SEMBODAI RUKMANI VARATHARAJAN ENGINEERING COLLEGE,

SEMBODAI, NAGAPATTINAM.

DEPARTMENT OF SCIENCE AND HUMANITIES

II- Semester – B.E (EEE)

EE6211 ELECTRIC CIRCUITS LAB

Prepared by,

Mr.R.Dhineshkumar M.E., AP/EEE

EE6211 ELECTRIC CIRCUITS LAB

LIST OF EXPERIMENTS

1. Experimental verification of Kirchhoff’s voltage and current laws
2. Experimental verification of network theorems (Thevenin, Norton, Superposition and maximum powertransfer Theorem).
3. Study of CRO and measurement of sinusoidal voltage, frequency and power factor.
4. Experimental 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 phases 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

INDEX

Ex. No	VERIFICATION OF KIRCHOFF'S CURRENT LAW & KIRCHOFF'S VOLTAGE LAW
Date:	

(a). VERIFICATION OF KIRCHOFF'S CURRENT LAW

AIM

To verify the kirchoff's current law for the given electrical circuit

APPARATUS REQUIRED

S.No	Name of the apparatus	Range	Type	Quantity
1	Regulated power supply	(0 - 30) V	Analog	1
2	Voltmeter	(0 - 30) V	MC	4
3	Resistor	1.2 kΩ,560Ω 3.3kΩ.	-	3
4	Bread board	-	-	1
5	Connecting wires	-	-	As Required

THEORY

The total current flowing towards a junction point is equal to the total current flowing away from that point. In other words, the algebraic sum of all current meeting at a point is zero.

$$\Sigma I \text{ at junction point} = 0$$

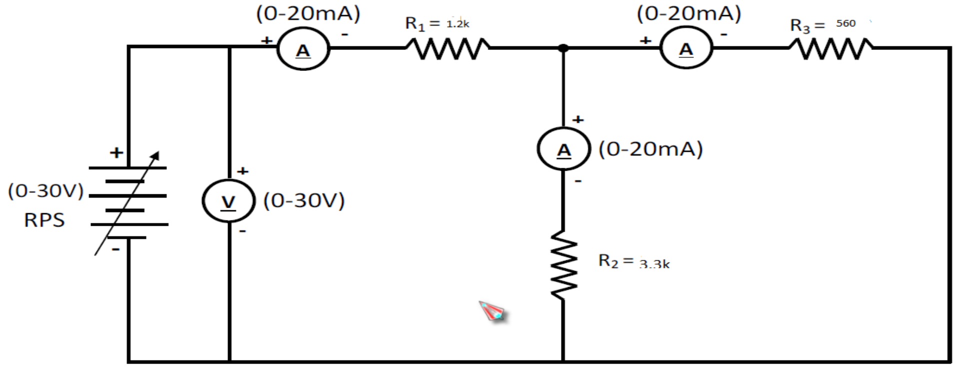
Note: Currents flowing towards a junction point are assumed to be positive while the currents flowing away from a junction point assumed to be negative.

.

PROCEDURE

1) Make the connections as per the circuit diagram.
2) Switch on the power supply.
3) Vary the RPS to a specified voltage and note down the corresponding ammeter readings.
4) Repeat the above step for various RPS voltages and tabulate the readings.

CIRCUIT DIAGRAM



TABULAR COLUMN:-

S.NO	Applied Voltage	Current I_1mA	Current I_2mA	Current I_3mA	Current I_1=I_2+I_3	
					Theoretical	**practical**

THEORETICAL CALCULATION:

RESULT:

Thus the Kirchoff's current law has been verified.

(b). VERIFICATION OF KIRCHOFF'S VOLTAGE LAW

AIM

To verify the kirchoff's voltage law for the given electrical circuit

APPARATUS REQUIRED:

S.No	Name of the apparatus	Range	Type	Quantity
1	Regulated power supply	(0 - 30) V	Analog	1
2	Voltmeter	(0 - 30) V	MC	4
3	Resistor	1.2 kΩ,560Ω 3.3kΩ.	-	3
4	Bread board	-	-	1
5	Connecting wires	-	-	As Required

THEORY

In any network, the algebraic sum of the voltage drops across the circuit elements of any closed path (loop or mesh) is equal to the algebraic sum of the e.m.fs in the path.

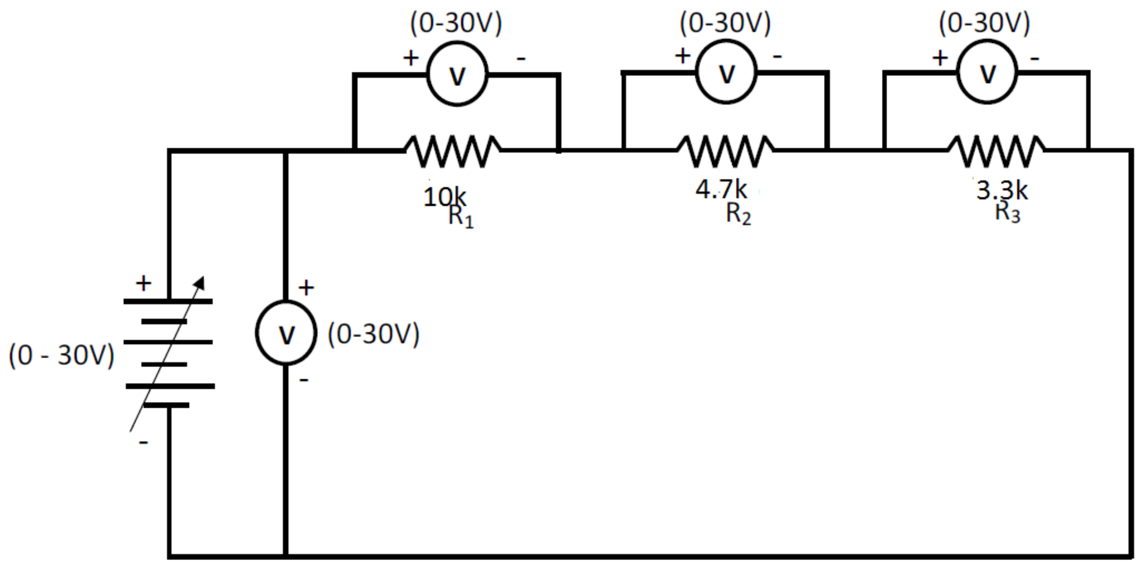
Around a closed path $\Sigma V = 0$

Note: While tracing a closed path, if we go from –ve marked terminal to +ve marked terminal, that voltage must be taken as positive, called **Potential rise** if it go from from +ve marked terminal to -ve marked terminal, that voltage must be taken as negative, called **Potential drop.**

PROCEDURE:

1) Make the connections as per the circuit diagram.
2) Switch on the power supply.
3) Vary the RPS to a specified voltage and note down the corresponding voltage readings across resistors.
4) Repeat the above step for various RPS voltages and tabulate the readings.

CIRCUIT DIAGRAM:



TABULAR COLUMN:-

S.No	Applied Voltage (V)	Voltage V_1 (V)	Voltage V_2 (V)	Voltage V_3 (V)	Total voltage $V_t = V_1+V_2+V_3$ (V)	
					Theoretical	Practical

THEORETICAL CALCULATION:

Review questions

1. Define current.
2. Define voltage
3. Define KCL.
4. Define KVL.
5. What are the limitations of KVL and KCL?

RESULT:

Thus Kirchoff's voltage law has been verified.

Ex. No	VERIFICATION OF NETWORK THEOREMS
Date:	

AIM

To verify the network theorems.

(i) Thevenin's theorem,
(ii) Norton's theorem,
(iii)Superposition theorem and
(iv)Maximum Power Transfer theorem

(a).THEVENIN'S THEOREM

APPARATUS REQUIRED

S.NO	Name of the apparatus	Range	Type	Quantity
1.	Regulated power supply	(0 - 30) V	Analog	1
2.	Ammeter	(0 - 30) mA	MC	1
3.	Voltmeter	(0 - 30) V	MC	1
4.	Resistor	560, 1. 8kΩ 220, 470	-	4
5	Bread board	-	-	-
6	Connecting wires	-		Required

THEORY

Any combination if linear bilateral circuit elements and active sources, regardless of the connection, connected to a given load Z_L, maybe replaced by a simple two port network consisting of a single voltage source (V_{th}) and a single impedance (Z_L). The load current equation is

$$I_L = V_{TH} / (Z_L + Z_{eq})$$

Where,

V_{TH} – Thevenin's equivalent voltage across Z_L in volts
Z_{eq}– Equivalent resistance across Z_L in ohms or Thevenin's impedance (Z_{th})
Z_L– Load impedance in ohms

PROCEDURE:

To determine Thevenin's voltage, V_{th}

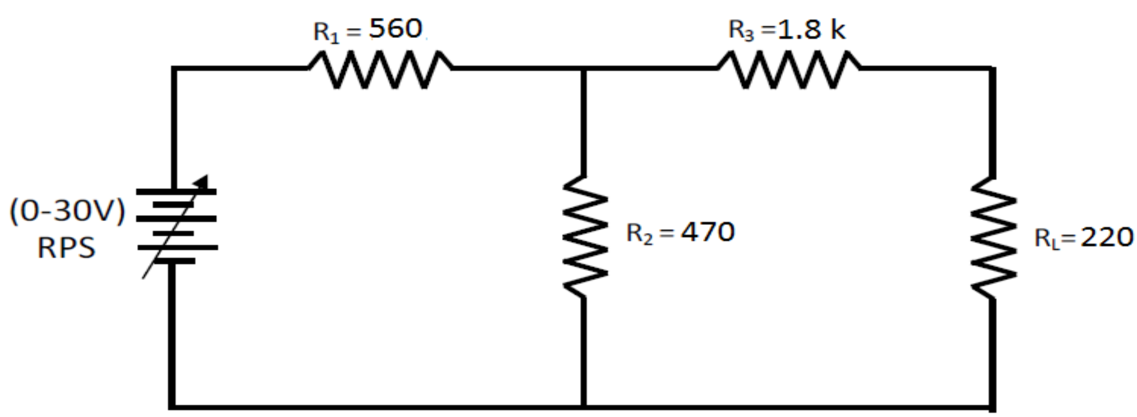
1) Make the connections as per the circuit diagram
2) Switch on the power supply

3) Vary the regulated power supply to a specified voltage and note down the corresponding voltmeter readings.
4) Repeat the previous step for different voltage by varying the RPS.
5) Switch off the power supply.

To determine of load current, I_L

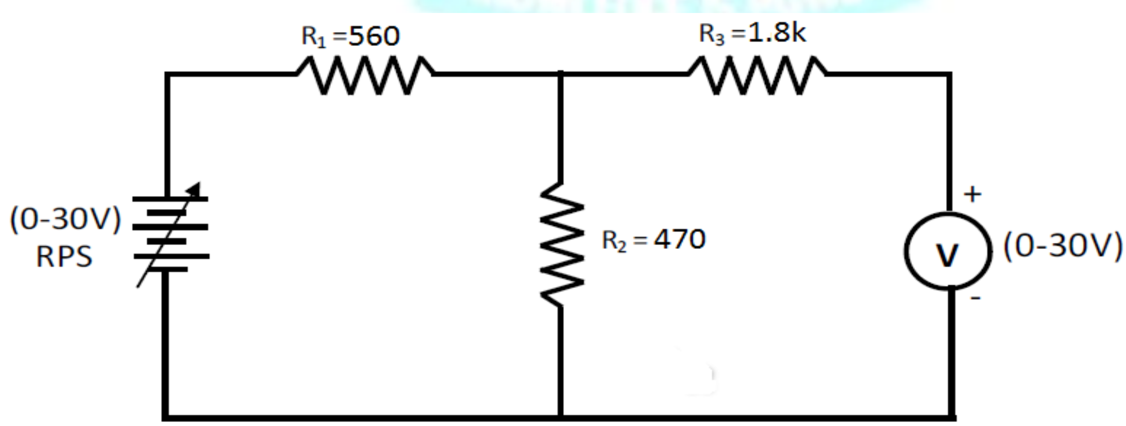
1) Make the connections as per the circuit diagram
2) Switch on the power supply
3) Vary the regulated power supply to a specified voltage and note down the corresponding ammeter readings.
4) Repeat the previous step for different voltage by varying the RPS.
5) Switch off the power supply.

