SRM VALLIAMMAI ENGINEERING COLLEGE
(An Autonomous Institution)

SRM Nagar, Kattankulathur – 603 203

DEPARTMENT OF
ELECTRONICS AND INSTRUMENTATION ENGINEERING

Lab Manual

III SEMESTER

1907305 CIRCUITS AND DEVICES LABORATORY

Regulation – 2019
Academic Year 2022-23 (ODD Semester)

Prepared by

N.Sowrirajan, Assistant Professor/ EIE

SYLLABUS

1907305 CIRCUITS AND DEVICES LABORATORY L T P C

0 0 4 2

COURSE OBJECTIVES

1. To simulate various electric circuits using Pspice/ Matlab/e-Sim / Scilab.
2. To gain practical experience on electric circuits and verification of theorems.
3. To facilitate the students to study the characteristics of various semiconductor devices.
4. To be exposed to the characteristics of basic electronic devices.
5. To provide practical knowledge on the analysis of regulators, amplifiers and oscillators

LIST OF EXPERIMENTS FOR CIRCUITS LAB

1. Simulation and experimental solving of electrical circuit problems using Kirchhoff's voltage and current laws.
2. Simulation and experimental solving of electrical circuit problems using Thevenin's theorem.
3. Simulation and experimental solving of electrical circuit problems using Norton's theorem.
4. Simulation and experimental solving of electrical circuit problems using Superposition theorem.
5. Simulation and experimental verification of Maximum Power transfer Theorem.
6. Simulation and Experimental validation of R-C electric circuit transience.
7. Simulation and Experimental validation of frequency response of RLC electric circuit.
8. Design and Simulation of series resonance circuit.

Minimum of five experiments to be offered from the list. Additional one or two experiments can be framed beyond the list or curriculum

LIST OF EXPERIMENTS FOR DEVICES LAB

1. Simulation and experimental Characteristion of Semiconductor diode and Zener diode.
2. Simulation and experimental Characteristion of a NPN Transistor under common emitter configurations.
3. Simulation and experimental Characterisation of FET and JFET(Draw the

equivalent circuit)

4. Simulation and experimental Characterisation of RC and LC phase shift oscillators.
5. Simulation and experimental Characterisation of Monostable and Astable multivibrators.
6. Simulation of Single Phase half-wave and full wave rectifiers with inductive and capacitive filters.
7. Characteristics of SCR and application as a controlled rectifier.

Minimum of five experiments to be offered from the list. Additional one or two experiments can be framed beyond the list or curriculum.

TOTAL : 60 PERIODS

COURSE OUTCOMES

1. Ability to analyse electrical circuits
2. Ability to apply circuit theorems
3. Ability to analyse transients.
4. Gain knowledge on the proper usage of various electronic equipment and simulation tools for design and analysis of electronic circuits.
5. Get hands-on experience in studying the characteristics of semiconductor devices.
6. Ability to analyze various electronic circuits such as voltage regulators, transistor amplifiers and oscillators

CO - PO and CO - PSO MAPPING:

CO	PO												PSO			
	1	2	3	4	5	6	7	8	9	10	11	12	1	2	3	4
CO 1	3	2										3	3			
CO 2	3		2												2	
CO 3			3	2										1	2	
CO 4				2								1	2		3	
CO 5				2								2	3		2	
CO 6				2								1	2		2	2

LIST OF EXPERIMENTS

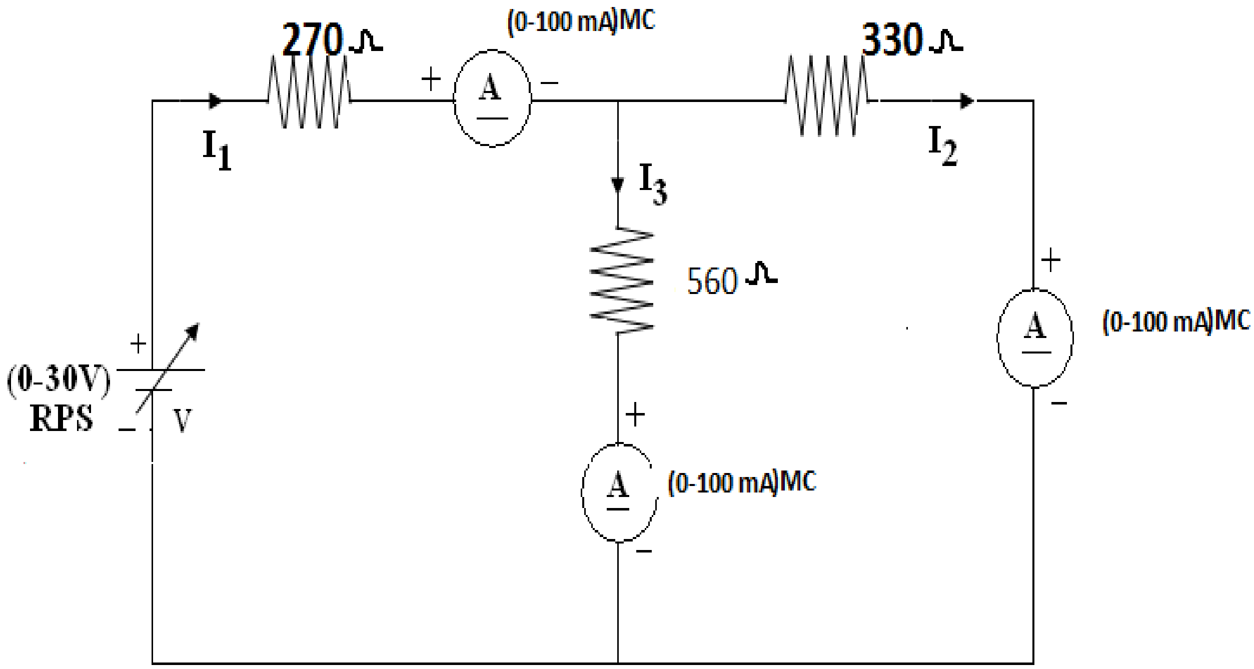
Exp. No	Experiments
CIRCUITS LAB (CYCLE-I)	
1	Simulation and experimental solving of electrical circuit problems using Kirchhoff's voltage and current laws.
2	Simulation and experimental solving of electrical circuit problems using Thevenin's theorem.
3	Simulation and experimental solving of electrical circuit problems using Norton's theorem.
4	Simulation and experimental solving of electrical circuit problems using Superposition theorem.
5	Simulation and experimental verification of Maximum Power transfer Theorem.
6	Simulation and Experimental validation of R-C electric circuit transience.
7	Simulation and Experimental validation of frequency response of RLC electric circuit. .
8	Design and Simulation of series resonance circuit.
DEVICES LAB (CYCLE-II)	
1	Simulation and experimental Characterization of Semiconductor diode and Zener diode.
2	Simulation and experimental Characterization of a NPN Transistor under common emitter configurations.
3	Simulation and experimental Characterization of FET and JFET
4	Simulation of Single Phase half-wave and full wave rectifiers with inductive and capacitive filters.
5	Simulation of passive filters
6	Simulation and experimental Characterization of RC and LC phase shift oscillators.
7	Simulation and experimental Characterization of Monostable and Astable multivibrators.
8	Characteristics of SCR and application as a controlled rectifier.
9	Simulation and experimental Characterization of UJT and generation of saw tooth Waveforms
Additional Experiment	
1	**CLIPPER AND CLAMPER CIRCUITS**

1907305 CIRCUITS AND DEVICES LABORATORY

CYCLE -I

CIRCUITS LAB

CIRCUIT DIAGRAM FOR KIRCHHOFF'S CURRENT LAW



OBSERVATION TABLE

S.No	V (Volts)	I_1 (mA)	I_2 (mA)	I_3 (mA)	$I_1 = I_2 + I_3$ (mA)

EXP.NO:

DATE:

SIMULATION AND EXPERIMENTAL VERIFICATION OF ELECTRICAL CIRCUIT PROBLEMS USING KIRCHHOFF'S VOLTAGE AND CURRENT LAWS

AIM:

To verify (i) Kirchhoff's current law (ii) Kirchhoff's voltage law

APPARATUS REQUIRED:

S.No	Name of the apparatus	Range	Type	Quantity
1	RPS			
2	Resistor			
3	Ammeter			
4	Voltmeter			
5	Bread board			
6	Connecting wires			

SOFTWARE REQUIRED:

Matlab 7.1

KIRCHHOFF'S CURRENT LAW:

THEORY:

The law states, "The sum of the currents entering a node is equal to sum of the currents leaving the same node". Alternatively, the algebraic sum of currents at a node is equal to zero.

THEORETICAL CALCULATION

S.No.	V (Volts)	I_1 (mA)	I_2 (mA)	I_3 (mA)	$I_1 = I_2 + I_3$ (mA)

MODEL CALCULATION:

The term node means a common point where the different elements are connected. Assume negative sign for leaving current and positive sign for entering current.

PROCEDURE:

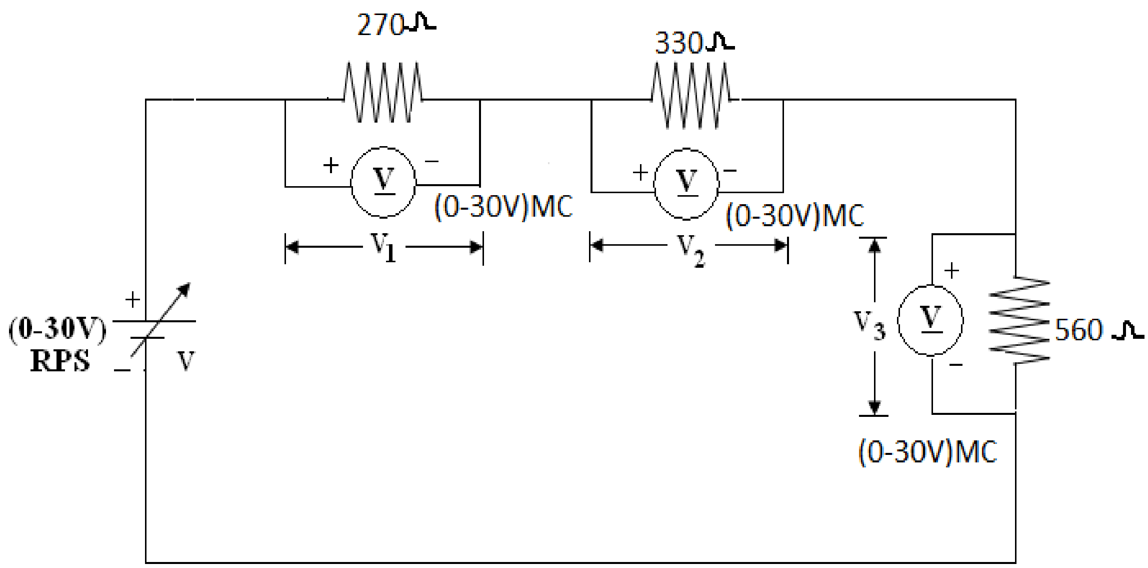
1. Connect the circuit as per the circuit diagram.
2. Switch on the supply.
3. Set different values of voltages in the RPS.
4. Measure the corresponding values of branch currents I_1, I_2 and I_3.
5. Enter the readings in the tabular column.
6. Find the theoretical values and compare with the practical values

FORMULA:

$\sum$ Currents entering a node = $\sum$ Currents leaving the node

$$I_1 = I_2 + I_3$$

CIRCUIT DIAGRAM FOR KIRCHHOFF'S VOLTAGE LAW:



OBSERVATION TABLE:

S.No.	V Volts	V_1 Volts	V_2 Volts	V_3 Volts	$V = V_1 + V_2 + V_3$ Volts

KIRCHHOFF'S VOLTAGE LAW:

THEORY:

The law states, "The algebraic sum of the voltages in a closed circuit/mesh is zero".

The voltage rise is taken as positive and the voltage drop is taken as negative.

PROCEDURE:

1. Connect the circuit as per the circuit diagram.
2. Switch on the supply.
3. Set different values of voltages in the RPS.
4. Measure the corresponding values of voltages (V_1, V_2 and V_3) across resistors R_1, R_2 and R_3 respectively.
5. Enter the readings in the tabular column.
6. Find the theoretical values and compare with the practical values.

FORMULA:

$$\sum \text{Voltages in a closed loop} = 0$$

$$V - V_1 - V_2 - V_3 = 0$$

THEORETICAL CALCULATION:

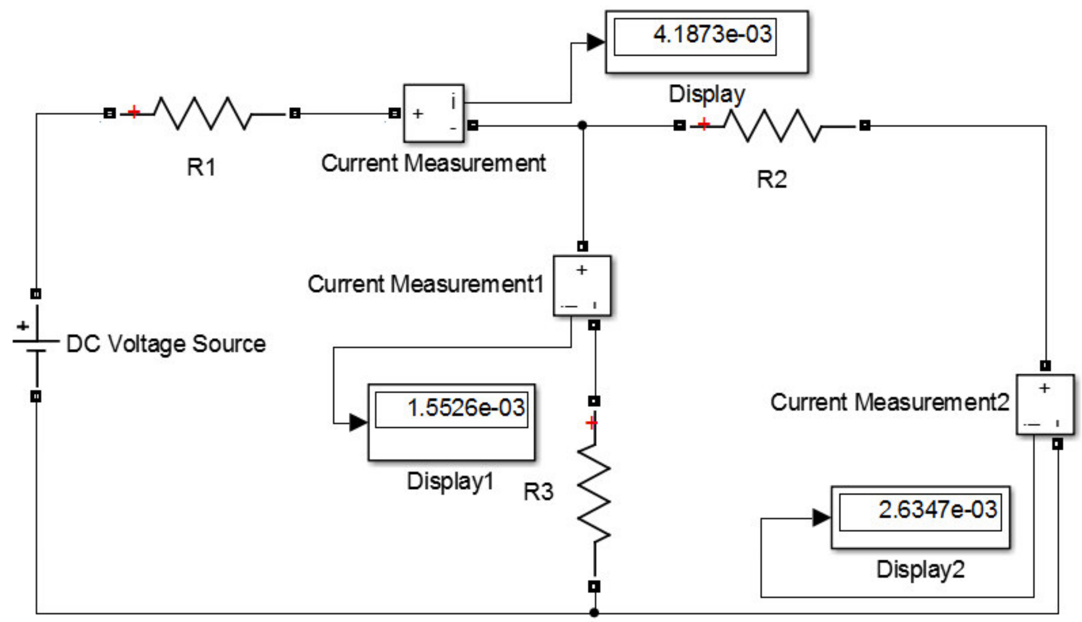
S.No.	V Volts	V_1 Volts	V_2 Volts	V_2 Volts	$V = V_1 + V_2 + V_3$ Volts

MODEL CALCULATION:

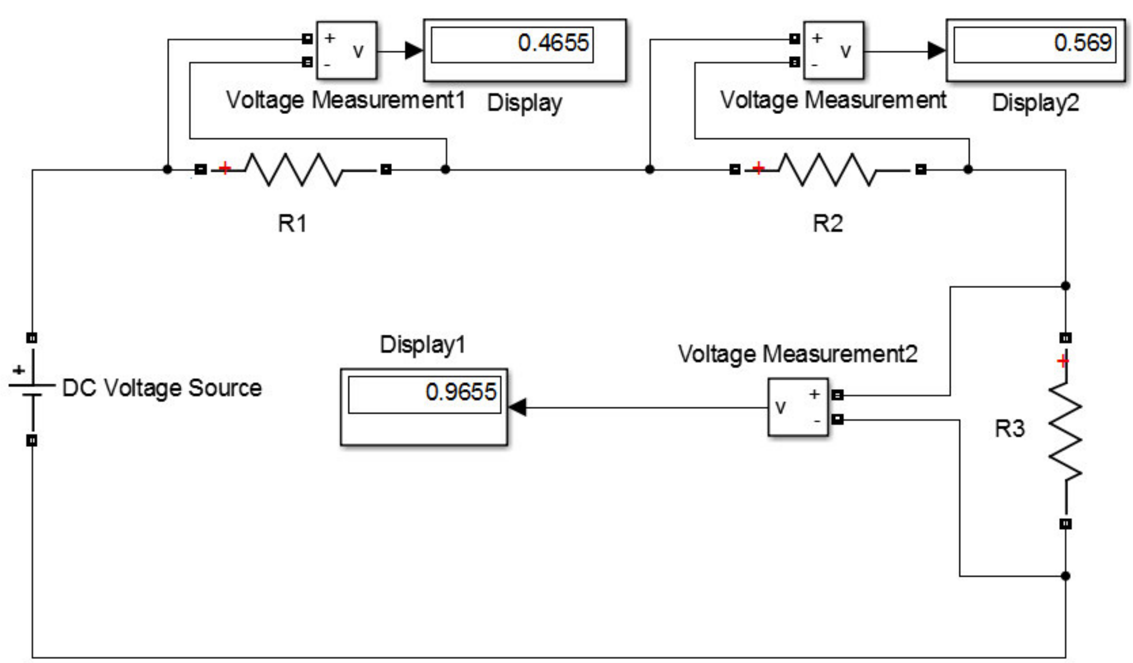
SIMULATION PROCEDURE:

1. Open a new MATLAB/SIMULINK model
2. Connect the circuit as shown in the figure
3. Debug and run the circuit
4. For different input voltages, record the current and voltages and verify with theoretical values.

SIMULATION CIRCUIT DIAGRAM FOR KIRCHHOFF'S CURRENT LAW:



SIMULATION DIAGRAM FOR KIRCHHOFF'S VOLTAGE LAW:



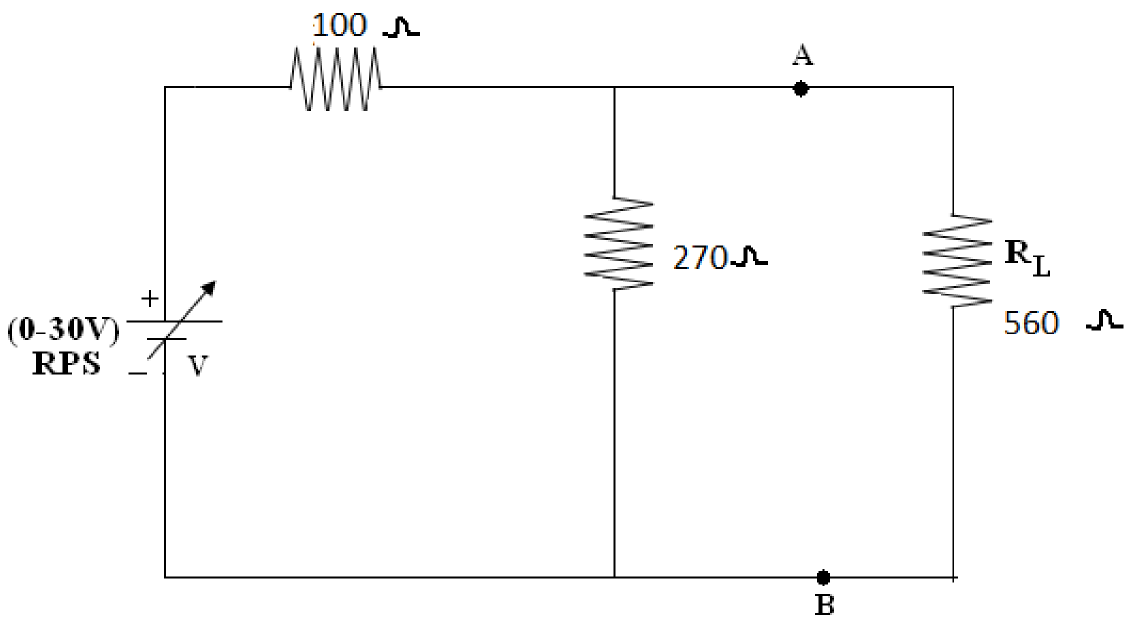
VIVA QUESTIONS:

1. State Kirchhoff's Voltage Law.
2. State Kirchhoff's Current Law.
3. What is current division rule?
4. What is voltage division rule?
5. Give the equivalent resistance when 'n' number of resistances is connected in series.
6. Give the equivalent resistance when 'n' number of resistances is connected in parallel

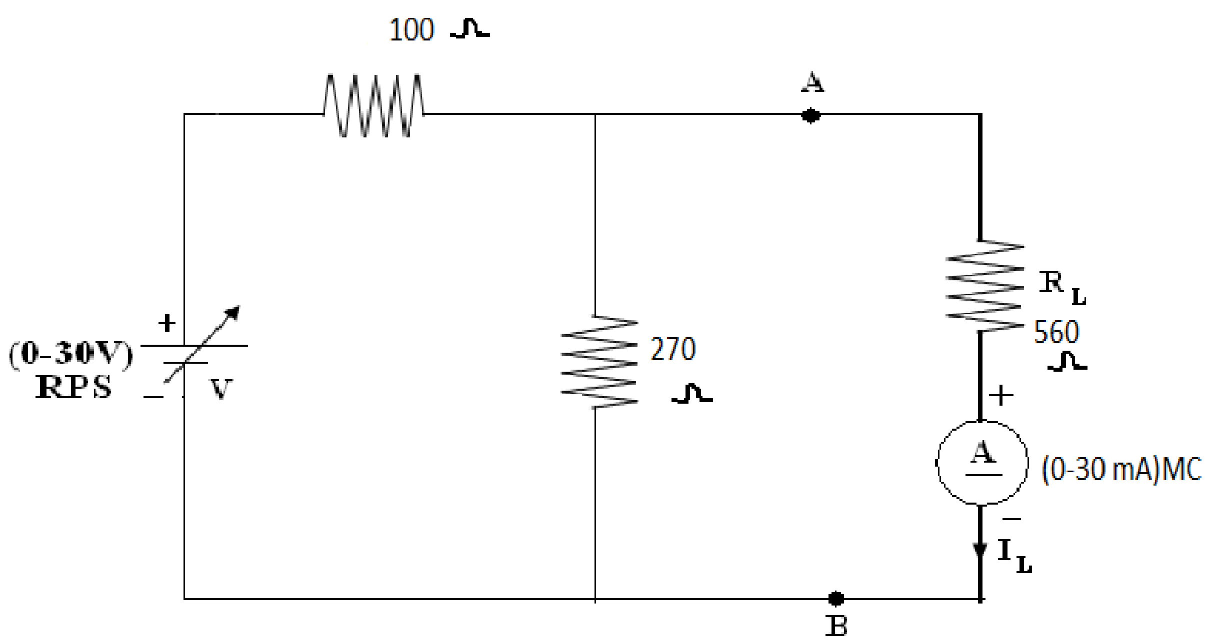
RESULT:

Thus the Kirchhoff's Current and Voltage laws are verified.

CIRCUIT DIAGRAM FOR THEVENIN'S THEOREM:



TO FIND LOAD CURRENT:



EXP.NO:

DATE:

SIMULATION AND EXPERIMENTAL VERIFICATION OF ELECTRICAL CIRCUIT PROBLEMS USING THEVENIN'S THEOREM

AIM:

To verify Thevenin's theorem.

