DEPARTMENT OF ELECTRICAL AND ELECTRONICS ENGINEERING

LAB MANUAL

EE 6211 ELECTRICAL CIRCUIT LABORATORY

(FOR II SEMESTER EEE)

LIST OF EXPERIMENTS

1. Experimental verification of Kirchhoff's voltage and current laws
2.Experimental verification of network theorems (Thevenin, Norton, Superposition and maximum power transfer Theorem).
3. Study of CRO and measurement of sinusoidal voltage, frequency and power factor.
4. Experiental determination of time constant of series R-C electric circuits.
5. Experimental determination of frequency response of RLC circuits.
6. Design and Simulation of series resonance circuit.
7. Design and Simulation of parallel resonant circuits.
8. Simulation of low pass and high pass passive filters.
9. Simulation of three phase balanced and unbalanced star, delta networks circuits.
10. Experimental determination of power in three phase circuits by two-watt meter method .
11. Calibration of single phase energy meter.
12. Determination of two port network parameters

LIST OF EXPERIMENTS

CYCLE -I

1. Experimental verification of Kirchhoff's voltage and current laws
2.Experimental verification of network theorems (Thevenin, Norton, Superposition and maximum
power transfer Theorem).
3. Study of CRO and measurement of sinusoidal voltage, frequency and
power factor.
4. Experiental determination of time constant of series R-C electric circuits.
5. Experimental determination of frequency response of RLC circuits.
6. Experimental determination of power in three phase circuits by two-watt meter method.

CYCLE –II

7. Calibration of single phase energy meter
8. Determination of two port network parameters
9. Design and Simulation of series resonance circuit
10. Design and Simulation of parallel resonant circuits.
11. Simulation of low pass and high pass passive filters.
12. Simulation of three phase balanced and unbalanced star, delta networks circuits

Exp No: 1B

Verification of Kirchoff's Law

Aim:

To verify Kirchoff's current law and Kirchoff's voltage law for the given circuit.

Apparatus required:

Sl.No	Name of the Apparatus	Range	Type	Qty
1.	Dual Regulated Power Supply	(0-30)V	MC	1No
2	Resistors (Fixed)	100Ω, 120Ω, 220Ω	-	Each 1 No
3.	Ammeter	(0-100)mA	MC	3 No
4.	Voltmeter	(0-20)V	MC	3 No
5.	Connecting wires			
	Bread board			

Statement:

Formula Used:

Kirchoff's Current law (KCL)

This law states the algebraic sum of current meeting at any junction in a circuit is zero.

$\Sigma\, i = 0$

Kirchoff's Voltage law (KVL)

This law states that the algebraic sum of electromotive forces plus the algebraic sum of voltages across the impedance in any closed electrical circuit is equal to zero.

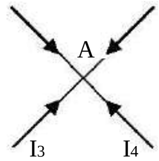
$\Sigma\, IR + \Sigma\, EMF = 0$

Theory

Kirchoff's Current law (KCL)

The algebraic sum of current meeting at any junction in a circuit is zero.

I_2 I_1



At any node the total current meeting the node is equal to the total current leaving the node. A node or junction is a common point in a network to which more than two circuit elements are connected.

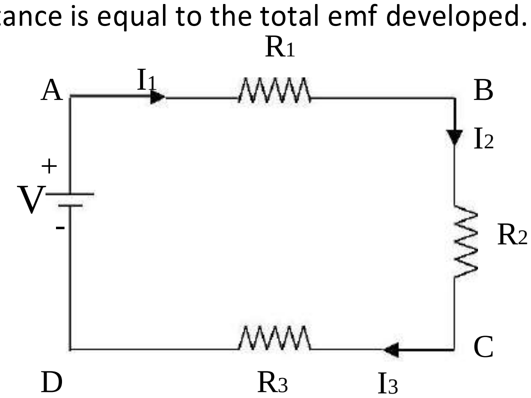
At node A currents I_1 and I_2 flows into the junctions and currents I_3 and I_4 flows away from junction.

Total current entering into the junction = total current leaving from the

junction $I_1 + I_2 = I_3 + I_4$

Kirchoff's Voltage law (KVL)

In any closed path of an electrical circuit the algebraic sum of product of current and resistance is equal to the total emf developed.



By the consideration of voltage drop as positive and voltage raise as negative, Applying KVL to loop ABCDA,

$I_1R_1 + I_2R_2 + I_3R_3 - V = 0$

(i.e) $I_1R_1 + I_2R_2 + I_3R_3 = V$

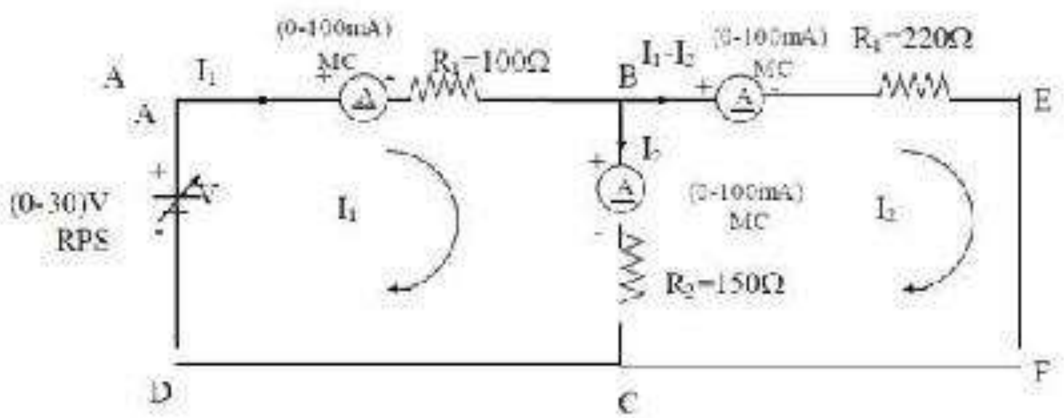
Precaution:

1. Short circuit and loose connection should be avoided
2. During trouble shooting RPS should be switched off.

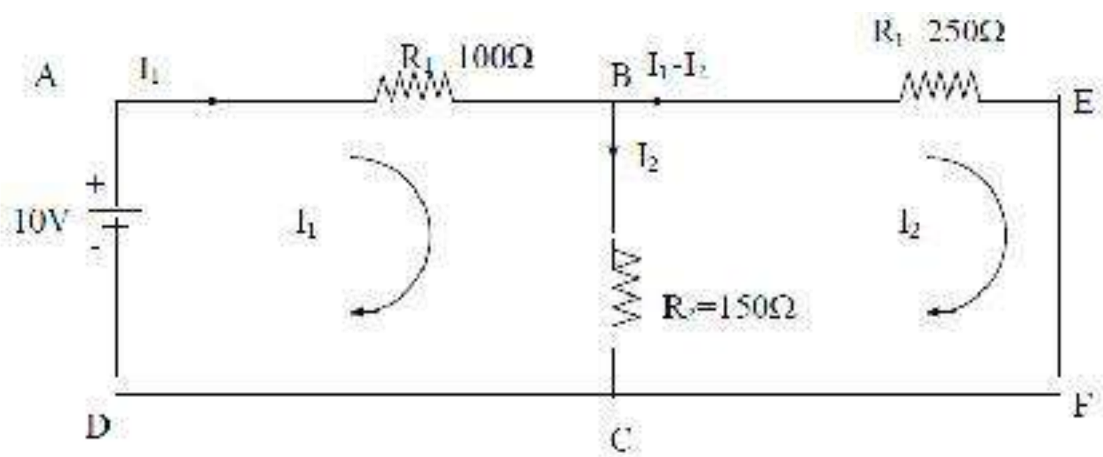
Procedure:

1. Make the connections as per the circuit diagram
2. Switch on the supply and set the required input voltages.
3. Note the corresponding ammeter and voltmeter readings
4. Reduce the input voltage to zero. Then switch off the supply.
5. Remove the connections.
6. Compare the theoretical and practical values.

Circuit diagram for Kirchoff's Current law



Theoretical Expression



Applied voltage =10V

Applying Kirchoff's Current law

For Loop ABCDA

$100I_1-120\ I_2=10$ -----(1)

For Loop BEFCB

$-220(I_1 - I_2)+120\, I_2 = 0$

$-220I_1+340I_2 = 0$ ----- (2)

Solving equation 1 & 2,

$I_1 = 0.056A = 56\ mA$

$I_2 = 0.036A = 36\ mA$

Current through R_1 $(I_1) = 56\ mA$

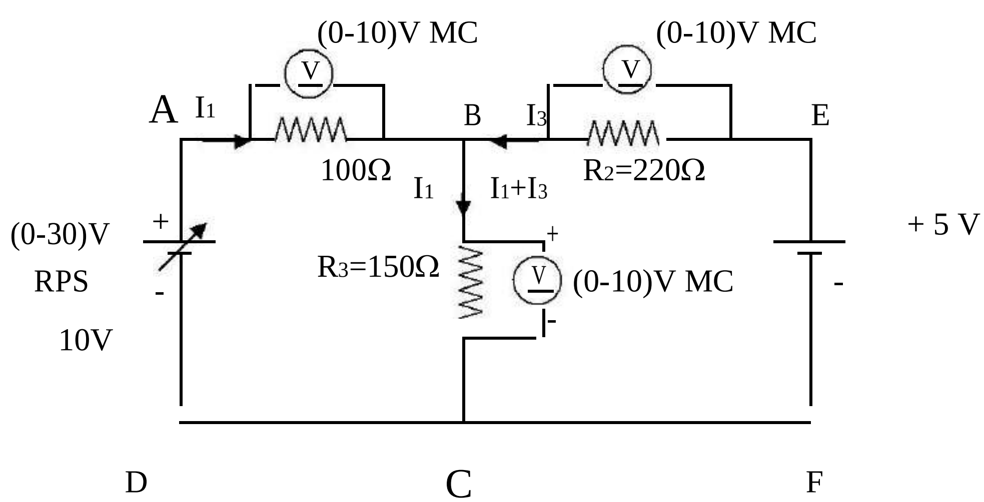
Current through R_2 $(I_2) = 36\ mA$

Current through R_3 $(I_1 - I_2) = 20\ mA$

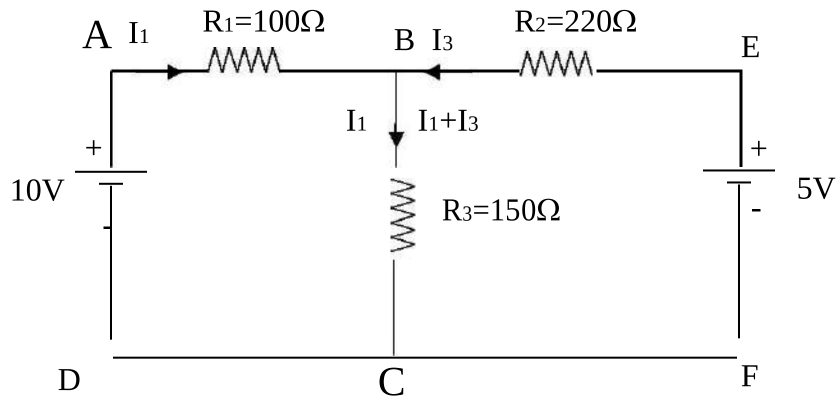
Tabular Column for Kirchoff's Current law

Sl. No	Theoritical Values			Practical Values		
	Current through R_1 I_1 (mA)	Current through R_2 I_2 (mA)	Current through R_3 I_3 (mA)	Current through R_1 I_1 (mA)	Current through R_2 I_2 (mA)	Current through R_3 I_3 (mA)

Circuit Diagram for Kirchoff's Voltage law



Theoretical Explanation



Applied voltage across source 1 = 10 V

Applied voltage across source 1 = 5 V

Applying Kirchoff's Voltage law

For loop ABCDA

$10- 100I_1 – 120(I_1+I_3) =0$

$220I_1 – 120\ I_3 = 10$ -----(1)

For loop BEFCB

$5-220I_3-120(I_1+I_3)=0$

$120I_1-34I_3 = 5$ -----(2)

Solving equation 1 & 2

I_1 = 0.0463 A= 46.3 mA

I_3 =-0.0016 A =-1.6mA

I_1+I_3 = 44.7 mA

Voltage across R_1 (V_1) = $I_1\ R_1$ = 0.0463 X 100

Voltage across R_2 (V_2) = $R_2\ I_2$ = $(I_1+I_3)I_2$ = 0.047X120 = 5.64 V

Voltage across R_3 (V_3) = $R_3\ I_3$ = I_3R_3 = -0.016X220 = - 0.352V

Tabular Column for Kirchoff's Voltage law

Sl.No	Theoritical Values			Practical Values		
	Voltage across R_1 Volts	Voltage across R_2 Volts	Voltage across R_3 Volts	Voltage across R_1 Volts	Voltage across R_2 Volts	Voltage across R_3 Volts

Voltage across source 1 = 10 V

Voltage across source 2 = 5 V

Result:

Thus the Kirchoff's Current law and Voltage law were verified and the theoretical values were compared with the practical values.

Exp No:2 A

Verification of Thevenin's Theorem

Aim:

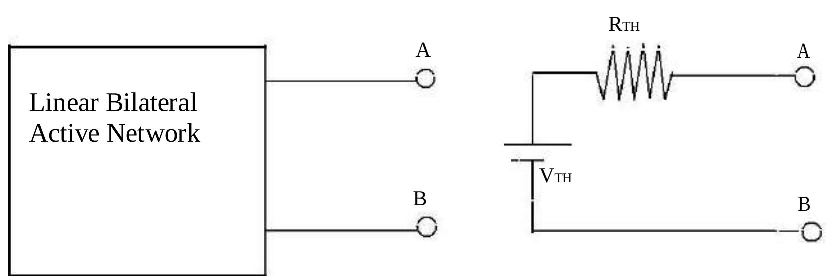
To practically verify the Thevenin's theorem for the network with the theoretical calculations.