CIRCUIT DIAGRAM 1:



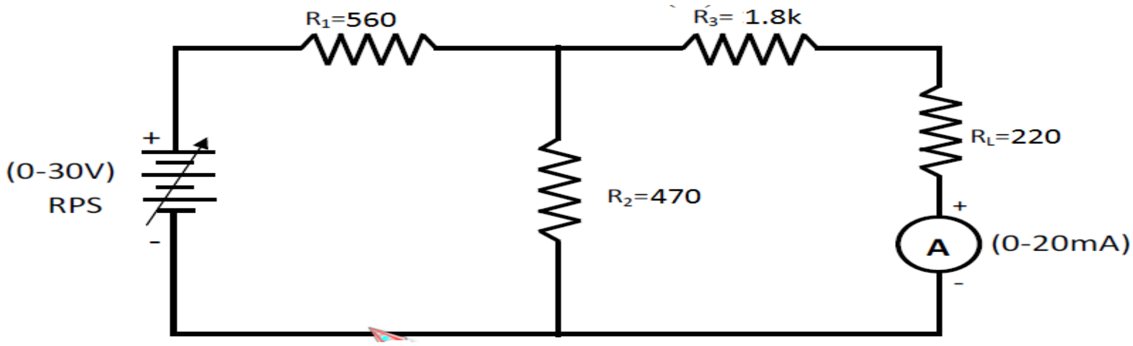
CIRCUIT DIAGRAM 2:

DETERMINATION OF THEVENIN VOLTAGE (Vth)



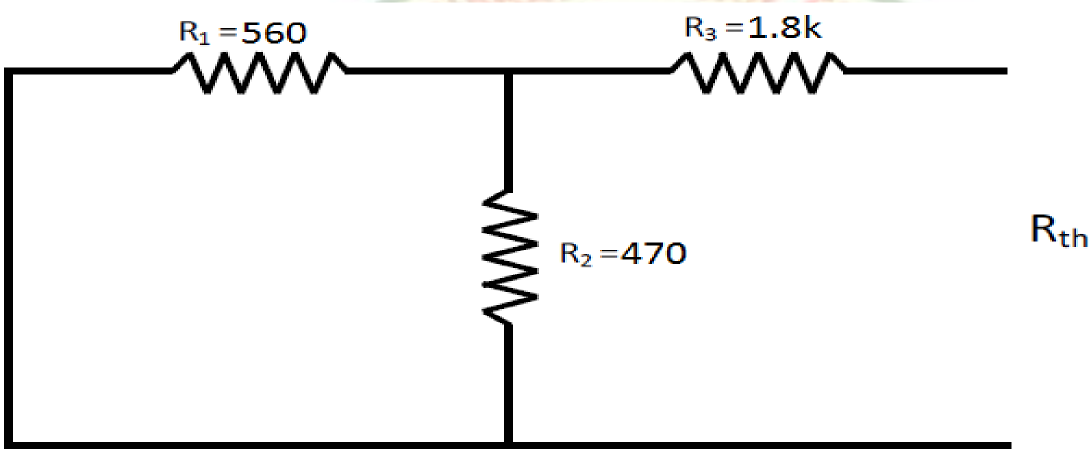
CIRCUIT DIAGRAM 3:

DETERMINATION OF LOAD CURRENT (IL)



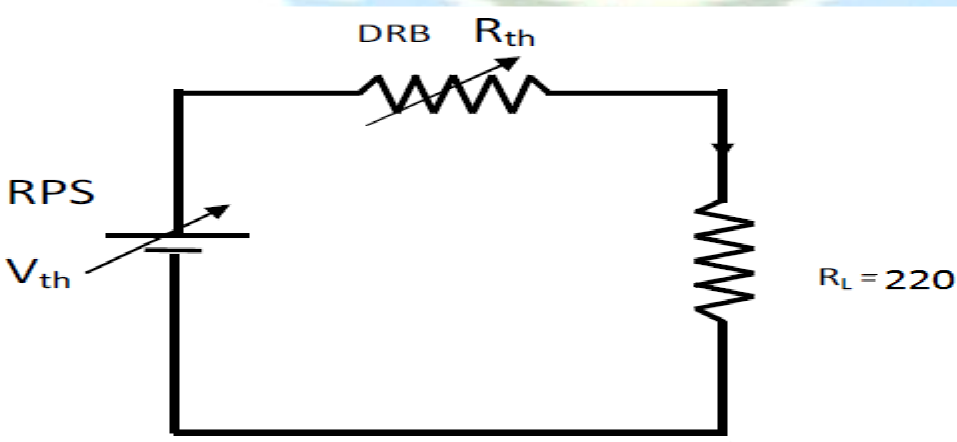
CIRCUIT DIAGRAM 4:

DETERMINATION OF Rth



CIRCUIT DIAGRAM5:

THEVENIN EQUIVALENT CIRCUIT:



TABULAR COLUMN:-

S.No	Applied Voltage (V)	Thevenin's Voltage (V_{Th})		Load current (IL)	
		Practical (V)	Theoretical (V)	Practical (I)	Theoretical (I)

THEORETICAL CALCULATION:

Thevenin's voltage, V_{th}= V [R2 / (R $_1$ + R $_2$)] Volts

The load current equation is $I_L = V_{TH} / (R_L+R_{eq})$

RESULT:

Thus the Thevenin's theorem was verified for the given electrical circuit.

Theoretical:

Vth =

Rth =

IL =

Practical:

Vth =

Rth =

IL =

(b).VERIFICATION OF NORTON THEOREM

AIM:

To verify the Norton theorem for the given electrical circuit

APPARATUS REQUIRED

S.NO	Name of the apparatus	Range	Type	Quantity
1.	Regulated power supply	(0 - 30) V	Analog	1
2.	Ammeter	(0 - 30) mA	MC	1
3.	Voltmeter	(0 - 30) V	MC	1
4.	Resistor	560, 1.8 kΩ 220, 470	-	4
5	Bread board	-	-	-
6	Connecting wires	-		Required

THEORY

Any linear active network with output terminals A & B can be replaced by an equivalent circuit with a single current source I in parallel with Rth(Thevenin equivalent resistance)

Where Rth is the equivalent resistance obtained by looking back the circuit through the open terminal A & B

FORMULAE

$I_L = I_{SC} * (R_{th} / (R_{th} + R_L))$

where, I_{SC}- Norton equivalent current source in amperes

I_L - Current through the load in amperes

R_{th}- Thevenin's equivalent resistance in ohms

R_L - Load resistance in ohms

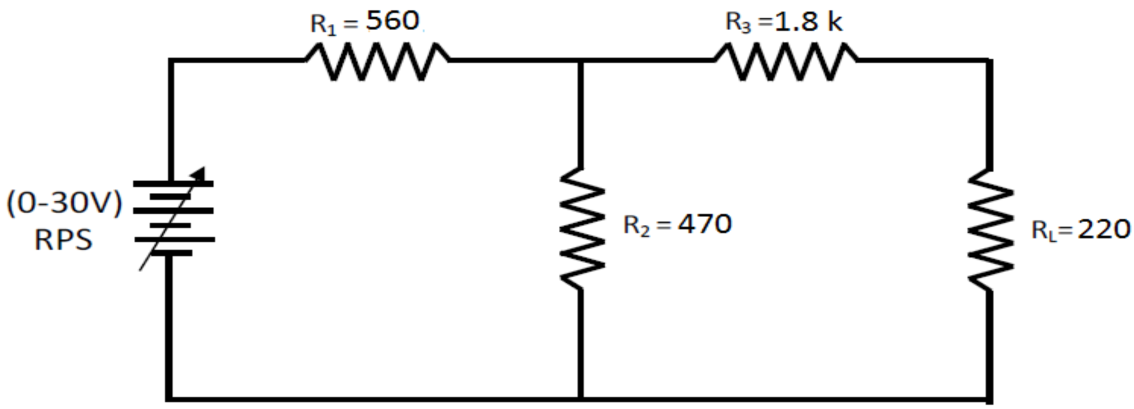
PROCEDURE :

1) The connections are given as per the circuit diagram
2) Switch on the power supply
3) The current in short circuited branch is noted using the ammeter
4) Tabulate the readings and check with the theoretical values.

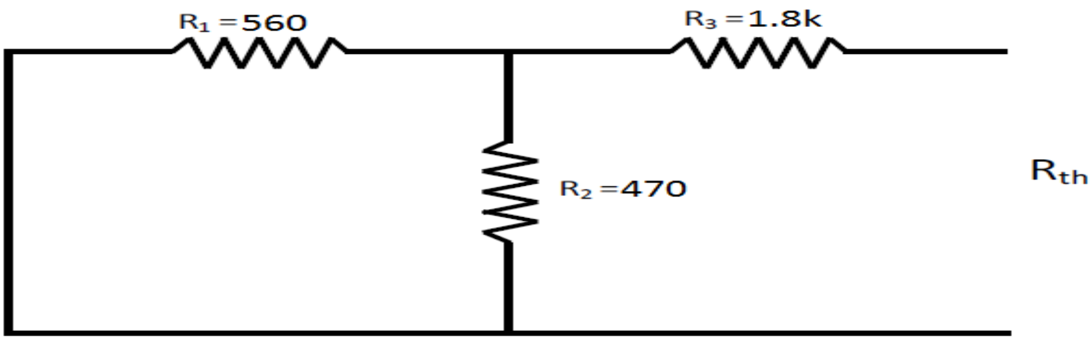
DETERMINATION OF LOAD CURRENT

1) Make the connections as per the circuit diagram.
2) Switch on the power supply.
3) Vary the RPS to a specified voltage and note the corresponding ammeter reading.
4) Repeat the above step for various RPS voltages and tabulate the reading.

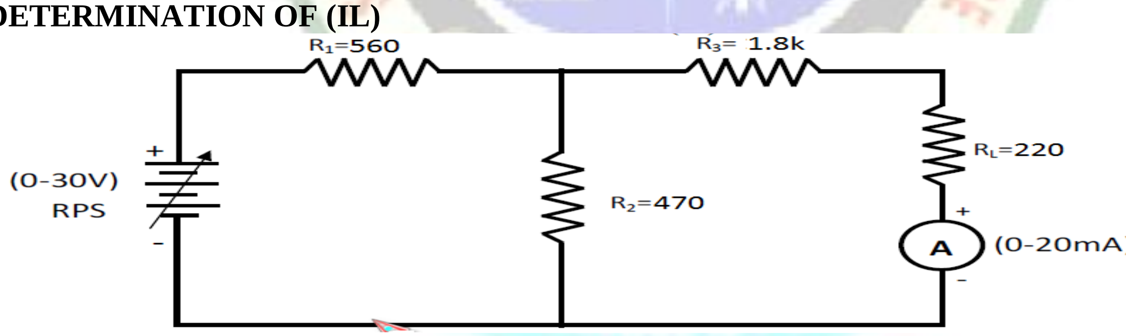
CIRCUIT DIAGRAM:I



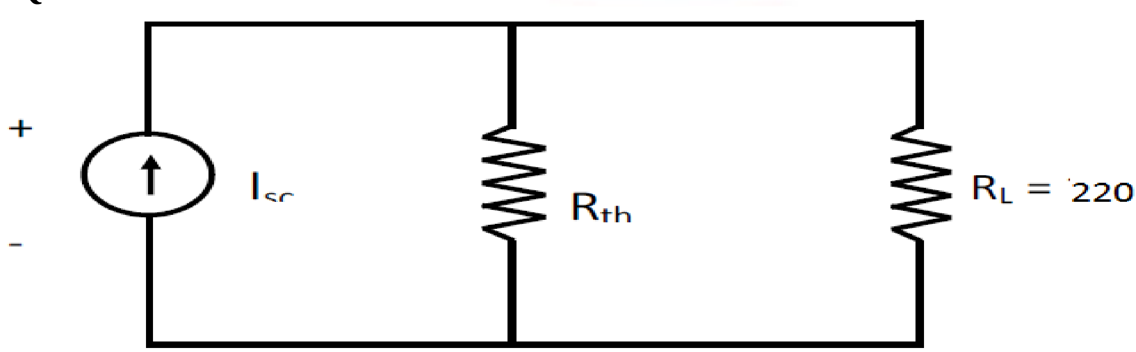
CIRCUIT DIAGRAM:II
DETERMINATION OF Rth:



DETERMINATION OF (IL)



EQUIVALENT CIRCUIT:



TABULAR COLUMN:-

S.No	Applied Voltage (V)	Isc		Load current (IL)	
		Practical (mA)	Theoretical (mA)	Practical (mA)	Theoretical (mA)

THEORETICAL CALCULATION:

$V_L = I_L * R_L$

$P_L = I_L^2 * R_L$

RESULT

Thus Norton theorem was verified for the given electrical circuit.

Theoretical:

$\mathbf{I_{sc}}$**=**

$\mathbf{R_{th}}$**=**

I_L =

Practical:

$\mathbf{I_{sc}}$**=**

$\mathbf{R_{th}}$**=**

I_L =

(C) VERIFICATION OF SUPERPOSITION THEOREM

AIM:

To practically verify superposition theorem for the given network with the theoretical calculation.