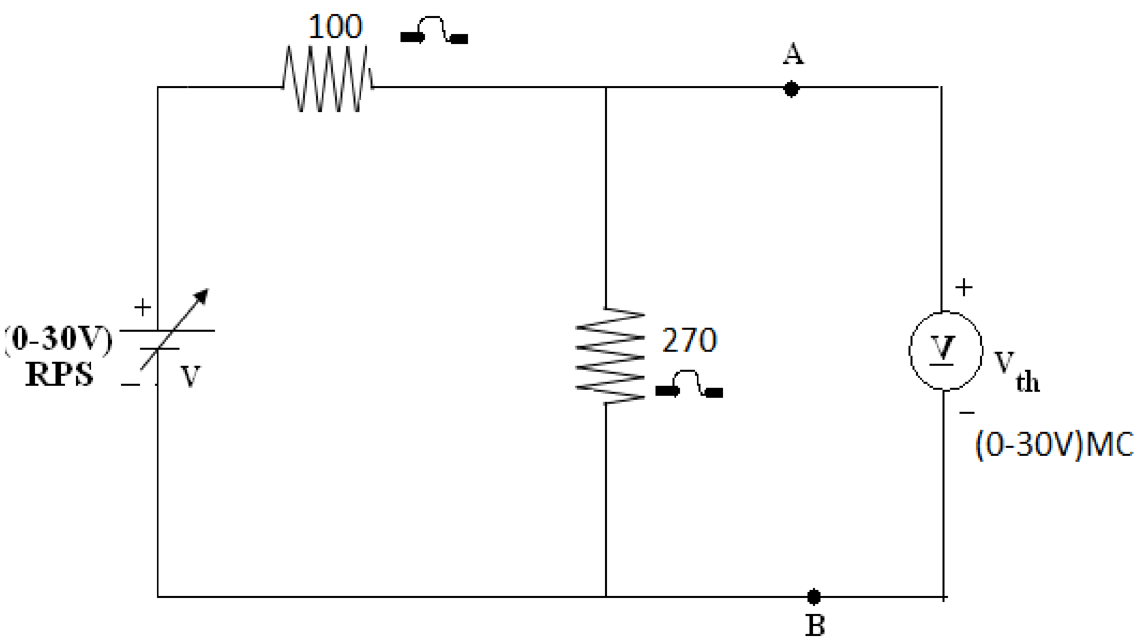
APPARATUS REQUIRED:

S.No no	Name of the Components / Equipment	Type/Range	Quantity required
1	Resistor		
2	Dc power supply		
3	Voltmeter		
4	Ammeter		
5	Wires		
6	Bread board		

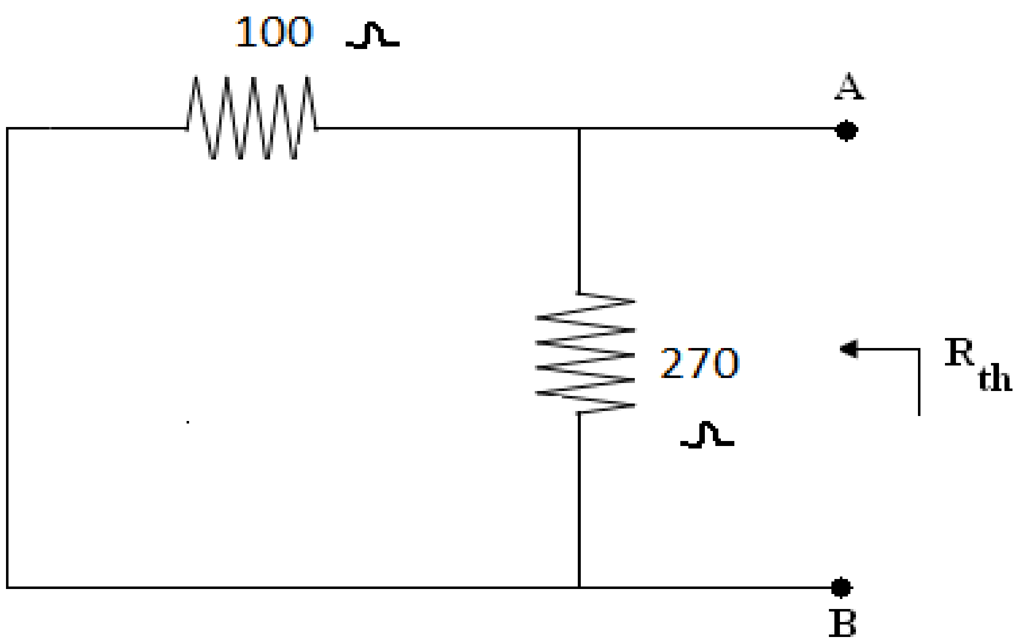
SOFTWARE REQUIRED:

Matlab 7.1

TO FIND V_{th}:



TO FIND R_{th}:

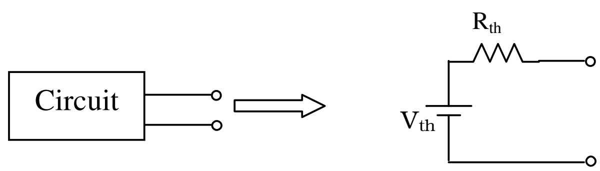


THEVENIN'S THEOREM:

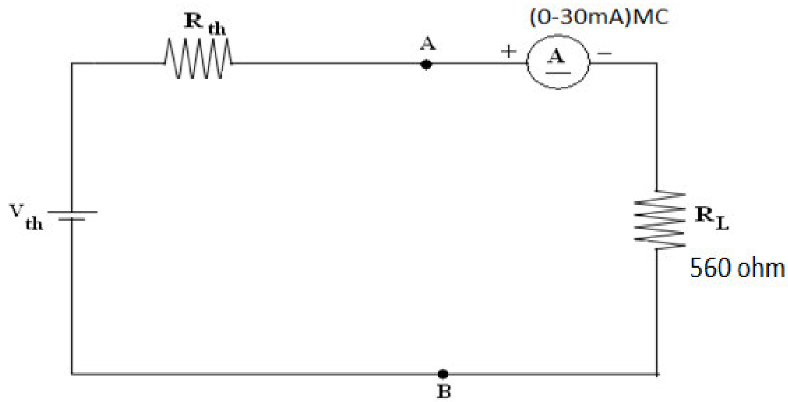
STATEMENT:

Any two-terminal linear network, composed of voltage sources, current sources, and resistors,

can be replaced by an equivalent two-terminal network consisting of an independent voltage source in series with a resistor. The value of voltage source is equivalent to the open circuit voltage (V_{th}) across two terminals of the network and the resistance is equal to the equivalent resistance (R_{th}) measured between the terminals with all energy sources replaced by their internal resistances.



THEVENIN'S EQUIVALENT CIRCUIT



OBSERVATION TABLE

S. No	V_{dc}	V_{th} (Volts)		R_{th} (Ω)		Current through Load Resistance I_L(mA)	
		Practical Value	Theoretical Value	Practical Value	Theoretical Value	Practical Value	Theoretical Value

PROCEDURE:

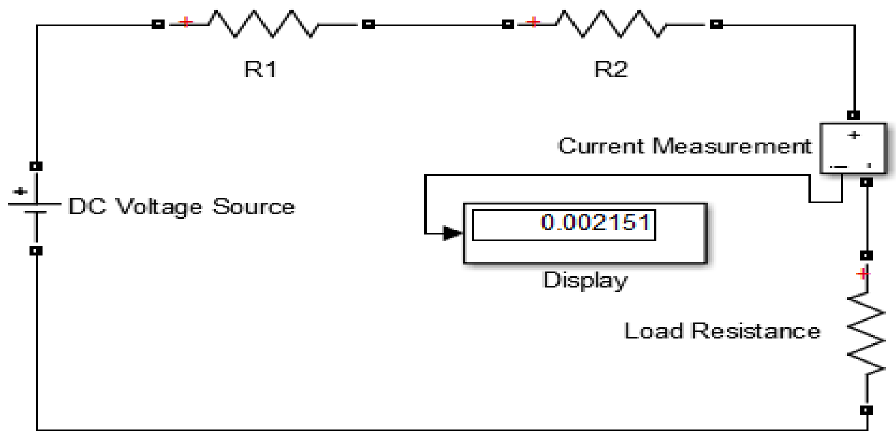
1. Give connections as per the circuit diagram.
2. Measure the current through R_L in the ammeter.
3. Open circuit the output terminals by disconnecting load resistance R_L.
4. Connect a voltmeter across AB and measure the open circuit voltage V_{th}.
5. To find R_{th}, replace the voltage source by short circuit.
6. Give connections as per the Thevenin's Equivalent circuit.
7. Measure the current through load resistance in Thevenin's Equivalent circuit.
8. Verify Thevenin's theorem by comparing the measured currents in Thevenin's Equivalent circuit with the values calculated theoretically.

SIMULATION PROCEDURE:

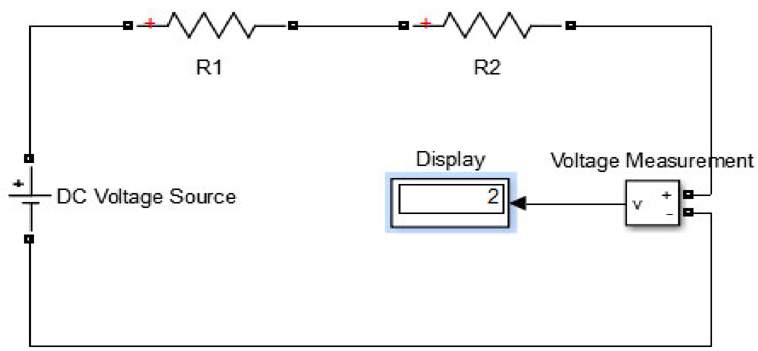
1. Open a new MATLAB/SIMULINK model
2. Connect the circuit as shown in the figure
3. Debug and run the circuit
4. For different input voltages, record the current and voltages and verify with theoretical values.

SIMULATION:

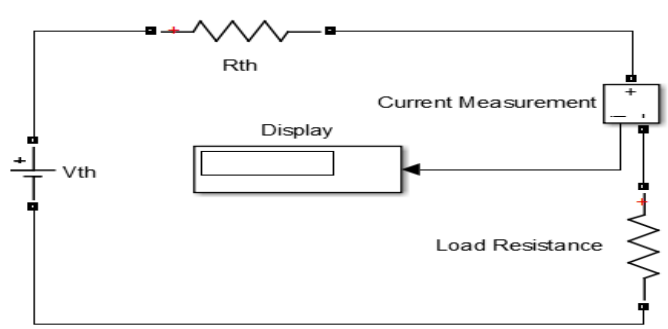
TO FIND LOAD CURRENT:



TO FIND V_{th}:



THEVENIN'S EQUIVALENT CIRCUIT:



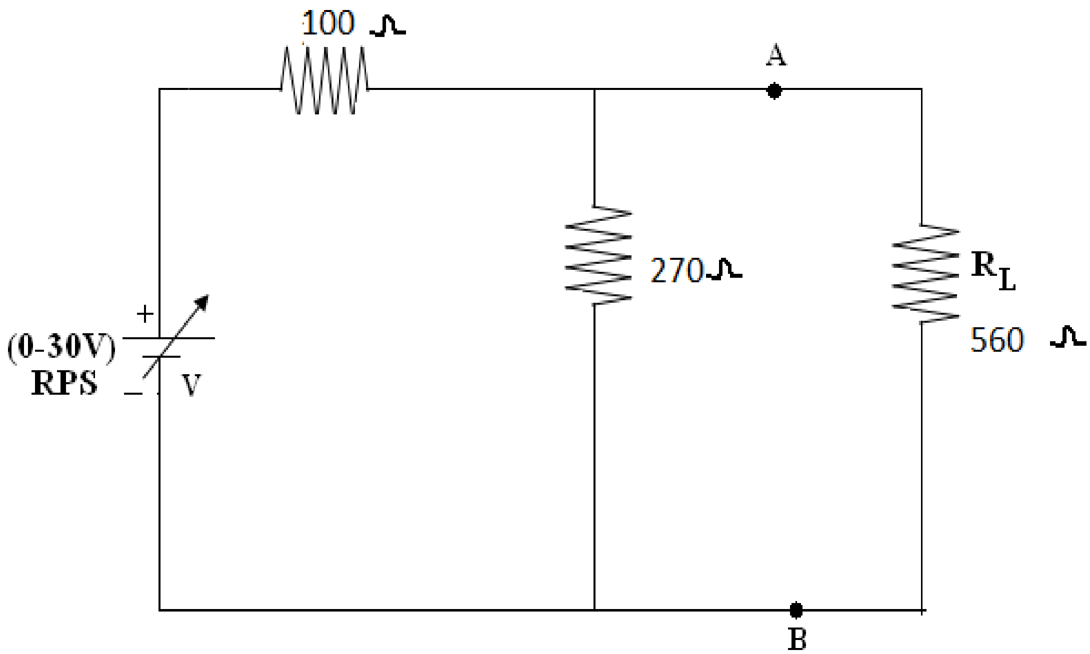
VIVA QUESTIONS:

1. What is meant by a linear network?
2. State Thevenin's Theorem.
3. How do you calculate thevenin's resistance?

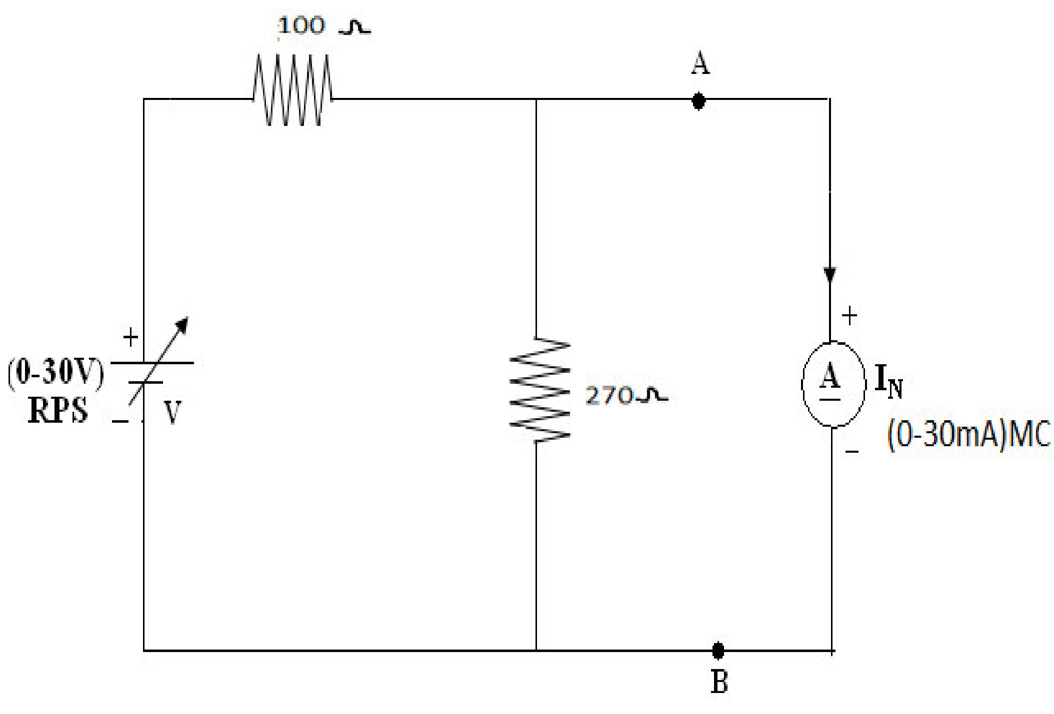
RESULT:

Thus the Thevenin's theorem was verified.

CIRCUIT DIAGRAM FOR NORTON'S THEOREM:



TO FIND NORTON'S CURRENT:



EXP.NO:

DATE:

SIMULATION AND EXPERIMENTAL VERIFICATION OF ELECTRICAL CIRCUIT PROBLEMS USING NORTON'S THEOREM

AIM:

To verify Norton's theorem.

APPARATUS REQUIRED:

S.No no	Name of the Components / Equipment	Type/Range	Quantity required
1	Resistor		
2	Dc power supply		
3	Voltmeter		
4	Ammeter		
5	Wires		
6	Bread board		

SOFTWARE REQUIRED:

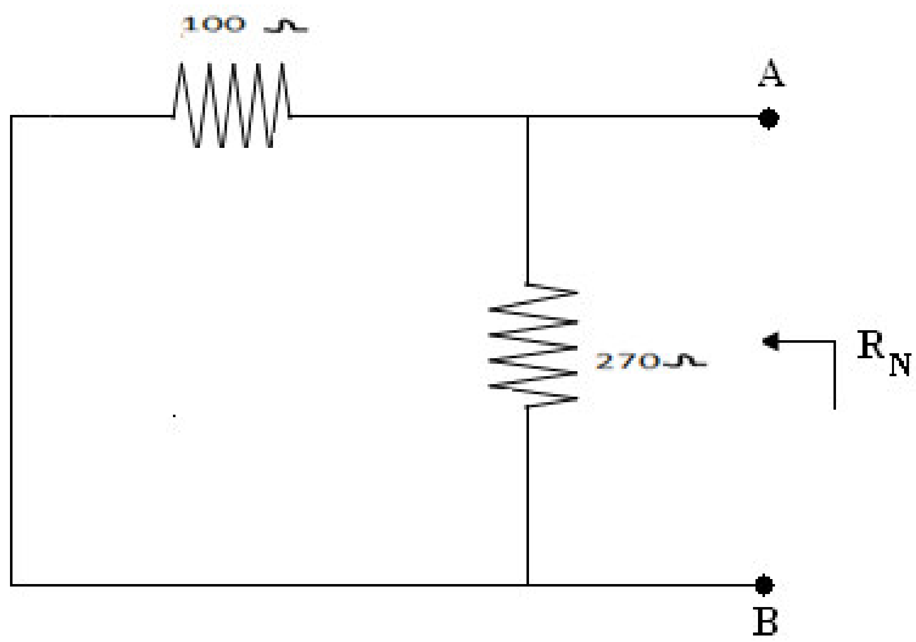
Matlab 7.1

NORTON'S THEOREM

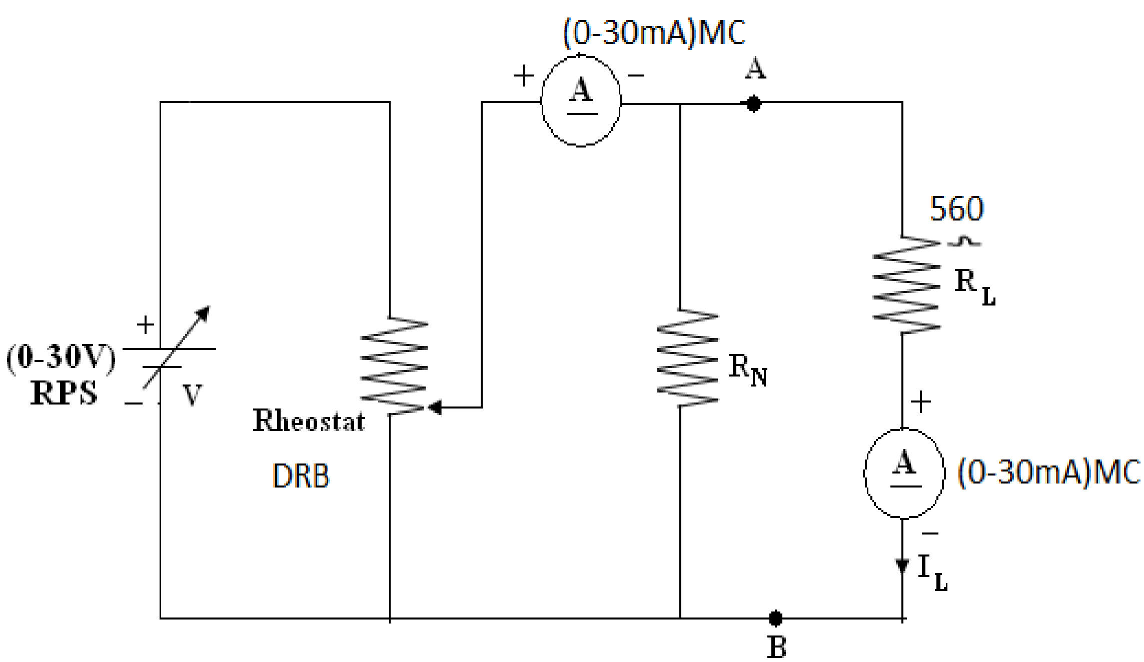
STATEMENT:

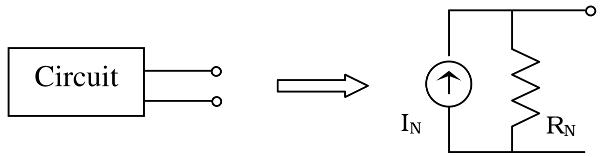
Any two-terminal linear network, composed of voltage sources, current sources, and resistors, can be replaced by an equivalent two-terminal network consisting of an independent current source in parallel with a resistor. The value of the current source is the short circuit current (I_N) between the two terminals of the network and the resistance is equal to the equivalent resistance (R_N) measured between the terminals with all energy sources replaced by their internal resistances.

TO FIND NORTON'S RESISTANCE:



NORTON'S EQUIVALENT CIRCUIT:





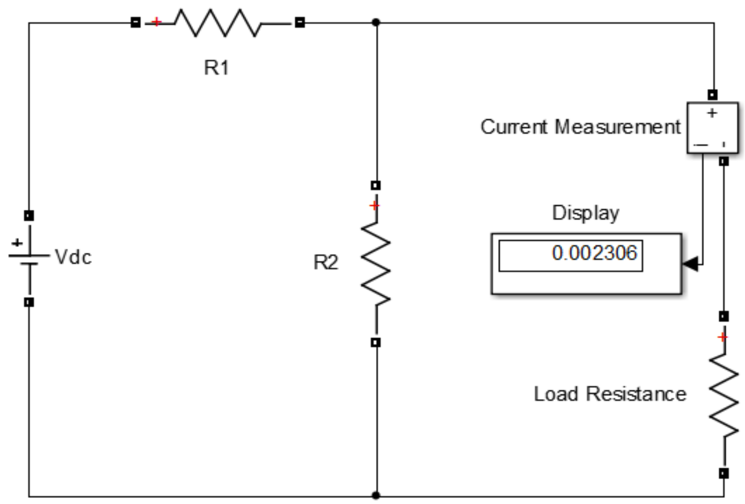
PROCEDURE:

1. Give connections as per the circuit diagram.
2. Measure the current through R_L in ammeter.
3. Short circuit A and B through an ammeter.
4. Measure the Norton current in the ammeter.
5. Find out the Norton's Resistance viewed from the output terminals.
6. Give connections as per the Norton's Equivalent circuit.
7. Measure the current through R_L.
8. Verify Norton's theorem by comparing currents in R_L directly and that obtained with the equivalent circuit.

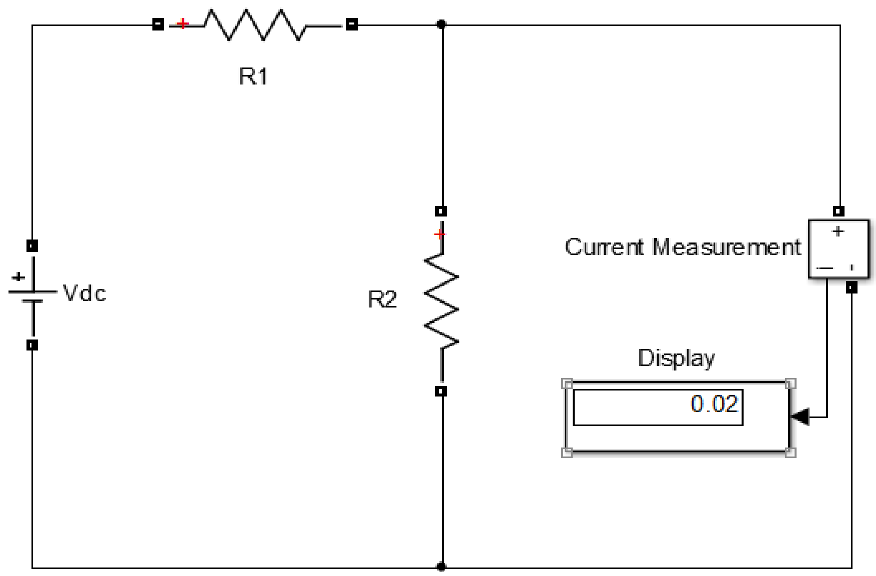
SIMULATION PROCEDURE:

1. Open a new MATLAB/SIMULINK model
2. Connect the circuit as shown in the figure
3. Debug and run the circuit
4. For different input voltages, record the current and voltages and verify with theoretical values.