Apparatus Required:

Sl.No	Name of the Apparatus	Range	Type	Qty
1.	Milli Ammeter	(0-20)mA	MC	1
2.	Resistors			
3.	RPS	(0-30)V	Single Mode	01
4.	Bread Board			
5.	Connecting Wires			

Statement:

Any linear bilateral network with two output terminals AB can be replaced by a simple equivalent circuit with single voltage source V_{th} (Thevinin voltage or Open circuit voltage) in series with a single resistor R_{th}(Thevenin resistance) or looking back resistance or impedance Z_{th} (Thevenin impedance in ac circuit) about the terminals AB.



R_{th} = Thevenin's Resistance (Equivalent resistance between A and B)in ohms
V_{th} = Thevenin's Resistance (Open circuit voltage between A and B) in volts
R_L = Load resistance connected between A and B in ohms
I_L = Load Current = $V_{th} / (R_{th} + R_L)$ in amps

Procedure:

For finding Thevenin's Voltage(V_{TH})

1. The connections are made as per the circuit diagram.
2. Remove the load resistor and find the open circuit voltage or thevenin's voltage by connecting suitable dc voltmeter.

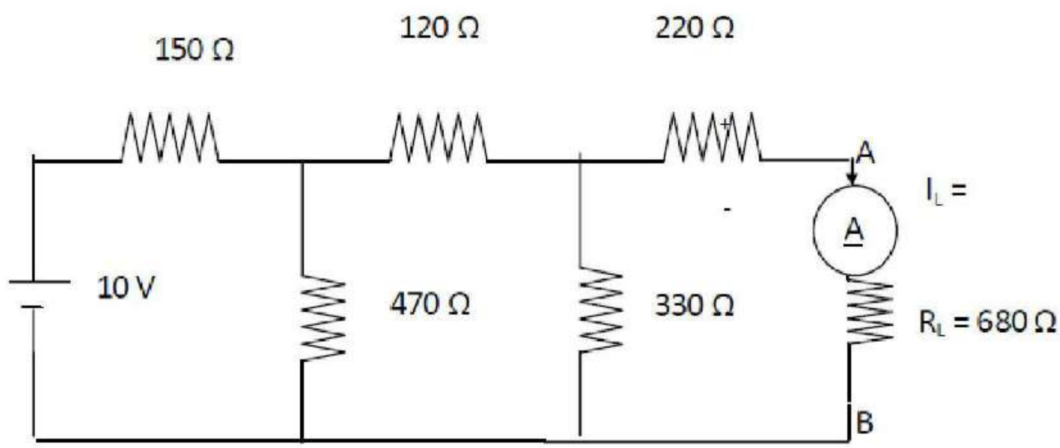
For Finding Thevenins Resistance(R_{th})

1. Remove the load resistor
2. Short circuit or kill the voltage source and open circuit the current source
3. calculate the resistance across the load terminals by connecting the ohmmeter across the load terminals.

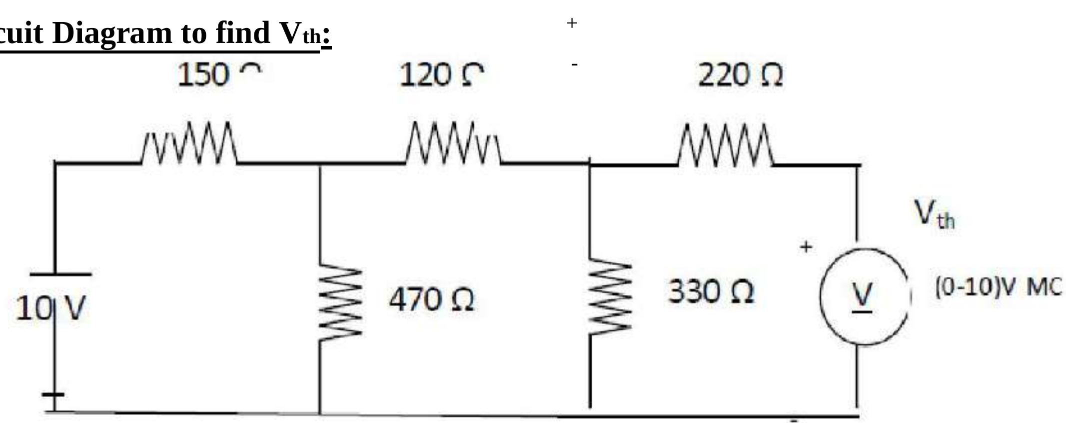
For finding load current from Thevenin equivalent circuit.

1. Draw the thevenins equivalent circuit
2. Find the load current (I_L) from the equivalent circuit by connecting an dc ammeter through the load resistor.

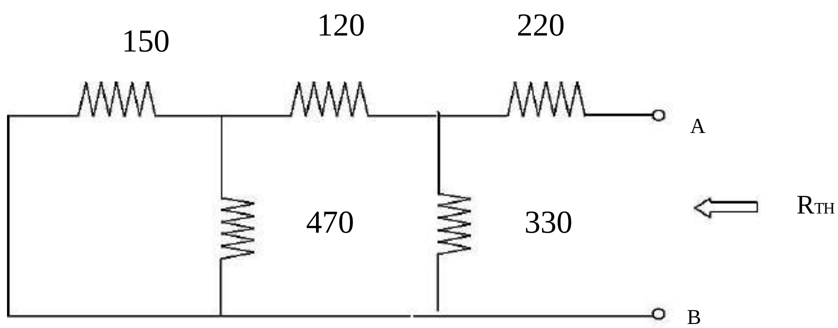
Circuit diagram to find Load Current:



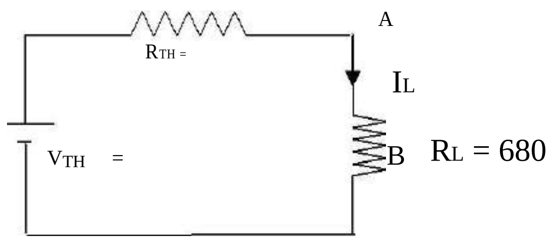
Circuit Diagram to find V_{th}:



Circuit Diagram to find R_{TH}:



Thevenin's Equivalent Circuit:



I_L=

Tabular Column

Sl No	Theoretical Value			Practical Value		
	Thevenin's Voltage V_{TH} (V)	Current flowing through R_L I_L	Thevenin's equivalent Resistance R_{TH}	Thevenin's Voltage V_{TH} (V)	Current flowing through R_L I_L mA	Thevenin's equivalent Resistance R_{TH}

Result:

Thus the Thevenins theorem was verified and the theoretical values were compared with the practical values

Exp No:2B

Verification of Norton's Theorem

Aim:

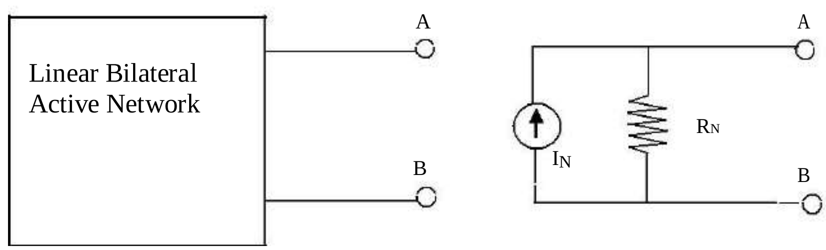
To practically verify the Norton's theorem for the network with the theoretical calculations.

Apparatus Required:

Sl.No	Name of the Apparatus	Range	Type	Qty
1.	Milli Ammeter	(0-20)mA	DC	1
2.	Resistors			
3.	RPS	(0-30)V	Single Mode	01
4.	Bread Board			
5.	Connecting Wires			

Statement:

Any linear bilateral network with two output terminals AB can be replaced by a simple equivalent circuit with single current source I_N or I_{sc}(Norton's current or Short circuit current) in parallel with a single resistor R_N(Norton resistance) or impedance Z_N (Norton impedance) about the terminals AB.



R_N = Nortons's Resistance (Equivalent resistance between A and B)
I_N = Norton's current (Short circuited path current through A and B)in milli amps
R_L = Load resistance connected between A and B in ohms
I_L = Load Current = (I_N X R_{th}) / (R_{th} + R_L) in milli amps

Procedure:

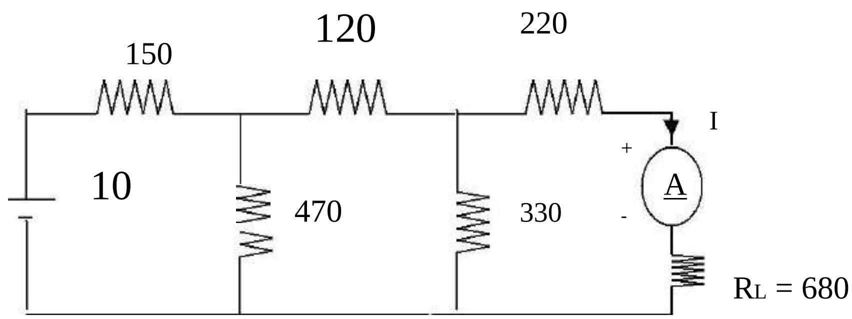
For Finding Norton's current source

1.In the given circuit find the current(I_L) through the load resistor.
2. Remove the load resistor and short circuit the path A and B
3. Measure the Norton's current by connecting the suitable dc ammeter. **For Finding** Nortons **(R_{SC}) or R_N**
1. Remove the load resistor AB
2. Short circuit or kill the voltage source and open circuit the current source
3. Calculate the resistance across the load terminals by connecting the ohmmeter across the load terminals.

For finding load current from Norton's equivalent circuit.

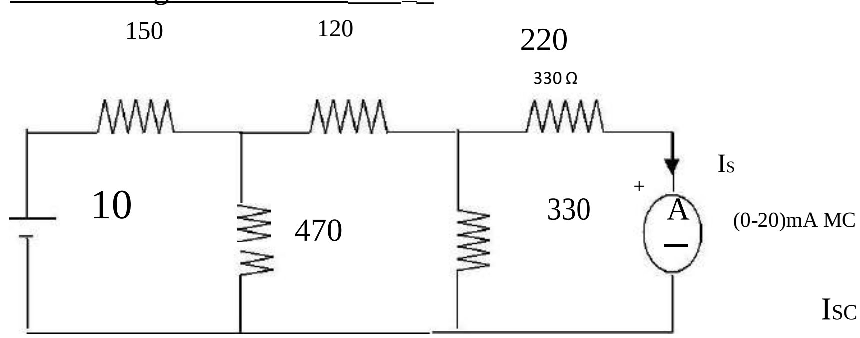
1. Draw the Norton's equivalent circuit
2. Find the load current (I_L) from the equivalent circuit by connecting suitable DC ammeter through it.

Basic Circuit diagram:



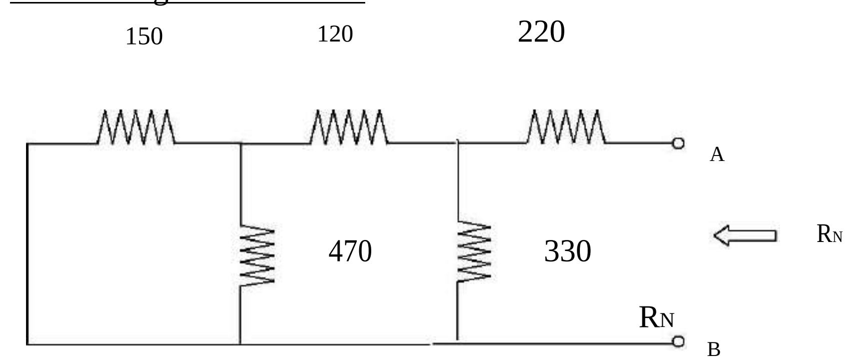
I_L =

Circuit Diagram to find I_{sc} or I_N :



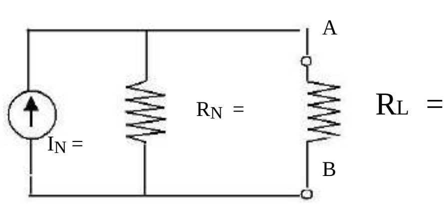
I_{SC} or I_N =

Circuit Diagram to find R_N:



=

Norton's Equivalent Circuit:



I_L =

Sl No	Theoretical Value		Practical Value	
	Norton's Current I_N (mA)	Equivalent Resistance R_N	Load current I_L (mA)	Equivalent Resistance R_N

Result:

Thus the Norton`s theorem was verified and the theoretical values were compared with the practical values.

Exp No : 2C

Verification of Super Position Theorem

Aim:
To practically verify the Super Position theorem for the network with the theoretical calculations.

Apparatus Required:

Sl.No	Name of the Apparatus	Range	Type	Qty
1.	Milli Ammeter	(0-20)mA	DC	1
2.	Resistors	2.2KΩ, 5.6KΩ		
3.	RPS	(0-30)V	Single Mode	01
4.	DRB			
5.	Bread Board			

Statement:

In a linear, lumped element, bilateral electric circuit that is energized by more than one sources the current in any resistor is equal to the algebraic sum of the separate currents in the resistor when each source acts separately.