APPARATUS REQUIRED:

S.NO	Name of the apparatus	Range	Type	Quantity
1.	Regulated power supply	(0 - 30) V	Analog	1
2.	Ammeter	(0 - 20) mA	MC	1
3.	Voltmeter	(0 - 30) V	MC	1
4.	Resistor	1KΩ, 1kΩ 2 kΩ	-	3
5	Bread board	-	-	-
6	Connecting wires	-		Required

THEORY:

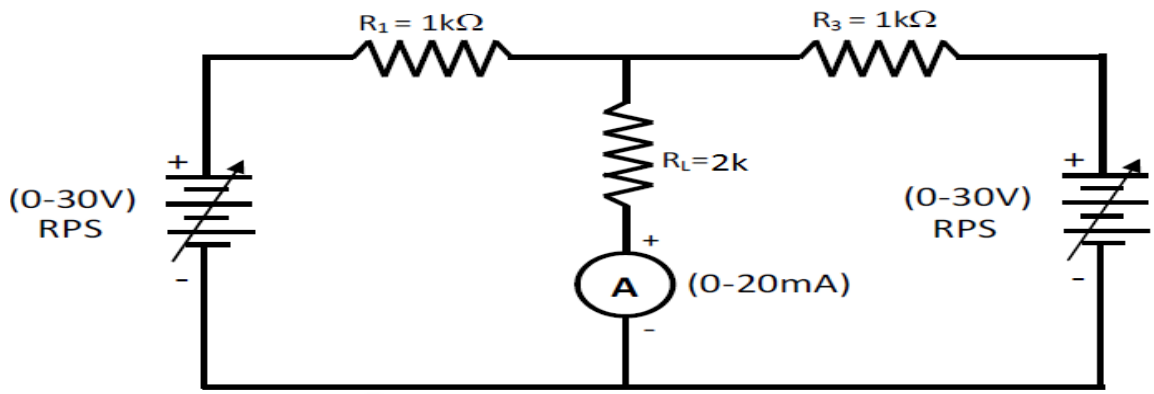
In a linear bilateral active network containing more than one source the total response obtained is algebraic sum of response obtained individually considering only one source at a time the source being suitable suppressed.

PROCEDURE:

1. The connection is made as per the circuit diagram.
2. With V_1 = 20V and V_2 = 0V observe the ammeter reading.
3. The above procedure repeated with V_1 = 0V and V_2 = 20V
4. The total response at the required terminal is obtained using sum of individual response.
5. Respect same procedure for different values of V_1 and V_2..

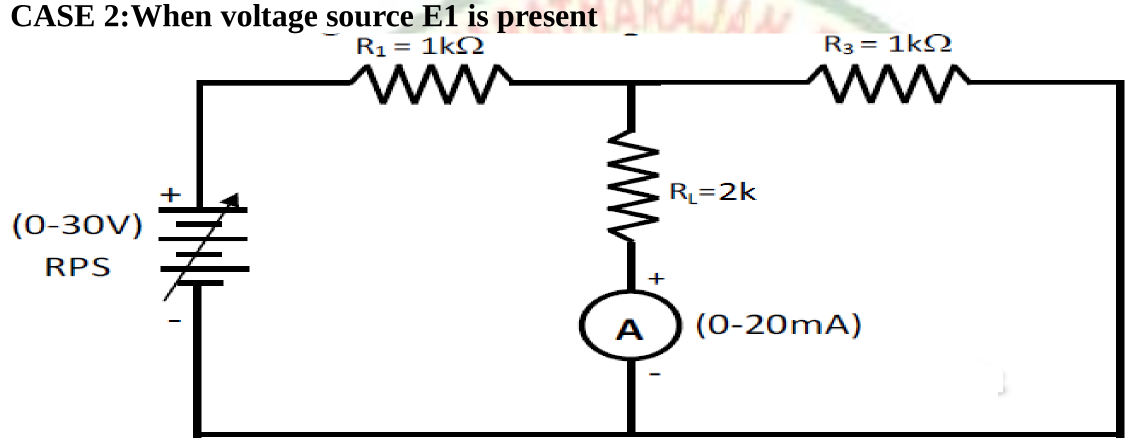
CIRCUIT DIAGRAM 1:

CASE 1:When both voltage sources E_1 and E_2 are present



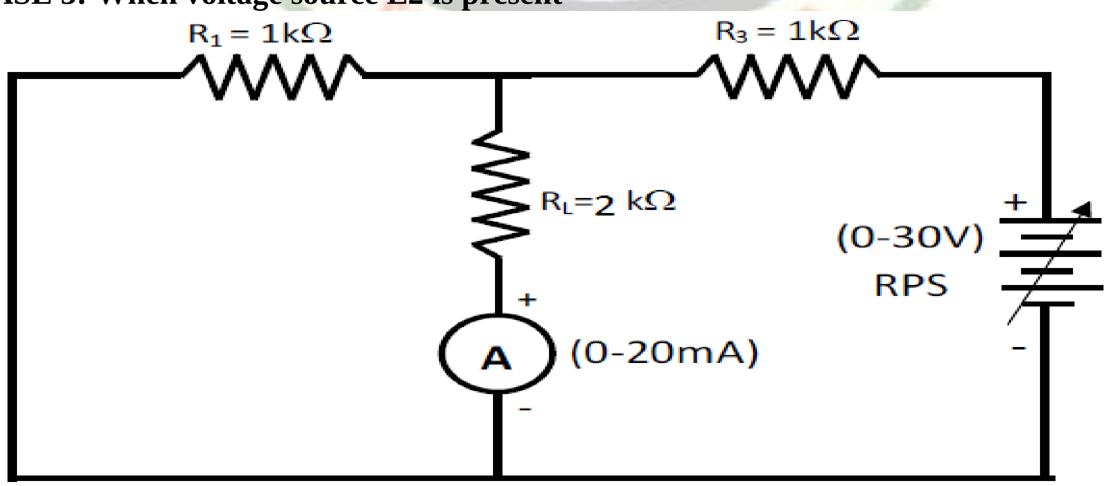
CIRCUIT DIAGRAM2:

CASE 2:When voltage source E1 is present



CIRCUIT DIAGRAM3:

CASE 3: When voltage source E2 is present



TABULAR COLUMN:-

CASE 1:When both voltage sources E1 and E2 are present.

S.No	Voltage E_1 (V)	Voltage E_2 (V)	Current I	
			Practical (I)	Theoretical (I)

CASE 2: When voltage source E_1 is present. [Circuit Diagram 2]

S.No	Voltage E_1 (V)	Current I	
		Practical (I)	Theoretical (I)

CASE 3:When voltage source E_1 is present. [Circuit Diagram 3]

S.No	Voltage E_2 (V)	Current I	
		Practical (I)	Theoretical (I)

THEORETICAL CALCULATION:

RESULT:

Thus superposition theorem was verified theoretically and experimentally.

(D).VERIFICATION OF MAXIMUMPOWER TRANSFER THEOREM

AIM:

To verify the maximum power transformation in purely passive circuit and the load resistance is variable.

APPARATUS REQUIRED:

S.NO	Name of the apparatus	Range	Type	Quantity
1.	Regulated power supply	(0 - 30) V	Analog	1
2.	Ammeter	(0 - 20) mA	MC	1
3.	Voltmeter	(0 - 30) V	MC	1
4.	Resistor	10KΩ, 1kΩ 22 kΩ, 3 kΩ	-	4
5	Bread board	-	-	-
6	Connecting wires	-		Required

THEORY

Maximum power will be delivered from the voltage source to a load, when the load resistance is equal to the internal resistance of the source.

The maximum power transferred to RL

$$P_{max} = V_g^2 / R_L$$

Where, Vg = Voltage across the load in Volts

RL = Load resistance in Ohm

PROCEDURE:

1. Connections are made as per the circuit diagram.
2. Remove the load resistor on the network
3. Calculated R_{TH} by substituting all sources with their internal resistances looking back at the network.
4. Calculate V_{TH}, the open circuit voltage between the terminals by replacing all the sources to their original position.

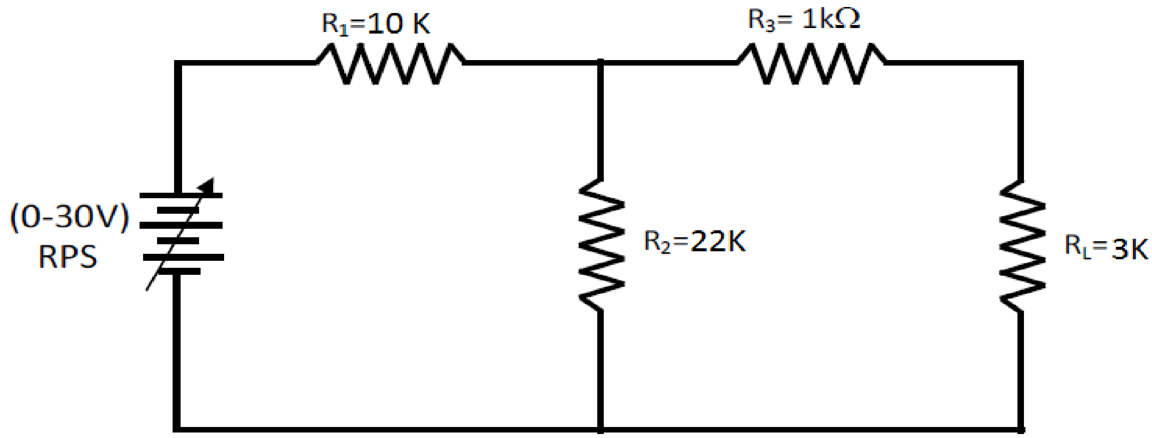
FORMULAE:

Maximum Power=$V_{th}{}^2/R_L$

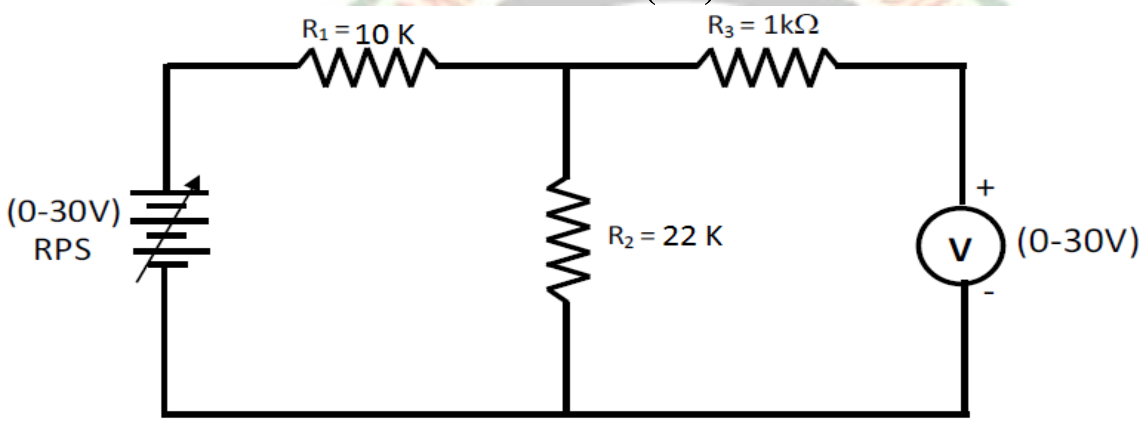
Where Vth - Thevenin voltage

R_L-Load resistor

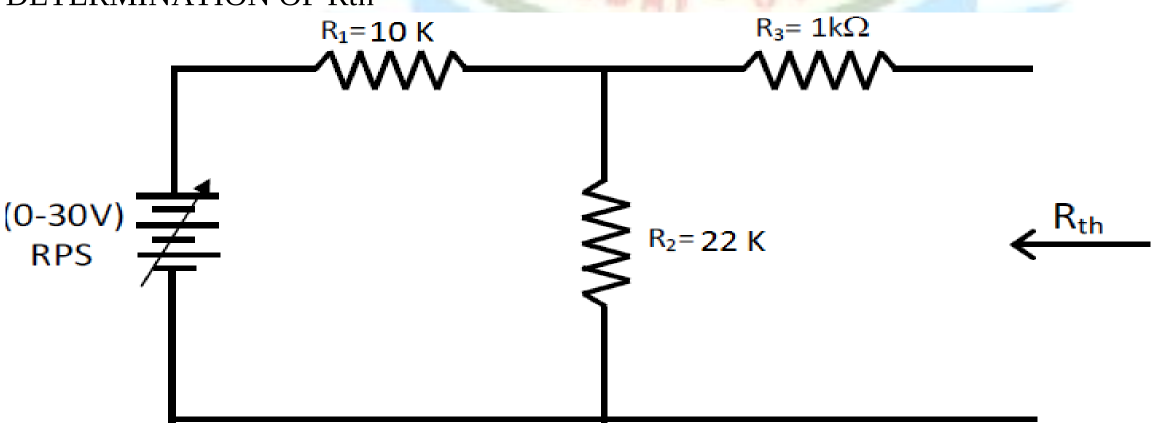
CIRCUIT DIAGRAM 1:



CIRCUIT DIAGRAM 2:
DETERMINATION OF THEVENIN VOLTAGE (Vth)



CIRCUIT DIAGRAM 3:
DETERMINATION OF Rth



TABULAR COLUMN:-

S.No	**Voltage (V)**	**Thevenin's voltage V_{th}**		**Maximum power delivered=**$[V_{th2}/R_L]$Watts
		Practical Voltage	**Theoretical Voltage**	

THEORETICAL CALCULATION:

Review questions

1. Define Thevenin's theorem.
2. Define Norton's theorem.
3. Define Superposition theorem.
4. Define Maximum Power Transfer theorem.