TO FIND LOAD CURRENT:



TO FIND NORTON'S CURRENT:



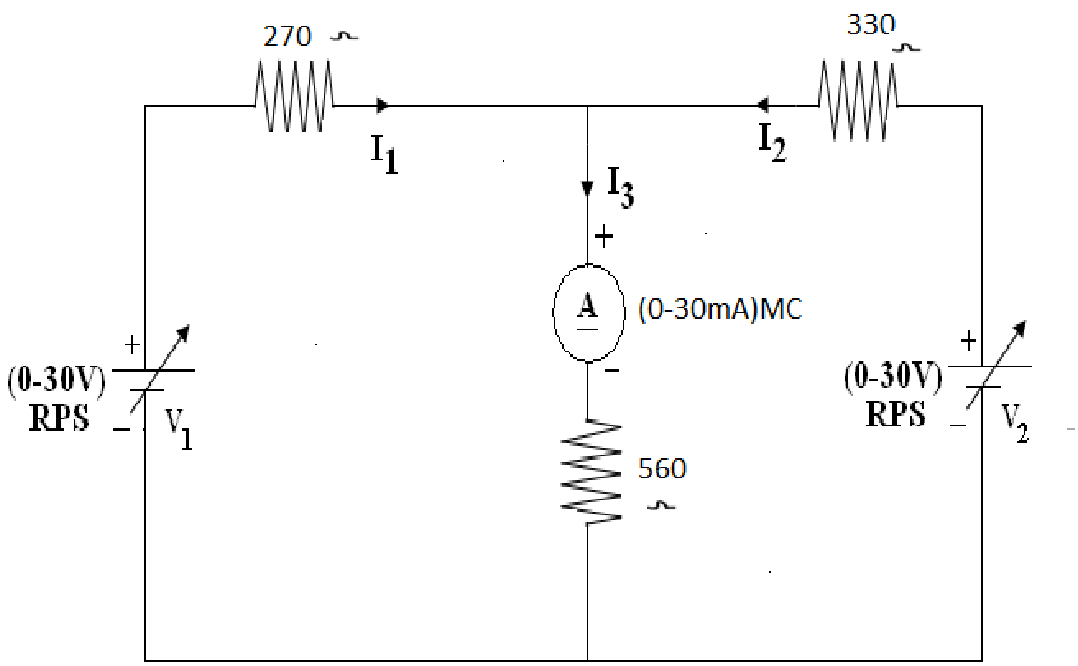
VIVA QUESTIONS:

1. How do you calculate Norton’s resistance?
2. State Norton’s Theorem.
3. Give the usefulness of Norton’s theorems.

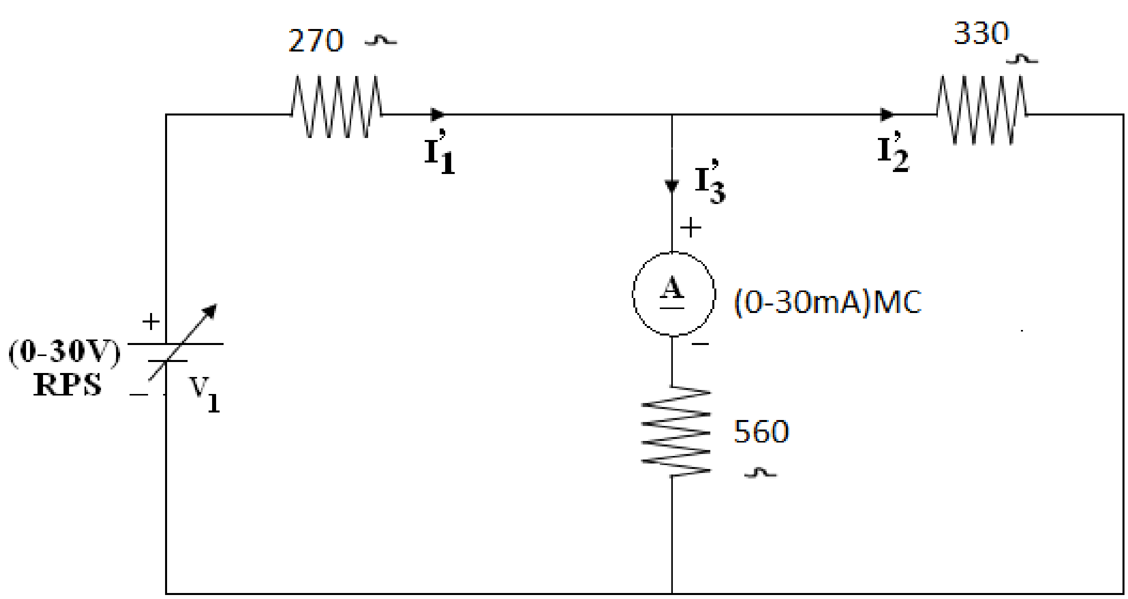
RESULT:

Thus the Norton’s theorem was verified.

CIRCUIT DIAGRAM FOR SUPERPOSITION THEOREM:



CIRCUIT DIAGRAM WITH V_1 ACTING INDEPENDENTLY:



EXP.NO:

DATE:

SIMULATION AND EXPERIMENTAL VERIFICATION OF ELECTRICAL CIRCUIT PROBLEMS USING SUPERPOSITION THEOREM

AIM:

To verify superposition theorem.

APPARATUS REQUIRED:

S.No	Name of the Components / Equipment	Type/Range	Quantity required
1	Resistor		
2	Dc power supply		
3	Voltmeter		
4	Ammeter		
5	Wires		
6	Bread board		

SOFTWARE REQUIRED:

Matlab 7.1

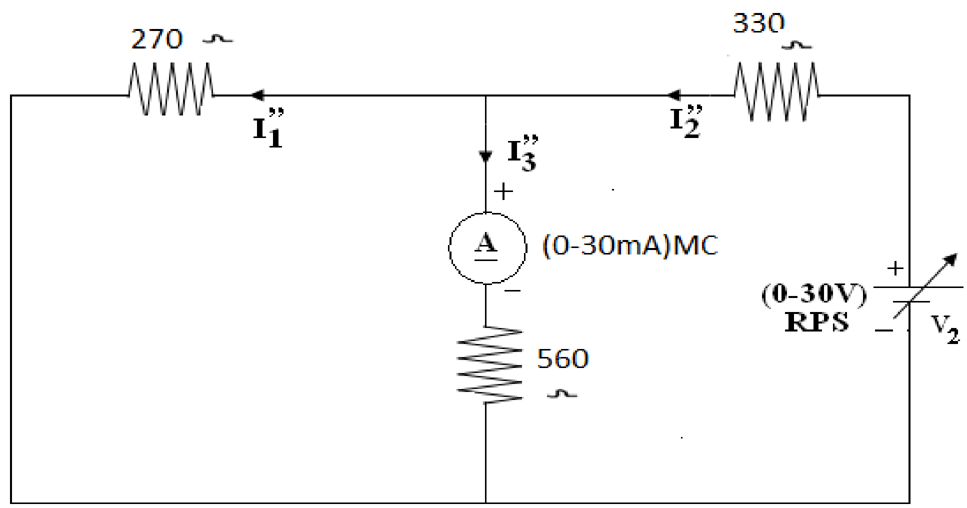
SUPERPOSITION THEOREM:

STATEMENT:

In any linear, bilateral network energized by two or more sources, the total response is equal to the algebraic sum of the responses caused by individual sources acting alone while the other sources are replaced by their internal resistances.

To replace the other sources by their internal resistances, the voltage sources are short- circuited and the current sources open- circuited.

CIRCUIT DIAGRAM WITH V_2 ACTING INDEPENDENTLY:



OBSERVATION TABLE:

Experimental Values:

V_1 (Volts)	V_2 (Volts)	I_3 (mA)

Theoretical Values:

V_1 (Volts)	V_2 (Volts)	I_3 (mA)

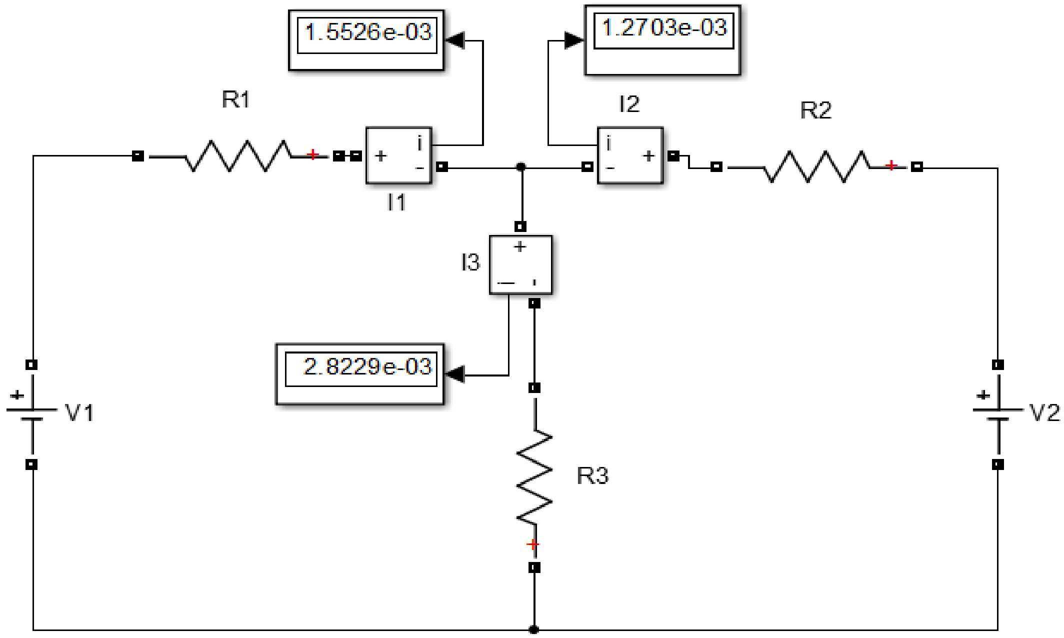
FORMULAE :

$$I_3' + I_3'' = I_3$$

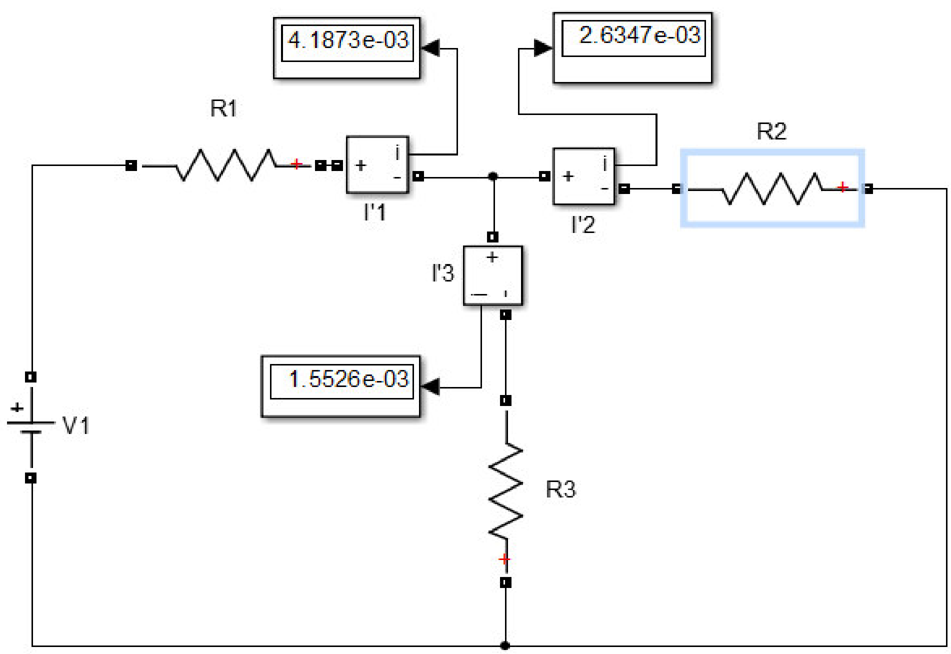
PROCEDURE :

1. Connections are made as per the circuit diagram given in Fig. 1.
2. Switch on the supply.
3. Note the readings of three Ammeters.
4. One of the voltage source V_1 is connected and the other voltage source V_2 is short circuited as given in Fig.2.
5. Note the three ammeter readings.
6. Now short circuit the voltage source V_1 and connect the voltage source V_2 as given in the circuit diagram of Fig. 3.
7. Note the three ammeter readings.
8. Algebraically add the currents in steps (5) and (7) above to compare with the current in step (3) to verify the theorem.
9. Verify with theoretical values.

SIMULATION DIAGRAM FOR SUPERPOSITION THEOREM:



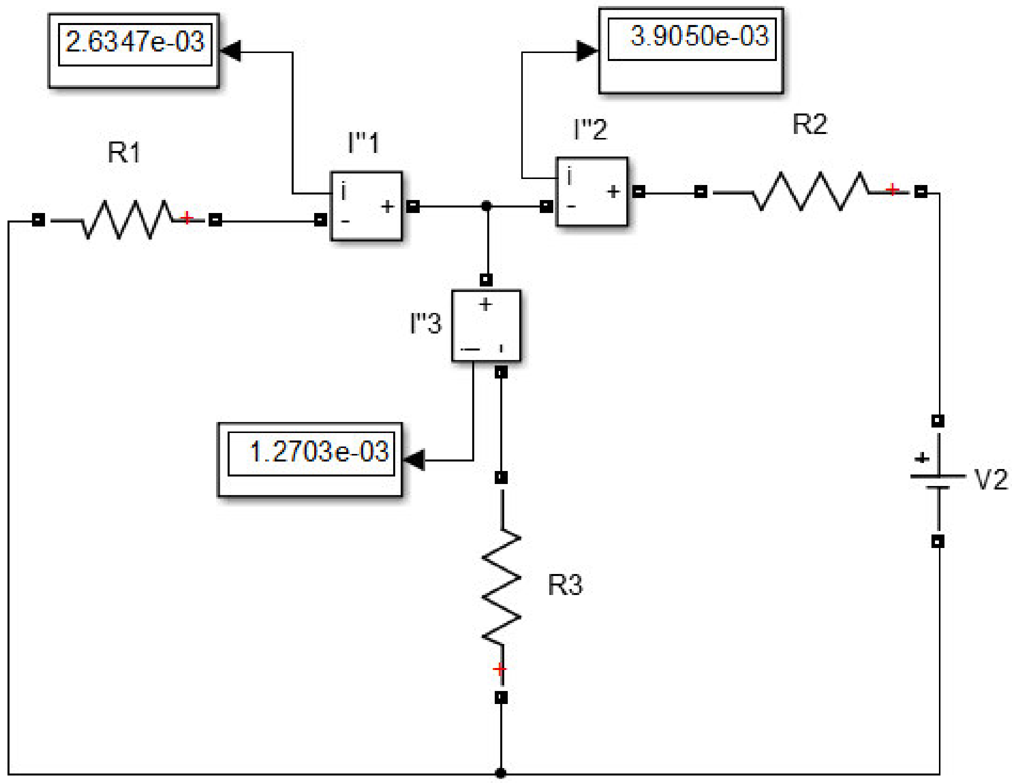
CIRCUIT DIAGRAM WITH V_1 ACTING INDEPENDENTLY:



SIMULATION PROCEDURE:

1. Open a new MATLAB/SIMULINK model
2. Connect the circuit as shown in the figure
3. Debug and run the circuit
4. For different input voltages, record the current and voltages and verify with theoretical values.

CIRCUIT DIAGRAM WITH V_2 ACTING INDEPENDENTLY:



VERIFICATION OF SUPERPOSITION THEOREM:

Practical:

S.No.	I_3 (mA)	I_3' (mA)	I_3'' (mA)	$I_3 = I_3' + I_3''$ (mA)

Theoretical:

S.No.	I_3 (mA)	I_3' (mA)	I_3'' (mA)	$I_3 = I_3' + I_3''$ (mA)

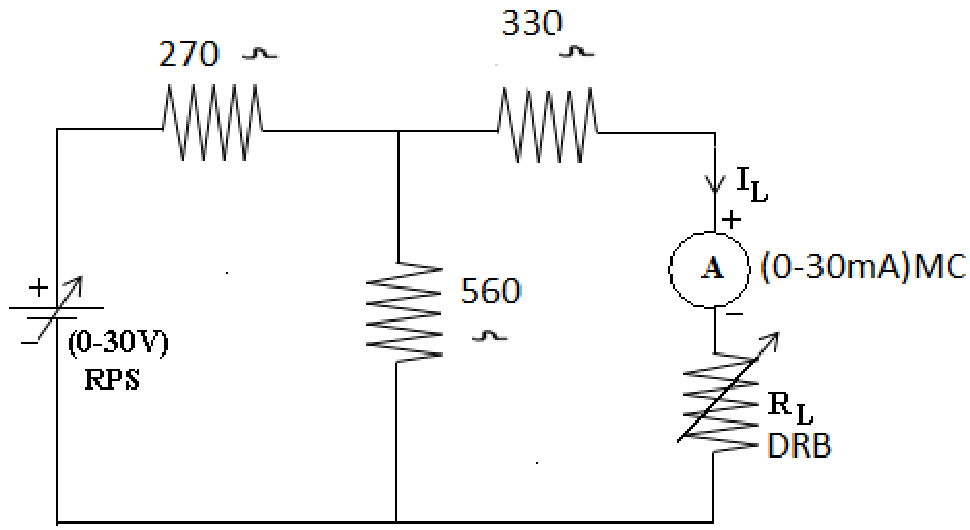
VIVA QUESTIONS:

1. State Superposition Theorem.
2. What is meant by a linear system?
3. Give the usefulness of Superposition Theorem.
4. How will you apply Superposition Theorem to a linear circuit containing both dependent and independent sources?
5. State the limitations of Superposition theorem.

RESULT:

Thus the Superposition theorem was verified.

CIRCUIT DIAGRAM FOR MAXIMUM POWER TRANSFER THEOREM:



OBSERVATION TABLE:

S.No.	R_L (kΩ)	I_L (mA)		$P = I^2R_L$ (mW)	
		Practical Value	Theoretical Value	Practical Value	Theoretical Value

EXP.NO:

DATE:

SIMULATION AND EXPERIMENTAL VERIFICATION OF ELECTRICAL CIRCUIT PROBLEMS USING MAXIMUM POWER TRANSFER THEOREM

AIM:

To verify maximum power transfer theorem.

APPARATUS REQUIRED:

S.No	Name of the Components / Equipment	Type/Range	Quantity required
1	Resistor		
2	Dc power supply		
3	Voltmeter		
4	Ammeter		
5	Wires		
6	Bread board		

SOFTWARE REQUIRED:

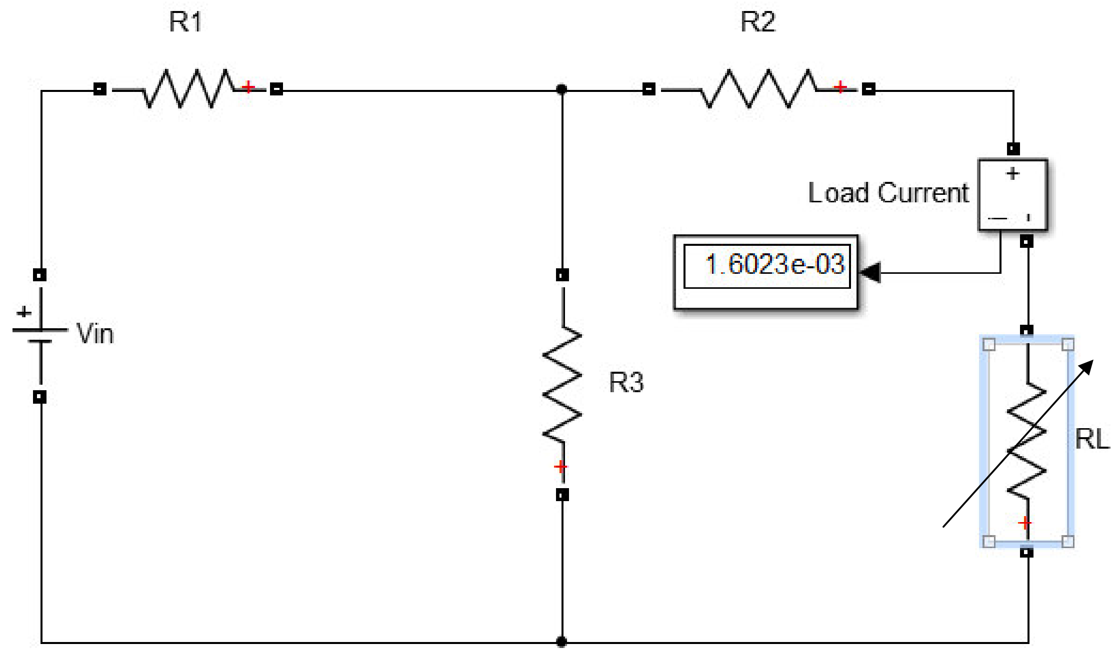
Matlab 7.1

MAXIMUM POWER TRANSFER THEOREM:

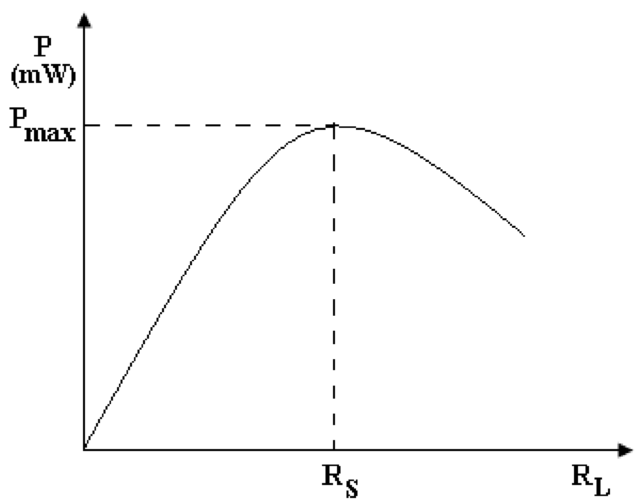
THEORY:

The Maximum Power Transfer Theorem states that maximum power is delivered from a source to a load when the load resistance is equal to source resistance.

SIMULATION DIAGRAM FOR MAXIMUM POWER TRANSFER THEOREM:



MODEL GRAPH:



MODEL CALCULATION:

PROCEDURE:

1. Find the Load current for the minimum position of the Rheostat theoretically.
2. Select the ammeter Range.
3. Give connections as per the circuit diagram.
4. Measure the load current by gradually increasing R_L .
5. Enter the readings in the tabular column.
6. Calculate the power delivered in R_L.
7. Plot the curve between R_L and power.
8. Check whether the power is maximum at a value of load resistance that equals source resistance.
9. Verify the maximum power transfer theorem.

SIMULATION PROCEDURE:

1. Open a new MATLAB/SIMULINK model
2. Connect the circuit as shown in the figure
3. Debug and run the circuit
4. For different input voltages, record the current and voltages and verify with theoretical values.

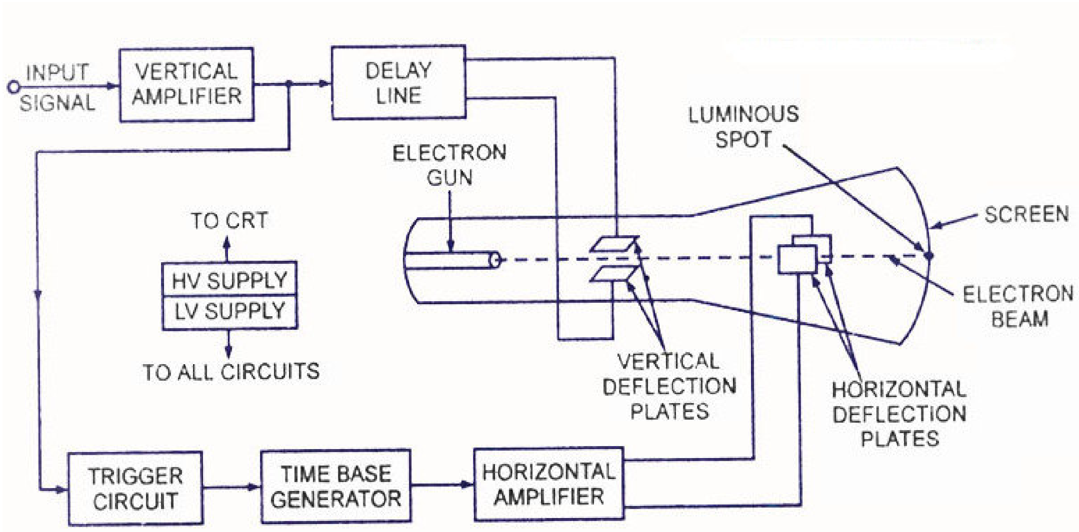
VIVA QUESTIONS:

1. Define Power. What is the unit of Power?
2. State Maximum Power Transfer Theorem

RESULT:

Thus the Maximum power transfer theorem was verified.

BLOCK DIAGRAM OF GENERAL PURPOSE CRO:



Block Diagram of a General Purpose CRO

EXP NO.:

DATE:

STUDY OF ANALOG AND DIGITAL OSCILLOSCOPES AND MEASUREMENT OF SINUSOIDAL VOLTAGE, FREQUENCY AND POWER FACTOR

AIM:

The aim of the experiment is to understand the operation of cathode ray oscilloscope (CRO) and to become familiar with its usage, also to perform an experiment using function generator to measure amplitude, time period, frequency & power factor of the time varying signals using a calibrated cathode ray oscilloscope.

APPARATUS REQUIRED:

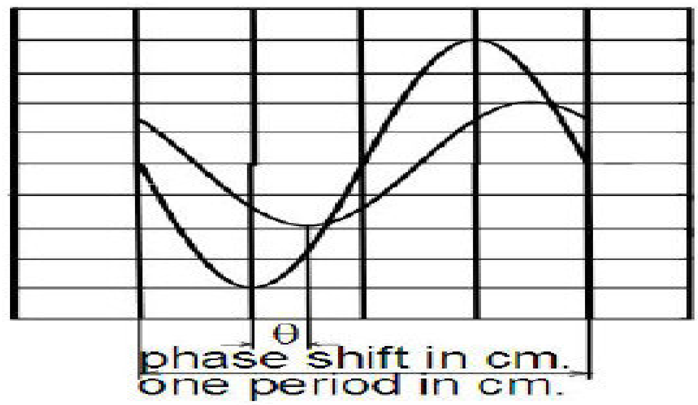
S.No	Name of the Components/Equipment	Qty
1.	CRO	1
2.	Function generator	2
3.	Probes	2

THEORY:

The cathode ray oscilloscope (CRO) provides a visual presentation of any waveform applied to the input terminal. The oscilloscope consists of the following major subsystems.