For the removal of the sources the following points should be considered,

1. Removal of ideal voltage source means short circuiting
2. Removal of ideal current source means open circuiting
3. Removal of practical voltage source and current source means replacing them by their respective internal resistances.

Procedure:

1. In the given circuit one voltage source is allowed to act and other voltage source is short circuited.
2. The Value of current (I_1) through the load resistor (R_L) is noted.
3. The first voltage source is short circuited and the second voltage source is allowed to act on the circuit.
4. The Value of current (I_2) through the load resistor (R_L) is noted.
5. Then by the Superposition theorem, the value of the current through the load resistor is calculated by using the formula,

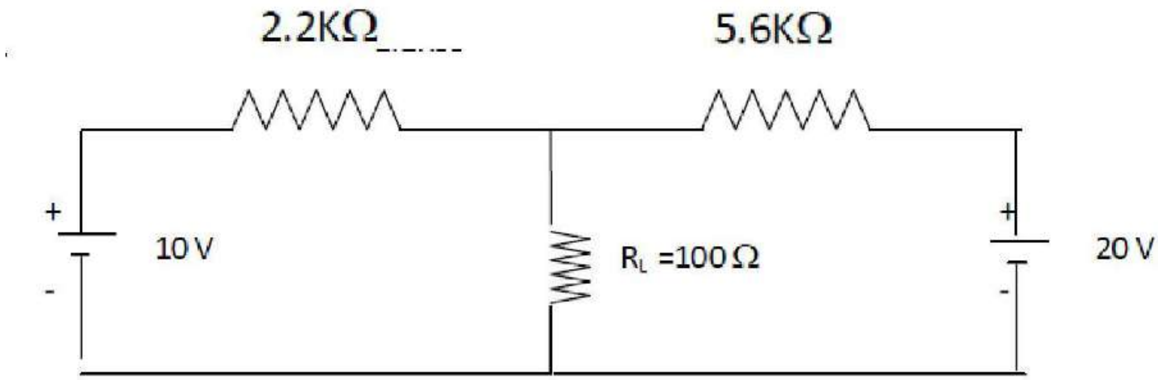
$$I_L = I_{L1} + I_{L2}.$$

(If both the currents are in the same direction then they are additive. If both are in opposite direction the load current is given by, I_L = higher value of current – lower value of current and

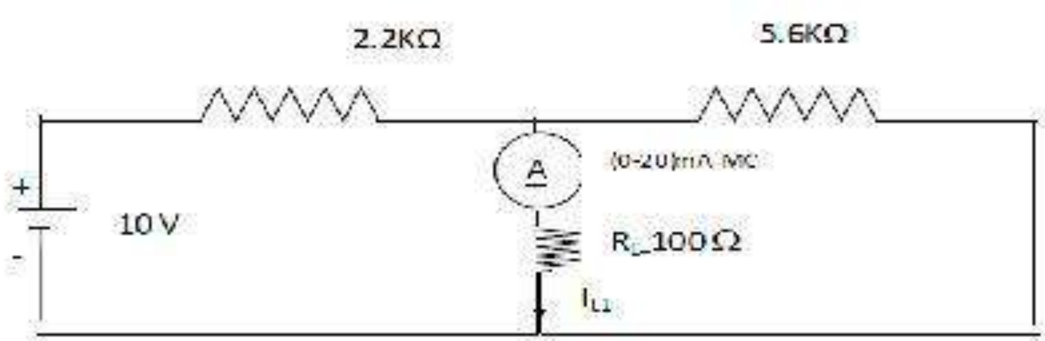
the load current will flow in the direction of higher value current)

6. The current I_{L2} is noted with two sources connected to the circuit.
7. According to Superposition theorem $I_{L1} = I_{L2}.$

<u>Practical Part:</u>



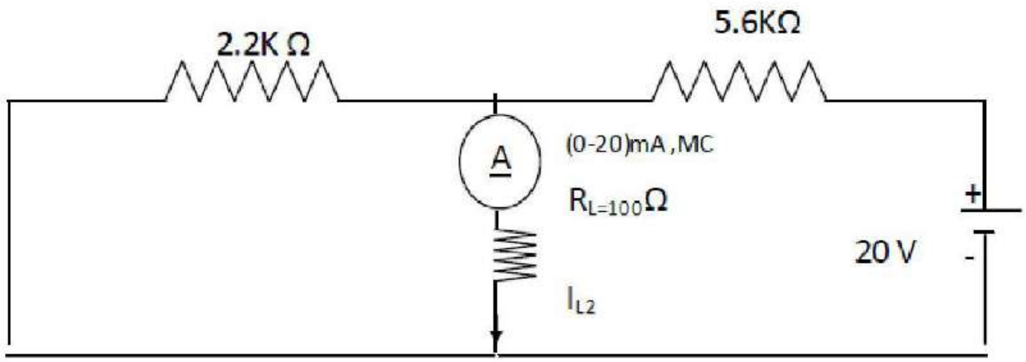
<u>STEP 1: Allowing first 10V source to act and de activating 20v source by short circuiting it</u>



Find the Current through R_L as(I_{L1}) =

Sl.No	Resistance R_L in kΩ	Current I_{L1} in mA

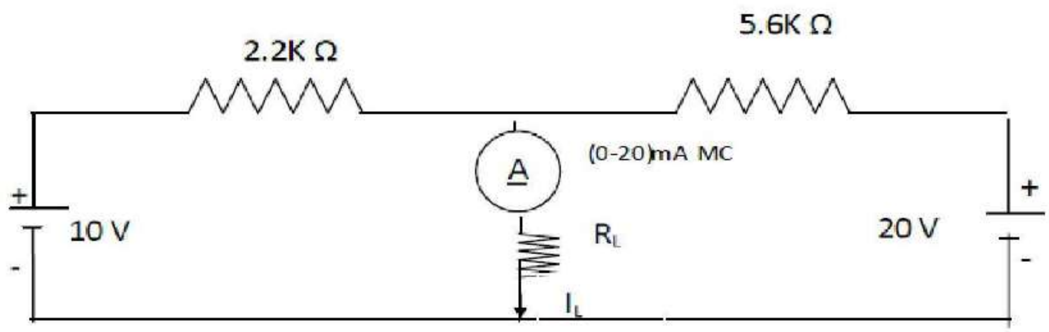
<u>STEP 2:</u>
<u>Allowing first 20V source to act and de activating 10v source by short circuiting it.</u>



Current through R_L as (I_{L2}) =

Sl.No	Resistance in R_L kΩ	Current I_{L2} in mA

STEP 3: When both the sources are acting



Current through R_L as (I_L) =

According to Superposition theorem $I_L = I_{L1} + I_{L2}$

Sl.No	Resistance in R_L kΩ	$I_{L2} + I_{L2}$ in mA

Result : Thus the Super Position theorem was verified and the theoretical values werecompared with the practical values.

Exp No : 2D

Verification of Maximum Power Transfer Theorem

Aim:

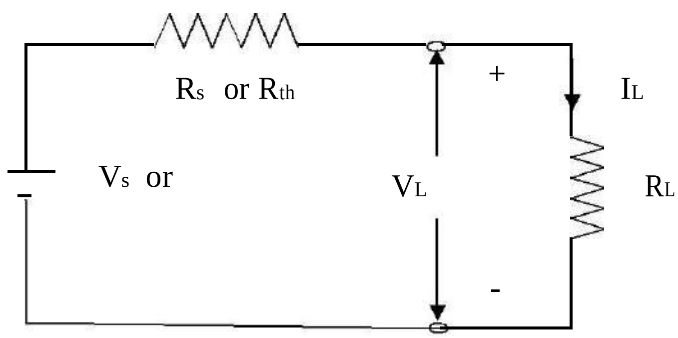
To practically verify the Maximum Power Transfer Theorem for the network with the theoretical calculations.

Apparatus Required:

Sl.No	Name of the Apparatus	Range	Type	Qty
1.	Milli Ammeter	(0-20)mA	MC	1
2	Voltmeter	(0-30) V	MC	1
3.	Resistors	220 Ω		2
4.	Resistors	330 Ω		1
5.	DRB			1
6.	RPS	(0-30) V	Single mode	1
7.	Bread Board			1
8	Connecting wires			

Statement:

It states that the power transferred from source to the load will be maximum, when the source resistance or impedance is equal to load resistance or impedance.



The current through the load resistor is obtained using equation, .

$$I_L = \frac{V_{Th}}{R_L + R_{Th}}$$

The current through the load resistor is obtained using equation,

$$P_L = (I_L)^2 \times R_L$$

DERIVING CONDITION FOR MAXIMUM POWER TRANSFER

$$P_L = \left(\frac{V_{Th}}{R_{Th} + R_L}\right)^2 \times R_L$$

When maximum power is transferred to the load resistor, the rate of change of power delivered with respect to load resistance is zero,

$$\frac{dP_L}{dR_L} = 0$$

$$\frac{dP_L}{dR_L} = (V_{Th})^2 \times \left[\frac{1}{(R_{Th} + R_L)^2} - \frac{2 \times R_L}{(R_{Th} + R_L)^3}\right]$$

Equating the derivative to zero,

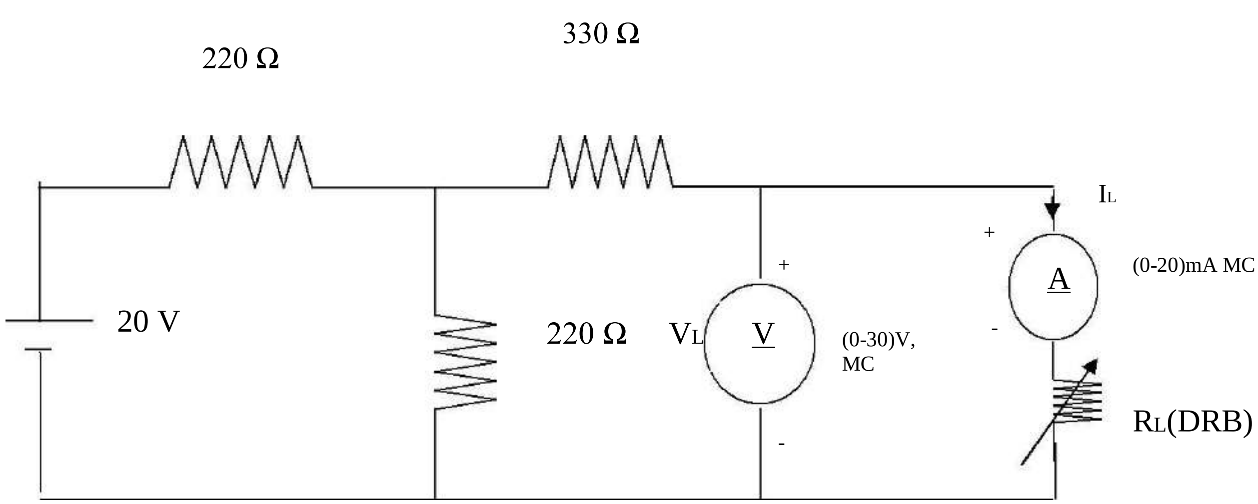
$$(V_{Th})^2 \times \left[\frac{1}{(R_{Th} + R_L)^2} - \frac{2 \times R_L}{(R_{Th} + R_L)^3}\right] = 0$$

From the above equation we can get

$R_L = R_{Th}$ or

$\boldsymbol{R_L = R_S}$

Basic Circuit diagram:

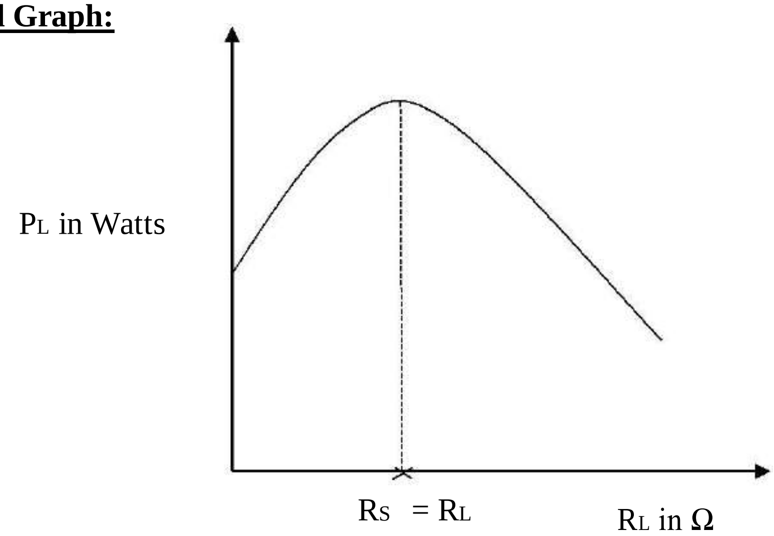


Tabulation:

Sl No	R_{TH} or R_S	R_L	I_L	$V_L = I_L \times R_L$	$P_L = I_L^2 \times R_L$

	(Ω)	(Ω)	(Amps)	(Volts)	(Watts)
1					
2					
3					
4					
5					

Model Graph:



Procedure:

1. Make the connections as per the circuit

diagram. 2.Find the value of Thevenin resistance

3. For different value of R_L ,note the values of V_L and I_L .

4. Calculate the power delivered to the load for each value of R_L.

5. Switch OFF the supply and disconnect the circuit.

6. Plot the graph between load resistance RL (X- axis) and Power absorbed PL (Y- axis)

Result :

Thus the maximum power transfer theorem was verified and the theoretical values were compared with the practical values.