RESULT:

Thus the maximum power transfer theorem was verified theoretically and experimentally.

Ex. No	EXPERIMENTAL DETERMINATION OF TIME CONSTANT OF SERIES R-C ELECTRIC CIRCUITS
Date:	

AIM

To determine the time constant of RC circuit.

APPARATUS REQUIRED:

S. NO.	COMPONENTS	TYPE/RANGE	QTY.
1	Regulated power supply	(0-15)V	2 Nos.
2	SPST (single pole – single throw switch)		
3	Resistor	100 Ω	2 No
4	Capacitor	0.01 μF	1 No
5	Stop watch		1 No
6	DPST		1 No

THEORY

It is the product of resistance of the circuit in Ohms and the capacitance in Farad. It is denoted by tau (τ)

$\tau = R * C$

It is the time required to charge the capacitor, through the resistor, by ≈ 63.2 percent of the difference between the initial value and final value or discharge the capacitor to ≈36.8 percent. This value is derived from the mathematical constant *e*, specifically $1 - e^{-1}$, more specifically as voltage to charge the capacitor versus time

Charging $V(t) = Vo(1 - e^{-t/\tau})$[1]

Discharging $V(t) = Vo(e^{-t/\tau})$

The time constant τ is related to the cutoff frequency f_c, an alternative parameter of the RC circuit, by

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

or, equivalently,

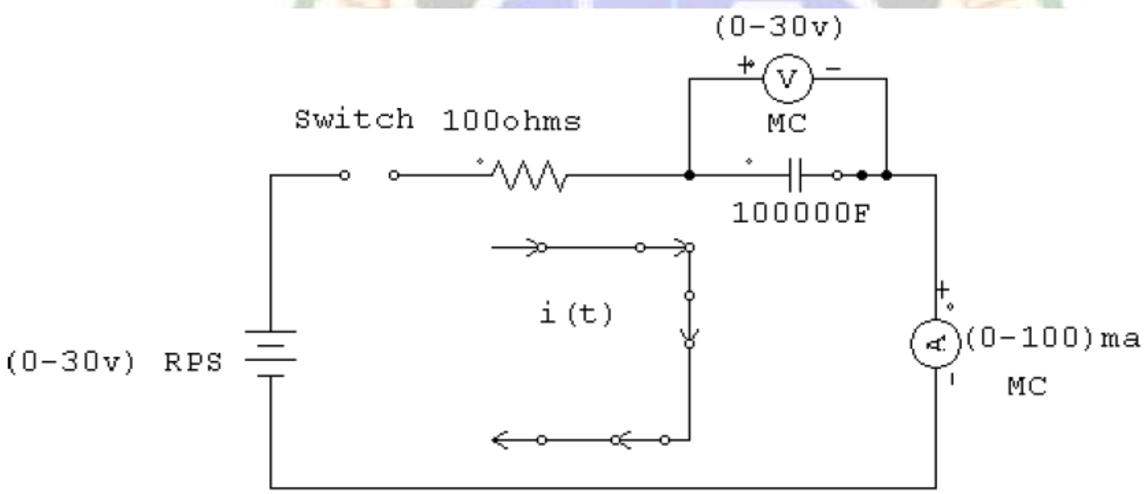
$$f_c = \frac{1}{2\pi RC} = \frac{1}{2\pi\tau}$$

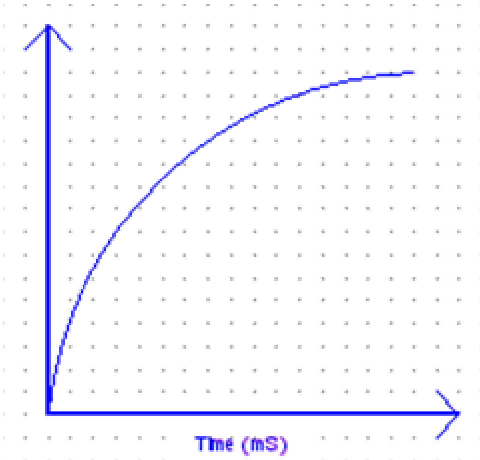
Where resistance in ohms and capacitance in farads yields the time constant in seconds or the frequency in Hz.

PROCEDURE:

1. Charge on capacitor is ‘o’ initially.
2. If there is a charge in it, short circuit the terminal then the charge will be dissipated.
3. Close the switch at t = 0
4. Simultaneously switch on the stop watch.
5. For every 2 seconds note down the voltage across capacitor until Voltmeter reaches 5 V.After reaching 15V allow 10 sec. for it.

RC – TRANSIENTS :-





TABULAR COLUMN:-

S. NO.	T(MS)	V (T) AMPS.

THEORETICAL CALCULATION:

Review questions

1. Define time constant.
2. Mention the applications of RC circuits.
3. What is meant by cut off frequency and formula?
4. Define angular frequency.

RESULT:

Thus, the transient response of RC circuits for dc input was obtained

Ex. No	EXPERIMENTAL DETERMINATION OF FREQUENCY RESPONSE OF RLC CIRCUITS
Date:	

(a) FREQUENCY RESPONSE OF SERIES RLC CIRCUIT

AIM

To determine and obtain the frequency response of a series RLC circuit.

APPARATUS REQUIRED:

Sl. No	Name of the apparatus	Range	Type	Quantity
1.	Decade Resistance Box	1 kΩ	-	**1**
2	Decade Inductance Box	250 mH	-	**1**
3	Decade Capacitance Box	1 μF	-	**1**
4	Function Generator	(0 - 3) MHz	-	**1**
5	C.R.O.		Analog	**1**
6	Bread Board		-	**1**
7	Connecting wires		-	Required

THEORY

An A.C. circuit is said to be in resonance with its power factor becomes unity at which the impedance of circuit becomes purely resistive. The frequency at which such condition occurs is called resonant frequency. At resonance the circuit current is maximum for series resonant.

FORMULAE

Resonant frequency, $F_0 = 1 / [2\pi \sqrt{LC}]$

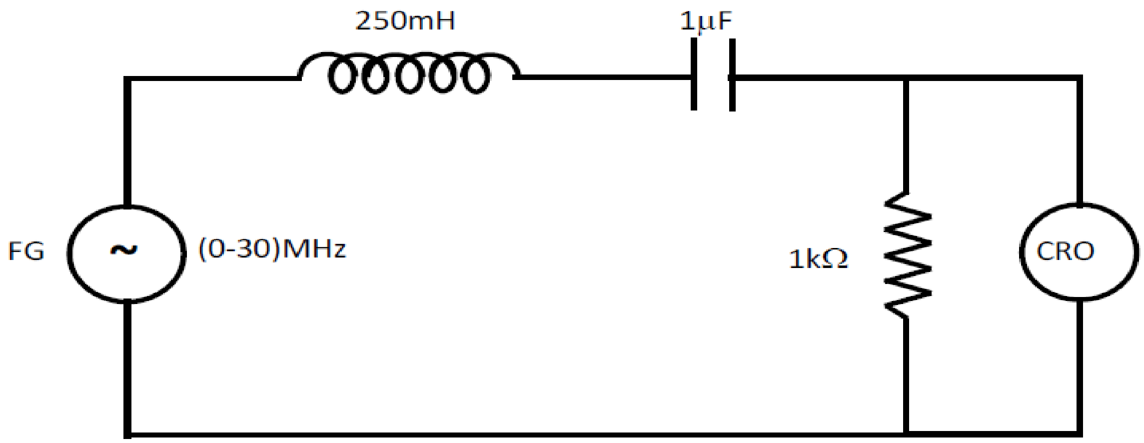
Band width = $F_2 - F_1$

Quality factor = $W_0 L / R$ Where, F_0 – Resonant frequency

PROCEDURE:

1. Make the connections as per the circuit diagram
2. Set the values of R, L & C
3. Frequency varied from 1kHz to 100 kHz in steps
4. At each step the frequency and voltage is noted down
5. Graph is drawn between frequency along X – axis and voltage along Y – axis

CIRCUIT DIAGRAM

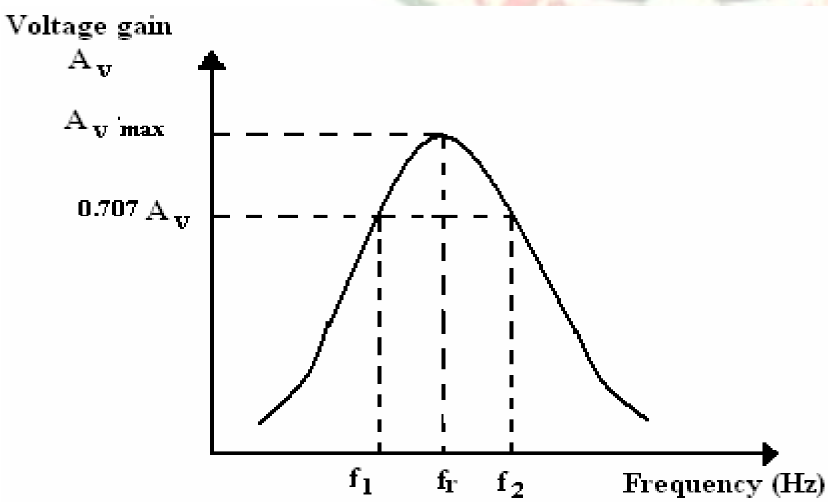


TABULAR COLUMN:-

Input voltage, Vin = …………..V

S.No	Frequency (Hz)	Voltage (Vo)	Voltage Gain = 20 log V_0/Vin

MODEL GRAPH:



F_1 – Lower cut off frequency in Hz

F_2 – Upper cut off frequency in Hz

THEORETICAL CALCULATION:

RESULT:

Thus the frequency response of series resonant circuit was obtained.

(B) FREQUENCY RESPONSE OF PARALLEL RLC CIRCUIT

AIM

To determine and obtain the frequency response of parallel R L C circuit

Sl. No	Name of the apparatus	Range	Type	Quantity
1.	Decade Resistance Box	1 kΩ	-	**1**
2	Decade Inductance Box	250 mH	-	**1**
3	Decade Capacitance Box	2 μF	-	**1**
4	Function Generator	(0 - 3) MHz	-	**1**
5	C.R.O.		Analog	**1**
6	Bread Board		-	**1**
7	Connecting wires		-	Required

THEORY

An A.C. circuit is said to be in resonance when its power factor becomes unity. The impedance of circuit at resonance becomes purely resistive. The frequency at which such a condition occurs is called resonant frequency.

The impedance is given by $Z = R + j (X_L - X_C)$

When the impedance is real, the | Z | is minimum. At resonance the power factor is unity

Therefore, Z = R and reactive part is zero. Thus $X_L - X_C = 0$

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

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

PROCEDURE

1. Make the connections as per the circuit diagram
2. Set the values of R, L & C
3. Frequency varied from 1kHz to 100 kHz in steps
4. At each step the frequency and voltage is noted down
5. Graph is drawn between frequency along X – axis and voltage along Y – axis

FORMULAE USED:

Resonant frequency, $f_0 = 1 / 2\pi \sqrt{LC}$

Band width = $F_2 - F_1$

Quality factor = $\omega_0 L / R$

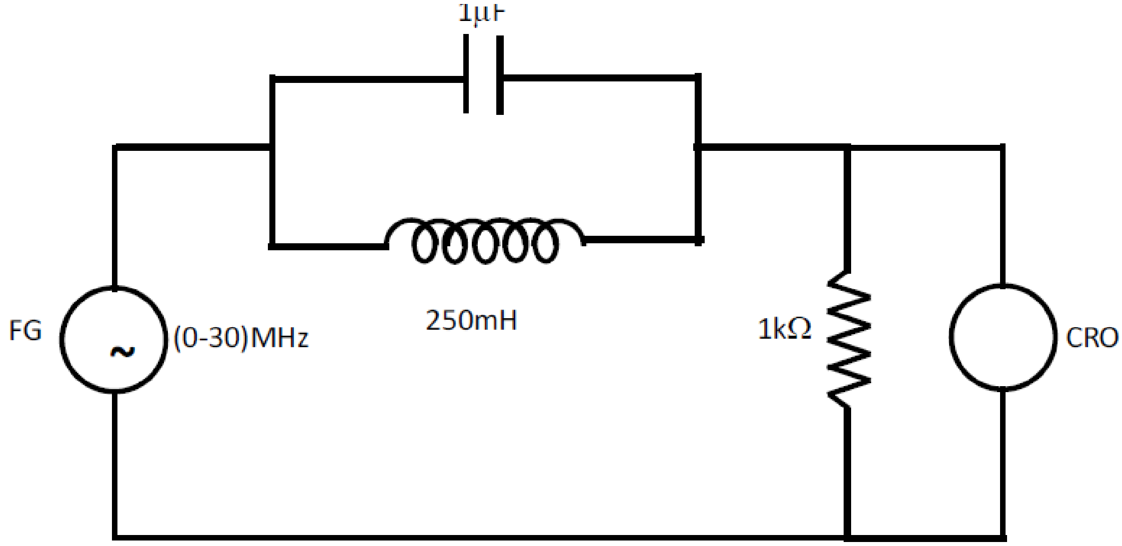
Where,

f_0 – Resonant frequency in Hz

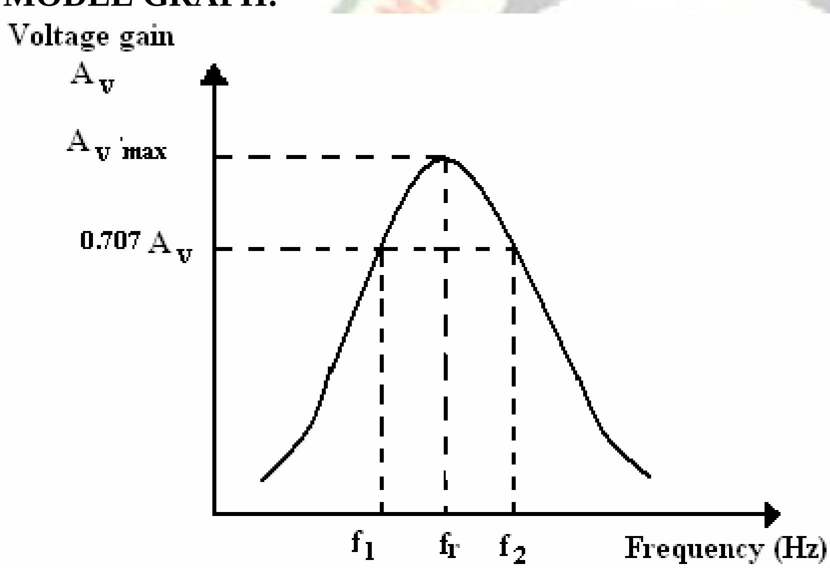
F_1 – Lower cut off frequency in Hz

F_2 – Upper cut off frequency in Hz

CIRCUIT DIAGRAM



MODEL GRAPH:



TABULAR COLUMN:-

Input voltage, Vin = …………..V

S.No	Frequency (Hz)	Voltage (Vo)	Voltage Gain = 20 log V_0/Vin

THEORETICAL CALCULATION:

Review questions

1. Define time constant.
2. Mention the applications of RC circuits.
3. What is meant by cut off frequency and formula?
4. Define angular frequency

RESULT

The frequency response of a parallel R.L.C. circuit was obtained.

Ex. No	DESIGN AND SIMULATION OF SERIES RESONANCE CIRCUIT
Date:	

AIM

To design and simulation of series resonance circuit using and matlab.

SOFTWARE REQUIRED

- Matlab
- Multisim-Pspice

PROCEDURE

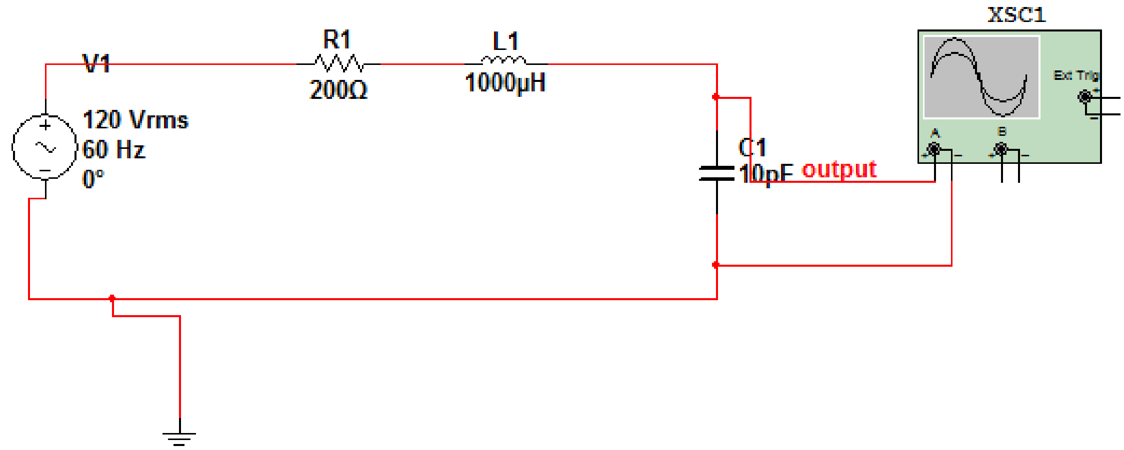
PSPICE-MULTISIM

1. Build the schematic shown in Figure 1.
2. Vm is an AC voltage source (VAC) from the source library. It needs to be set for 1 volt.
3. **L1** is an ideal inductor from the Analog Library. Set for 1000mH.
4. **R** is an ideal resistor from the Analog Library. Set value to {Rx}. Next add part named "Parameters". Then double click on part to enter edit mode. Click on new column, name = Rx, value= 200. Then click on column, select display and click on name and value.
5. **C1** is an ideal capacitor from the Analog library. Change the value to 40pF.

MULTISIM SIMULATION PROFILE SETTINGS

1. **Do analysis setup**
   i. At top of screen click on Multisim
   ii. Click on New Simulations Profile
   iii. Type name of profile that you wish.
   iv. Under Analysis tab, select AC sweep from the Analysis type pull down menu.
   v. Under AC Sweep Type
2. **Select Logarithmic and Decade as shown.**
   i. Click the simulate icon in tool bar and one tool bar will open and go to analyses and select the ac analyses
   ii. Then select the frequency parameter value
      a. Start freq = 100
      b. End freq = 10Meg
      c. Points/Decade = 101
   iii. Then add the output parameter to workspace.
   iv. Simulate the given below diagram.

SIMULATION DIAGRAM



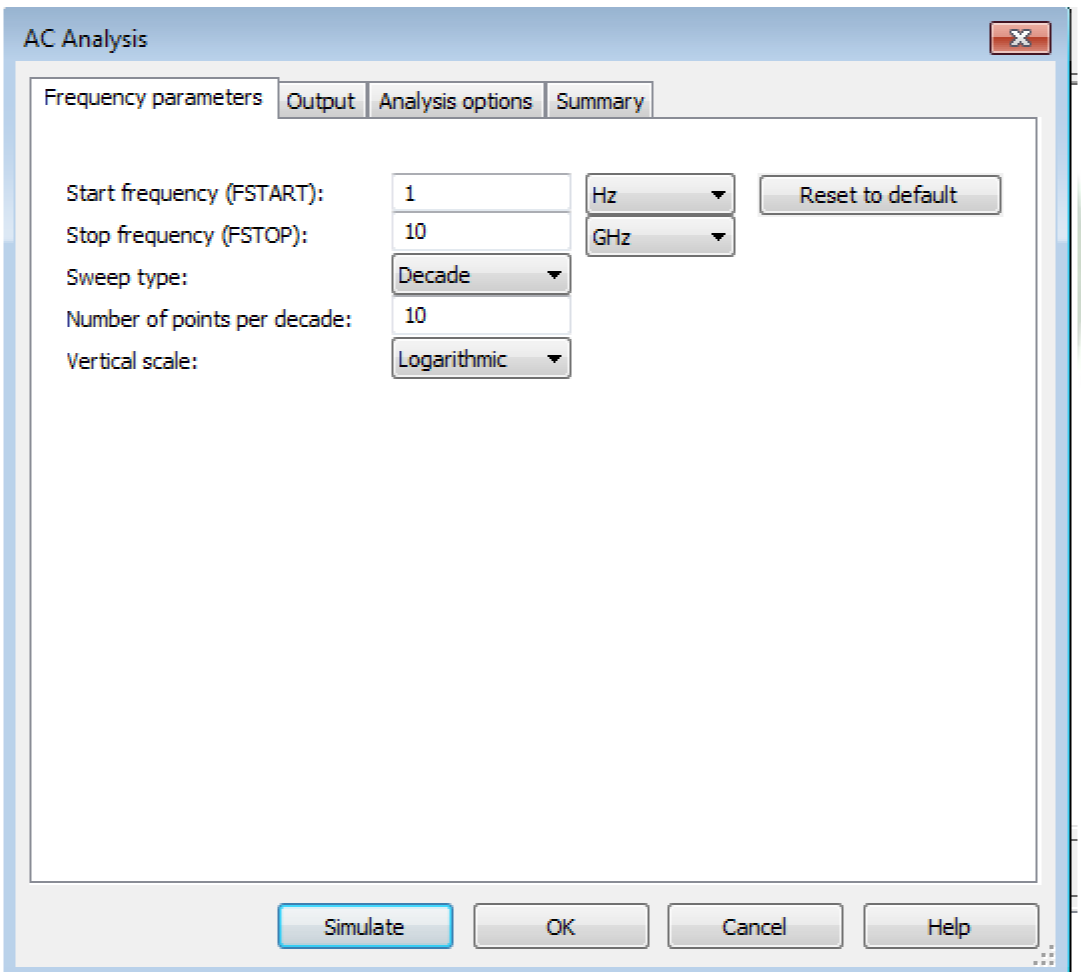


Fig. Simulation Profile Settings

MATLAB

Input impedance of series RLC tank circuit

```
disp('starting the function of Zinput_seriesRLC1');
%define all the component values and units for Tank
Vm=1; %volts
R=200; %ohms
C=40e-12; %Farads
L=1000e-6; %Henrys
%define the input impedance
Zin_numb=[L*C R*C 1];
Zin_de=[0 C 0];
Zinput=tf(Zin_numb,Zin_de)
figure(1)
bode(Zinput)
title('Input impedance of series RLC tank circuit')
%calculating important parameters of the tank
[z,p,k]=zpkdata(Zinput,'v');
wo=sqrt(1/L/C)
Beta=R/L
Q=wo/Beta
disp(' finished the function of Zinput_seriesRLC1');
```

Review questions

1. Define resonance.
2. What is resonant frequency?
3. State the condition for resonant frequency.
4. What are the applications of series resonant circuit?

RESULT

Thus the series resonance circuit was designed and simulated usingMultisimand Matlab.

Ex. No	DESIGN AND SIMULATION OF PARALLEL RESONANCE CIRCUIT
Date:	

Aim

To design and simulate the parallel resonance circuit.

SOFTWARE REQUIRED

- Matlab
- Multisim-Pspice

PROCEDURE

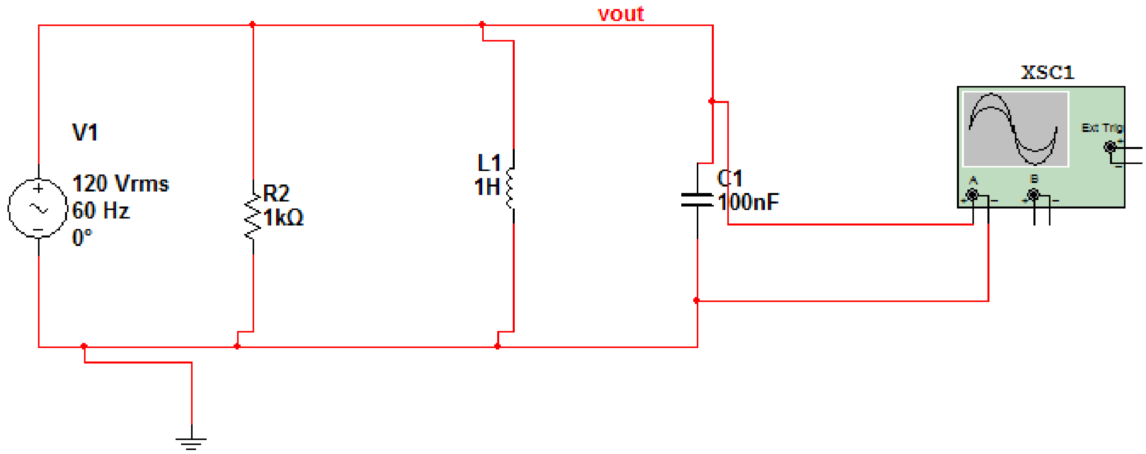
PSPICE-MULTISIM

1. Build the schematic shown in Figure 1.
2. Vm is an AC voltage source (VAC) from the source library. It needs to be set for 1 volt.
3. **L1** is an ideal inductor from the Analog Library. Set for 1000mH.
4. **R** is an ideal resistor from the Analog Library. Set value to {Rx}. Next add part named "Parameters". Then double click on part to enter edit mode. Click on new column, name = Rx, value= 200. Then click on column, select display and click on name and value.
5. **C1** is an ideal capacitor from the Analog library. Change the value to 40pF.