- Cathode ray tube (CRT)
- Vertical amplifier
- Horizontal amplifier
- Sweep Generator
- Trigger circuit
- Associated power supply

It can be employed to measure quantities such as peak voltage, frequency, phase difference, pulse width, delay time, rise time, and fall time.



S.No	Type of wave	Time period (T)	Amplitude	Theoretical Frequency	Practical Frequency
1.					
2.					
3.					

CATHODE RAY TUBE:

The CRT is the heart of the CRO providing visual display of an input signal waveform. A CRT contains four basic parts:

- An electron gun to provide a stream of electrons.
- Focusing and accelerating elements to produce a well define beam of electrons.
- Horizontal and vertical deflecting plates to control the path of the electron beam.
- An evacuated glass envelope with a phosphorescent which glows visibly when struck by electron beam.

A Cathode containing an oxide coating is heated indirectly by a filament resulting in the release of electrons from the cathode surface. The control grid which has a negative potential, controls the electron flow from the cathode and thus control the number of electron directed to the screen. Once the electron passes the control grid, they are focused into a tight beam and accelerated to a higher velocity by focusing and accelerating anodes. The high velocity and well defined electron beam then passed through two sets of deflection plates.

The First set of plates is oriented to deflect the electron beam vertically. The angle of the vertical deflection is determined by the voltage polarity applied to the deflection plates. The electron beam is also being deflected horizontally by a voltage applied to the horizontal deflection plates. The tube sensitivity to deflecting voltages can be expressed in two ways that are deflection factor and deflection sensitivity.

The deflected beam is then further accelerated by very high voltages applied to the tube with the beam finally striking a phosphorescent material on the inside face of the tube. The phosphor glows when struck by the energetic electrons.

CONTROL GRID:

Regulates the number of electrons that reach the anode and hence the brightness of the spot on the screen.

FOCUSING ANODE:

Ensures that electrons leaving the cathode in slightly different directions are focused down to a narrow beam and all arrive at the same spot on the screen.

ELECTOR GUN:

Cathode, control grid, focusing anode, and accelerating anode.

DEFLECTING PLATES:

Electric fields between the first pair of plates deflect the electrons horizontally and an electric field between the second pair deflects them vertically. If no deflecting fields are present, the electrons travel in a straight line from the hole in the accelerating anode to the center of the screen, where they produce a bright spot. In general purpose oscilloscope, amplifier circuits are needed to increase the input signal to the voltage level required to operate the tube because the signals measured using CRO are typically small. There are amplifier sections for both vertical and horizontal deflection of the beam.

VERTICAL AMPLIFIER:

Amplify the signal at its input prior to the signal being applied to the vertical deflection plates.

HORIZONTAL AMPLIFIER:

Amplify the signal at its input prior to the signal being applied to the horizontal deflection plates.

SWEEP GENERATOR:

Develop a voltage at the horizontal deflection plate that increases linearly with time.

OPERATION:

The four main parts of the oscilloscope CRT are designed to create and direct an electron beam to a screen to form an image. The oscilloscope links to a circuit that directly connects to the vertical deflection plates while the horizontal plates have linearly increasing charge to form a plot of the circuit voltage over time. In an operating cycle, the heater gives electrons in the cathode enough energy to escape. The electrons are attracted to the accelerating anode and pulled through a control grid that regulates the number of electrons in the beam, a focusing anode that controls the width of the beam, and the accelerating anode itself. The vertical and horizontal deflection plates create electric field that bend the beam of electrons. The electrons finally hit the fluorescent screen which absorbs the energy from the electron beam and emits it in the form of light to display an image at the end of the glass tube.

PRECAUTIONS:

1. Do not leave a 'bright spot' on the screen for any length of time.
2. Do not apply signals that exceed the scopes voltage rating.
3. Do not try make accurate measurements on signals whose frequency is outside the scope's frequency specifications.
4. Be aware that the scope's input circuitry can cause loading effects on the circuitry under test-use correct probe for the work.

PRODEDURE:

1. Measurement of Voltage Using CRO : A voltage can be measured by noting the Y deflection produced by the voltage; using this deflection in conjunction with the Y-gain setting, the voltage can be calculated as follows : V = (no. of boxes in cm.) x (selected Volts/cm scale)

2 .Measurement of Current and Resistance Using a CRO: Using the general method, a correctly calibrated CRO can be used in conjunction with a known value of resistance R to determine the current I flowing through the resistor.

3 Measurement of Frequency Using a CRO: A simple method of determining the frequency of a signal is to estimate its periodic time from the trace on the screen of a CRT. However this method has limited accuracy, and should only be used where other methods are not available. To calculate the frequency of the observed signal, one has to measure the period, i.e. the time taken for 1 complete cycle, using the calibrated sweep scale. The period could be calculated by T = (no. of squares in cm) x (selected Time/cm scale) Once the period T is known, the frequency is given by f (Hz)= 1/T(sec)

4. Measurement of Phase: The calibrated time scales can be used to calculate the phase shift between two sinusoidal signals of the same frequency. If a dual trace or beam CRO is available to display the two signals simultaneously (one of the signals is used for synchronization), both of the signals will appear in proper time perspective and the amount of time difference between the waveforms can be measured. This, in turn can be utilized to calculate the phase angle θ, between the two signals.
Referring to the fig below the phase shift can be calculated by the formula;

$$\theta° = \frac{phase\ shift\ in\ cm}{one\ period\ in\ cm}$$

MEASUREMENT OF PF:

The power factor is calculated by the formula

pf=VICOS θ.

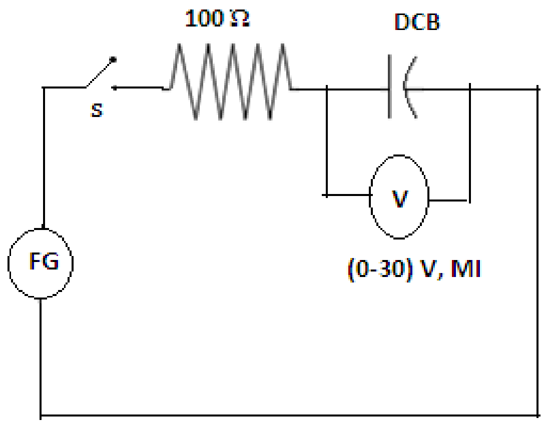
VIVA QUESTIONS:

1. What is a CRO?
2. How can we measure the voltage using a CRO?
3. Explain the different parts of the CRO
4. Explain the operation of a CRO.

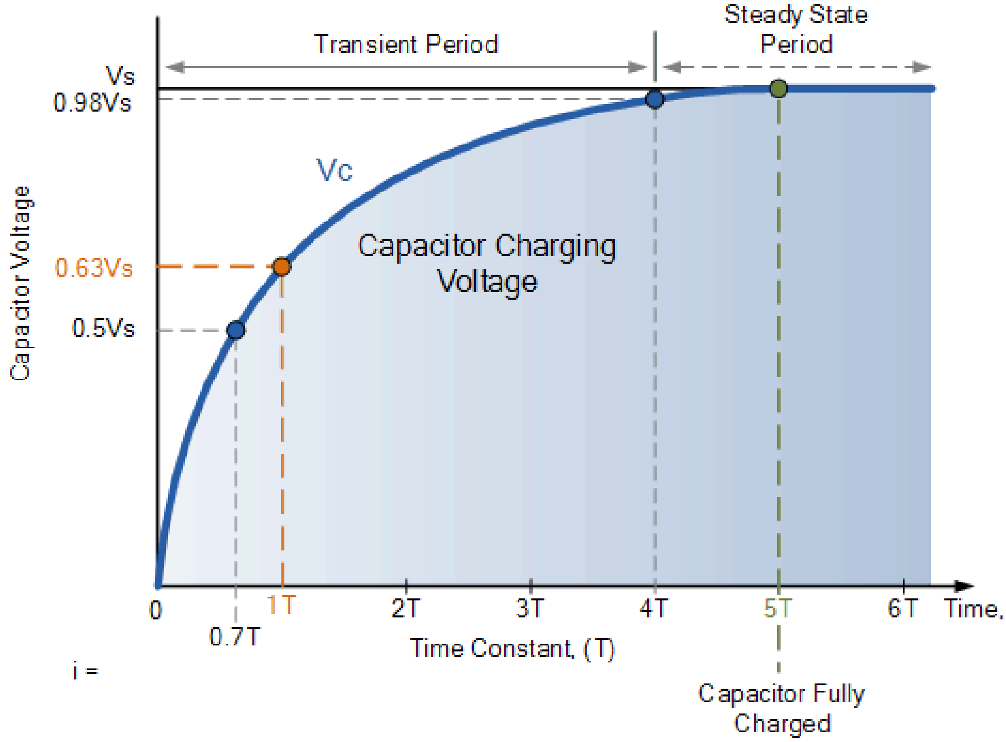
RESULT:

Thus the Analog and digital oscilloscopes were studied and measurement of sinusoidal voltage, frequency and power factor was done.

CIRCUIT DIAGRAM FOR RC TRANSIENT:



MODEL GRAPH:



EXP NO. :

DATE :

SIMULATION AND EXPERIMENTAL VALIDATION OF R-C ELECTRIC CIRCUIT TRANSIENTS

AIM:

To find the time constant of series R-C electric circuits

SOFTWARE REQUIRED:

PSpice Lite

APPARATUS REQUIRED:

S.No.	Name of the Components/Equipment	Range/Type	Quantity required
1	Resistor	100 Ω	1
2	Function generator	-	1
3	Voltmeter	(0-30)V MI	1
4	Decade capacitance box	-	1
5	Wires	Single strand	Few nos
6	Bread board		1

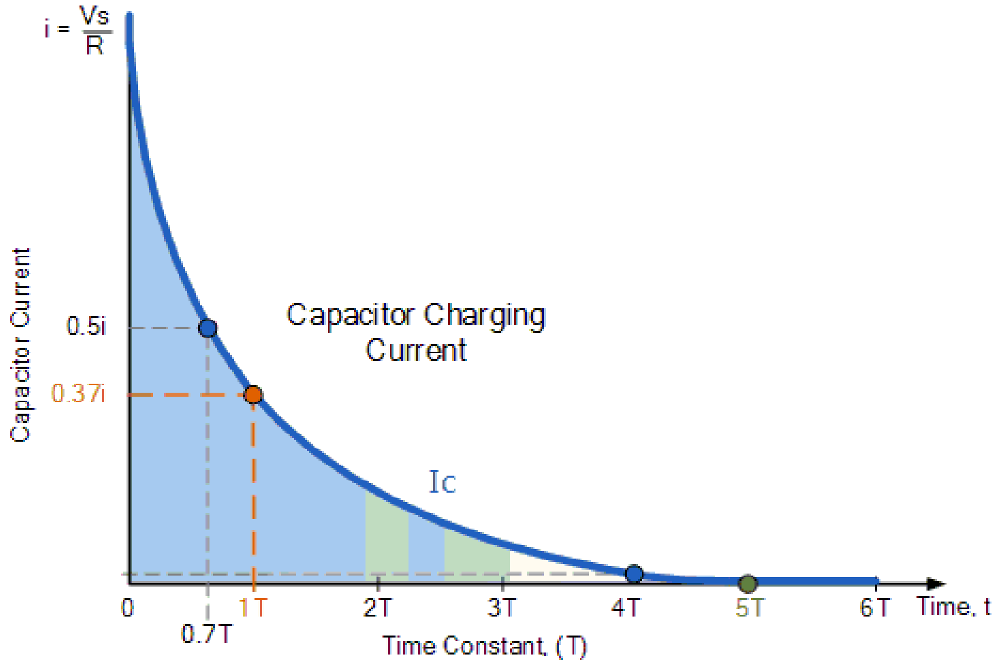
THEORY:

RC CIRCUIT:

Consider a series RC circuit as shown. The switch is in open state initially. There is no charge on condenser and no voltage across it. At instant t=0, switch is closed.

Immediately after closing a switch, the capacitor acts as a short circuit, so current at the time of switching is high. The voltage across capacitor is zero at $t = 0^+$ as capacitor acts as a short circuit, and the current is maximum given by,

$i = V/R$ Amps



OBSERVATION TABLE:

S.No.	Frequency (Hz)	Time (s)	Voltage across the capacitor V_C (v)

MODEL CALCULATION:

This current is maximum at t=0+ which is charging current. As the capacitor starts charging, the voltage across capacitor V_C starts increasing and charging current starts decreasing. After some time, when the capacitor charges to V volts, it achieves steady state. In steady state it acts as an open circuit and current will be zero finally.

Charging current and voltage in capacitor are given as below,

$$I_C = \frac{V_{in}}{R} e^{\frac{-t}{RC}} \qquad V_C = V_{in}(1 - e^{\frac{-t}{RC}})$$

$$V_C = V_{in}(1 - e^{-1})$$

The term RC in equation of V_C or I_C is called Time constant and denoted by τ, measured in seconds.

When, $t = RC = \tau$ then,

$$V_C = 0.632V_{in}$$

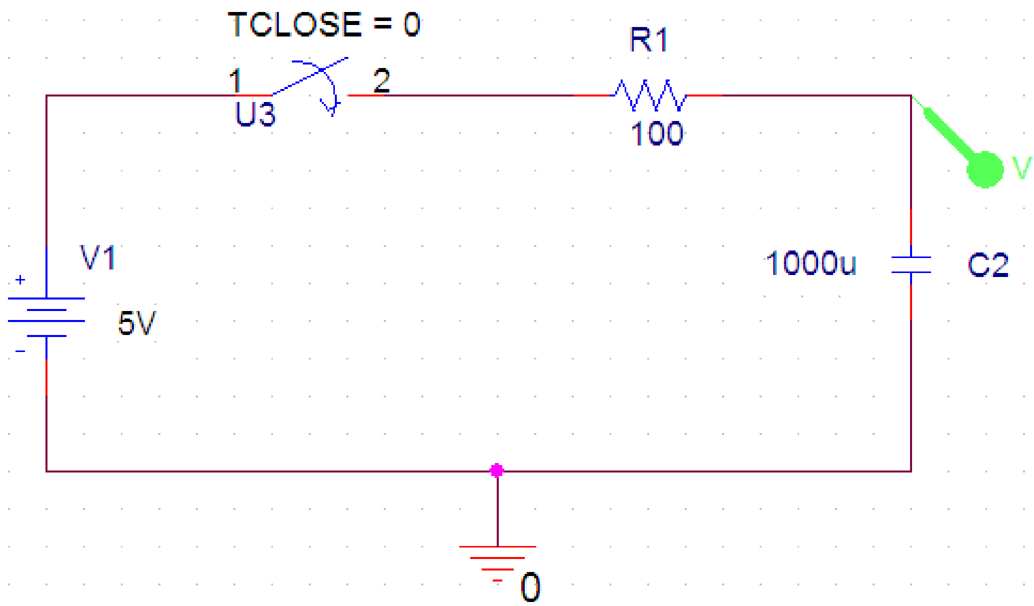
So time constant of series RC circuit is defined as time required by the capacitor voltage to rise from zero to 0.632 of its final steady state value during charging.

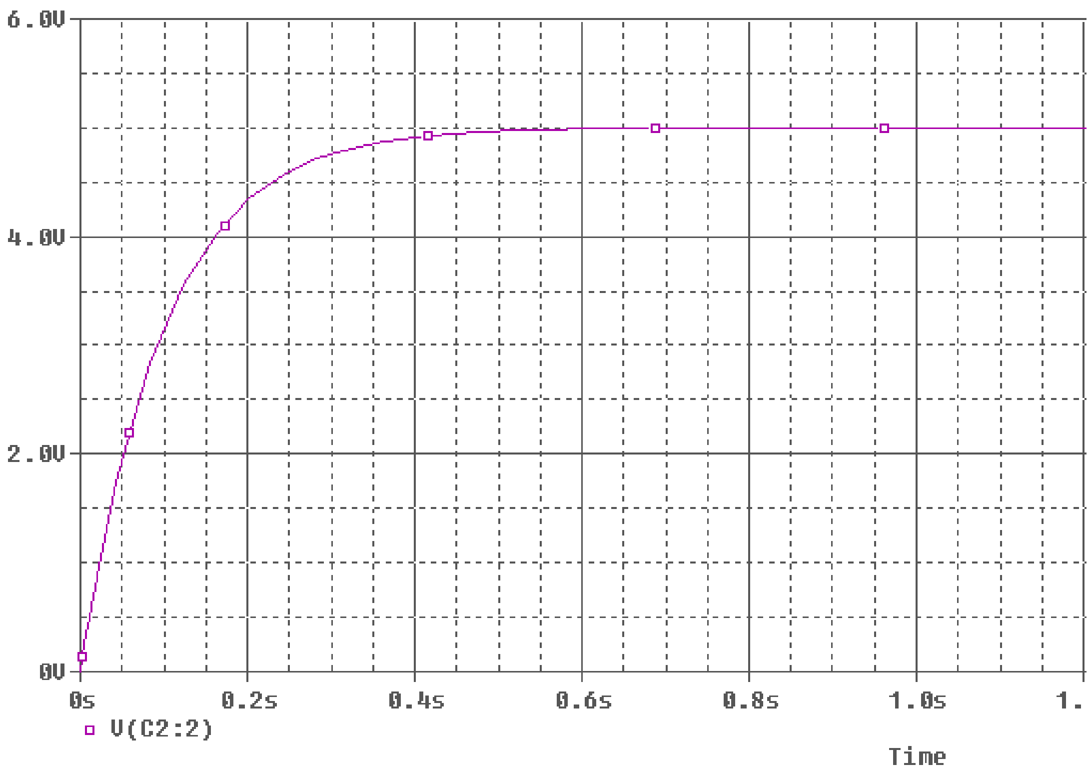
Thus, time constant of RC circuit can be defined as time seconds, during which voltage across capacitor (stating from zero) would reach its final steady state value if its rate of change was maintained constant at its initial value throughout charging period.

PROCEDURE:

1. Make the connections as per the circuit diagram.
2. Vary the frequency by using function generator.
3. For different frequencies tabulate the value of voltage across the capacitor .
4. Calculate the time period.
5. Plot the graph for time period Vs voltage across the capacitor.

SIMULATION DIAGRAM:





SIMULATION PROCEDURE:

1. Open a new PSpice CAPTURE project.
2. Connect the circuit as shown in the figure.
3. Create simulation profile and run the model

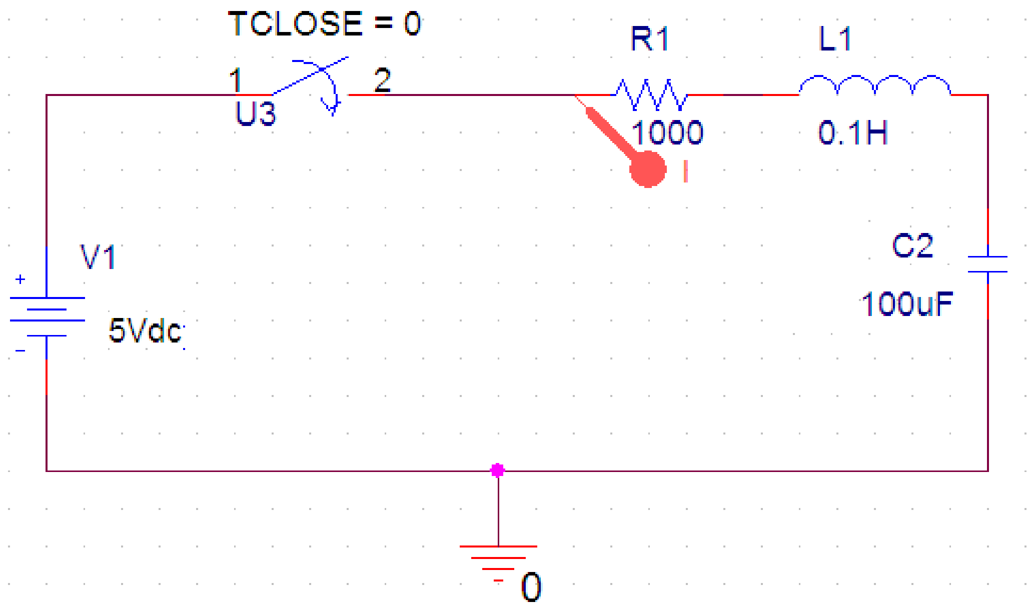
VIVA QUESTIONS:

1. Differentiate steady state and transient state.
2. What is meant by transient response?
3. Define the time constant of a RL Circuit.
4. Define the time constant of a RC Circuit.
5. What is meant by forced response?

RESULT:

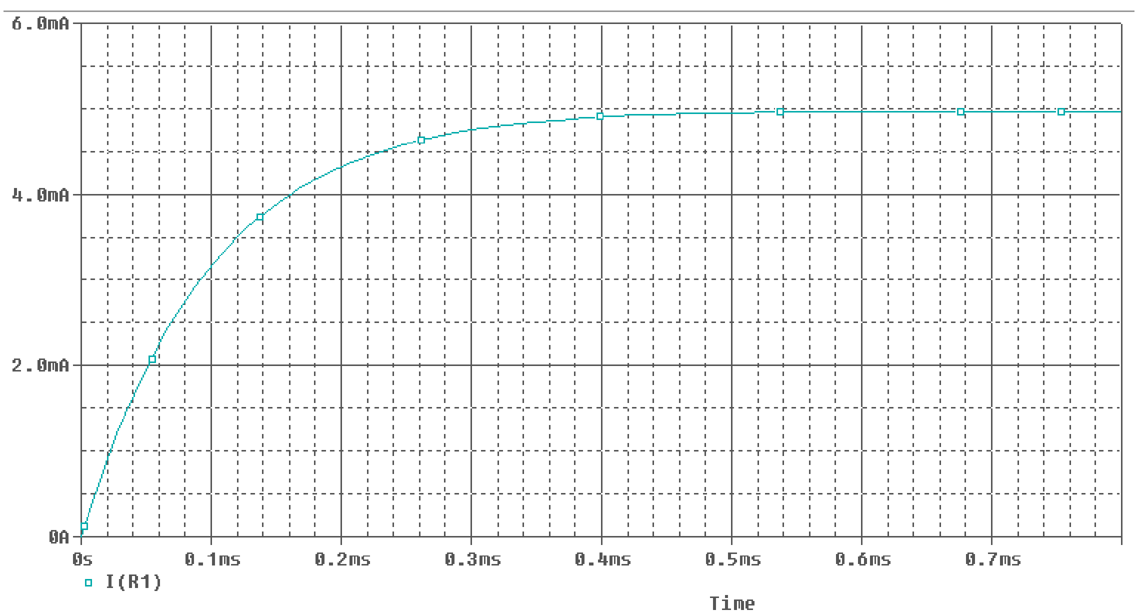
Thus the transient responses of RC circuit are found practically.

SIMULATION CIRCUIT DIAGRAM:



OUTPUT WAVEFORM:

Case (i):



EXP NO.:

DATE :

SIMULATION AND EXPERIMENTAL VALIDATION OF FREQUENCY RESPONSE OF RLC ELECTRIC CIRCUIT

AIM:

To simulate and find the frequency response of RLC electric circuits.

SOFTWARE REQUIRED:

PSpice Lite

APPARATUS REQUIRED:

S.No.	Name of the Components/Equipment	Range/Type	Quantity required
1	Resistor	1000 Ω	1
2	Function generator	-	1
3	Voltmeter	(0-30)V MI	1
4	Decade capacitance box	-	1
5	Decade Inductance box	-	1
6	Wires	Single strand	Few nos
7	Bread board		1

THEORY:

RLC CIRCUIT:

Consider a series RLC circuit as shown. The switch is in open state initially. There is no charge on condenser and no voltage across it. At instant t=0, switch is closed.

Immediately after closing a switch, the capacitor acts as a short circuit, so current at the time of switching is high. The voltage across capacitor is zero at $t= 0^+$ as capacitor acts as a short circuit, and the current is maximum given by,

$i = V/R$ Amps

OBSERVATION TABLE:

S.No.	Frequency (Hz)	Time (s)	Voltage across the capacitor V_C (v)

MODEL CALCULATION:

This current is maximum at t=0+ which is charging current. As the capacitor starts charging, the voltage across capacitor V_C starts increasing and charging current starts decreasing. After some time, when the capacitor charges to V volts, it achieves steady state. In steady state it acts as an open circuit and current will be zero finally.

Laplace transform of current flowing through the circuit is,

$$I(s)=\frac{V/L}{s^2+\frac{R}{L}s+\frac{1}{LC}}$$

Case (i):

If $\left[\frac{R}{2L}\right]^2 > \frac{1}{LC}$

The roots are real and distinct. The current is over damped.

Case (ii):

If $\left[\frac{R}{2L}\right]^2 = \frac{1}{LC}$

The roots are equal. The current is critically damped.