1) The maximum power delivered to the load is _______________

2) The value of $R_S = R_L =$ __________

Experiment No: Date:

5. Experimental determination of frequency response of RLC circuits.

Aim:

To study the frequency response of RLC circuits

Apparatus required:

S.No	Name of the apparatus	Range	Type	Quantity
1	Function generator	(0-3) MHz	-	1
2	Resistor	1KΩ	-	1
3	Capacitor	0.1µF	-	1
4	Inductor	10mH	-	1
5	Bread board	-	-	1

Theory:

Resonance circuits are one of the most important circuits used electrical and electronic circuits. Theycan be found in various forms such as in AC mains filters, noise filters and also in radio and television tuning circuits producing a very selective tuning circuit for the receiving of the different frequency channels. In a series RLC circuit, when the resistor, inductor and capacitor are connected in series, there is a frequency point where the inductive reactance of the inductor becomes equal in value to the capacitive reactance of the capacitor. In other words,$X_L = X_C$. The point at which this occurs is called the **ResonantFrequency** point, (f_r) of the circuit, and as we are analysing a series RLC circuit this resonance frequencyproduces a **Series Resonance**. In a parallel RLC circuit, when the resistor, inductor and capacitor are connected in parallel, parallel resonance is produced.

Inductive reactance : $X_L = 2\pi fL = \omega L$

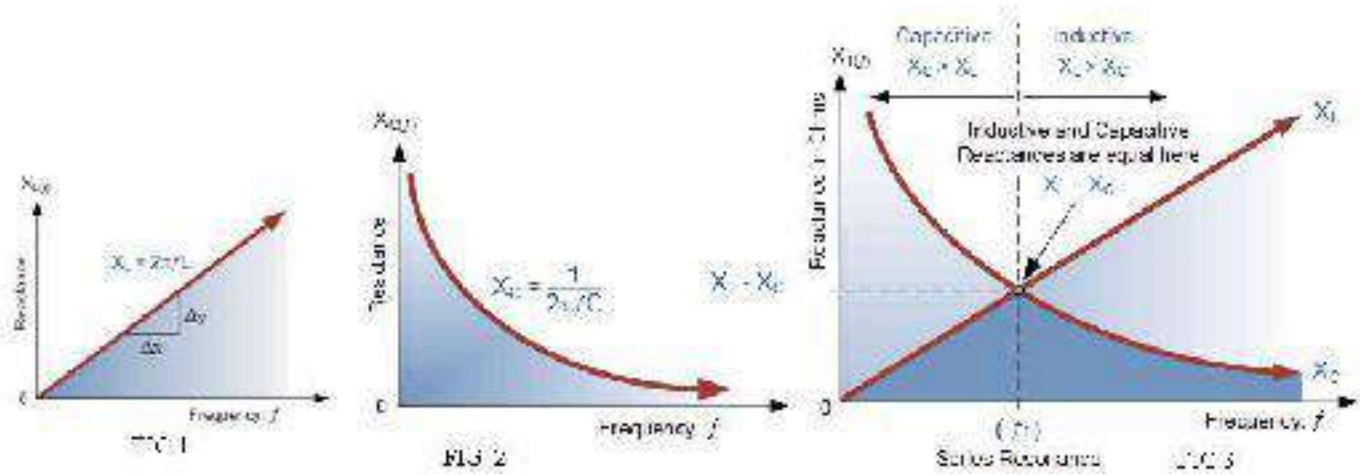
Capacitive reactance : $X_C = 1 / (2\pi fC) = 1/ \omega C$

When $X_L > X_C$, the circuit is inductive, Total circuit reactance $X_T = X_L - X_C$

When $X_L < X_C$, the circuit is capacitive, Total circuit reactance $X_T = X_C - X_L$

Total circuit impedance, $Z = (R^2 + (X_T)^2)^{1/2}$

As the frequency approaches zero or DC, the inductors reactance would decrease to zero, causing the opposite effect acting like a short circuit. This means that inductive reactance is "**Proportional**" to frequency and is small at low frequencies and high at higher frequencies and this demonstrated in the curve of FIG 1:



The graph of inductive reactance against frequency is a straight line linear curve. The inductive reactance value of an inductor increases linearly as the frequency across it increases. Therefore, inductive reactance is positive and is directly proportional to frequency ($X_L \propto f$). The same is also true for the capacitive reactance formula above but in reverse. If either the **Frequency** or the **Capacitance** is increased the overall capacitive reactance would decrease. As the frequency approaches infinity the capacitors reactance would reduce to zero causing the circuit element to act like a perfect conductor of 0Ω's. But as the frequency approaches zero or DC level, the capacitors reactance would rapidly increase up to infinity causing it to act like a very large resistance acting like an open circuit condition. This means then that capacitive reactance is "**Inversely proportional**" to frequency for any given value of capacitance and this shown in FIG 2. The graph of capacitive reactance against frequency is a hyperbolic curve. The Reactance value of a capacitor has a very high value at low frequencies but quickly decreases as the frequency across it increases. Therefore, capacitive reactance is negative and is inversely proportional to frequency ($X_C \propto f^{-1}$)

<u>Resonant frequency:</u>

Electrical resonance occurs in an AC circuit when the two reactances which are opposite and equal cancel each other out as $X_L = X_C$ and the point on the graph at which this happens is where the two reactance curves cross each other. In both series and parallel resonant circuits, the resonant frequency, f_r point can be calculated as follows.

$$X_L = X_C \quad \Rightarrow \quad 2\pi f L = \frac{1}{2\pi f C}$$

$$f^2 = \frac{1}{2\pi L \times 2\pi C} = \frac{1}{4\pi^2 LC}$$

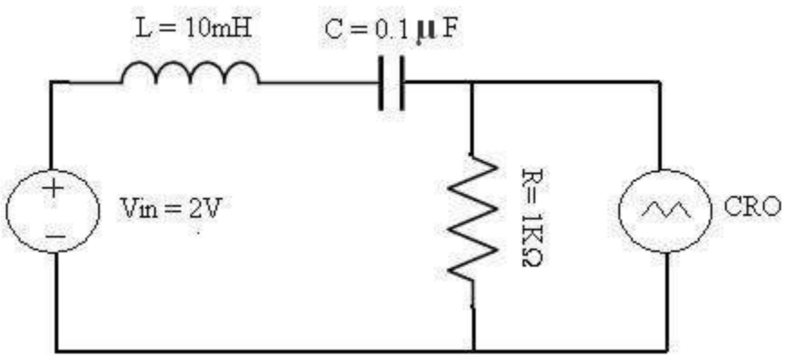
$$f = \sqrt{\frac{1}{4\pi^2 LC}}$$

$$\therefore \; f_r = \frac{1}{2\pi\sqrt{LC}} \text{ (Hz)} \quad \text{or} \quad \omega_r = \frac{1}{\sqrt{LC}} \text{ (rads)}$$

Bandwidth of a Resonance Circuit

The frequency response of the circuits current magnitude above, relates to the "sharpness" of the resonance in a series resonance circuit. The sharpness of the peak is measured quantitatively and is called the **Qualityfactor, Q** of the circuit. The quality factor relates the maximum or peak energy stored in the circuit (thereactance) to the energy dissipated (the resistance) during each cycle of oscillation meaning that it is a ratio of resonant frequency to bandwidth and the higher the circuit Q, the smaller the bandwidth, $Q = f_r / BW$. As the bandwidth is taken between the two -3dB points, the **selectivity** of the circuit is a measure of its ability to reject any frequencies either side of these points. A more selective circuit will have a narrower bandwidth whereas a less selective circuit will have a wider bandwidth. The selectivity of a series resonance circuit can be controlled by adjusting the value of the resistance only, keeping all the other components the same, since $Q = (X_L \text{ or } X_C)/R$.

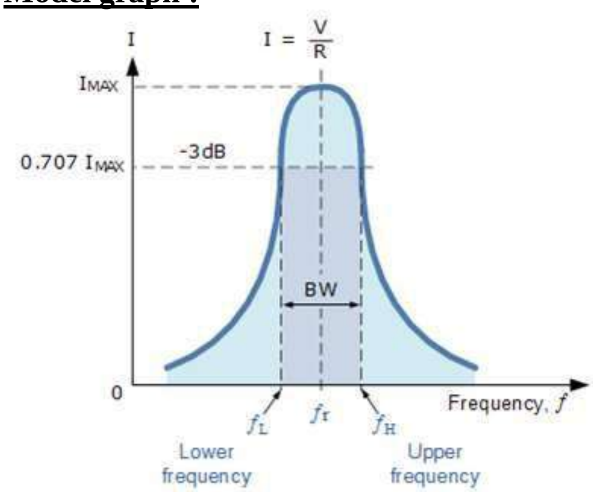
Circuit diagram:

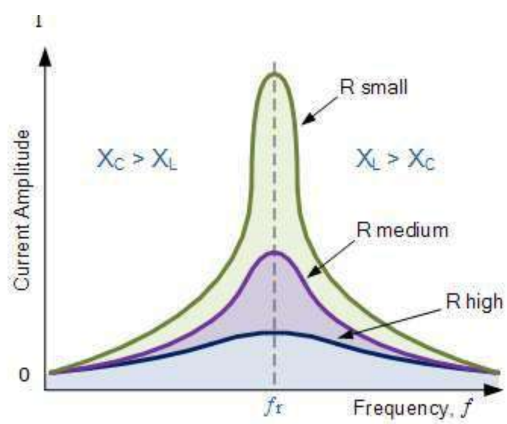


Procedure :

1. The connections are made as per the circuit diagram
2. Set the input voltage of function generator to 2V
3. Vary the frequency in audio range (20Hz to 30 KKz) and note down the amplitude of the output voltage on the CRO screen and tabulate in the tabular column.

Model graph :





Formulae used:

Inductive reactance : $X_L = 2\pi fL = \omega L$

Capacitive reactance : $X_C = 1 / (2\pi fC) = 1/ \omega C$

When $X_L > X_C$, the circuit is inductive, Total circuit reactance $X_T = X_L - X_C$
When $X_L < X_C$, the circuit is capacitive, Total circuit reactance $X_T = X_C - X_L$
Total circuit impedance, $Z = (R^2 + (X_T)^2)^{1/2}$

The relationship between resonance, bandwidth, selectivity and quality factor for a series resonance circuit being defined as:

1) Resonant Frequency, (f_r)

$$X_L = X_C \quad \Rightarrow \quad 2\pi f L = \frac{1}{2\pi f C}$$

$$f^2 = \frac{1}{2\pi L \times 2\pi C} = \frac{1}{4\pi^2 LC}$$

$$f = \sqrt{\frac{1}{4\pi^2 LC}}$$

$$\therefore \; f_r = \frac{1}{2\pi\sqrt{LC}} \,(\text{Hz}) \quad \text{or} \quad \omega_r = \frac{1}{\sqrt{LC}} \,(\text{rads})$$

2) Current, (I)

$$\text{at } \omega_r \quad Z_T = \text{min}, \quad I_S = \text{max}$$

$$I_{max} = \frac{V_{max}}{Z} = \frac{V_{max}}{\sqrt{R^2 + (X_L - X_C)^2}} = \frac{V_{max}}{\sqrt{R^2 + \left(\omega_r L - \frac{1}{\omega_r C}\right)^2}}$$

3) Quality Factor, (Q)
For a series resonant circuit,

$$Q = \frac{\omega_r L}{R} = \frac{X_L}{R} = \frac{1}{\omega_r CR} = \frac{X_C}{R} = \frac{1}{R}\sqrt{\frac{L}{C}}$$

For a parallel resonant circuit,

$$Q = \frac{R}{X_L} = \frac{R}{2\pi f L}$$

4) Bandwidth, (BW)

$$BW = \frac{f_r}{Q},\quad f_H - f_L,\quad \frac{R}{L}\,(\text{rads})\ \text{or}\ \frac{R}{2\pi L}\,(\text{Hz})$$

5) Lower cut-off frequency, (f_L)

$$f_L = f_r - \frac{1}{2}BW$$

6) Upper cut-off frequency, (f_H)

$$f_H = f_r + \frac{1}{2}BW$$

Tabular column:

S.No	Frequency (Hz)	Vo

Result:

Thus the frequency response of RLC circuits are obtained.

Experiment No: Date:

6. Experimental determination of power in three phase circuits by two-watt meter method.

Aim:

To conduct a suitable experiment on a 3Φ phase load connected in star or delta to measure 3Φ power and power factor using 2 wattmeter method

Objectives:

1. To study the working of wattmeter
2. To accurately measure 3Φ power
3. To accurately measure power factor
4. To study the concept of star connected load and delta connected load

Apparatus required:

S.No	Name of the apparatus	Range	Type	Quantity
1	Ammeter	(0-10)A	MI	-
2	Voltmeter	(0-600)V	MI	-
3	3Φ Autotransformer	(0-440)V, 50Hz	CLOSED	-
4	Wattmeter	600V, 10A,UPF	MI	-
5	3Φ load	1KW	Resistive	-
6	Connecting wires	-	-	Required

Formula used:

1. Total power, $P=W_1+W_2$
2. $\Phi = \tan^{-1} \sqrt{3}\,(W_1 - W_2)/(W_1 + W_2)$
3. Power factor = cos Φ

Theory:

In a three phase, star or delta three wire system, under balanced or unbalanced conditions, with any power factor, the two-wattmeter method is a practical and commonly used method of measuring total three phase power. In this method, the three meter potential coil terminals at 0 is kept joined, but is removed from the neutral of the system, the readings of all wattmeters will be unchanged, because the wattmeter potential coils themselves form a balanced Y-connected circuit and so the voltage across every potential coil remains unchanged. This method of measurement is called the "floating neutral" method and is accurate on a three-phase three-wire or four-wire system regardless of power factor or load unbalance.