MULTISIM SIMULATION PROFILE SETTINGS

1. **Do analysis setup**
   i. At top of screen click on Multisim
   ii. Click on New Simulations Profile
   iii. Type name of profile that you wish.
   iv. Under Analysis tab, select AC sweep from the Analysis type pull down menu.
   v. Under AC Sweep Type
2. **Select Logarithmic and Decade as shown.**
   i. Click the simulate icon in tool bar and one tool bar will open and go to analyses and select the ac analyses
   ii. Then select the frequency parameter value
      d. Start freq = 100
      e. End freq = 10Meg
      f. Points/Decade = 101
   iii. Then add the output parameter to workspace. Simulate the given below diagram.

SIMULATION DIAGRAM



MATLAB

INPUT IMPEDANCE OF PARALLEL RLC TANK CIRCUIT

```
function[Zinput]=Zinput_parallelRLC1()
disp('Starting the function of Zinput_seriesRLC1');
Im =0.0001;
R=20000;
C=100e-09;
L=0.1;
Zinductor=tf([L 0],[0,1]);
Zcapacitor=tf([0 1], [C 0]);
Zinput=1/(1/R+1/Zcapacitor+1/Zinductor)
figure(1)
bode(Zinput)
title('Input impedance of parallel RLC tank circuit')
[z,p,k]=zpkdata(Zinput,'v');
w0=sqrt(1/L/C)
Beta=1/R/C
Q=w0/Beta
disp('finished the function of Zinput_seriesRLC1');
```

RESULT

Thus the parallel resonance circuit was designed and simulated using Multiaim andMatlab.

Ex. No	SIMULATION OF LOW PASS AND HIGH PASS FILTERS
Date:	

Aim

To design and simulate the low pass and high pass filters.

SOFTWARE REQUIRED

- Pspice-Multisim

Theory

Low pass filter

A low-pass filter is a filter that passes low-frequencysignals and attenuates (reduces the amplitude of) signals with frequencies higher than the cutoff frequency. The actual amount of attenuation for each frequency varies depending on specific filter design. It is sometimes called a high-cut filter, or treble cut filter in audio applications

One simple low-pass filter circuit consists of a resistor in series with a load, and a capacitor in parallel with the load. The capacitor exhibits reactance, and blocks low-frequency signals, forcing them through the load instead. At higher frequencies the reactance drops, and the capacitor effectively functions as a short circuit.

High pass filter

A high-pass filter (HPF) is an electronic filter that passes high-frequencysignals but attenuates (reduces the amplitude of) signals with frequencies lower than the cutoff frequency. The actual amount of attenuation for each frequency varies from filter to filter. 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.

Formula

The combination of resistance and capacitance gives the time constant of the filter $\tau = RC$. The break frequency, also called the turnover frequency or cutoff frequency (in hertz), is determined by the time constant:

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

or equivalently (in radians per second):

$$\omega_c = \frac{1}{\tau} = \frac{1}{RC}$$

PROCEDURE

1. LOW PASS PASSIVE FILTER AND HIGH PASS FILTER

a) Build the schematic shown in Figure 1 and figure 2.
b) Apply the VAC, set VAC to 1.
c) R is an ideal resistor from the Analog Library. Set value to 1k
d) C is an ideal capacitor from the Analog library. Change the value to 0.1u.

This is a classical low pass filter with RC cut off frequency (-3db) that can be estimated by the formula fc=(6.28*R*C), and in our case fc=1 / (6.28*0.1*1k)=1.59khz, where we express the capacitances in uF,resistance in kohm and frequency in khz

2. MULTISIM SIMULATION PROFILE SETTINGS

a) Choose AC Sweep/Noise in the Analysis type menu
b) Set the Start Frequency at 10, the End Frequency at 1Meg and the Points/Decade at 10
c) Make sure Logarithmic is selected and set to Decade
d) Click OK

CIRCUIT DIAGRAM

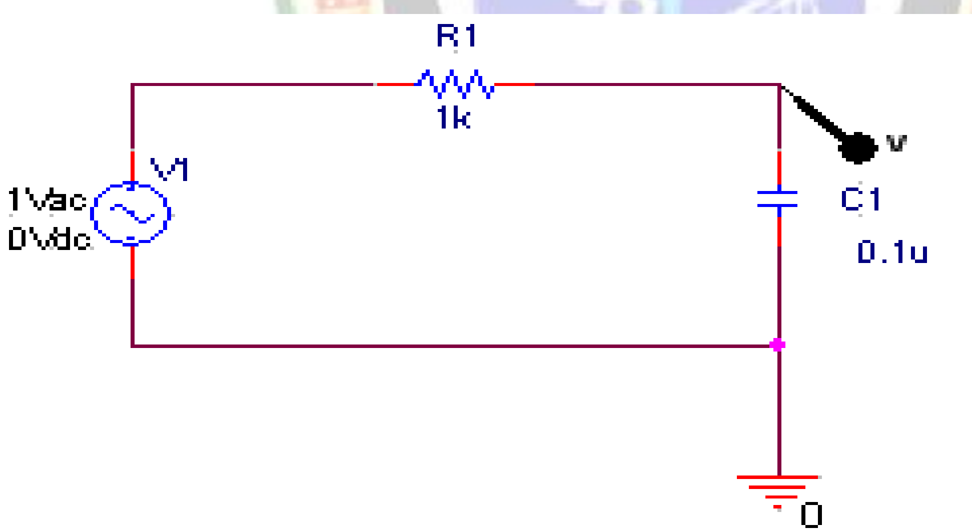


Fig.1.Low Pass Passive Filter

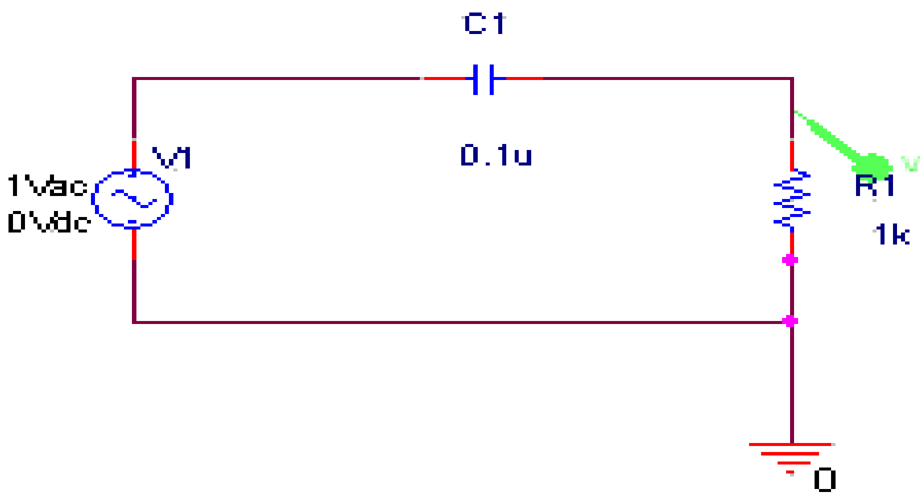


Fig.2.High Pass Passive Filter

Review questions

1. Define filter.
2. What is low pass filter?
3. What is high pass filter?
4. Define angular frequency.
5. What is cut off frequency?

Result

Thus the passive low pass and high pass filter was designed and simulated using Pspice-Multisim.

Ex. No	DETERMINATION OF POWER IN THREE PHASE CIRCUITS BY TWO-WATT METER METHOD
Date:	

Aim

To determine the power in the three phase circuit by two wattmeter method

Apparatus required

S.NO	NAME OF THE APPRATUS	RANGE	QUANTITY
1	wattmeter	600V,10A.UPF	2
2	3 Phase Auto transformer	-	1
3	AC Ammeter	(0-10)A MI	1
4	Voltmeter	(0-600)V MI	1
5	Connecting wires	-	Required
6	Resistive Load	3 phase	1

THEORY

Kirchoff's laws tell us the following about a three-wire circuit:

1) If two of the three currents are known, the third must be equal to the sum of the other two but opposite in direction or sign. Thus, if one measures the instantaneous current in two branches of a three-wire circuit, one can determine the instantaneous value of the third.

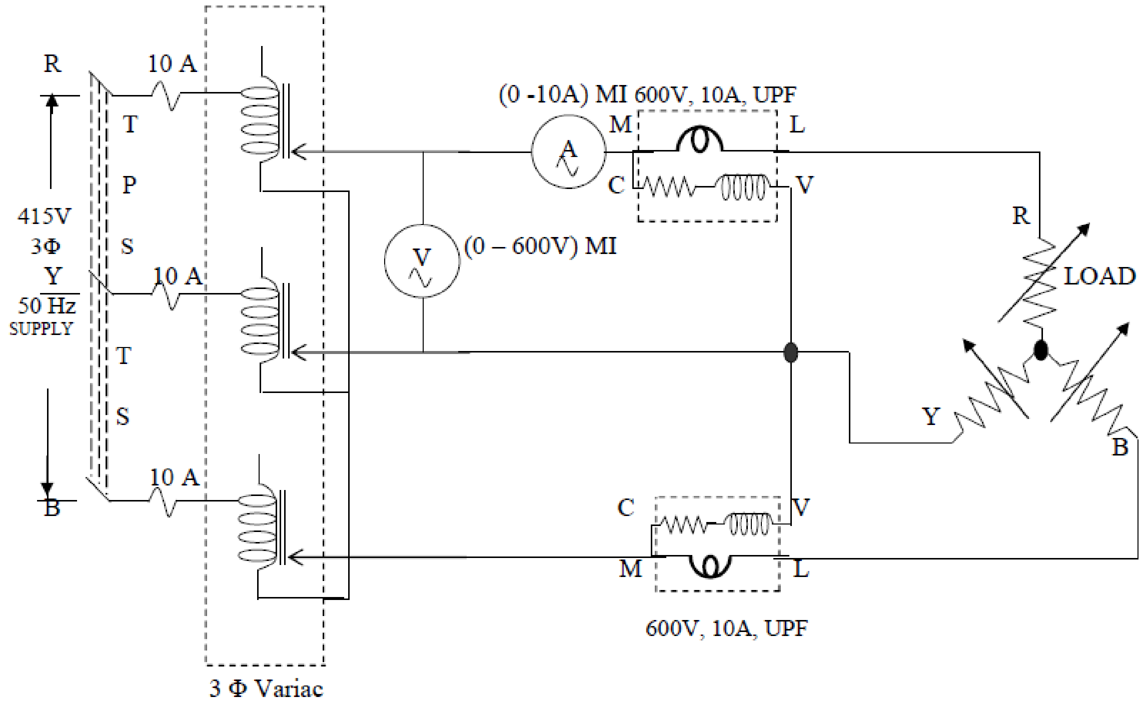
2) If two of the three voltages are known, the third must be equal to the sum of the other two but opposite in direction or sign. Thus, if one measures the instantaneous voltage between two pairs of lines, one can determine the instantaneous value of the third pair.

From these two laws one can infer that measuring two of the currents and two of the voltages in a three-wire circuit will be sufficient to measure the total power.

Procedure

1. Connections are made as per the circuit diagram
2. Set the voltage to its rated value.
3. Set the load and note down the corresponding meter readings
4. Repeat step 3 for various load ranges.
5. Compare the measure values with the practical calculations.

CIRCUIT DIAGRAM: **RESISTIVE LOAD**



TABULATION

S. No	Load voltage (V_L)	Load current (I_L)	Wattmeter1 reading (W_1)	Wattmeter2 reading (W_2)	Total power $P=W_1+W_2$	Theoretical Power $P=\sqrt{3}V_LI_L$
Unit	Volts	Amps	watts	watts	Watts	Watts

Model Calculation

Review questions

1. Define two wattmeter method.
2. Define three phase power.
3. Define wattmeter.
4. Define power factor.
5. Explain why it is necessary to potential coil circuit purely resistive in wattmeters?
6. Give the expression for 3phase power.
7. What is LPF Wattmeter?

RESULT:

The Power of the given experiment is measured by using two wattmeter methods

Ex. No	CALIBRATION OF SINGLE PHASE ENERGY METER
Date:	

AIM

To calibrate the single phase energy meter by direct loading.

APPARATUS REQUIRED

S.NO	NAME OF THE APPRATUS	RANGE	QUANTITY
1	Single-Phase Energy meter		**1**
2	Wattmeter	(300V,10A LPF)	1
3	Stopwatch		1
4	M.I Ammeter	(0-5)A	1
5	M.I Voltmeter	(0-300)V	1
6	Connecting wires		Required

FORMULA TO BE USED:

1. True energy = W*t
2. Energy Recorded = No of revolution /Energy meter constant.
3. %error = (True energy- Energy recorded)/True energy

PROCEDURE:

1. Connections are given as per the circuit diagrams.
2. Switch on the power supply.
3. Vary the load and keep one particular position.
4. Note down the wattmeter readings.
5. Determine the time require to complete the revolution of energy meter.
6. From that find out the actual energy consumed, energy recorded and percentage of error.