Case (iii):

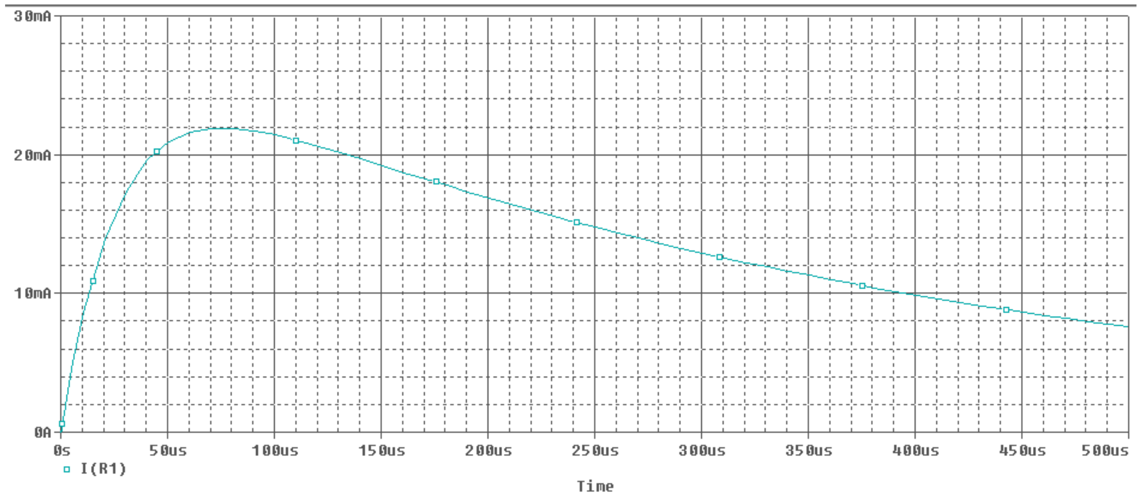
If $\left[\frac{R}{2L}\right]^2 < \frac{1}{LC}$

The roots become complex conjugate. The current is oscillatory in nature.

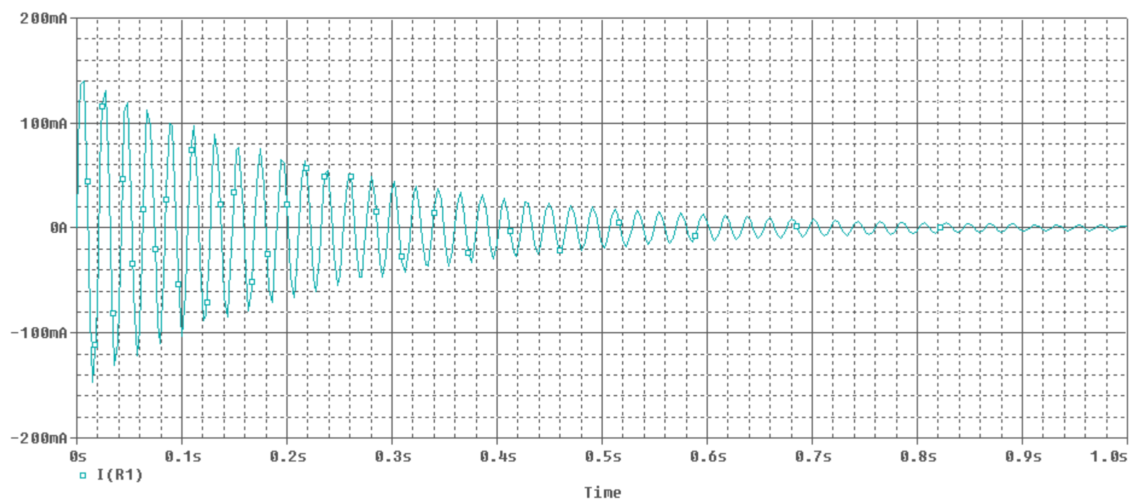
PROCEDURE:

1. Make the connections as per the circuit diagram
2. Vary the frequency by using function generator
3. For different frequencies tabulate the value of voltage across the capacitor
4. Calculate the time period
5. Plot the graph for time period Vs voltage across the capacitor.

Case (ii):



Case (iii):



SIMULATION PROCEDURE:

4. Open a new PSpice CAPTURE project.
5. Connect the circuit as shown in the figure.
6. Create simulation profile and run the model

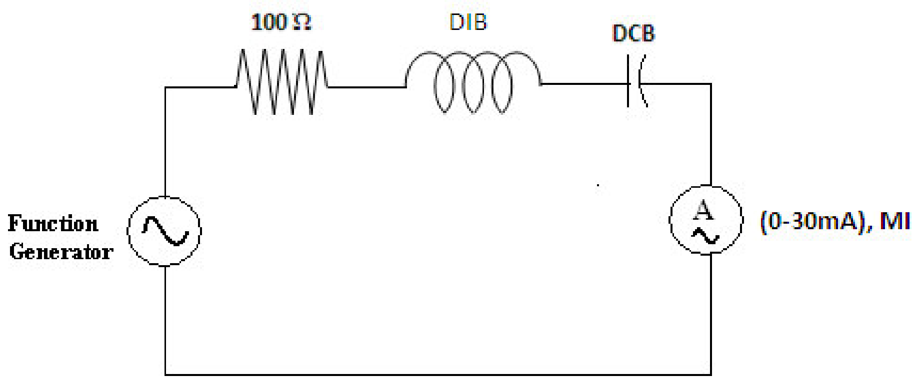
VIVA QUESTIONS

1. What is meant by transient response?
2. Define the time constant of a RL Circuit.
3. Define the time constant of a RC Circuit.
4. What is meant by forced response?

RESULT:

Thus the transient responses of RLC circuit are found practically.

CIRCUIT DIAGRAM FOR SERIES RESONANCE:



OBSERVATION TABLE:

S.No.	Frequency in Hz	Output Current in mA

MODEL CALCULATION:

EXP NO.:

DATE :

DESIGN AND SIMULATION OF SERIES RESONANCE CIRCUIT

AIM:

To plot the current Vs frequencies graph of series resonant circuits and hence measure their bandwidth, resonant frequency and Q factor.

SOFTWARE REQUIRED:

PSpice 9.1 Lite

APPARATUS REQUIRED:

S.No.	Name of the Components/Equipment	Type	Range	Quantity required
1	Function Generator	-	-	1
2	Resistor	-	100 Ω	1
3	Decade Inductance Box	-	-	1
4	Decade Capacitance Box	-	-	1
5	Ammeter	MI	(0-30) mA	1
6	Connecting Wires	Single strand	-	Few nos

THEORY:

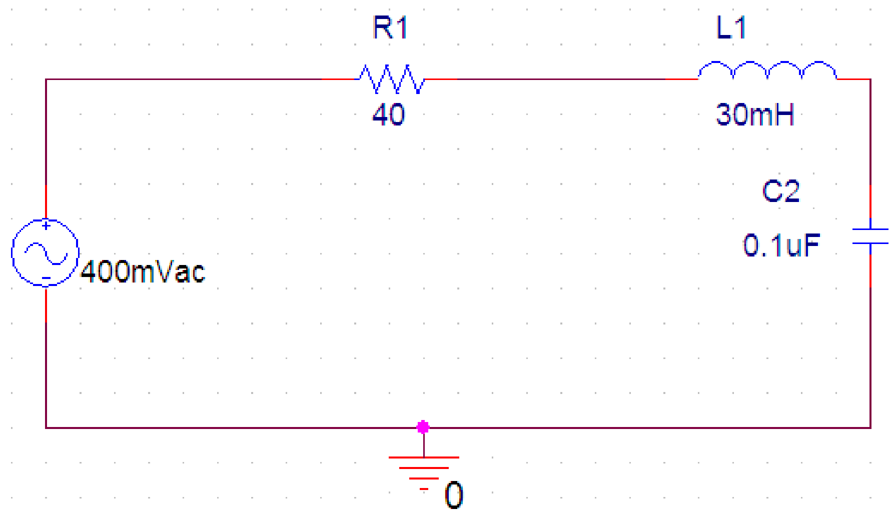
A circuit is said to be in resonance when applied voltage V and current I are in phase with each other. Thus at resonance condition, the equivalent complex impedance of the circuit consists of only resistance (R) and hence current is maximum. Since V and I are in phase, the power factor is unity.

The complex impedance

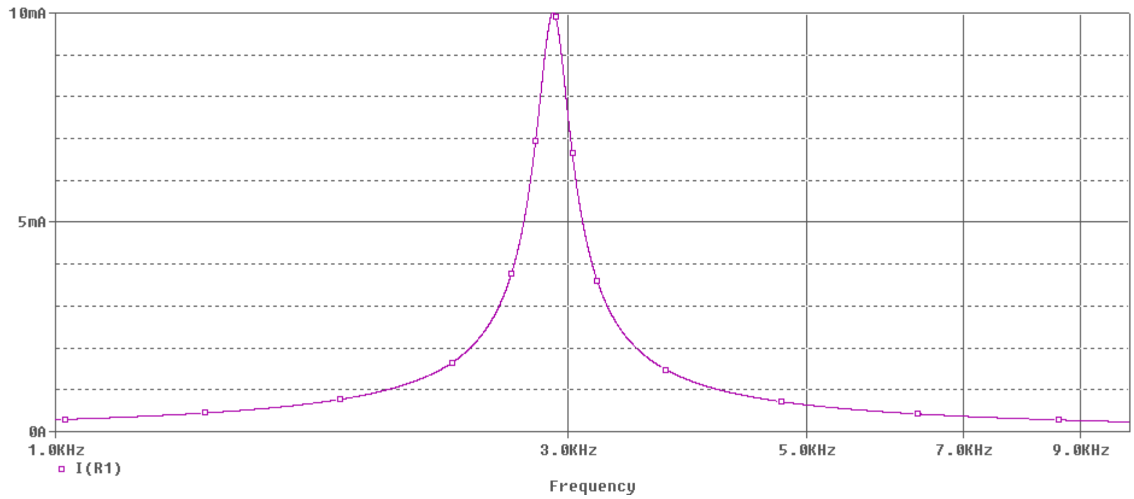
$$Z = R + j\,(X_L - X_C)$$

Where $X_L = \omega L$

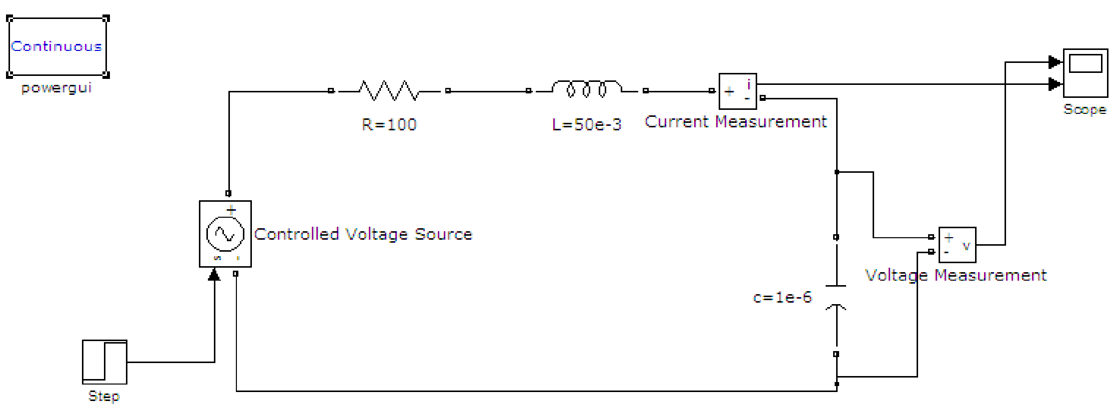
PSpice SIMULATION:



OUTPUT WAVWFORM:



MATLAB SIMULATION:



$X_C = 1/\omega C$

At resonance, $X_L = X_C$ and hence Z= R

BANDWIDTH OF A RESONANCE CIRCUIT:

Bandwidth of a circuit is given by the band of frequencies which lies between two points on either side of resonance frequency, where current falls through 1/1.414 of the maximum value of resonance. Narrow is the bandwidth, higher the selectivity of the circuit.

As shown in the model graph, the bandwidth AB is given by $f_2 - f_1$. f_1 is the lower cut off frequency and f_2 is the upper cut off frequency.

Q - FACTOR:

In the case of a RLC series circuit, Q-factor is defined as the voltage magnification in the circuit at resonance. At resonance, current is maximum. $I_o = V/R$.

The applied voltage $V = I_oR$

Voltage magnification = $V_L/V = I_oX_L$

In the case of resonance, high Q factor means not only high voltage, but also higher sensitivity of tuning circuit. Q factor can be increased by having a coil of large inductance, not of smaller ohmic resistance.

$Q = \omega L / R$

FORMULAE USED:

Resonant frequency $f_r = \frac{1}{2\pi\sqrt{LC}}$ Hz

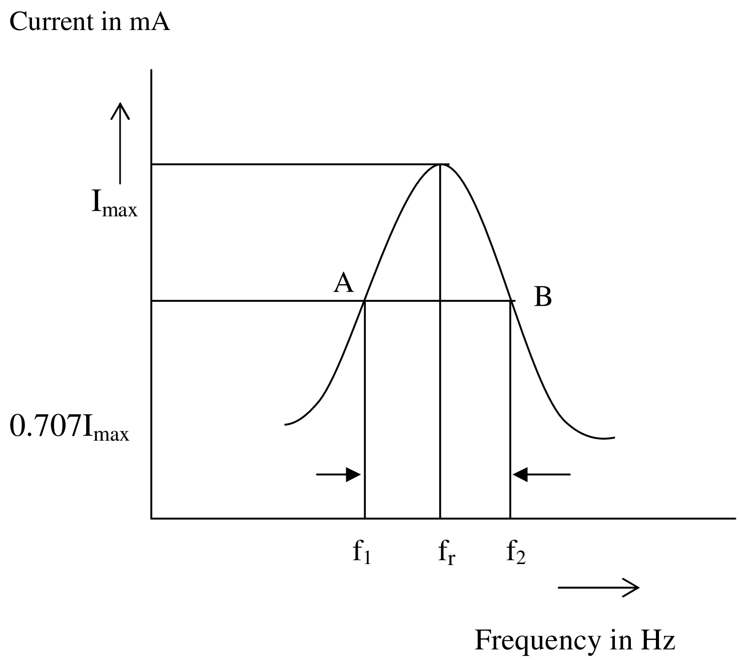
Bandwidth $BW = f_2 - f_1$

Quality Factor = $\frac{f_r}{BW}$

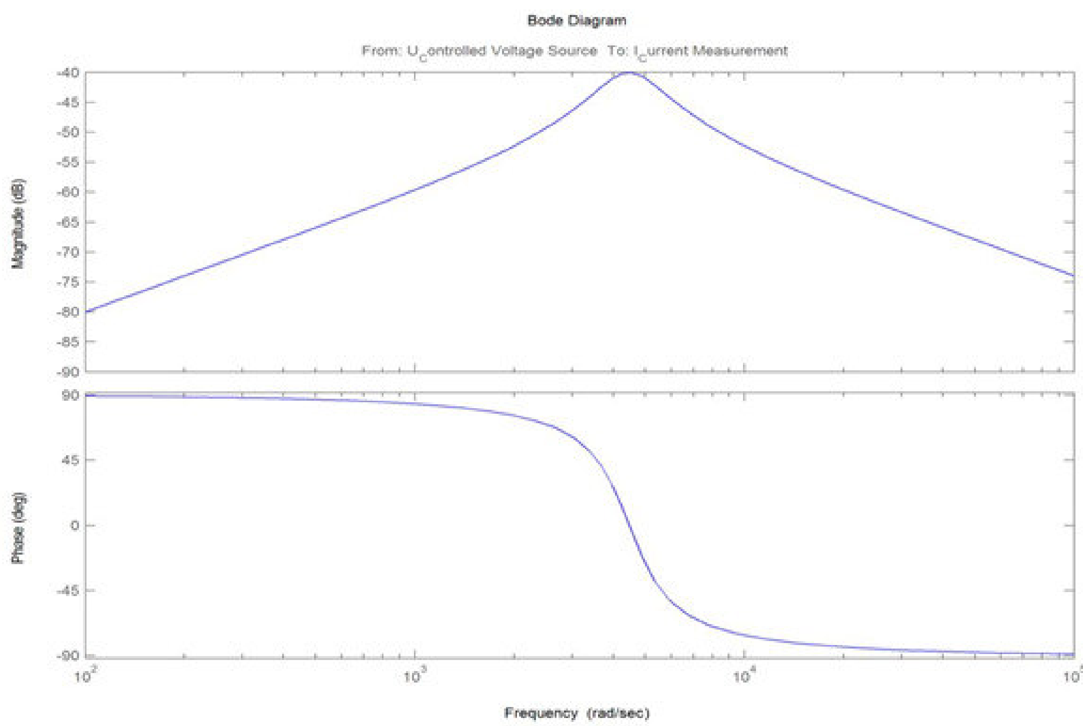
PROCEDURE:

1. Connect the circuit as per the circuit diagram.
2. Vary the frequency and note down the corresponding meter reading.
3. Draw the current Vs frequency curve and measure the bandwidth, resonant frequency and Q factor.

MODEL GRAPH FOR SERIES RESONANCE



PLOT OF MAGNITUDE & PHASE ANGLE OF CURRENT FOR VARIOUS FREQUENCIES:



SIMULATION PROCEDURE:

1. Open a new MATLAB/SIMULINK model or PSpice CAPTURE project.
2. Connect the circuit as shown in the figure.
3. Debug and run the circuit.
4. By double clicking the power gui plot the value of current for the different values of frequencies (for MATLAB Simulink).
5. For PSpice CAPTURE run the model create simulation profile and run the model.

VIVA QUESTIONS:

1. Define Bandwidth.
2. Define Quality factor.
3. What is meant by selectivity?
4. Give the significance of Q- factor.
5. What is meant by resonance?
6. What are the characteristics of a series resonant circuit?
7. What will be the power factor of the circuit at resonance?

RESULT:

Thus the current Vs frequency graphs of series and parallel resonant circuits were plotted and the bandwidth, resonant frequency and Q factor were measured. They were found to be

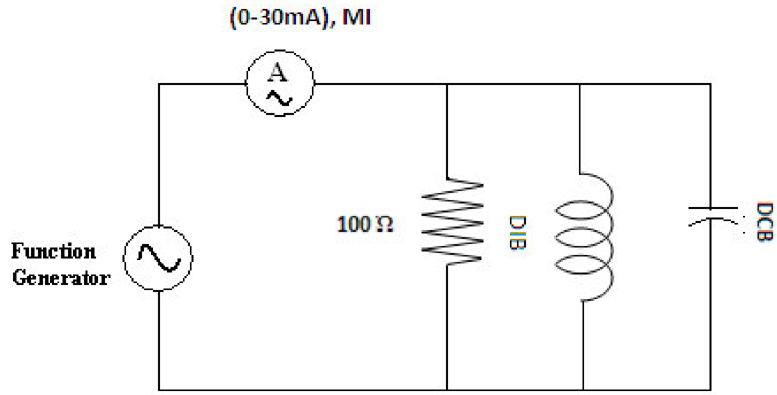
(a) Series resonance

Resonant frequency = ____________

Bandwidth = ____________

Q- Factor = ____________

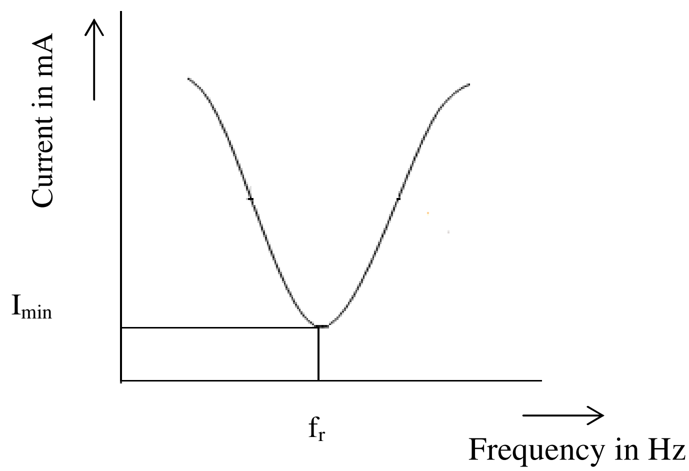
CIRCUIT DIAGRAM FOR PARALLEL RESONANCE:



OBSERVATION TABLE:

S.No.	Frequency in	Output

MODEL GRAPH FOR PARALLEL RESONANCE:



EXP NO. :

DATE :

DESIGN AND SIMULATION OF PARALLEL RESONANT CIRCUITS

AIM:

To plot the magnitude & phase angle of current for various frequencies for the given RLC parallel circuit.

SOFTWARE REQUIRED:

Matlab 7.1 or PSpice 9.1 Lite

APPARATUS REQUIRED:

S.No.	Name of the Components/Equipment	Type	Range	Quantity required
1	Function Generator	-	-	1
2	Resistor	-	100 Ω	1
3	Decade Inductance Box	-	-	1
4	Decade Capacitance Box	-	-	1
5	Ammeter	MI	(0-30) mA	1
6	Connecting Wires	Single strand	-	Few nos

THEORY:

A circuit is said to be in resonance when applied voltage V and current I are in phase with each other. Thus at resonance condition, the equivalent complex impedance of the circuit consists of only resistance (R) and hence current is maximum. Since V and I are in phase, the power factor is unity.

The complex impedance

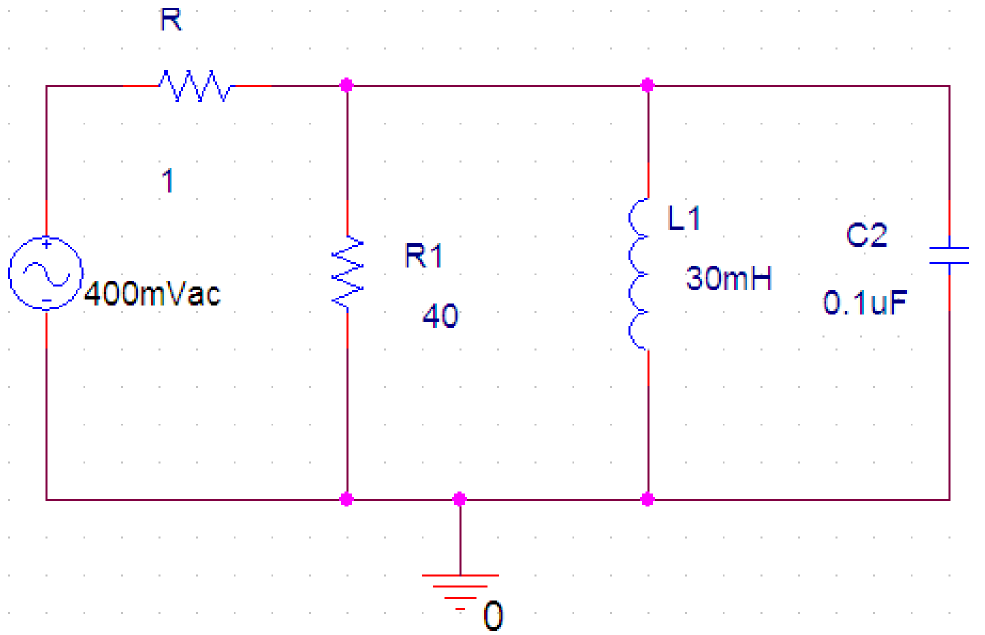
$Z = R + j\,(X_L - X_C)$

Where $X_L = \omega L$

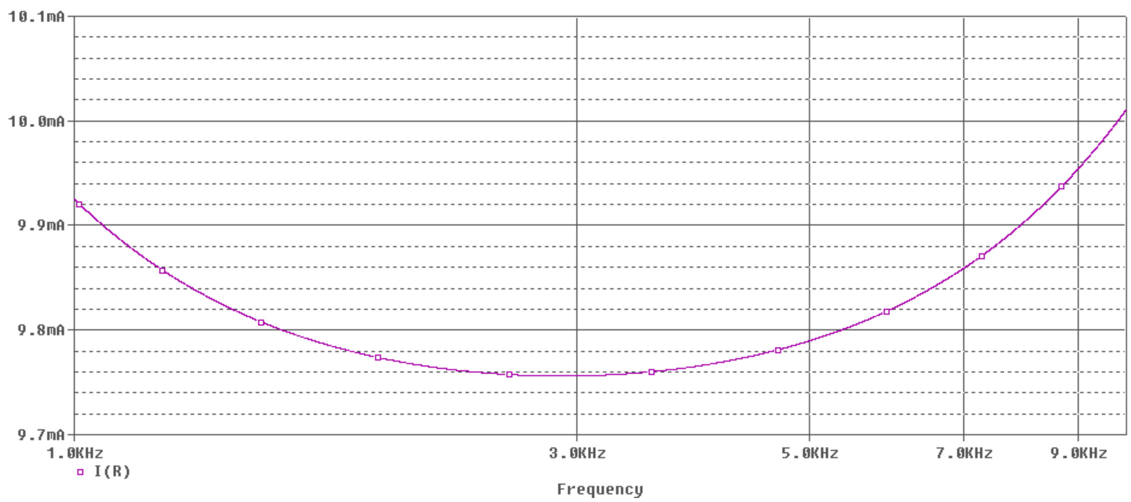
$X_C = 1/\omega C$

At resonance, $X_L = X_C$ and hence $Z = R$

PSpice SIMULATION:



OUTPUT WAVEFORM:



BANDWIDTH OF A RESONANCE CIRCUIT:

Bandwidth of a circuit is given by the band of frequencies which lies between two points on either side of resonance frequency, where current falls through 1/1.414 of the maximum value of resonance. Narrow is the bandwidth, higher the selectivity of the circuit. As shown in the model graph, the bandwidth AB is given by $f_2 - f_1$. f_1 is the lower cut off frequency and f_2 is the upper cut off frequency.