In this method we have two types of connections
(a)Star connection of loads
(b)Delta connection of loads.

For star connected load clearly the reading of one wattmeter is product phase current(I_1) and voltage difference(V_2-V_3). Similarly the reading of another wattmeter is the product of phase current(I_2) and thevoltagedifference (V_2-V_3). Thus the total power of the circuit is the sum of the readingsof both the wattmeters. Mathematically we can write,

$$P = P_1+P_2 = I_1(V_1+V_2)+I_2(V_2-V_3)$$

2-Wattmeter is preferable compared to 3-Wattmeter method as ultimately the power calculated in both the methods are similar. Blondel's Theorem states that you can have one less wattmeter than there are conductors supplying a balanced or unbalanced load. In either case, the sum of the wattmeter readings will give you the total power of the load.

Procedure:

1. Connections are made as per the circuit diagram.
2. Supply switch is closed and readings of ammeter and wattmeter are noted. If one of the wattmeter reads negative, then its potential coils.
3. The above procedure is repeated for different values of inductive coil. Care should be taken that the current should not exceed 10A during the experiment.

Tabulation:

S.No	Load current I (Amps)	Wattmeter reading W1	Wattmeter reading W2	Total Power W1+W2	PF(cosΦ)

Calculation:

Result:

Thus the power is measured and the power factor is calculated using 2-wattmeter method.

Experiment No: Date:

7. CALIBRATION OF SINGLE PHASE ENERGY METER

Aim:

To calibrate the given single phase energy meter at unity and other power factors.

Objective:

1. To study the working of energy meter
2. To accurately calibrate the meter at unity and other power factor.
3. To study the % of error for the given energy meter.

Apparatus required:

S.No	Name of the apparatus	Range	Type	Quantity
1	Energy meter			1
2	Ammeter	(0-10)A	MI	1
3	Voltmeter	(0-300)V	MI	1
4	Autotransformer	(0-270)V, 50Hz		1
5	Wattmeter	300V, 10A, UPF		1
6	Single phase load	1KW	Resistive	1
7	Wires			Required

Theory:

The total power consumed by a load during an interval of time is ENERGY.

$E = P \times T$

Where E=Energy, P= Power, T=Time

If the voltages & currents not constant & have n-values over the time t, then

$$E = \sum_{i=1}^{n} Pi.ti$$

$$= \sum_{i=1}^{n} Vi.Ii.ti$$

It can also be expressed as continuous integral of Power i.e.,

$$E = \int_0^t P dt$$

$$= \int_0^t V.I.dt$$

The unit of energy is Watt second or joule. But its commercial unit is Kilowatt-hours or KWh which is defined as the energy consumed by a load of 1000 watts over a period of one hour.

Energy Meter

Induction type energy meters are most commonly form of an A. c. KWh meter used to measure the energy consumed in any a.c. circuit in a prescribed period when supply voltage and frequency are constant, in day today life & in industrial installation. Energy meter is an integrating instrument which measure the total quantity of electrical energy supplied to the circuit in a given period. These meters measure electrical energy in Kilowatt hours.

PRINCIPLE: The Basic principle of induction type energy meter is electromagnetic induction. When analternating current flows through two suitably located coils (Current coil & Potential Coil) produces rotating magnetic field which is cut by the metallic disc Suspended near to the coils, thus, an e.m.f. is induced in the thin Aluminum disc which circulates eddy currents in it. By the interaction of Rotating magnetic field & eddy currents, torque is developed & causes the disc to rotate. This is the same principle which is applied in the single-phase induction motors.

Construction: An Indction type single phase energy meter, has following main parts of the operatingmechanism:

1. Driving System
2. Moving System
3. Braking System
4. Registering System

DRIVING SYSTEM develops torque to rotate the moving system. It consists of two electromagnets one isformed by current coil & other one is by voltage coil or pressure coil.

MOVING SYSTEM essentially consists of an aluminum mounted on the spindle which is supported byPivot-jewel Bearing system. Since there is not control spring, the disc makes continuous revolution under the action the deflecting torque.

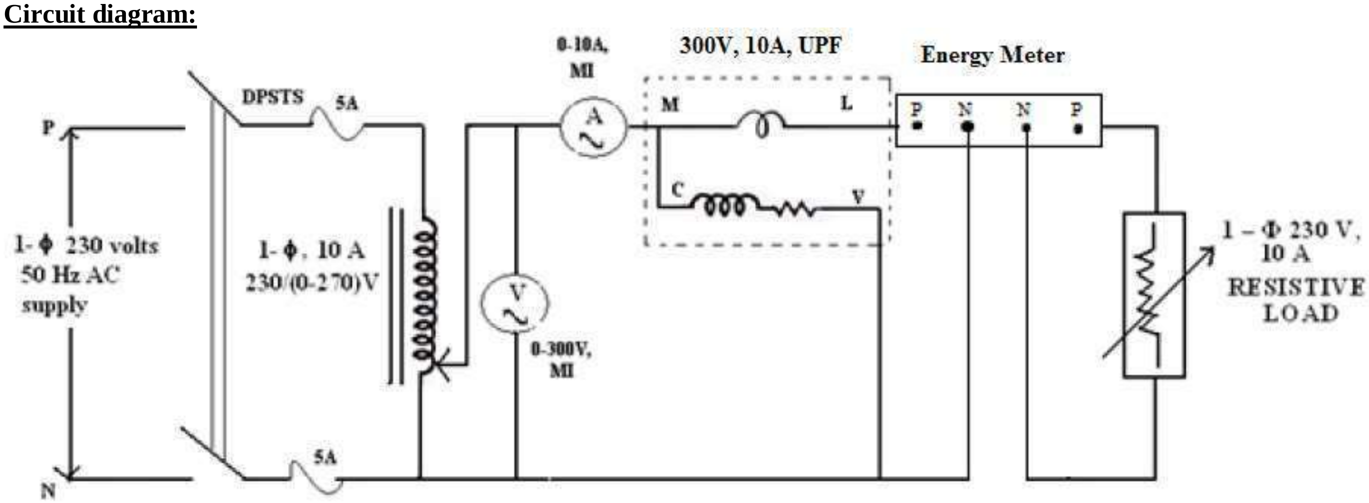
BRAKING SYSTEM consists of a permanent magnet of C shaped covering a part of rotating disc toprovide braking torque. By changing the position of breaking magnet, the Flux linkage with the disc can be changed, this torque is opposite to driving torque.

REGISTERING SYSTEM keeps the record of energy consumed by load through worm wheel or pinion gear mounted with spindle of moving disc.

WORKING

When the energy meter is connected in the circuit, the current coil carries the load current and the pressure coil carries the current proportional to the supply voltage. The magnetic field produced by the SERIES magnet (series coil) is in phase with the line current & the magnetic field produced by the shunt magnet (pressure coil) is in quadrature with the applied voltage (since the coil is highly inductive). Thus, a phase difference exists between the fluxes produced by the two coils. This sets up a rotating field which interacts with the disc and produces a driving torque and, thus, disc starts rotating. The number of revolutions made by the disc depends upon the energy passing through the meter. The spindle is geared to the recording mechanism so that electrical energy consumed in the circuit is directly registered in KWh. The speed of the disc is adjusted by adjusting the position of the breaking magnet. For example, if the energy meter registers less energy than the energy actually consumed in the circuit, then the speed of disc has to be increased which is obtained by shifting the magnet nearer to the centre of the Disc and vice-versa.

Circuit diagram:



Formulae used:

At constant angular speed the power $VICos\varphi$ is proportional to the angular speed in r.p.s. Let K be the meter constant of energy meter, which is the number of revolution per KWh energy consumption. When connected to measure energy, if disc makes R number of revolution in t seconds. Then the reading of energy meter is:

$Et = R/K$

Let KW= Power in Kilowatt from wattmeter reading.

R= No. of revolution made by disc in 't' Sec.

$$K = \frac{revolution}{KWh}$$

Energy recorded by meter under test $(Et) = \frac{R}{K} KWh$

Let the wattmeter reading be Kw watts of energy calculated from the wattmeter & stop watch is given by Energy consumed by wattmeter (Es) = $\frac{Kw \times t}{3600}$

Percentage Error = $\left[\frac{Et - Es}{Es}\right] \times 100$

Tabulation:

S.No	Load current I (Amps)	Wattmeter reading (Wa)	Indicated reading (Wi)	Time taken (sec)	% error

Result:

Thus the single phase energy meter is calibrated.

Experiment No: Date:

8. DETERMINATION OF TWO PORT NETWORK PARAMETERS

Aim:

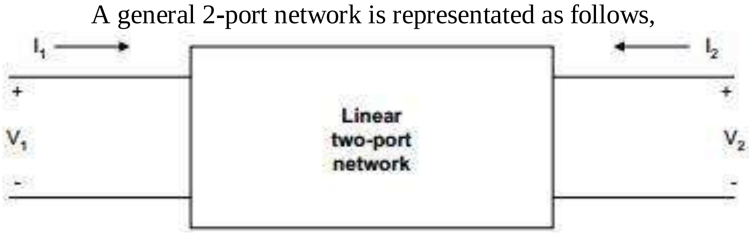
To obtain experimentally Z parameters and Y parameters of a given twoport network.

Apparatus required:

S.No	Name of the equipment	Range	Type	Quantity
1	Regulated Power Supply (RPS)	0-30V	Dual	1
2	Ammeter	0-30mA	MC	2
3	Voltmeter	0-30V	MC	2
4	Resistor	1KΩ, 220Ω	-	1,2
5	Bread board	-	-	1
6	Connecting wires	-	-	Required

Theory:

A general 2-port network is representated as follows,



I_1 and V_1 are input current and voltage, respectively. Also, I_2 and V_2 are output current and voltage, respectively. It is assumed that the linear two-port circuit contains no independent sources of energy and that the circuit is initially at rest (no stored energy). Furthermore, any controlled sources within the linear two-port circuit cannot depend on variables that are outside the circuit.

A two port network can be described by z-parameters as,

$$V_1 = z_{11} I_1 + z_{12} I_2$$

$$V_2 = z_{21} I_1 + z_{22} I_2$$

In matrix form,

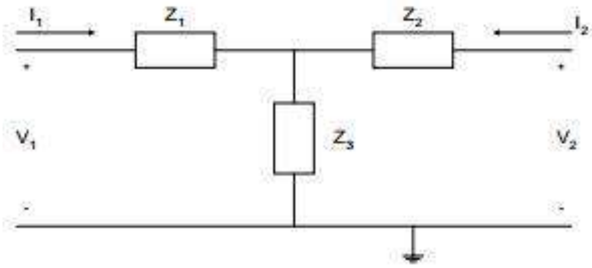
$$\begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix}$$

The z-parameter can be found as follows

$$z_{11} = \frac{V_1}{I_1}\bigg|_{I_2=0} \quad z_{12} = \frac{V_1}{I_2}\bigg|_{I_1=0} \quad z_{21} = \frac{V_2}{I_1}\bigg|_{I_2=0} \quad z_{22} = \frac{V_2}{I_2}\bigg|_{I_1=0}$$

The z-parameters are also called open-circuit impedance parameters since they are obtained as a ratio of voltage and current and the parameters are obtained by open-circuiting port 2 ($I_2 = 0$) or port1 ($I_1 = 0$). The following example shows a technique for finding the z-parameters of a simple circuit.

For the T-network, the Z parameters can be determined as follows,



Using KVL,

$$V_1 = Z_1 I_1 + Z_3(I_1 + I_2) = (Z_1 + Z_3)I_1 + Z_3 I_2$$

$$V_2 = Z_2 I_2 + Z_3(I_1 + I_2) = (Z_3)I_1 + (Z_2 + Z_3)I_2$$

thus

$$\begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} Z_1 + Z_3 & Z_3 \\ Z_3 & Z_2 + Z_3 \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix}$$

The z-parameters are,

$$[Z] = \begin{bmatrix} Z_1 + Z_3 & Z_3 \\ Z_3 & Z_2 + Z_3 \end{bmatrix}$$

Y-parameters:

A two-port network can also be represented using y-parameters. The describing equations are,

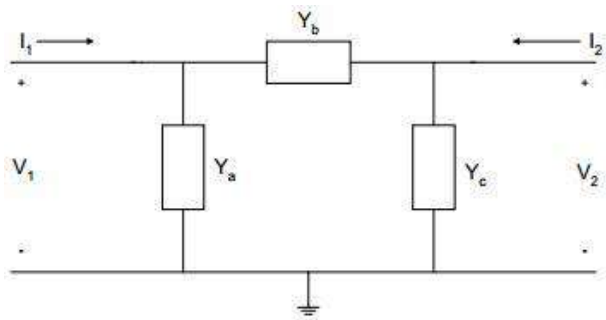
$$I_1 = y_{11}V_1 + y_{12}V_2$$

$$I_2 = y_{21}V_1 + y_{22}V_2$$

where V_1 and V_2 are independent variables
and I_1 and I_2 are dependent variables

The y-parameters can be found as follows:

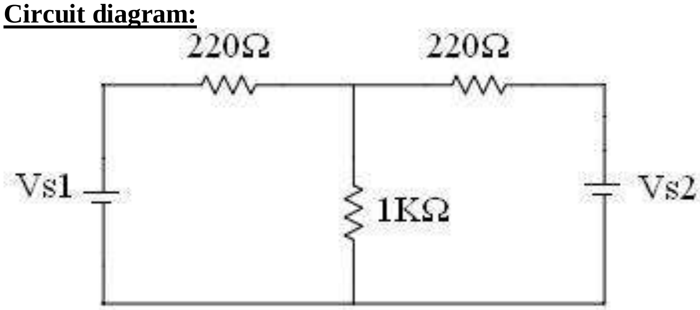
$$y_{11} = \frac{I_1}{V_1}\bigg|_{V_2=0} \quad y_{12} = \frac{I_1}{V_2}\bigg|_{V_1=0} \quad y_{21} = \frac{I_2}{V_1}\bigg|_{V_2=0} \quad y_{22} = \frac{I_2}{V_2}\bigg|_{V_1=0}$$



Using KCL we get the Y-parameters as follows,

$$[Y] = \begin{bmatrix} Y_a + Y_b & -Y_b \\ -Y_b & Y_b + Y_c \end{bmatrix}$$

Circuit diagram:



To calculate Z_{11} and Z_{21}, take Vs1 = 5V and place a voltmeter (0-30V)MC in Vs2
To calculate Z_{12} and Z_{22}, take Vs2= 5V and place a voltmeter (0-30V)MC in Vs1

Procedure:

1. Connections are made as per the circuit diagram.
2. In order to calculate Z_{11}, the output terminals are kept open via a voltmeter. Supply is given to input port. Note the readings of ammeter as I_1 and voltmeter as V_1. Similarly the values of other z-parameters and y-parameters can be obtained.
3. Tabulate the calculated values of z-parameters and y-parameters.