THEORY

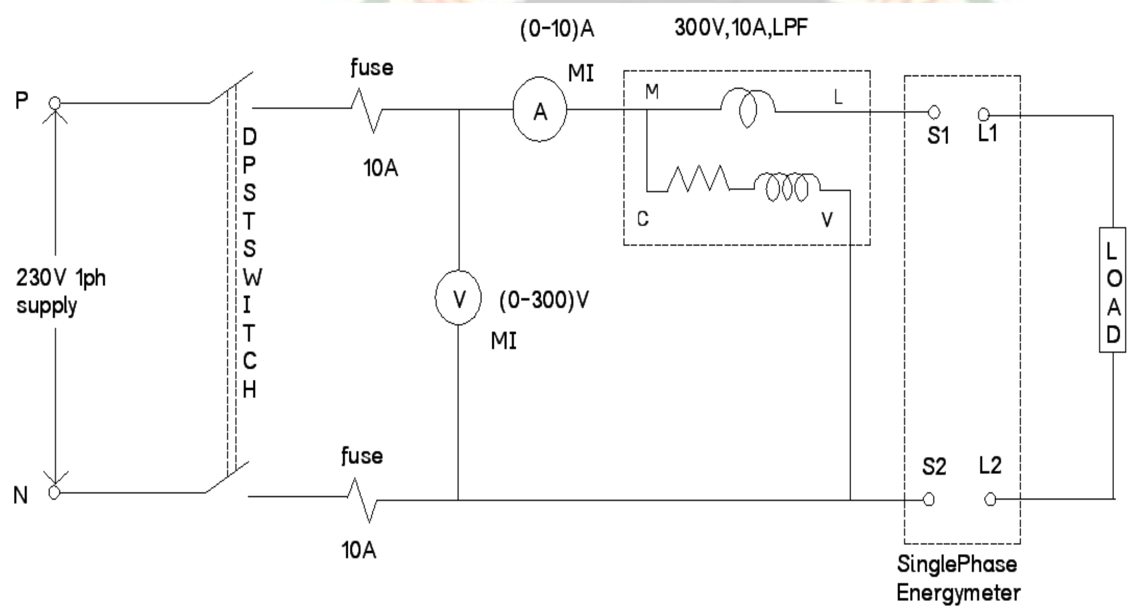
The calibration is the procedure for determining the correct values of measurand by comparing with the standard ones. The standard of device with which comparison ismade is called as a standard instrument. The standard instrument which is unknown and it is said to be calibrated is called test instrument .Thus in calibration; test instrument is compared with the standard instrument. There are two fundamental methodologies for obtaining the comparison between the test instrument and standard instrument

Thesemethodologies are

- Direct Comparison
- Indirect Comparison

The calibration offers a guarantee to the device or instrument that is operating withrequired accuracy under the stipulated environmental conditions. It creates the confidenceof using the properly calibrated instrument, in user's mind. The periodic calibration ofinstrument is very much necessary.The calibration procedure involves the steps like visual inspection for variousdefects, installations according to the specifications, zero adjustment, etc….

CIRCUIT DIAGRAM



TABULAR COLUMN:

S.No	True power KW	No of revolution	Time	True energy kWh	Energy recorded kWh	% error

MODEL CALCULATION

Review questions

1. What is creeping?
2. Which torque is absent in energy meter? Why?
3. Define energy meter constant.

RESULT:

Thus the given single phase energy meter is calibrated with actual energy consumption and found out the error

Ex. No	DETERMINATION OF TWO PORT NETWORK PARAMETERS
Date:	

AIM

To calculate and verify 'Z' ,‘Y’ , ABCD, and H parameters of two-port network.

APPARATUS REQUIRED

SL.NO.	NAME OF THE COMPONENT	SPECIFICATIONS	QUANTITY
1	Resistors	1K	2
		2K	1
2	Regulated Power Supply (RPS)	0-30 V	1
3	Voltmeter	0-20V	1
4	Ammeter	0-20 mA	1
5	Bread Board		1

THEORY:

In **Z parameters** of a two-port, the input & output voltages V1 & V2 can be expressed in terms of input & output currents I1 & I2. Out of four variables (i.e V1, V2, I1, I2) V1& V2 are dependent variables whereas I1 & I2 are independent variables. Thus,

V1 = Z11I1+ Z12 I2 -----(1)
V2 = Z21I1 + Z22 I2 -----(2)

Here Z11 & Z22 are the input & output driving point impedances while Z12 & Z21 are the reverse & forward transfer impedances.

In **Y parameters** of a two-port, the input & output currents I1 & I2 can be expressed in terms of input & output voltages V1 &V2 . Out of four variables (i.e I1, I2, V, V2) I1& I2 are dependent variables whereas V1 & V2 are independent variables.

I1 = Y11V1 + Y12V2 ------(3)
I2 = Y21V1 + Y22V2 -------(4)

Here Y11 & Y22 are the input & output driving point admittances while Y12 & Y21are thereverse & forward transfer admittances

ABCD parameters are widely used in analysis of power transmission engineering where they are termed as "CircuitParameters". ABCD parameters are also known as "Transmission Parameters". In these parameters, the voltage &current at the sending end terminals can be expressed in terms of voltage & current at the receiving end.Thus,

V1 = AV 2 + B (-I2) ---------(5)
I1 = CV2 + D (-I2) -----------(6)

Here "A" is called reverse voltage ratio, "B" is called transfer impedance "C" is called transfer admittance & "D" is called reverse current ratio.

In **'h' parameters** of a two port network, voltage of the input port and the current of the output port are expressed in terms of the current of the input port and the voltage of the output port. Due to this reason, these parameters are called as 'hybrid' parameters, i.e. out of four variables (i.e. V1, V2, I1, I2) V1, I2 are dependent variables.
Thus,

V1= h11I1 + h12V2 ------------- (1)
I2 = h21I1 + h22V22 ----------- (2)

H11 and H22 are input impedance and output admittance.
H21 and H12 are forward current gain and reverse voltage gain
PROCEDURE:

Z Parameters

1. Connect the circuit as shown in fig. & switch 'ON' the experimental board.
2. First open the O/P terminal & supply 5V to I/P terminal. Measure O/P Voltage & I/P Current.
3. Secondly, open I/P terminal & supply 5V to O/P terminal. Measure I/P Voltage & O/P current using multi-meter.
4. Calculate the values of Z parameter using Equation (1) & (2).
5. Switch 'OFF' the supply after taking the readings.

Y-Parameter

1. Connect the circuit as shown in fig. & switch 'ON' the experimental board.
2. First short the O/P terminal & supply 5V to I/P terminal. Measure O/P & I/P current
3. Secondly, short I/P terminal & supply 5V to O/P terminal. Measure I/P& O/P current using multi-meter.
4. Calculate the values of Y parameter using Eq. (1) & (2).
5. Switch 'off' the supply after taking the readings.

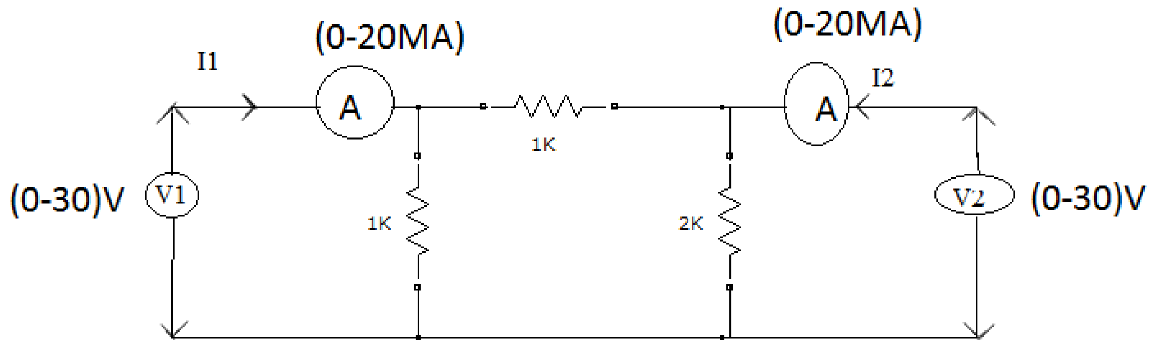
ABCD Parameter

1. Connect the circuit as shown in fig. & switch 'ON' the experimental board.
2. First open the O/P terminal & supply 5V to I/P terminal. Measure O/P voltage & I/P current
3. Secondly, short the O/P terminal & supply 5V to I/P terminal. Measure I/P& O/P current using multi-meter.
4. Calculate the A, B, C, & D parameters using the Eq. (1) & (2).
5. Switch 'off' the supply after taking the readings

H Parameter

1. Connect the circuit as shown in fig. & switch 'ON' the experimental board.
2. Short the output port and excite input port with a known voltage source Vs. So that V1 = Vs and V2 = 0.We determine I1 and I2 to obtain h11 and h21.
3. Input port is open circuited and output port is excited with the same voltage source Vs. So that V2 = VS and I1 = 0, we determine I2 and V1 to obtain h12 and h22.
4. Switch 'off' the supply after taking the readings.

CIRCUIT DIAGRAM



OBSERVATION TABLE:

Z Parameters

S.No	When i/p is open ckt			When o/p is open ckt		
	V2	V1	I2	V2	V1	I1

Y Parameters

S.No	When i/p is short ckt			When o/p is short ckt		
	V2	I1	I2	V1	I1	I2

ABCD Parameters

S.No	When o/p is short ckt			When i/p is short ckt		
	V1	I1	I2	V2	V1	I2

H Parameters

S.No	When o/p is open ckt			When o/p is shortckt		
	V1	V2	I1	V1	I2	I1

SAMPLE CALCULATION:

Z PARAMETER:

1. When O/P is open circuited i.e. I2 = 0

   Z11 = V1/I1 , Z21 =V2 /I1.

2. When I/P is open circuited i.e. II = 0

   Z12 = V1/I2 , Z22 = V2 /I2.

Y PARAMETER:

1. When O/P is short circuited i.e. V2 = 0

   Y11 = I1/V1 Y21 = I2 /V1

2. When I/P is short circuited i.e. VI = 0

   Y12 = I1/V2 Y22 = I2 /V2.

ABCD PARAMETER:

1. When O/P is open circuited i.e. I2 = 0

   A = V1/V2 C = I1 /V2

2. When O/P is short circuited i.e. V2 = 0

   B = -V1/I2 D = -I1 /I2

H PARAMETER:

1. When O/P is short circuited i.e. V2 = 0

   $h_{11} = V_1/I_1$ $h_{21} = I_2/I_1$

2. When I/P is open circuited i.e. II = 0

   $h_{12} = V_1/V_2$ $h_{22} = I_2/V_2$

Review questions

1. What do you mean by a port?
2. What are two port networks?
3. What are the parameters of a two port network?
4. What is purpose of calculating two port network parameters?

RESULT:

Thus the various parameters of the two port network has been calculated and verified.

Ex. No	SIMULATION OF THREE PHASE BALANCED AND UNBALANCED STAR, DELTA NETWORKS CIRCUITS
Date:	

Aim

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

Apparatus required

S. No	Name of the apparatus	Type	Range	Qty

Theory

A three-phase network can be seen as a special connection of three single phase or simple AC circuits. Three-phase networks consist of three simple networks, each having the same amplitude and frequency, and a 120° phase difference between adjacent networks.

A three-phase system may be arranged in delta (Δ) or star (Y) (also denoted as wye in some areas). A wye system allows the use of two different voltages from all three phases, such as a 230/400V system which provides 230V between the neutral and any one of the phases, and 400V across any two phases.

Balanced loads

Generally, in electric power systems, the loads are distributed as evenly as is practical between the phases

For all phases and the instantaneous currents are

$$I_{L1} = I_P \sin(\theta - \varphi)$$
$$I_{L2} = I_P \sin\left(\theta - \frac{2}{3}\pi - \varphi\right)$$

$$I_{L3} = I_P \sin\left(\theta - \frac{4}{3}\pi - \varphi\right)$$

Unbalanced systems

The analysis of unbalanced cases is greatly simplified by the use of the techniques of symmetrical components. An unbalanced system is analyzed as the superposition of three balanced systems, each with the positive, negative or zero sequence of balanced voltages.