Q - FACTOR:

In the case of a RLC series circuit, Q-factor is defined as the voltage magnification in the circuit at resonance. At resonance, current is maximum. $I_o = V/R$.

The applied voltage $V = I_oR$

Voltage magnification = $V_L/V = I_oX_L$

In the case of resonance, high Q factor means not only high voltage, but also higher sensitivity of tuning circuit. Q factor can be increased by having a coil of large inductance, not of smaller ohmic resistance.

$Q = \omega L / R$

FORMULAE USED:

Resonant frequency $f_r = \dfrac{1}{2\pi\sqrt{LC}}$ Hz

Bandwidth $BW = f_2 - f_1$

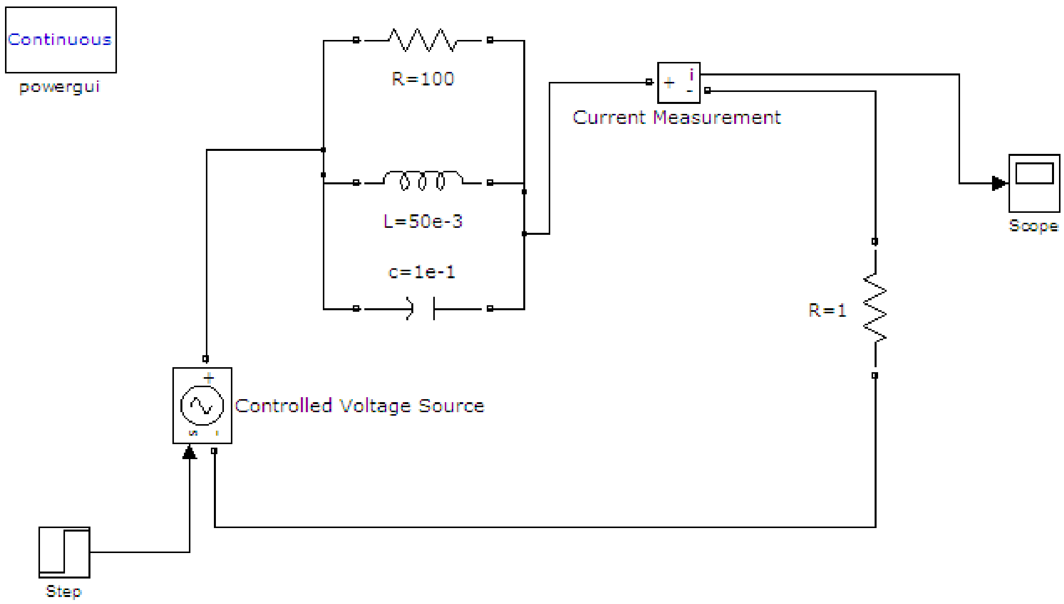
Quality Factor = $\dfrac{f_r}{BW}$

PROCEDURE:

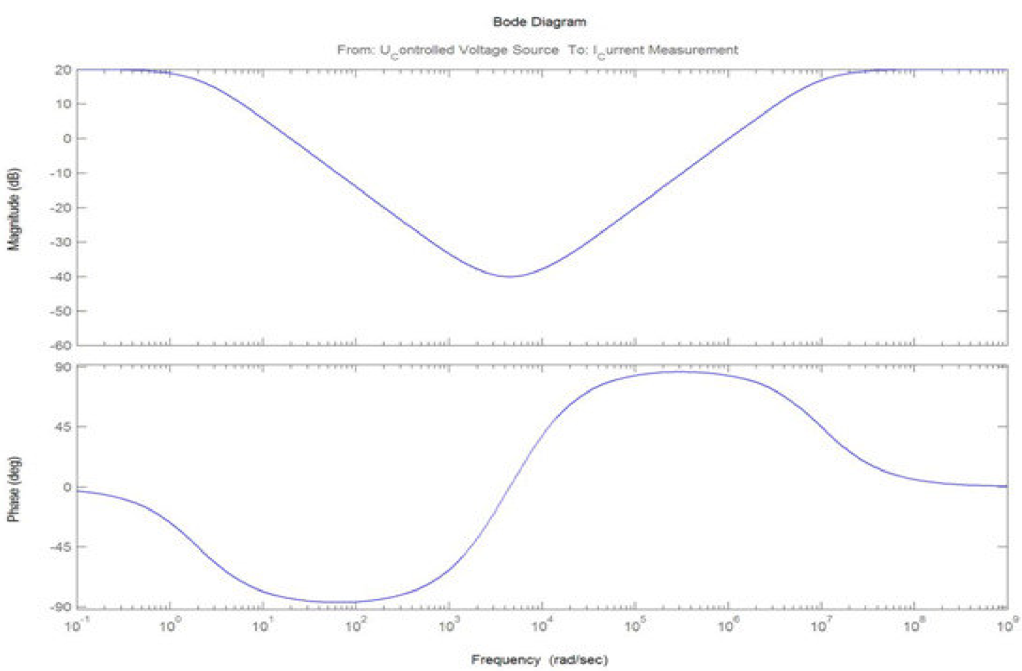
1. Connect the circuit as per the circuit diagram.
2. Vary the frequency and note down the corresponding meter reading.
3. Draw the current Vs frequency curve and measure the bandwidth, resonant frequency and Q factor.

.

MATLAB SIMULATION:



PLOT OF MAGNITUDE & PHASE ANGLE OF CURRENT FOR VARIOUS FREQUENCIES:



SIMULATION PROCEDURE:

1. Open a new MATLAB/SIMULINK model or PSpice CAPTURE project.
2. Connect the circuit as shown in the figure.
3. Debug and run the circuit.
4. By double clicking the power gui plot the value of current for the different values of frequencies (for MATLAB Simulink).
5. For PSpice CAPTURE run the model create simulation profile and run the model

VIVA QUESTIONS:

1. Define Bandwidth.
2. Define Quality factor.
3. What is meant by selectivity?
4. Give the significance of Q- factor.
5. What is meant by resonance?
6. What are the characteristics of a parallel resonant circuit?
7. What will be the power factor of the circuit at resonance?

RESULT:

Thus the current Vs frequency graphs of series and parallel resonant circuits were plotted and the bandwidth, resonant frequency and Q factor were measured. They were found to be

(a) Parallel resonance

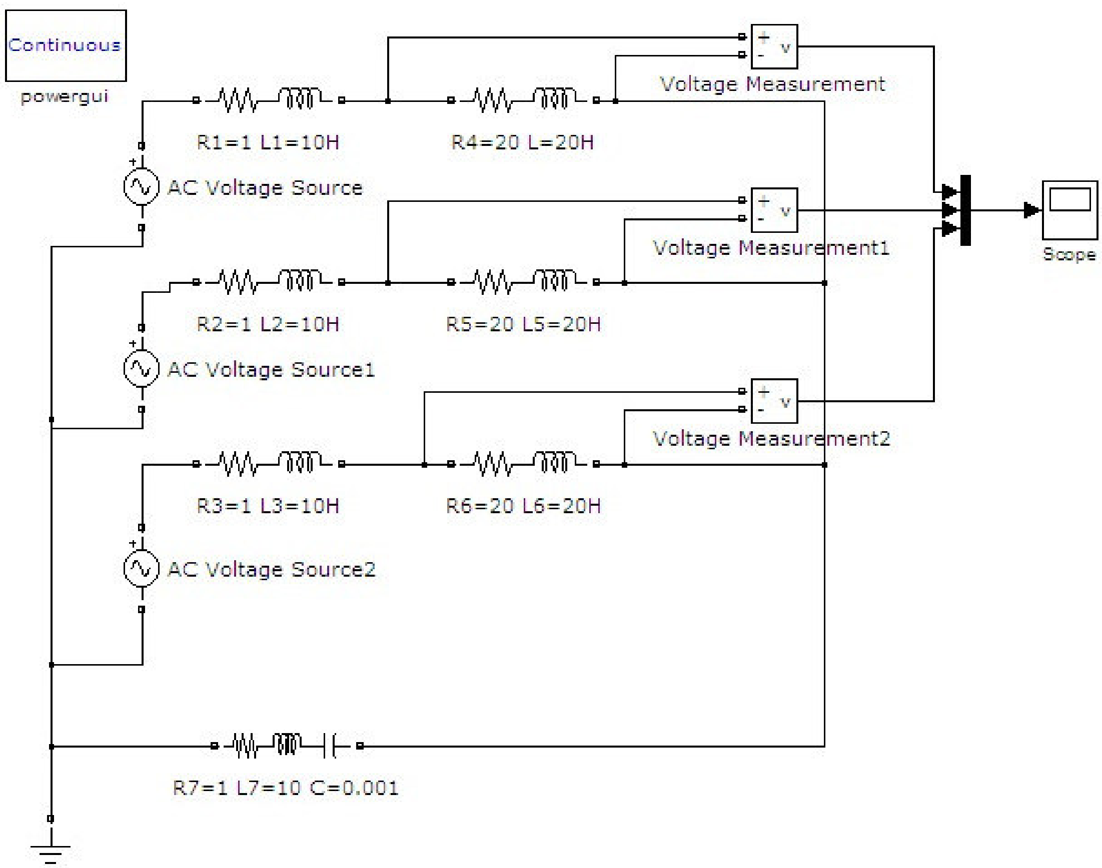
Resonant frequency = ____________

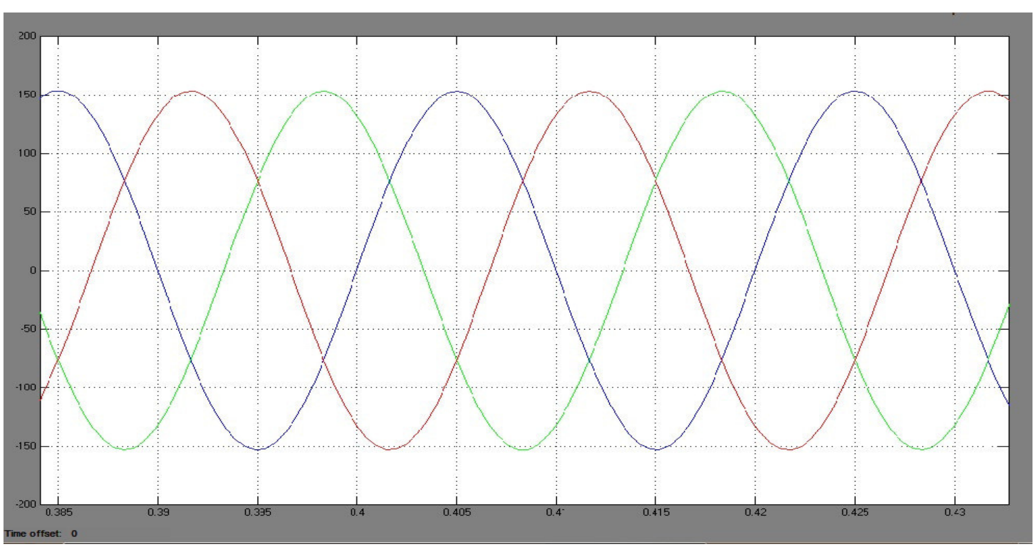
Bandwidth = ____________

Q- Factor = ____________

SIMULATTION DIAGRAM:

3 Φ BALANCED STAR CONNECTED NETWORK:





EXP NO.:

DATE :

SIMULATION OF THREE PHASE BALANCED AND UNBALANCED STAR, DELTA NETWORKS CIRCUITS

AIM:

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

SOFTWARE REQUIRED:

Matlab 7.1

THEORY:

BALANCED THREE- PHASE CIRCUIT:

Balanced phase voltages are equal in magnitude and are out of phase with each other by 120°.The phase sequence is the time order in which the voltages pass through their respective maximum values. A balanced load is one in which the phase impedances are equal in magnitude and in phase.

POSSIBLE LOAD CONFIGURATIONS:

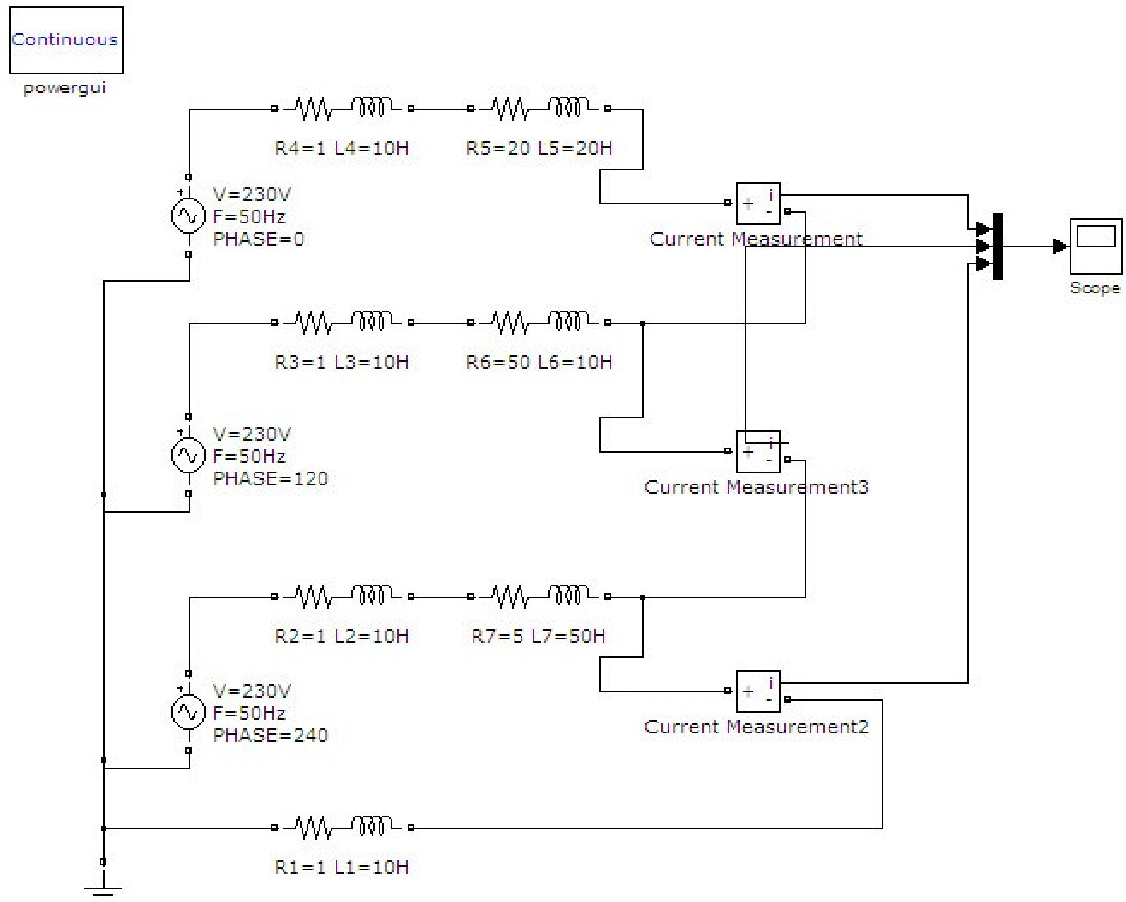
Four possible connections between source and load:

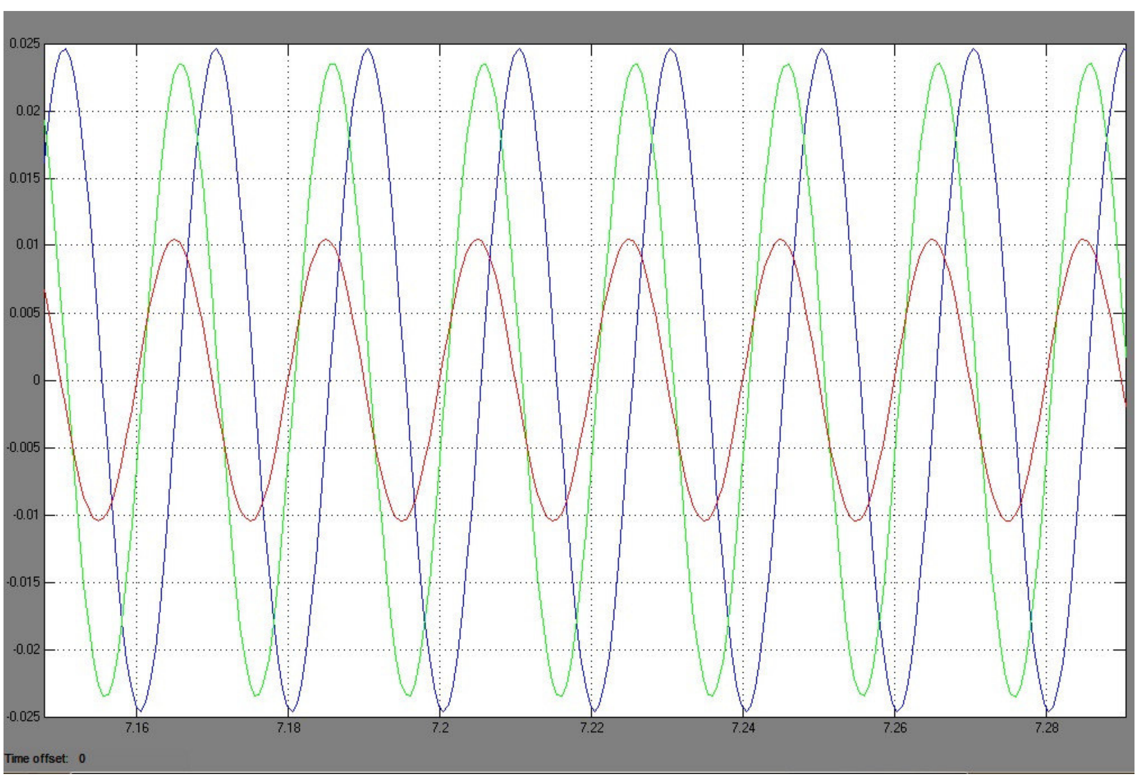
1. Y-Y connection (Y-connected source with a Y-connected load)
2. Y-Δ connection (Y-connected source with a Δ-connected load)
3. Δ-Δ connection
4. Δ-Y connection

UNBALANCED THREE- PHASE CIRCUIT:

An unbalanced system is due to unbalanced voltage sources or an unbalanced load. To calculate power in an unbalanced three-phase system requires that we find the power in each phase. The total power is not simply three times the power in one phase but the sum of the powers in the three phases.

3 Φ UNBALANCED DELTA CONNECTED NETWORK:





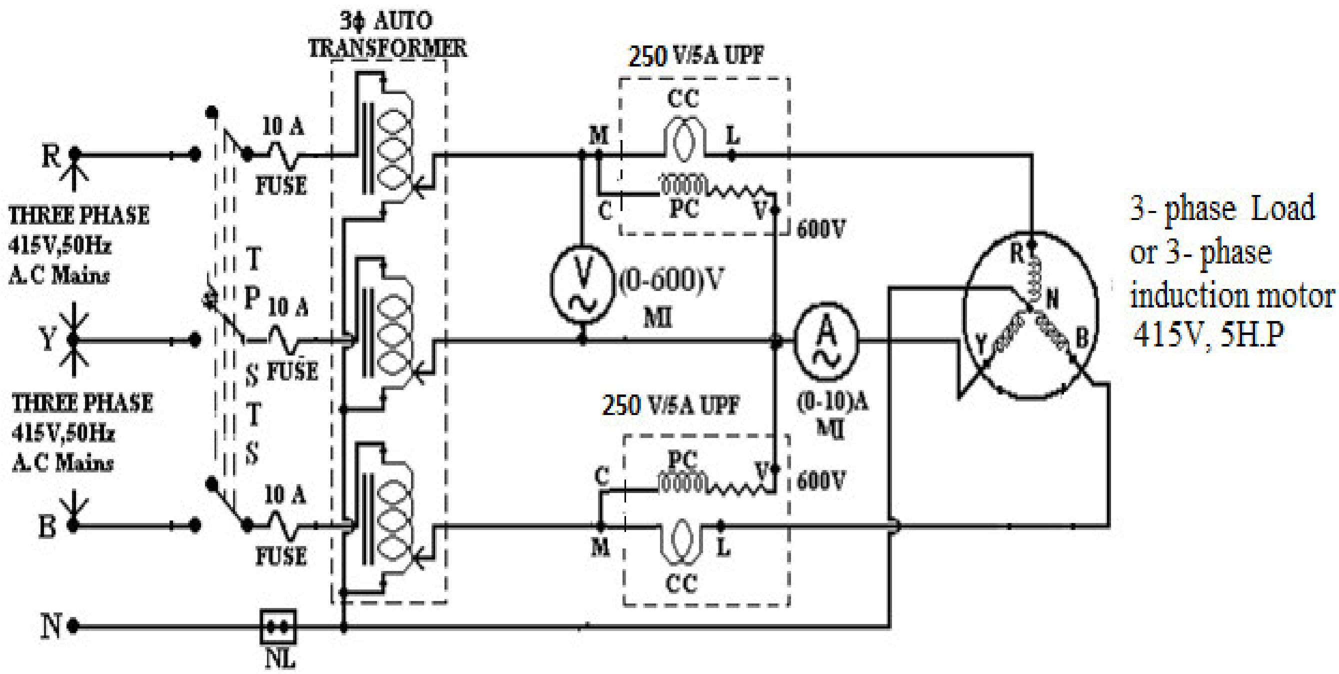
VIVA QUESTIONS:

1. What do you meant by balanced circuit?
2. List the possible load configuration?
3. What is mean by unbalanced circuit?

RESULT:

Thus the three phase balanced and unbalanced star, delta network circuits were simulated and verified.

CIRCUIT DIAGRAM:



EXP NO.:

DATE :

EXPERIMENTAL DETERMINATION OF POWER IN THREE PHASE CIRCUITS BY TWO-WATT METER METHOD

AIM:

To determine the power in three-phase balanced and unbalanced circuit using two-watt meter method.

APPARATUS REQUIRED:

SLNO	NAME OF ITEM	SPECIFICATION	QUANTITY
1.	3-phase Auto transformer	20 Amp. 440v 50 Hz	1
2.	Ammeter	MI(0-10A)	1
3.	Voltmeter	MI(0-600V)	1
4.	Wattmeter	250v, 5A	2
5.	3- phase Load or 3- phase induction motor	415V, 5H.P	1
6	Connecting wires	-	Few

THEORY:

Two wattmeter method can be employed to measure power in a 3- phase,3 wire star or delta connected balance or unbalanced load. In this method, the current coils of the watt meters are connected in any two lines say R and Y and potential coil of each watt meters is joined across the same line and third line i.e. B. Then the sum of the power measured by two watt meters W1 and W2 is equal to the power absorbed By the 3- phase load

PROCEDURE:

1. Connect the Voltmeter, Ammeter and Watt meters to the load through 3ϕ Auto transformer as shown fig and set up the Autotransformer to Zero position.
2. Switch on the 3ϕ A.C. supply and adjust the autotransformer till a suitable voltage.

OBSERVATION TABLE:

S. No	Voltmeter reading V_L	Ammeter reading I_L	Wattmeter reading (watts)				Total power P	Reactive power Q	Power factor
	(V)	(A)	W_1 observed	W_1 Actual	W_2 Observed	W_2 actual	(watts)	(watts)	

MODEL CALCULATION:

3. Note down the readings of watt meters, voltmeter& ammeter

4. Vary the voltage by Autotransformer and note down the Various readings.

5. Now after the observation switch off and disconnect all the Equipment or remove the lead wire.

FORMULAE USED:

1. Total power or Real power $P = \sqrt{3} V_L I_L COS\phi = W_{1actual} + W_{2actual}$
2. Reactive power of load= $Q=\sqrt{3}(W_{1actual} - W_{2actual})$
3. $\tan \phi = [\sqrt{3}(W_{1actual} - W_{2actual})]/[W_{1actual} + W_{2actual}]$
4. Power factor=$\cos \phi$

PRECAUTION & SOURCES OF ERROR:

1. Proper currents and voltage range must be selected before putting the instruments in the circuit.
2. If any Wattmeter reads backward, reverse its pressure coil connection and the reading as negative.
3. As the supply voltage Fluctuates it is not possible to observe the readings correctly.