Table:

	Z-Parameters				Y-Parameters			
	Z_{11}	Z_{12}	Z_{21}	Z_{22}	Y_{11}	Y_{12}	Y_{21}	Y_{22}
Theoritical								
Practical								

Result:

Thus the two port network parameters were obtained.

Experiment No: Dat e:

9. Design and Simulation of series resonance circuit.

Aim:

To study the frequency response of series resonance circuit.

Apparatus required:

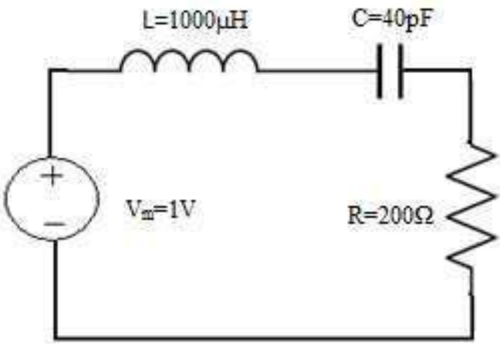
1. Personal computer
2. PSPICE/MATLAB Software

Theory:

The series RLC circuit can be designed and simulated using PSPICE. PSpice is a SPICE analogcircuitand digital logicsimulationprogram forMicrosoft Windows. The name is an acronym for PersonalSPICE - SPICE itself being an acronym for Simulation Program with Integrated Circuit Emphasis. PSpice was initially developed by MicroSim and is used in electronic design automation. The company was bought by OrCAD, which was subsequently purchased by Cadence Design Systems. During its development, PSpice has evolved into an analog mixed signal simulator. The software, now developed towards more complex industry requirements, is integrated in the complete systems design flow in OrCAD and Cadence Allegro. It includes features such as analysis of a circuit with automatic optimization, encryption, a model editor, support for parameterized models, auto-convergence and checkpoint restart, several internal solvers, a magnetic part editor, and support for Tabrizi core model for non-linear cores. This lab will explore some of the following aspects of the series RLC circuit using PSPICE

- Input impedance
- Current
- Voltage

Circuit diagram:



Procedure:

1. Build the schematic shown in the circuit diagram V_m is an AC voltage source (V_{AC}) from the source library. It needs to be set for 1 volt. L1 is an ideal inductor from the Analog Library. Set for 1000µH. R is an ideal resistor from the Analog Library. Set value to {R_x}.

2. Next add part named "Parameters". Then double click on part to enter edit mode. Click on new column, name = Rx, value = 200Ω. Then click on column, select display and click on name and value. C1 is an ideal capacitor from the Analog library. Change the value to 40pF.
3. 1. Do analysis setup

   a. At top of screen click on Pspice

   b. Click on New Simulations Profile

   c. Type name of profile that you wish.

   d. Under Analysis tab, select AC sweep from the Analysis type pull down menu.

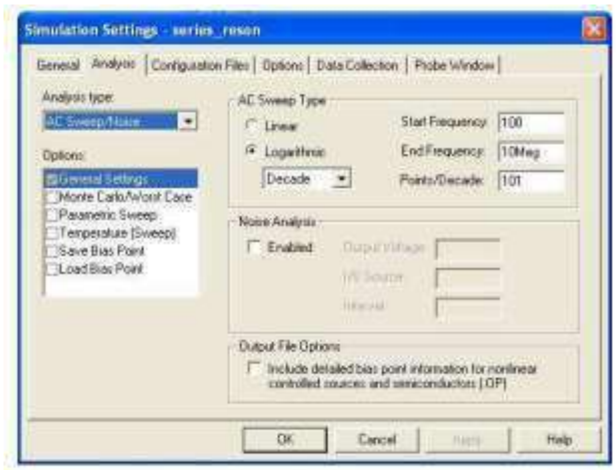
   e. Under AC Sweep Type

   Select Logarithmic and Decade as shown in figure.

   i. Start freq = 100

   ii. End freq = 10Meg

   iii. Points/Decade = 101

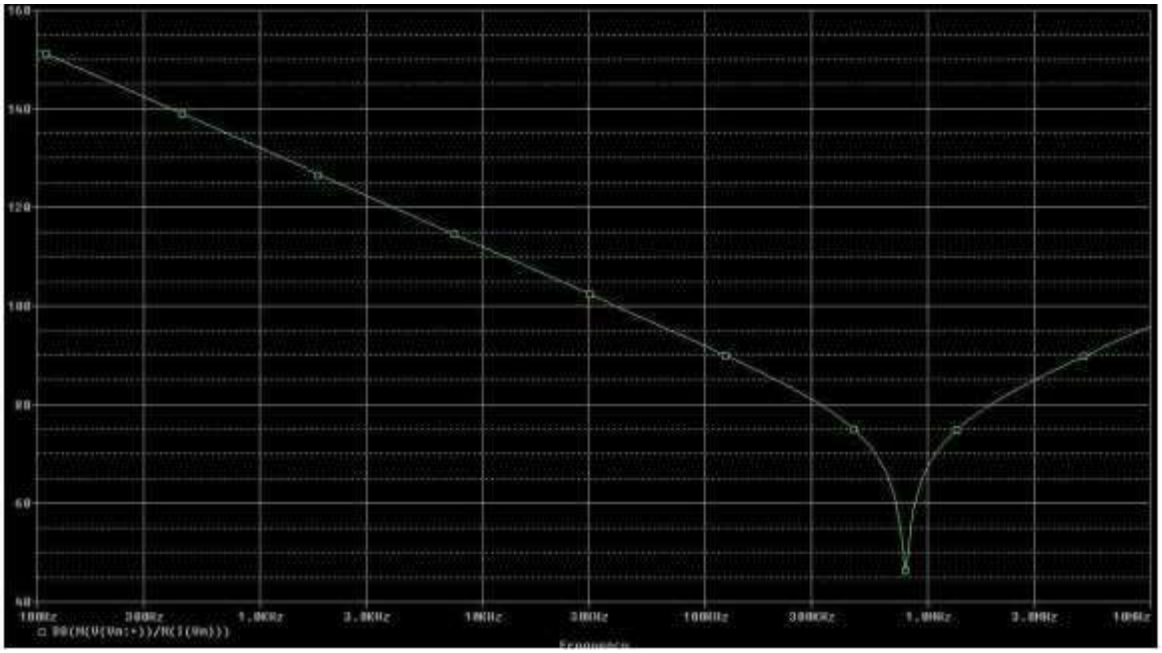


   f. Then click the run Pspice button. (Looks like a play button)

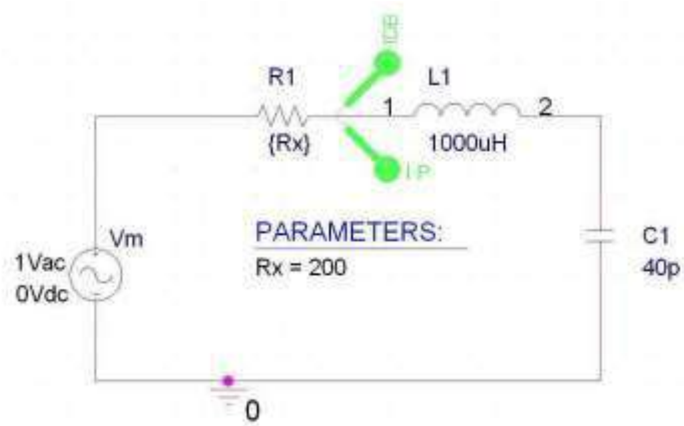
   g. After running, look at schematic file and click on trace, add trace.

   h. Next Select Db() on left, select M() on left, select V(Vm:+), then divide by M(I(Vm)).

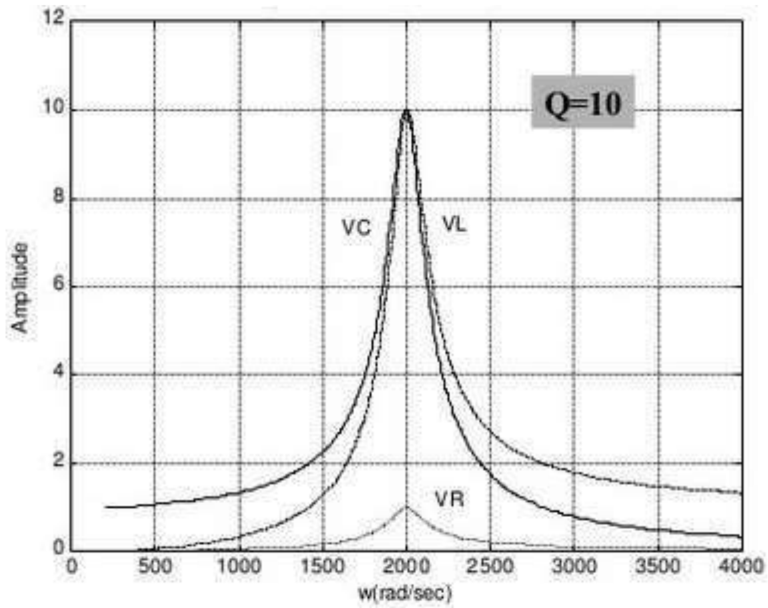
   The figure below is the result of input impedance of series RLC tank circuit,



Next, we want to measure the total inductor current of RLC series resonance circuit. Use the same circuit as above, and from the Pspice button, Markers, Advanced, select “db magnitude of current marker” and “Phase of Current marker”, and place in series next to L1.



The simulated result of voltage and current across resistive load in a series RLC circuit will be dumpbell shaped as shown in figure below,



Result:
Thus the series RLC circuit was designed and simulated using PSPICE.

Experiment No: Date:

10. Design and Simulation of parallel resonance circuit.

Aim:

To study the frequency response of parallel resonance circuit.

Apparatus required:

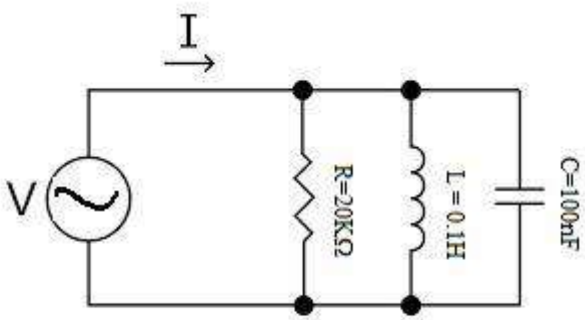
3. Personal computer
4. PSPICE/MATLAB Software

Theory:

The series RLC circuit can be designed and simulated using PSPICE. PSpice is a SPICE analogcircuitand digital logicsimulationprogram forMicrosoft Windows. The name is an acronym for PersonalSPICE - SPICE itself being an acronym for Simulation Program with Integrated Circuit Emphasis. PSpice was initially developed by MicroSim and is used in electronic design automation. The company was bought by OrCAD, which was subsequently purchased by Cadence Design Systems. During its development, PSpice has evolved into an analog mixed signal simulator. The software, now developed towards more complex industry requirements, is integrated in the complete systems design flow in OrCAD and Cadence Allegro. It includes features such as analysis of a circuit with automatic optimization, encryption, a model editor, support for parameterized models, auto-convergence and checkpoint restart, several internal solvers, a magnetic part editor, and support for Tabrizi core model for non-linear cores. This lab will explore some of the following aspects of the series RLC circuit using PSPICE

- Input impedance
- Current
- Voltage

Circuit diagram:

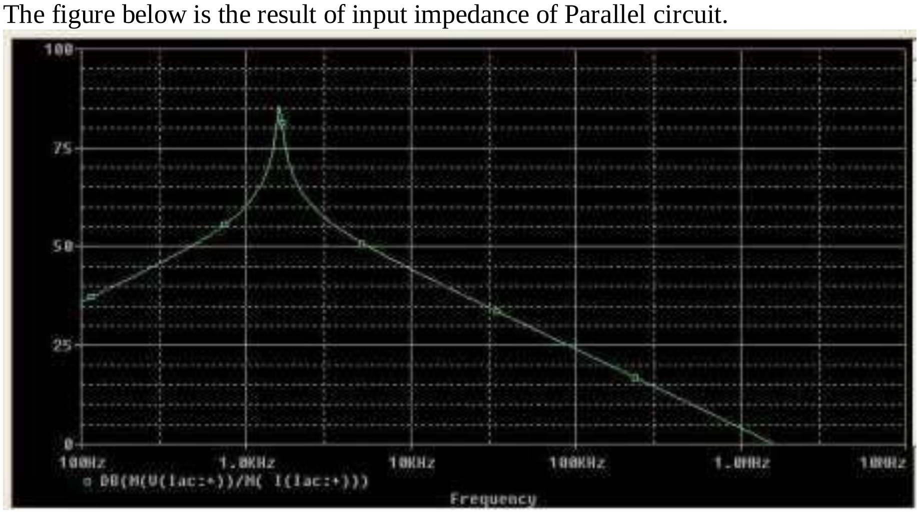


Procedure:

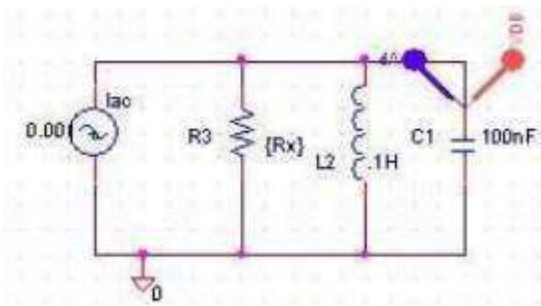
1. Draw the circuit given above
2. Apply the I_{AC}, because we want to plot the frequency response
3. Set AC_{MAG} =0.001 in I_{AC}
4. Do analysis setup

a. On the ORCAD Capture CIS文
menu select new simulation profile
b. Choose AC Sweep/Noise in the Analysis type menu
c. Set the Start Frequency at 100, the End Frequency at 10Meg and the Points/Decade at 101
d. Make sure Logarithmic is selected and set to Decade
e. Click OK

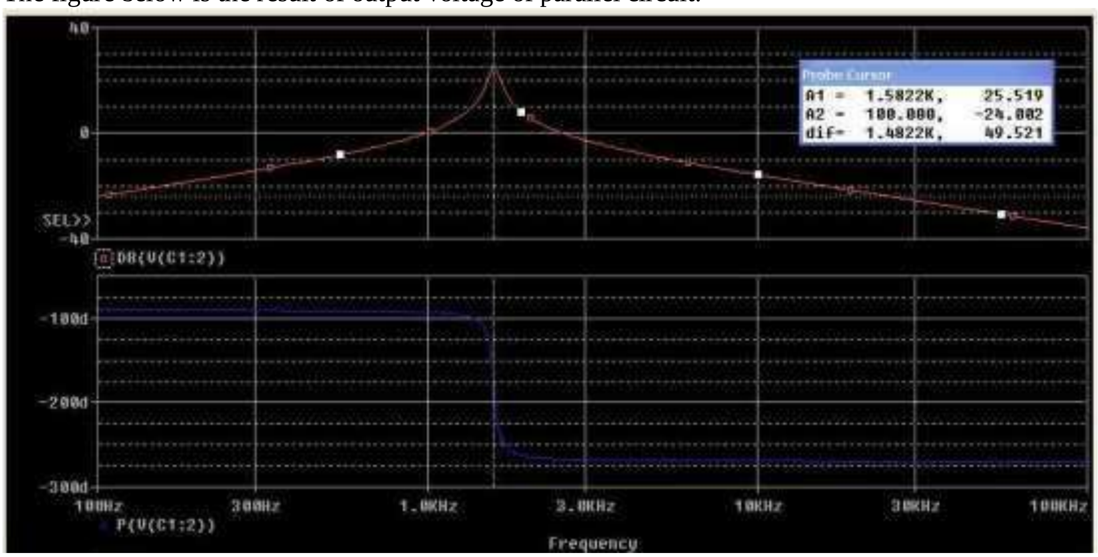
The figure below is the result of input impedance of Parallel circuit.



Next, we want to run the simulation of the output voltage of the parallel circuit. Use the same circuit as above and place the "db magnitude of voltage marker" and the "phase of voltage marker" in series next to output capacitor.
(Note: Markers are in the PSpice menu)



The figure below is the result of output voltage of parallel circuit.



Similarly the values of current can be identified using current marker.

Result:
Thus the parallel RLC circuit was designed and the voltage & current was simulated.

Experiment No: Date:

11. Simulation of low pass and high pass passive filters.

Aim:

To simulate low pass and high pass filters

Apparatus required:

1. Personal computer
2. PSPICE/MATLAB Software

Theory:

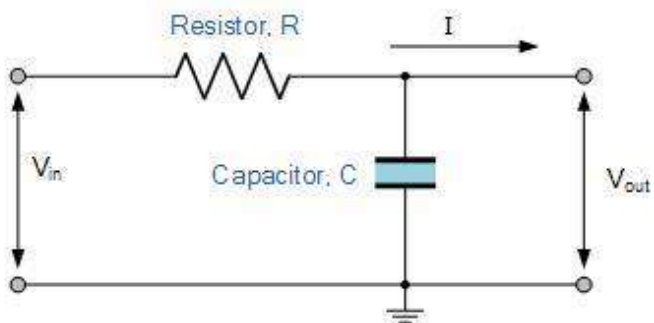
A low-pass filter is a filter that passes signals with a frequency lower than a certain cutoff frequency and attenuates signals with frequencies higher than the cutoff frequency. The amount of attenuation for each frequency depends on the filter design. The filter is sometimes called a high-cut filter, or treble cut filter in audio applications. A low-pass filter is the opposite of a high-pass filter. A band-pass filter is a combination of a low-pass and a high-pass filter.

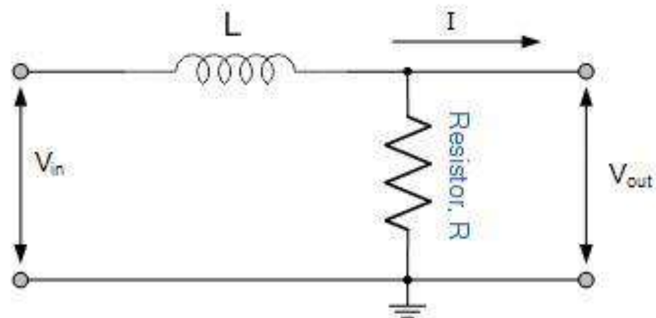
Low-pass filters exist in many different forms, including electronic circuits (such as a hiss filter used in audio), anti-aliasing filters for conditioning signals prior to analog-to-digital conversion, digital filters for smoothing sets of data, acoustic barriers, blurring of images, and so on. The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

A high-pass filter is an electronic filter that passes signals with a frequency higher than a certain cutoff frequency and attenuates signals with frequencies lower than the cutoff frequency. The amount of attenuation for each frequency depends on the filter design. A high-pass filter is usually modeled as a linear time-invariant system. It is sometimes called a low-cut filter or bass-cut filter. High-pass filters have many uses, such as blocking DC from circuitry sensitive to non-zero average voltages or radio frequency devices. They can also be used in conjunction with a low-pass filter to produce a bandpass filter.

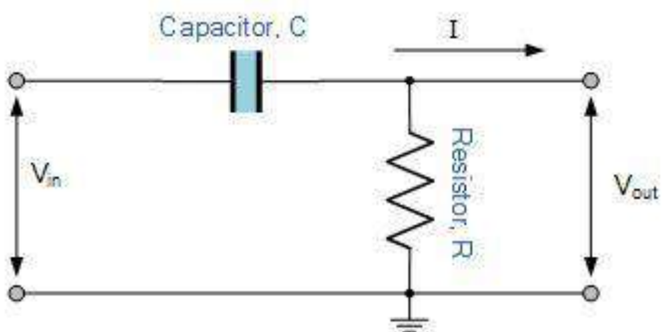
<u>Circuit diagram:</u>

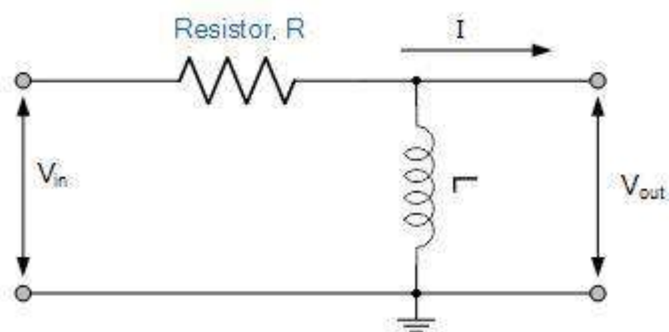
<u>Low Pass Filter:</u>



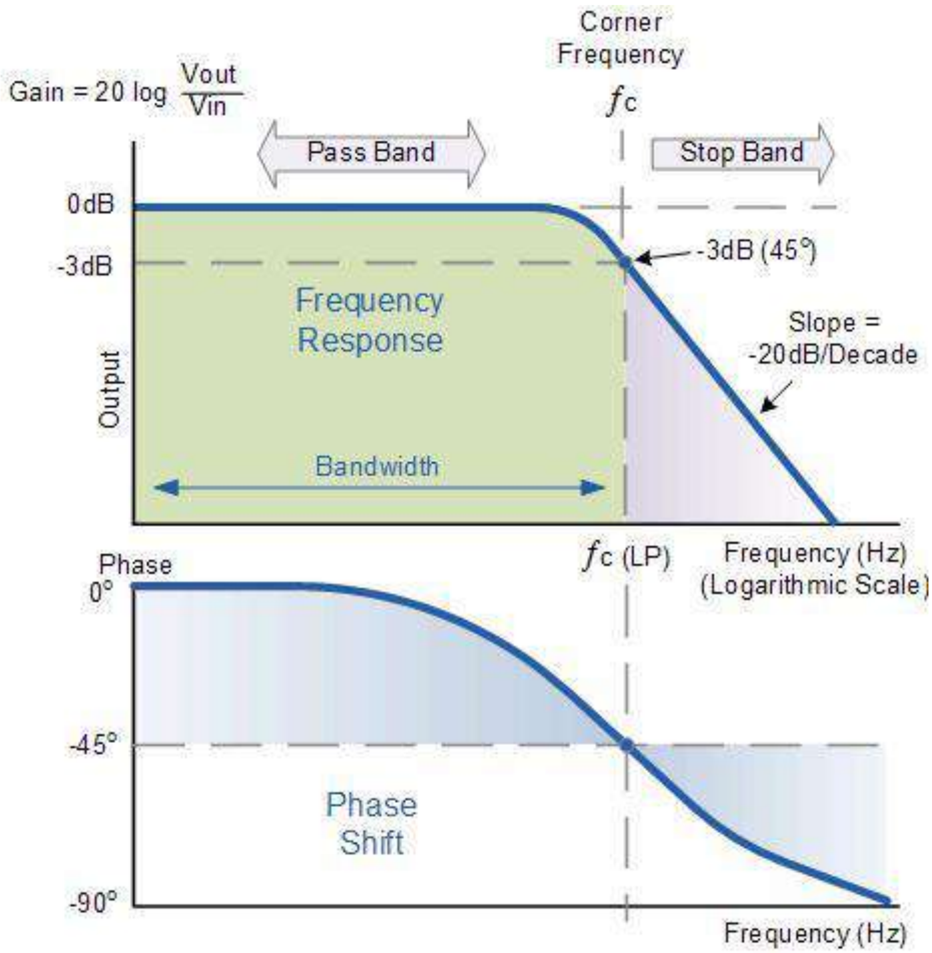


<u>High Pass Filter:</u>

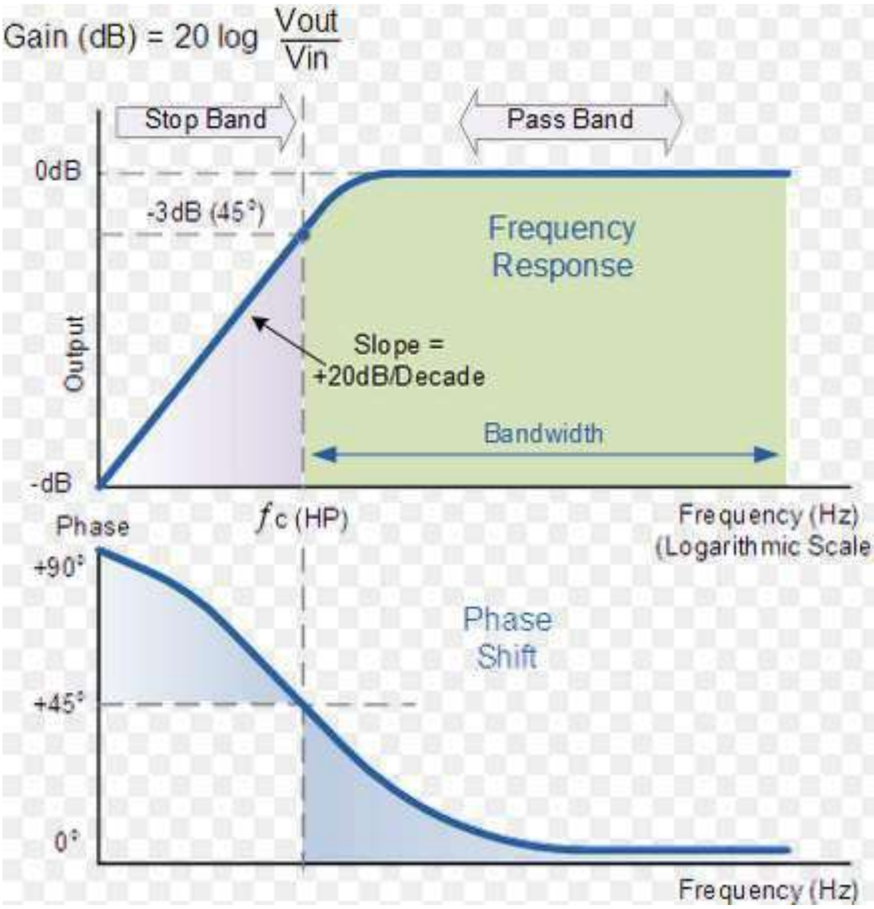




<u>Model graph:</u>
<u>Low Pass Filter:</u>



High Pass Filter



Formulae :

$V_{out} = V_{in}\ X_C/(R+X_C)$ OR $V_{out} = V_{in}\ X_L/(R+X_L)$

$X_C = 1/(2\pi fC)$

$X_L = 2\pi fL$

Cut-off frequency $f_C = 1/(2\pi RC)$

Procedure:

1. Simulate the circuit using PSPICE.
2. Summarize the results.
3. Attach the report of the results.

Result:

Thus the low pass and high pass filter of the frequency response curve was measured,

Low Pass Filter Cut-off frequency fc =

High Pass Filter Cut-off frequency fc =

Experiment No: Dat e:

12. Simulation of three phases balanced and unbalanced star, delta networks circuits

Aim:

To simulate low pass and high pass filters using PSPICE.