When specifying wiring sizes in a three-phase system, we only need to know the magnitude of the phase and neutral currents. The neutral current can be determined by adding the three phase currents together as complex numbers and then converting from rectangular to polar co-ordinates. If the three phase RMS (Root Mean Square) currents are I_{L1}, $I_{L2}$and I_{L3}, the neutral RMS current is:

$$I_{L1} + I_{L2} * \cos\frac{2}{3}\pi + j * I_{L2} * \sin\frac{2}{3}\pi + I_{L3} * \cos\frac{4}{3}\pi + j * I_{L3} * \sin\frac{4}{3}\pi$$

Which resolves to

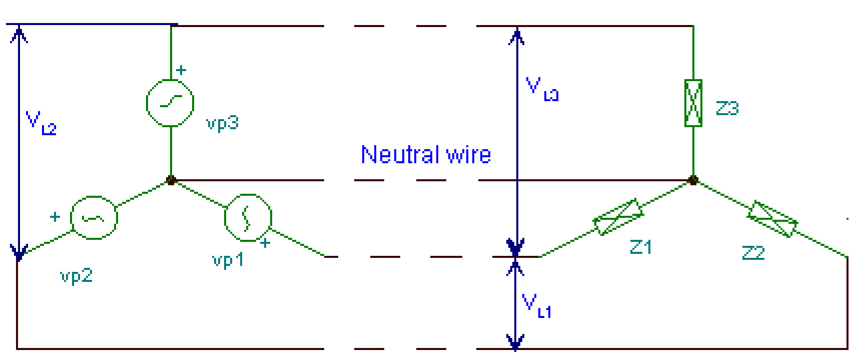
$$I_{L1} - I_{L2} * 0.5 - I_{L3} * 0.5 + j * \frac{\sqrt{3}}{2} * (I_{L2} - I_{L3})$$

The polar magnitude of this is the square root of the sum of the squares of the real and imaginary parts, which reduces to

$$\sqrt{I_{L1}^2 + I_{L2}^2 + I_{L3}^2 - I_{L1} * I_{L2} - I_{L1} * I_{L3} - I_{L2} * I_{L3}}$$

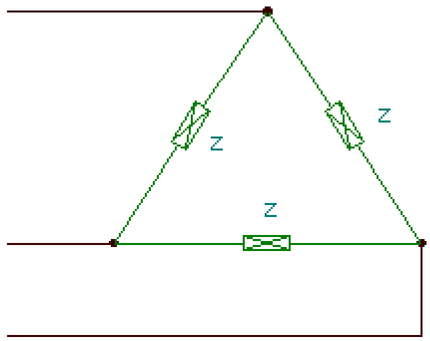
Three phase Star Connected network

The Wye or Y-connection, where the negative terminals of each generator or load are connected to form the neutral terminal. This results in a three-wire system, or if a neutral wire is provided, a four-wire system.



Three phase Delta Connected network

The delta or D-connection of three phases is achieved by connecting the three loads in series forming a closed loop. This is only used for three-wire systems



Circuit diagram

Output Waveforms

Procedure

1. Open a new model in MATLAB simulink
2. Draw the circuit model as given.
3. Run the simulation.
4. Make the observations from the output waveforms.
5. Compare the values with the practical calculations.

Review questions

1. What is three phase system?
2. What are the types of connections in three phase system?
3. Define star and delta connection schemes.
4. What is the difference between balanced and unbalanced networks?

Result

Ex. No	STUDY OF CRO AND MEASUREMENT OF SINUSOIDAL VOLTAGE, FREQUENCY AND POWER FACTOR
Date:	

Objective

- To introduce the basic structure of a cathode-ray Oscilloscope.
- To get familiar with the use of different control switches of the device.
- To visualize an ac signal, measure the amplitude and the frequency

Theory

Cathode-ray Oscilloscope

Fluorescent screen (see Figure 1). When the cathode is heated (by Theory Cathode-ray Oscilloscope applying a smallpotential difference across its terminals), it emits electrons. Having a potential difference between the cathode and theanode (electrodes), accelerate the emitted electrons towards the anode, forming an electron beam, which passes to fallon the screen. When the fast electron beam strikes the fluorescent screen, a bright visible spot is produced. The grid,which is situated between the electrodes, controls the amount of electrons passing through it thereby controlling theintensity of the electron beam. The X&Y-plates, are responsible for deflecting the electron beam horizontally andvertically.

A sweep generator is connected to the X-plates, which moves the bright spot horizontally across the screenand repeats that at a certain frequency as the source of the signal. The voltage to be studied is applied to the Y-plates.The combined sweep and Y voltages produce a graph showing the variation of voltage with time, as shown in Fig. 2.

Alternating current (ac)

An ac signal can be of different forms: sinusoidal, square, or triangular. The sinusoidal is the most popular
type, which is the natural output of the rotary electricity generators. An ac voltage source can be represented by

$$\varepsilon(t) = \varepsilon_m \sin(wt + \phi)$$

whereεm is the maximum output voltage value, ω =2π*f* (*f* is the frequency), and φ is the phase shift.

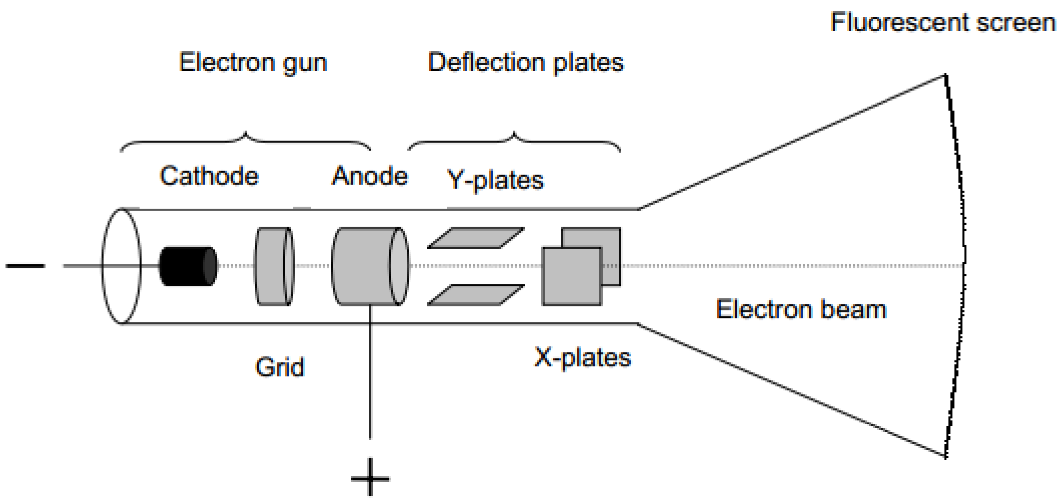


Fig.1.Basic structure of CRO

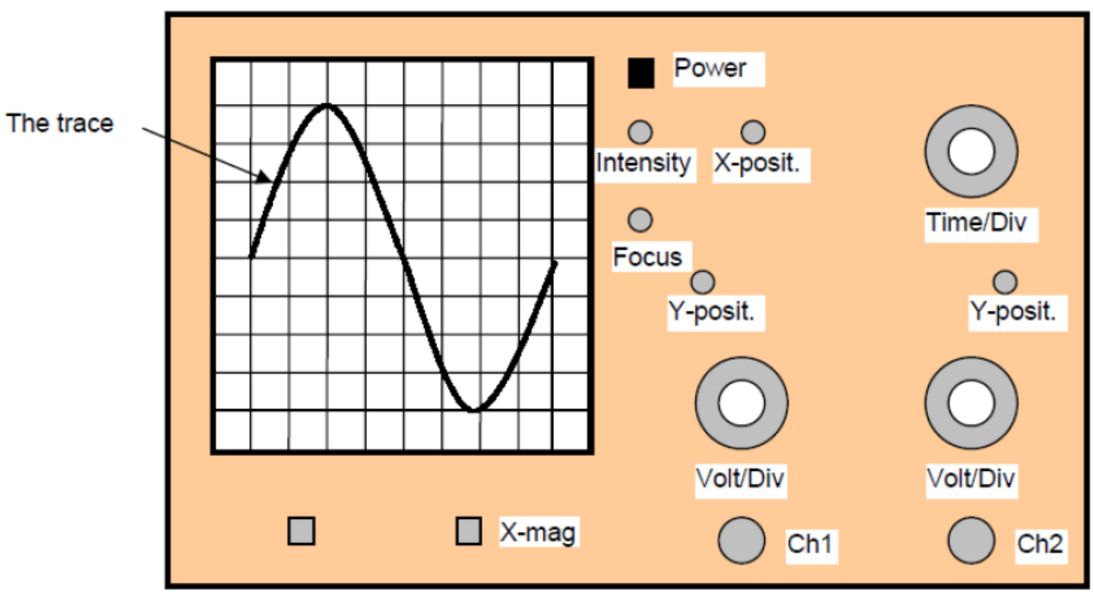


Fig.2. Front Panel of CRO\

Block diagram

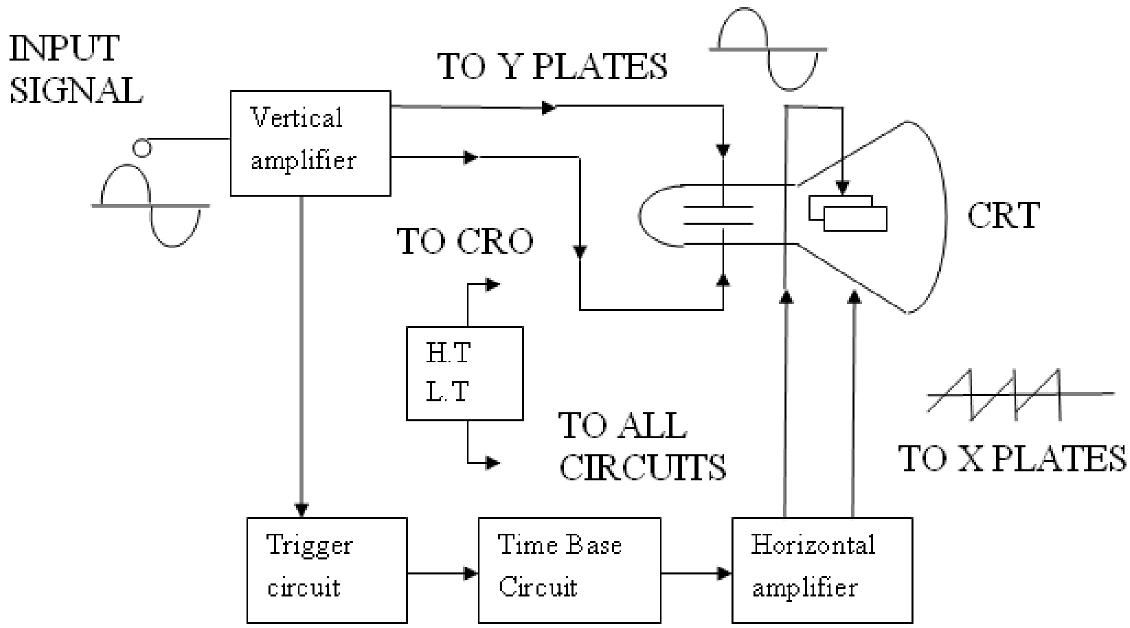


Table 1

Frequency (Hz)	**Period (T)Sec**	**F(Hz)**	**V_{p-p}(V)**	**V_{rms}(V)**
200				
X				
1000				
Y				
2000				

Vrms(multimeter)=

PROCEDURE

PART ONE

1. Turn on the Oscilloscope, wait a couple of seconds to warm up, then the trace will show up on the screen.
2. Adjust the intensity and the focus of the trace.
3. Use the X &Y-post. knobs to center the trace horizontally and vertically.
4. Connect a cable to Ch1 socket.
5. Turn on the Heath kit.
6. Connect the cable from Ch1 of the CRO to the SIN connector of the Heathkit, via a piece of wire.
7. A signal will appear on the screen.
8. Make sure that the inner red knobs of the Volt/Div and the Time/Div are locked clockwise.
9. Set the frequency of the generator to 200 Hz.
10. Adjust the Volt/Div and the Time/Div knobs so that you get a suitable size signal (from 1-2 wavelengths filling most of the screen vertically).
11. Count the number of vertical squares lying within the signal, then calculate the peak
    to peak value as:
    Vp-p= No. vertical Div x Volt/Div
12. Calculate Vrms value, record in Table I:
    Vrms= Vp-p / 2.sqr root(2)
13. Measure Vrms using the multimeter (connect the probes of the multimeter to the SIN
    and the GND connectors).
14. Calculate the period T, record in Table I:
    T = No. horizontal Div. × Time/Div
15. Calculate the frequency, *f*=1/T, record in the table.
16. Repeat steps 10-14 for the frequency values as in the table

Part two

1. Connect the cable from Ch1 to the upper connector of the line frequency of the Heathkit.
2. Adjust the Volt/Div and the Time/Div knobs so that you get a suitable size signal (from 1-2 wavelengths filling most of the screen vertically).
3. Calculate the peak to peak voltage value.
4. Calculate Vrms value.
5. Measure *Vrms*using the multimeter.
6. Measure the period *T*, then calculate the frequency.

Vp-p=

Vrms=

Vrms(multimeter)=
T=
f=

Review questions
1. What is the purpose of the grid, and X&Y-plates?
2. For a certain ac input signal, if the Volt/Div knob is set to alower value, what effect does this have on the size of the signal on the screen?
3. The X-mag button magnifies the signal horizontally; is this button used for high or low frequency signal? Why?
4. What is the physical meaning of the root-mean-square value of an ac signal?

Result

Thus the CRO basic structure, measurement of voltage and frequency was studied.