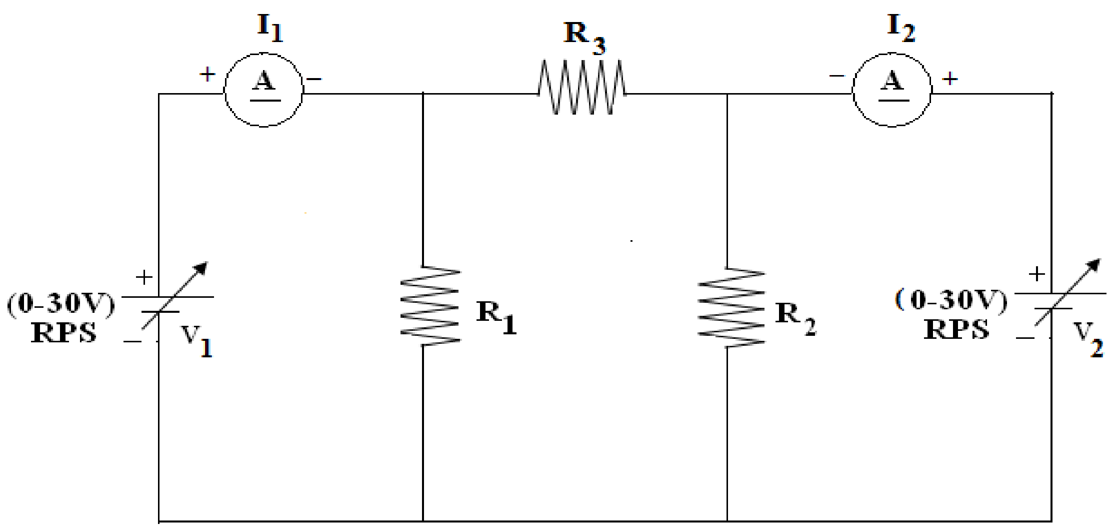
VIVA QUESTIONS:

1. What are the various types of wattmeter?
2. How many coils are there in wattmeter?
3. What is meant by real power?
4. What is meant by apparent power?

RESULT:

The power measured in the 3-phase circuit and there corresponding power factors are in observation table.

CIRCUIT DIAGRAM FOR TWO- PORT NETWORK



OBSERVATION TABLE:

S.No	V_1 (Volts)	V_2 (Volts)	I_1 (Amps)	I_2 (Amps)	Z_{11} (Ohms)	Z_{12} (Ohms)	Z_{22} (Ohms)	Z_{21} (Ohms)

EXP NO.:

DATE :

DETERMINATION OF TWO PORT NETWORK PARAMETERS

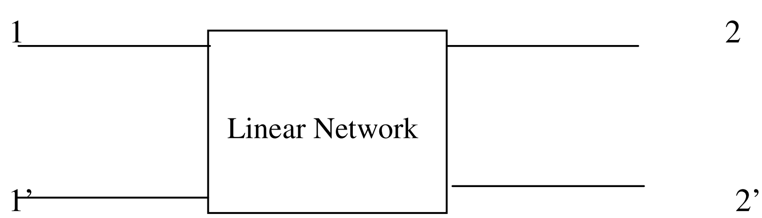
AIM:

To determine the two port parameters for the given electric circuit.

APPARATUS REQUIRED:

S.NO	Name of the Apparatus/Component	Range	Type	Quantity
1	Ammeter		MC	
2	Power Supply			
3	Voltmeter		MC	
4	Connecting wires			
5	Resistors			
6	Breadboard			

THEORY:



The terminal pair where the signal enters the network is called as the INPUT PORT and the terminal pair where it leaves the network is called as the OUTPUT PORT. V_1& I_1 are measured at the Input terminals and V_2 & I_2 are measured at the Output terminals. The two port network parameters express the inter relationship between V_1, I_1,V_2 and I_2 They are Z- parameters, Y- parameters, H-parameters, ABCD parameters and image parameters.

MODEL CALCULATION:

The Impedance parameters are also called as Z parameters.

$$V_1 = Z_{11}I_1 + Z_{12}I_2 \qquad \text{(i)}$$

$$V_2 = Z_{21}I_1 + Z_{22}I_2 \qquad \text{(ii)}$$

where, Z_{11}, $Z_{12,}$ Z_{22} and Z_{21} are constants of the network called Z parameters.

When I_2=0, (Open circuit the output terminal)

$$Z_{11}=V_1/I_1 \qquad \text{(iii)}$$

$$Z_{21}=V_2/I_1 \qquad \text{(iv)}$$

When I_1=0, (Open circuit the Input terminal)

$$Z_{12}=V_1/I_2 \qquad \text{(v)}$$

$$Z_{22}=V_2/I_2 \qquad \text{(vi)}$$

PROCEDURE:

1. Connect the circuit as per the circuit diagram.
2. Open circuit the output terminal (2,2').
3. Vary the power supply to a fixed value and note down the ammeter and voltmeter readings.
4. Open circuit the Input terminal (1,1').
5. Vary the power supply to a fixed value and note down the ammeter and voltmeter readings.
6. Tabulate the readings and calculate the Z parameters.

VIVA QUESTIONS:

1. What is meant by a two-port network?
2. Give the use of two-port network model.
3. What are impedance parameters?
4. What are admittance parameters?
5. What are hybrid parameters?
6. What are ABCD parameters? Mention their significance.

RESULT:

Thus the two port parameters are measured for the given electric circuit.

1907305 DEVICES LABORATORY

CYCLE -II

DEVICES LAB

EXP. 1. A P-N JUNCTION DIODE CHARACTERISTICS

Objective:

To study the Volt-Ampere Characteristics of Silicon P-N Junction Diode and to find cut-in voltage, static and dynamic resistances.

Hardware Required:

S. No	Apparatus	Type	Range	Quantity
01	PN Junction Diode	1N4001		1
02	Resistance		1/2 watt rating, 1k ohm, 10% tolerance,	1
03	Regulated power supply		(0 – 30V), 2A Rating	1
04	Ammeter	MC	(0-30)mA, (0-500)μA	1
05	Voltmeter	MC	(0 – 1)V, (0 – 30)V	1
06	Bread board and connecting wires			

Introduction:

Donor impurities (pentavalent) are introduced into one-side and acceptor impurities into the other side of a single crystal of an intrinsic semiconductor to form a p-n diode with a junction called depletion region (this region is depleted off the charge carriers). This region gives rise to a potential barrier V_γ called **Cut- in Voltage**. This is the voltage across the diode at which it starts conducting. The P-N junction can conduct beyond this Potential.

The P-N junction supports uni-directional current flow. If +ve terminal of the input supply is connected to anode (P-side) and –ve terminal of the input supply is connected to cathode (N- side), then diode is said to be forward biased. In this condition the height of the potential barrier at the junction is lowered by an amount equal to given forward biasing voltage.

Both the holes from p-side and electrons from n-side cross the junction simultaneously and constitute a forward current (**injected minority current** – due to holes crossing the junction and entering N-side of the diode, due to electrons crossing the junction and entering P-side of the diode). Assuming current flowing through the diode to be very large, the diode can be approximated as short-circuited switch. If –ve terminal of the input supply is connected to anode (p-side) and +ve terminal of the input supply is connected to cathode (n-side) then the diode is said to be reverse biased. In this condition an amount equal to reverse biasing voltage increases the height of the potential barrier at the junction.

Both the holes on p-side and electrons on n-side tend to move away from the junction thereby increasing the depleted region. However the process cannot continue indefinitely, thus a small current called **reverse saturation current** continues to flow in the diode. This small current is due to thermally generated carriers. Assuming current flowing through the diode to be negligible, the diode can be approximated as an open circuited switch.

The volt-ampere characteristics of a diode explained by following equation:

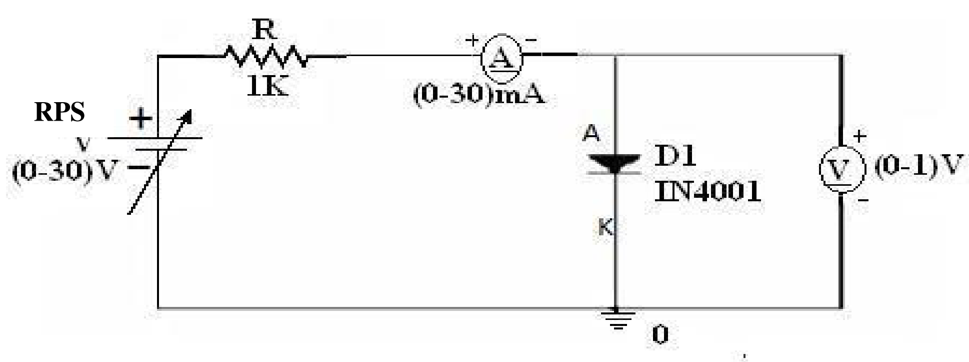
$I = I_o(\exp(V/\eta V_T)-1)$ I=current flowing in the diode I_o=reverse saturation current V=voltage applied to the diode

V_T=volt-equivalent of temperature=kT/q=T/11,600=26mV(@ room temp). η=1 (for Ge) and 2 (for Si)

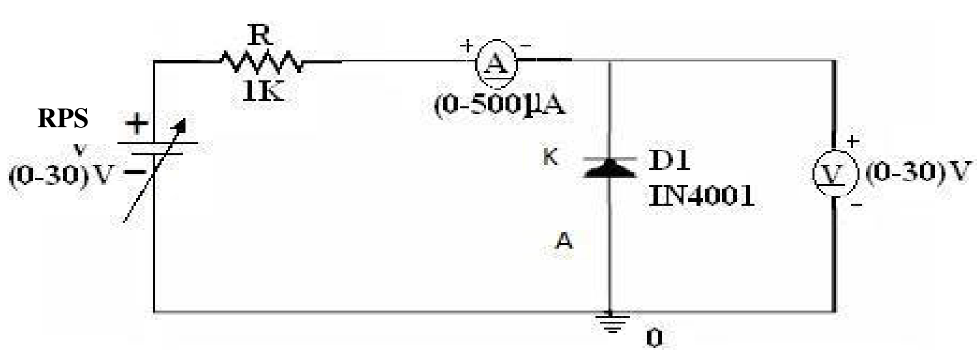
Germanium diode has smaller cut-in-voltage than Silicon diode. The reverse saturation current in Ge diode is larger in magnitude when compared to silicon diode.

Circuit diagram:

Forward Bias



Reverse Bias



Precautions:

1. While doing the experiment do not exceed the ratings of the diode. This may lead to damage of the diode.
2. Connect voltmeter and Ammeter in correct polarities as shown in the circuit diagram.
3. Do not switch **ON** the power supply unless you have checked the circuit connections as per the circuit diagram.

Characteristics of PN junction diode:

1. Breakdown voltage can be traded with switching speed. A reduction in recombination lifetime through addition of suitable impurities will increase leakage current. This can be countered by decreasing diode area which however will lead to reduced forward current rating unless doping is increased. This will lead to a reduced breakdown voltage.
2. The breakdown voltage and reverse recovery are also related together in more direct manner. Regions which have higher doping also have a lower recombination lifetime so that a lower breakdown voltage diode is likely to have lower lifetime and better switching speeds. So a **single diode** cannot meet the diverse applications.

<u>Procedure:</u>

Forward Biased Condition:

1. Connect the PN Junction diode in forward bias i.eAnode is connected to positive of the power supply and cathode is connected to negative of the power supply .
2. Use a Regulated power supply of range (0-30) V and a series resistance of 1kΏ.
3. For various values of forward voltage (Vf) note down the corresponding values of forward current (If).

Reverse biased condition:

4. Connect the PN Junction diode in Reverse bias i.e; anode is connected to negative of the power supply and cathode is connected to positive of the power supply.
5. For various values of (Vr) note down the corresponding values of reverse current (Ir).

Tabular column:

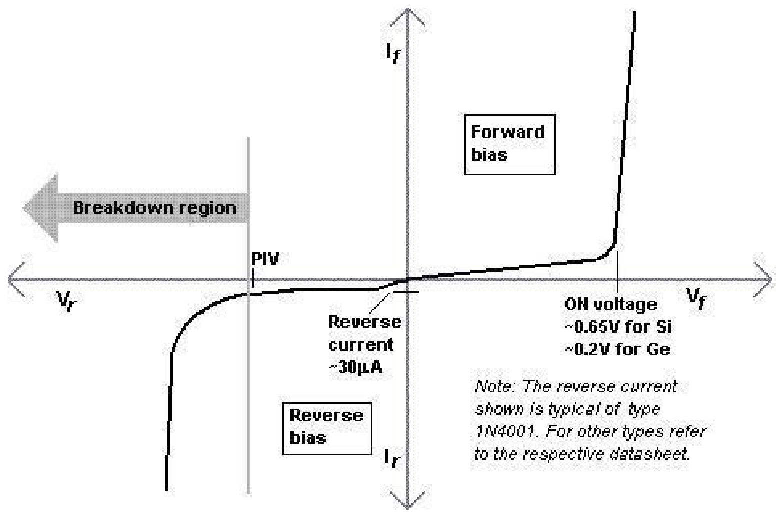
Forward Bias:

S. No	Vf (volts)	If (mA)

Reverse Bias:

S. No	Vr (volts)	Ir (μA)

Model Graph:

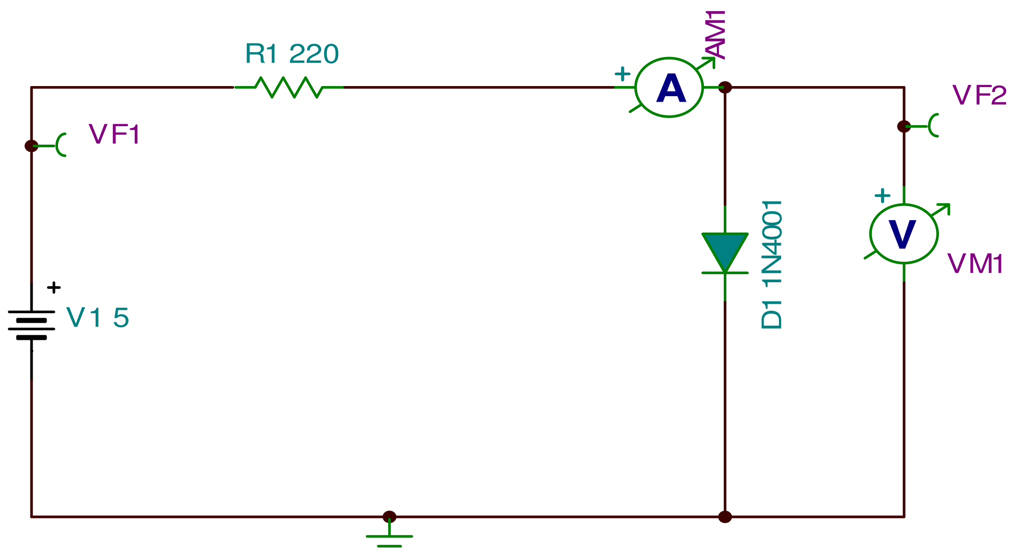


Result:

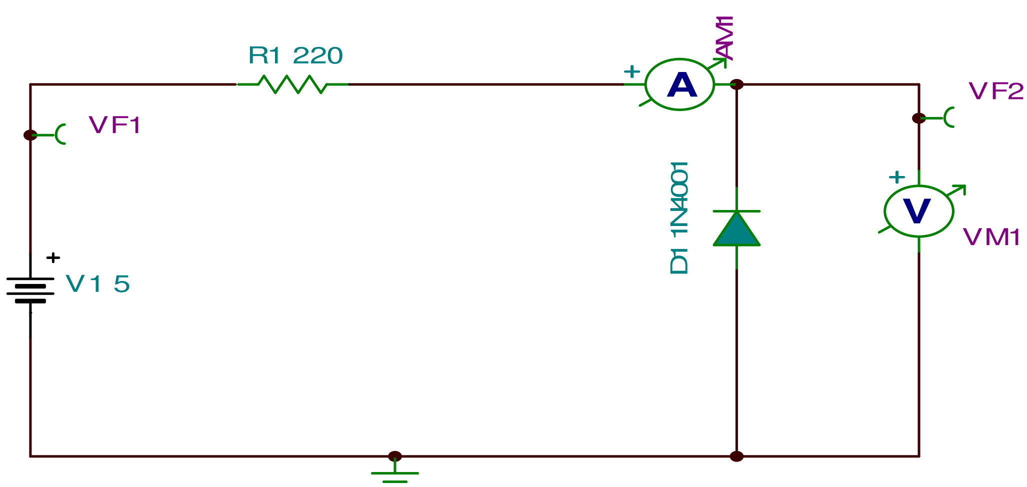
Thus the VI characteristic of PN junction diode was verified.

SIMULATION CIRCUITS:

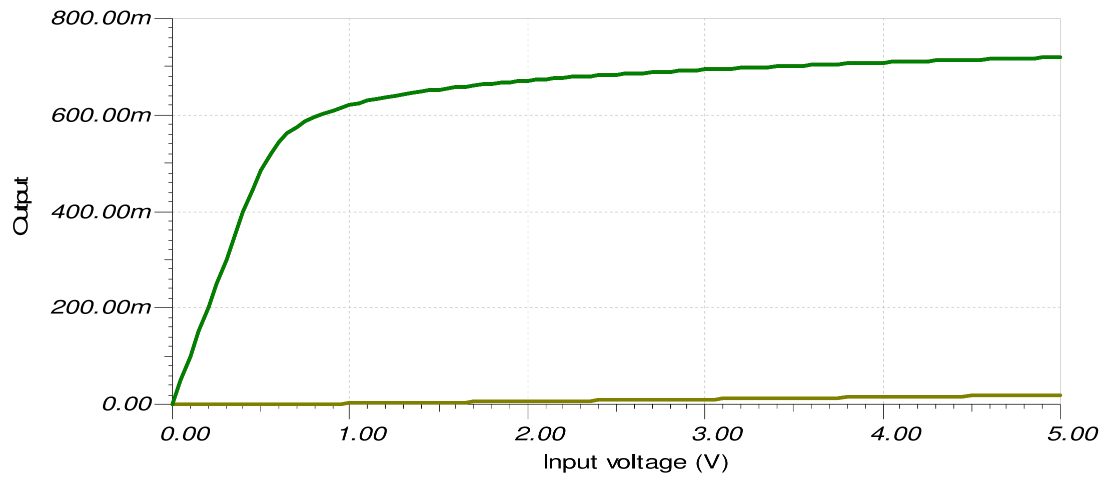
Forwardbias:



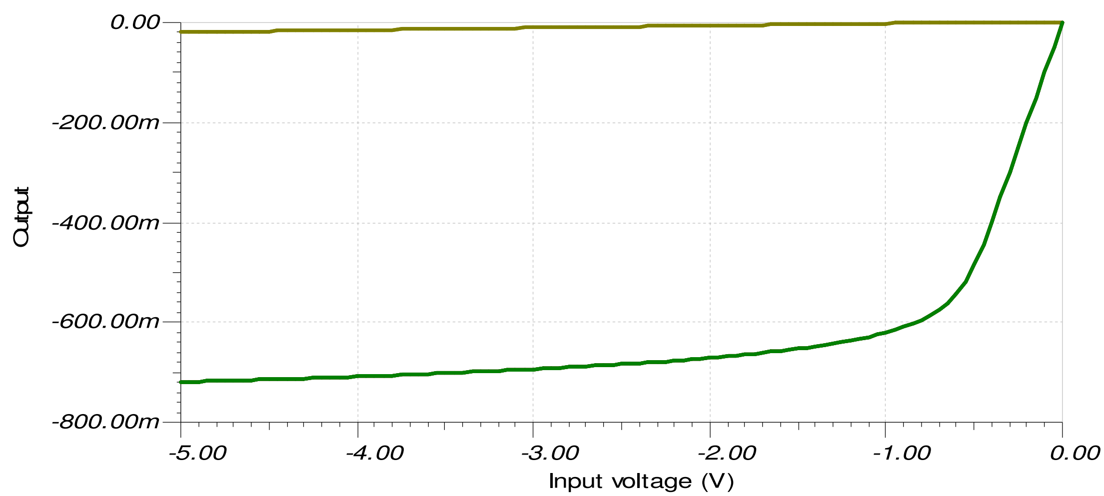
Reverse bias:



Simulation output :
Forward bias



Reverse bias



Questions:

1. What is the need for doping?
2. How depletion region is formed in the PN junction?
3. What is break down voltage?
4. What is cut-in or knee voltage? Specify its value in case of Ge or Si?
5. What are the differences between Ge and Si diode?
6. What is the relationship between depletion width and the concentration of impurities?

EXP 1.B: **ZENER DIODE CHARACTERISTICS**

Objective:

To study the Volt-Ampere characteristics of Zener diode and to measure the Zener break down voltage.

Hardware Required:

S.No	Apparatus	Type	Range	Quantity
01	Zener Diode	IZ 6.2		1
02	Resistance		1k ohm, 10% tolerance, 1/2 watt rating	1
03	Regulated power supply		(0 – 30V), 2A rating	1
04	Ammeter	MC	(0-30)mA	1
05	Voltmeter	MC	(0 – 1)V, (0 – 10)V	1
06	Bread board and connecting wires			

Introduction:

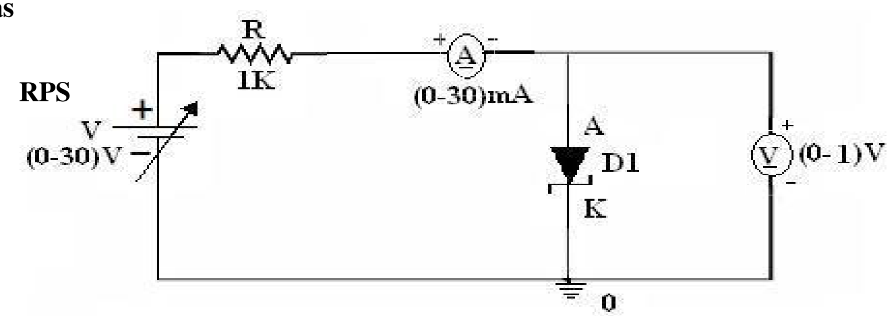
An ideal P-N Junction diode does not conduct in reverse biased condition. A **Zener diode** conducts excellently even in reverse biased condition. These diodesoperate at a precise value of voltage called break down voltage. A **Zener diode** when forward biased behaves like an ordinary P-N junction diode. A **Zener diode** when reverse biased can either undergo **avalanche breakdown** or **Zener breakdown**.

Avalanche breakdown:-If both p-side and n-side of the diode are lightlydoped, depletion region at the junction widens. Application of a very large electric field at the junction may rupture covalent bonding between electrons. Such rupture leads to the generation of a large number of charge carriers resulting in **avalanchemultiplication**.

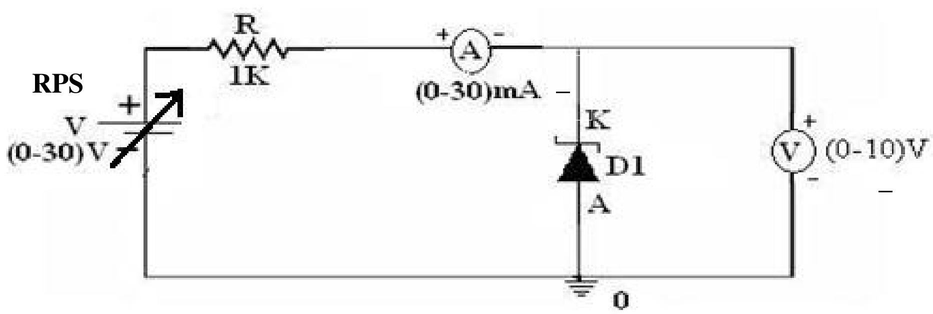
Zener breakdown:-If both p-side and n-side of the diode are heavily doped,depletion region at thejunction reduces. Application of even a small voltage at the junction ruptures covalent bonding and generates large number of charge carriers. Such sudden increase in the number of charge carriers results in **Zener mechanism**.

Circuit diagram:

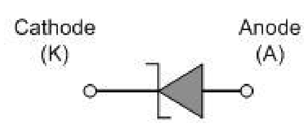
Forward Bias



Reverse Bias



Zener Diode Symbol:



Precautions:

1. While doing the experiment do not exceed the ratings of the diode. This may lead to damage of the diode.
2. Connect voltmeter and Ammeter in correct polarities as shown in the circuit diagram.
3. Do not switch **ON** the power supply unless you have checked the circuit connections as per the circuit diagram.

Characteristics of Zener Diode:

For IZ6.2 Zener diode,

Forward Bias:

At a given (constant) diode current, V exhibits an approximately linear shift in the VI -characteristic due to the combined effect of the temperature dependences of both I_S and V_T

Typically, the VI -characteristic shifts approximately -2 mV/°C.