Apparatus required:

1. Personal computer
2. PSPICE/MATLAB Software

Theory:

There are two types of systems available in electric circuit, one is single phase circuits and the other is three phase circuits. In single phase circuit, there will be only one phase, i.e the current will flow through only one wire and there will be one return path called neutral line to complete the circuit. So in single phase minimum amount of power can be transported. Here the generating station and load station will also be single phase. This is an old system using from previous time.

In 1882, new invention has been done on polyphase system, that more than one phase can be used for generating, transmitting and for load system. Three phase circuit is the polyphase system where three phases are send together from the generator to the load. Each phase are having a phase difference of 120°, i.e 120° angle electrically. So from the total of 360°, three phases are equally divided into 120° each. The power in three phase system is continuous as all the three phases are involved in generating the total power.

The three phases can be used as single phase each. So if the load is single phase, then one phase can be taken from the three phase circuit and the neutral can be used as ground to complete the circuit. There are various reasons for this question because there are numbers of advantages over single phase circuit. The three phase system can be used as three single phase line so it can act as three single phase system. The three phase generation and single phase generation is same in the generator except the arrangement of coil in the generator to get 120° phase difference. The conductor needed in three phase circuit is 75% that of conductor needed in single phase circuit. Also the instantaneous power in single phase system falls down to zero as in single phase we can see from the sinusoidal curve but in three phase system the net power from all the phases gives a continuous power to the load.

In three phase circuit, connections can be given in two types:

1.Star connection

2.Delta connection

Star connection:

In star connection, there is four wire, three wires are phase wire and fourth is neutral which is taken from the star point. Star connection is preferred for long distance power transmission because it is having the neutral point. In this we need to come to the concept of balanced and unbalanced current in power system. When equal current will flow through all the three phases, then it is called as balanced current and the circuit is called a balanced circuit. When the current will not be equal in any of the phase, then it is unbalanced current and such a circuit is called unbalanced circuit. During balanced condition there will be no current flowing through the neutral line and hence there is no use of the neutral terminal. But when there is unbalanced current flowing in the three phase circuit, neutral is having a vital role. It will take the unbalanced current to the ground and protect the transformer. Unbalanced current affects transformer and it may also cause damage to the transformer. Under such cases, star connection is preferred for long distance transmission.

$$E_{line} = \sqrt{3}E_{phase} \text{ and } I_{line} = I_{phase}$$

Delta connection:

In delta connection, there is three wires alone and no neutral terminal is taken. Normally delta connection is preferred for short distance due to the problem of unbalanced current in the circuit.

$$E_{line} = E_{phase} \text{ and } I_{line} = \sqrt{3}I_{phase}$$

In three phase circuit, star and delta connection can be arranged in four different ways-

- Star-Star connection
- Star-Delta connection
- Delta-Star connection
- Delta-Delta connection

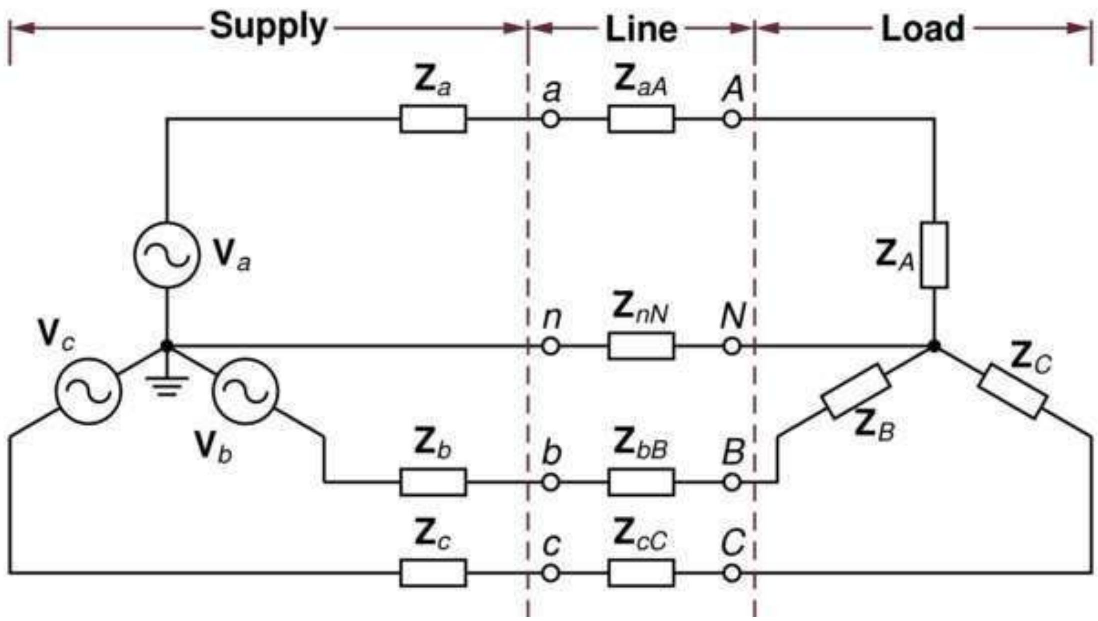
But the power is independent of the circuit arrangement of the three phase system. The net power in the circuit will be same in both star and delta connection. The power in three phase circuit can be calculated from the equation below,

$$P_{total} = 3 \times E_{phase} \times I_{phase} \times \text{power factor}$$

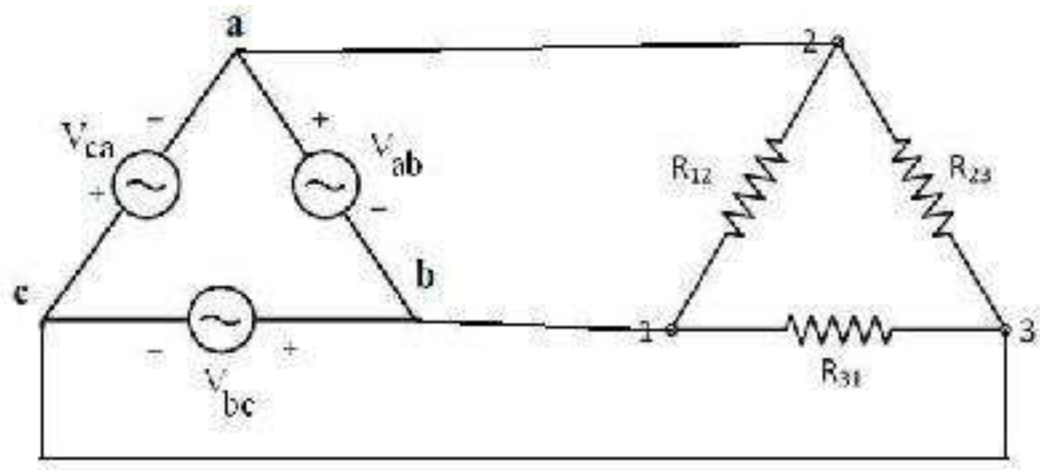
Since there is three phases, so the multiple of 3 is made in the normal power equation. Power factor is a very important factor in three phase system and some times due to certain error, it is corrected by using capacitors.

Circuit Diagram:

Star Connection:



Delta connection:



Procedure:

1. Simulate the circuit shown in circuit diagram using PSPICE.
2. Place the voltage and current markers on the locations where the output is to be taken.
3. Obtain the simulated results and attach the generated reports of voltage and current readings.

Result:

Thus the 3 phase balanced and unbalanced star and delta connected circuits were simulated and the results were obtained.

Experiment No: Date:

13. Design and simulation of band pass filter using PSPICE

Aim:

To simulate low pass and high pass filters using PSPICE.

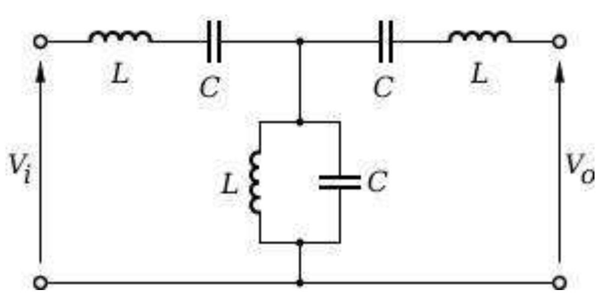
Apparatus required:

1. Personal computer
2. PSPICE/MATLAB Software

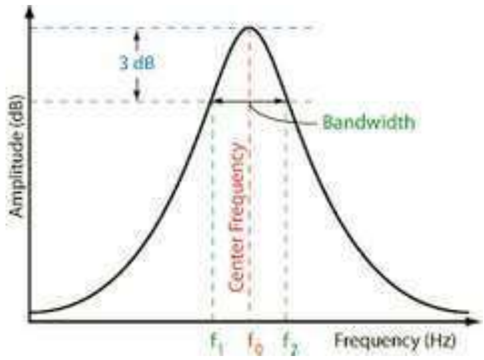
Theory:

A band-pass filter is a device that passes frequencies within a certain range and rejects (attenuates) frequencies outside that range. An ideal bandpass filter would have a completely flat passband (e.g. with no gain/attenuation throughout) and would completely attenuate at all frequencies within the passband. Additionally, the transition out of the passband would be instantaneous in frequency. In practice, no bandpass filter is ideal. The filter does not attenuate all frequencies outside the desired frequency range completely; in particular, there is a region just outside the intended passband where frequencies are attenuated, but not rejected. This is known as the filter roll-off, and it is usually expressed in dB of attenuation per octave or decade of frequency. Generally, the design of a filter seeks to make the roll-off as narrow as possible, thus allowing the filter to perform as close as possible to its intended design. Often, this is achieved at the expense of pass-band or stop-band ripple. The bandwidth of the filter is simply the difference between the upper and lower cutoff frequencies. The shape factor is the ratio of bandwidths measured using two different attenuation values to determine the cutoff frequency, e.g., a shape factor of 2:1 at 30/3 dB means the bandwidth measured between frequencies at 30 dB attenuation is twice that measured between frequencies at 3 dB attenuation. Optical band-pass filters are common in photography and theatre lighting work. These filters take the form of a transparent coloured film or sheet.

Circuit diagram:

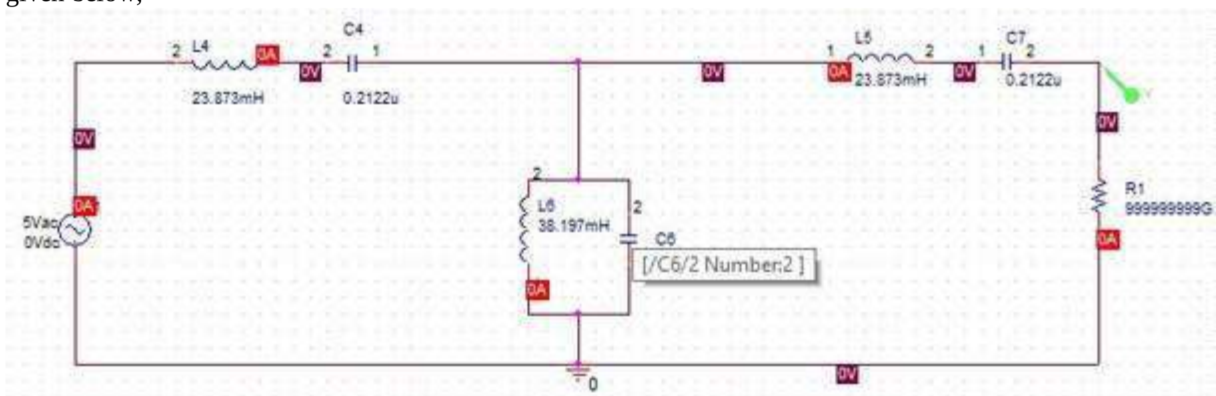


Model graph:



Procedure:

1. Simulate the circuit given below,



2. Place the voltage and current markers on the locations where the output is to be taken.
3. Obtain the simulated results and attach the generated reports of voltage and current readings.

Result :

Thus the band pass filter was designed using PSPICE and the outputs were generated.