Controlled Voltage Source
Resistor
Capacitor
Voltage Sensor
PS-Simulink Converter
Solver Configuration
Inductor
Electrical Reference
Critically Damped
Step1
Simulink-PS Converter1
Controlled Voltage Source1
Resistor1
Capacitor1
Voltage Sensor1
PS-Simulink Converter1
Solver Configuration1
Inductor1
Electrical Reference1
Scope
StepRLC
StepRLC
Over Damped
Step2
Simulink-PS Converter2
Controlled Voltage Source2
Resistor2
Capacitor2
Voltage Sensor2
PS-Simulink Converter2
Solver Configuration2
Inductor2
Electrical Reference2



PROCEDURE:

1. Make the connections as shown in connection diagram.
2. Observe the output waveforms across RLC.
3. Change the value of resistance such that the output obtained at each oscilloscope is

i) Critically damped.

ii) Under damped.

iii) Over damped.

RESULT:

The critically damped, under damped, damped response is observed for an RLC network in the simulation environment.

- The response to various inputs can be simulated.
- The response of any system designed can be simulated to verify its performance and design