Reverse Bias:

The temperature dependence of the reverse current is that of IS alone, which changes exponentially as a function of temperature. Typically, I_S approx. doubles for every 10 °C increase in Temperature. These variations may lead to significant changes in the operation of a circuit over a large temperature range and, in many applications, requires compensation strategies to be implemented in the design of some circuits.

Procedure:

Forward Biased Condition:

1. Connect the Zener diode in forward bias i.e; anode is connected to positive of the power supply and cathode is connected to negative of the power supply as in circuit
2. Use a Regulated power supply of range (0-30) V and a series resistance of 1kΏ.
3. For various values of forward voltage (V_f) note down the corresponding values of forward Current (I_f).

Reverse biased condition:

1. Connect the Zener diode in Reverse bias i.e; anode is connected to negative of the power supply and cathode is connected to positive of the power supply as in circuit.
2. For various values of reverse voltage (Vr) note down the corresponding values of reverse current (Ir).

Tabular column:

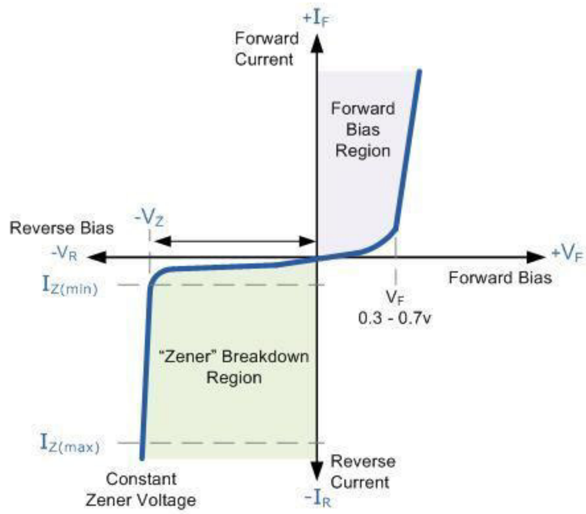
Forward Bias:

S. No	Vf (volts)	If (mA)

Reverse Bias:

S. No	Vr (volts)	Ir (mA)

Model Graph:



Result:

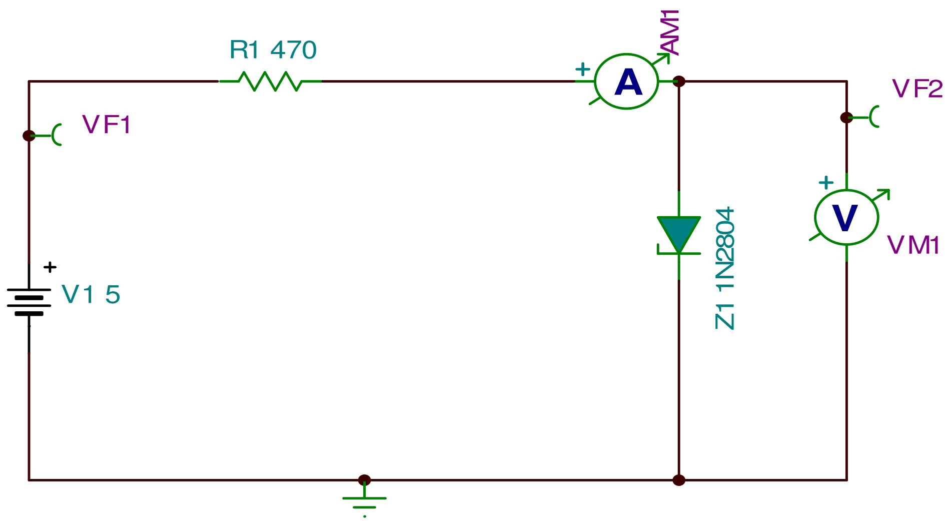
The Zener diode characteristics have been verified and the following parameters were calculated.

i) Cut in voltage= ……… V

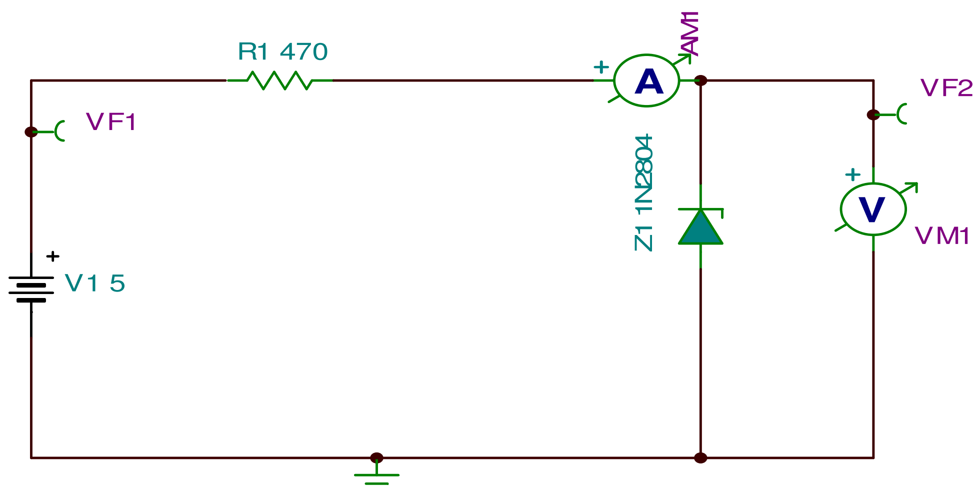
ii) Break down voltage= ……… V

Simulation circuits:

Forward bias:

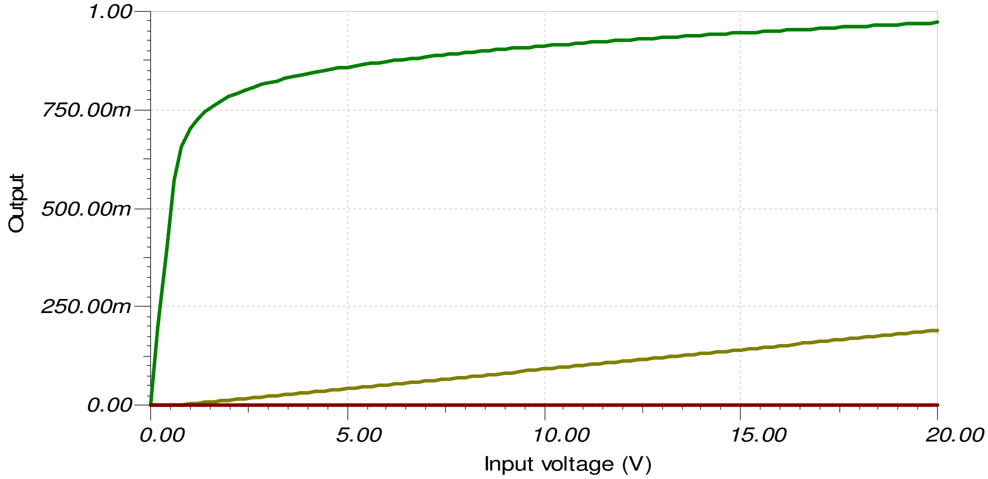


Reverse bias:

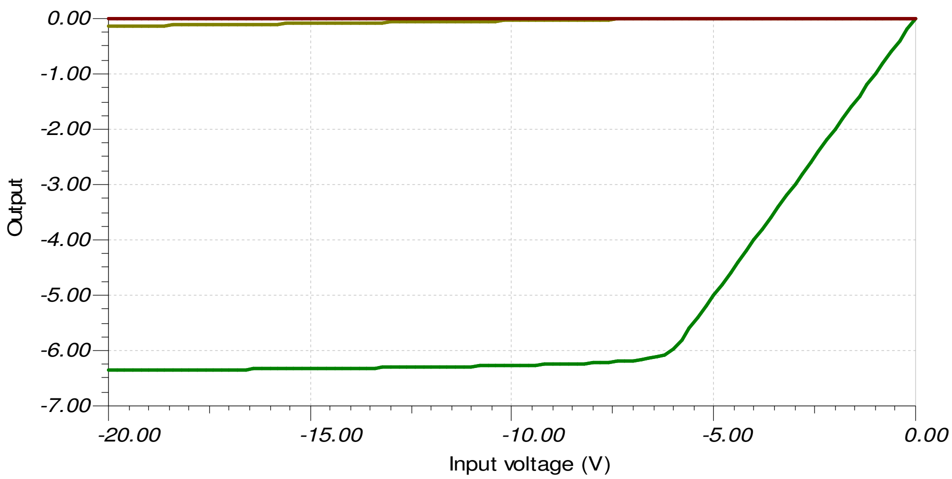


Simulation output:

Forward bias



Reverse bias



Questions:

1. Explain the concept of Zener breakdown?
2. How depletion region gets thin by increasing doping level in Zener diode?
3. State the reason why an ordinary diode suffers avalanche breakdown rather than Zener breakdown?
4. Give the reasons why Zener diode acts as a reference element in the voltage regulator circuits.
5. What type of biasing must be used when a Zener diode is used as a regulator?

EXP. 2: COMMON EMITTER CONFIGURATION OF BJT

Objective:

To study the input and output characteristics of a bipolar junction transistor in Common Emitter configuration and to measure h-parameters

Hardware Required:

S. No	Apparatus	Type	Range	Quantity
01	Transistor	BC147		1
02	Resistance		1k ohm, 10% tolerance, 1/2 watt rating	2
03	Regulated power supply		(0 – 30V), 2A Rating	2
04	Ammeter	MC	(1-30)mA, (0-500)µA	1
05	Voltmeter	MC	(0 – 1)V, (0 – 30)V	1
06	Bread board and connecting wires			

Introduction:

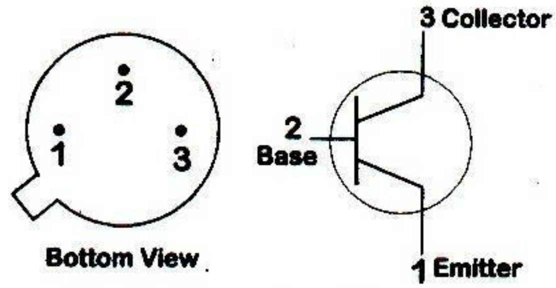
Bipolar junction transistor (BJT) is a 3 terminal (emitter, base, collector) semiconductor device. There are two types of transistors namely NPN and PNP. It consists of two P-N junctions namely emitter junction and collector junction.

In Common Emitter configuration the input is applied between base and emitter and the output is taken from collector and emitter. Here emitter is common to both input and output and hence the name common emitter configuration.

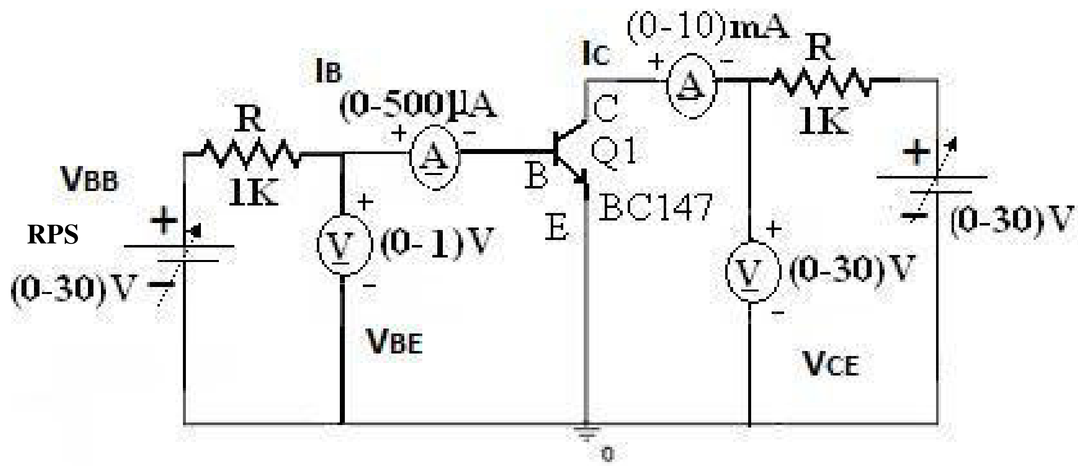
Input characteristics are obtained between the input current and input voltage taking output voltage as parameter. It is plotted between V_{BE} and I_B at constant V_{CE} in CE configuration.

Output characteristics are obtained between the output voltage and output current taking input current as parameter. It is plotted between V_{CE} and I_C at constant I_B in CE configuration.

Pin Assignment:



Circuit Diagram:



Precautions:

1. While doing the experiment do not exceed the ratings of the transistor. This may lead to damage the transistor.
2. Connect voltmeter and Ammeter in correct polarities as shown in the circuit diagram.
3. Do not switch **ON** the power supply unless you have checked the circuit connections as per the circuit diagram.
4. Make sure while selecting the emitter, base and collector terminals of the transistor.

Procedure:

Input Characteristics

1. Connect the transistor in CE configuration as per circuit diagram
2. Keep output voltage V_{CE} = 0V by varying V_{CC}.
3. Varying V_{BB} gradually, note down both base current I_B and base - emitter voltage (V_{BE}).
4. Repeat above procedure (step 3) for various values of V_{CE}

Output Characteristics

1. Make the connections as per circuit diagram.
2. By varying V_{BB} keep the base current I_B = 20μA.
3. Varying V_{CC} gradually, note down the readings of collector-current (I_C) and collector- emitter voltage (V_{CE}).
4. Repeat above procedure (step 3) for different values of I_E

Tabular Column:

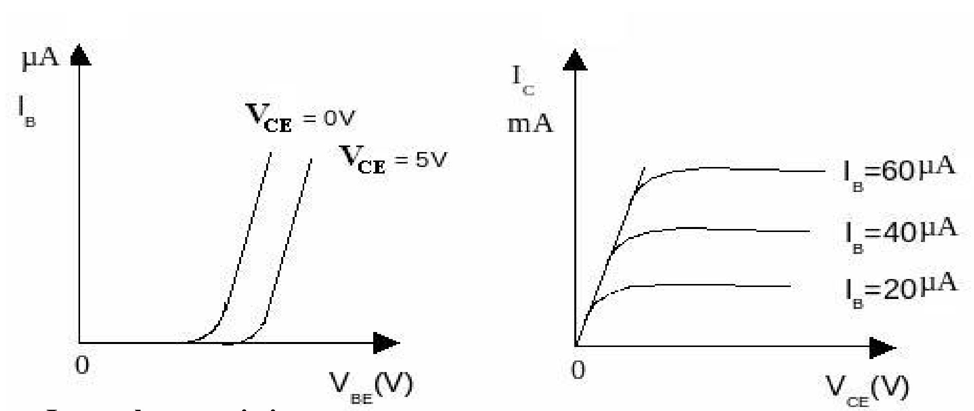
Input Characteristics:

V_{CE} = 0 V		V_{CE} = 4V	
V_{BE} (volts)	I_B (mA)	V_{BE} (volts)	I_B (mA)

Output Characteristics:

I_B = 30 µA		I_B = 60 µA	
V_{CE} (volts)	Ic (mA)	V_{CE} (volts)	Ic (mA)

Model Graph:



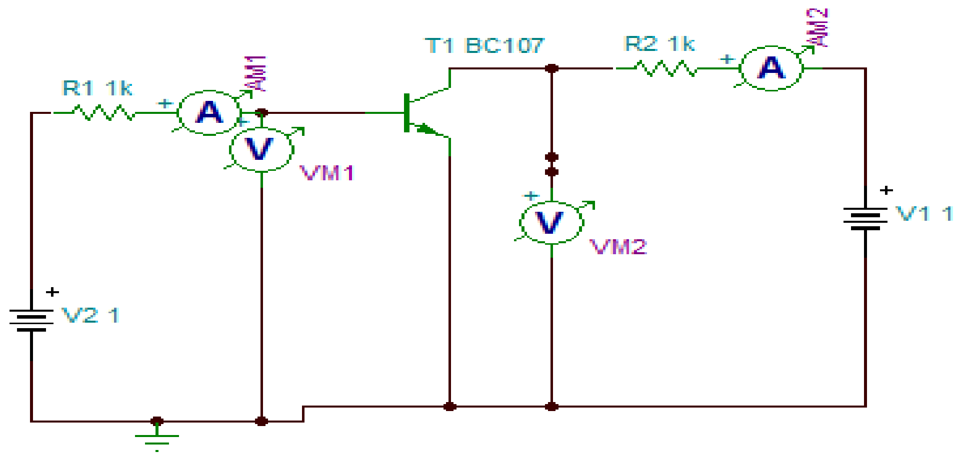
Input characteristics

Output characteristic

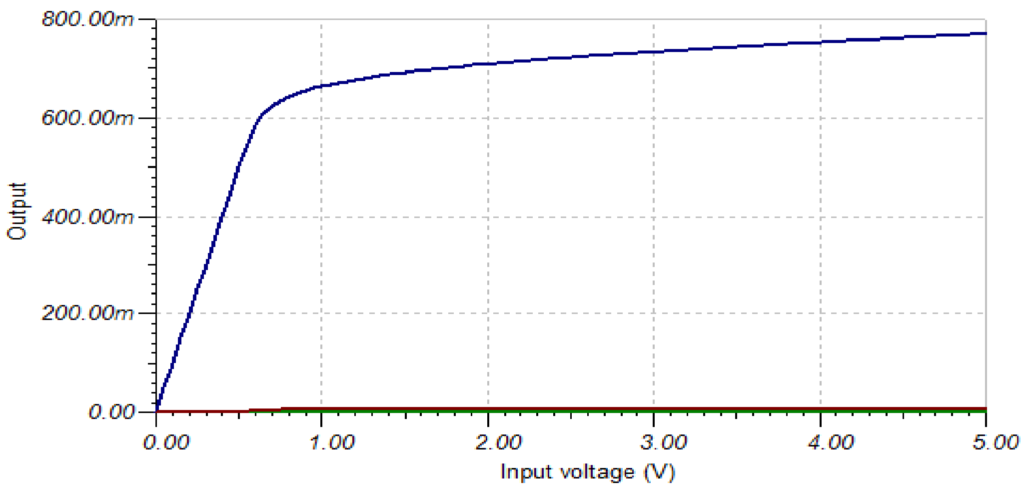
Result:

The common emitter characteristics were studied and analyzed

Simulation Circuit



Output



Questions

1. Why is base width small?
2. Why is Silicon transistor more commonly used compared to Germanium transistor?
3. What is base width modulation?
4. The junction capacitance across collector to base junction is much lower than that across base to emitter junction. Why?
5. What is the difference between diffusion capacitance and transition capacitance?
6. What is the voltage across the collector to emitter terminal when the transistor is in (i) saturation (ii) cut-off (iii) active region?

EXP. 3 : **JFET CHARACTERISTICS**

Objective:

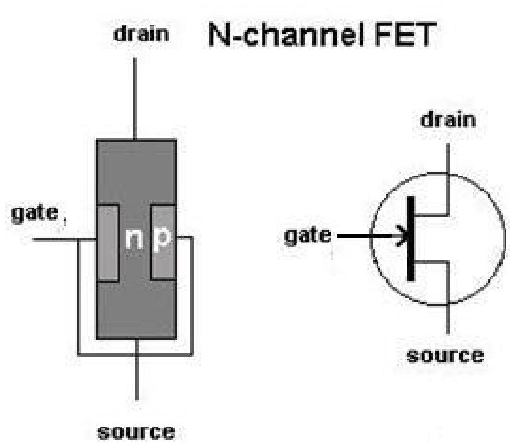
To study drain and transfer characteristics of a FET and measure the parameters

Hardware Required:

S. No	Apparatus	Type	Range	Quantiy
01	JFET	BFW10		1
02	Resistance		1k ohm, 10% tolerance, ½ watt rating	1
03	Regulated power supply		(0 – 30V), 2A Rating	1
04	Ammeter	MC	(0-30)mA	1
05	Voltmeter	MC	(0 – 30)V	2
06	Bread board and connecting wires			

Introduction:

The field effect transistor (FET) is made of a bar of N type material called the SUBSTRATE with a P type junction (the gate) diffused into it. With a positive voltage on the drain, with respect to the source, electron current flows from source to drain through the CHANNEL. If the gate is made negative with respect to the source, an electrostatic field is created which squeezes the channel and reduces the current.

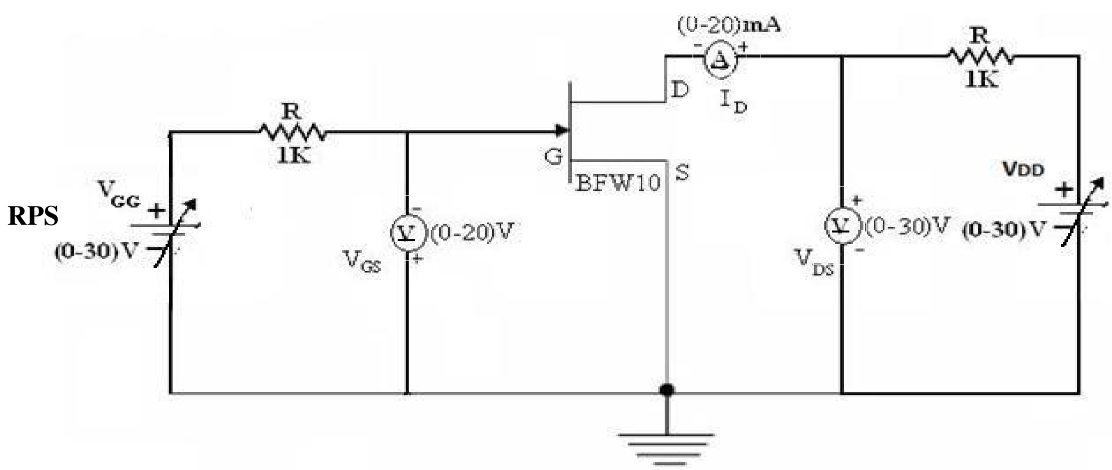


If the gate voltage is high enough the channel will be "pinched off" and the current will be zero. The FET is voltage controlled, unlike the transistor which is current controlled. This device is sometimes called the junction FET or IGFET or JFET.

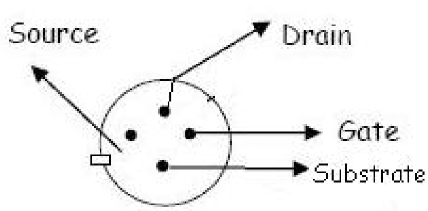
If the FET is accidentally forward biased, gate current will flow and the FET will be destroyed. To avoid this, an extremely thin insulating layer of silicon oxide is placed between the gate and the channel.

The device is then known as an insulated gate FET, or IGFET or metal oxide semiconductor FET (MOSTFET) Drain characteristics are obtained between the drain to source voltage (VDS) and drain current (ID) taking gate to source voltage (VGS) as the parameter. Transfer characteristics are obtained between the gate to source voltage (VGS) and Drain current (ID) taking drain to source voltage (VDS) as parameter.

Circuit diagram:



Pin assignment of JFET (BFW10):



BFW10 – Silicon- N channel Depletion mode FET manufactured by MOTOROLA.

Characteristics of JFET:

In BFW10 JFET,

1. The transconductance g_m of JFET at zero gate –source voltage is in the range of 0.1 to 10mA/V. (Since drain current is proportional to g_m, for more transconductance MOSFETs can be preferred).
2. Gate leakage current of JFET is in the range of 100µA to 10nA, which consumes power (whereas MOSFET has 100nA to 10pA).
3. Greater susceptibility to damage in its handling.

Precautions:

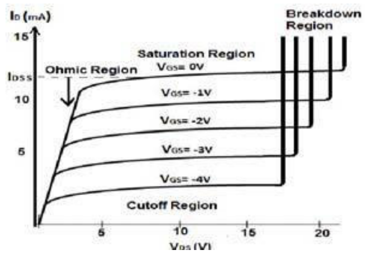
1. While doing the experiment do not exceed the ratings of the FET. This may lead to damage the FET.
2. Connect voltmeter and Ammeter in correct polarities as shown in the Circuit diagram.
3. Do not switch ON the power supply unless you have checked the Circuit connections as per the circuit diagram.
4. Make sure while selecting the Source, Drain and Gate terminals of the FET.

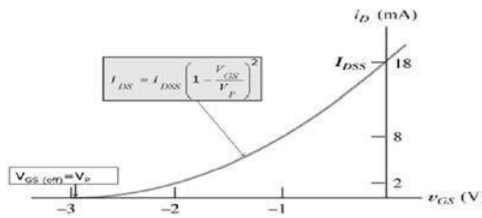
Tabular Column:

Drain Characteristics				**Transfer Characteristics**			
V_{GS} =		**V_{GS} =**		**V_{DS} =**		**V_{DS} =**	
V_{DS}(V)	**I_D(mA)**	**V_{DS}(V)**	**I_D(mA)**	**V_{GS} (V)**	**I_D(mA)**	**V_{GS} (V)**	**I_D(mA)**

Model Graph:

Drain Characteristics **Transfer Characteristics:**





Graph (Instructions):

1. Plot the drain characteristics by taking V_{DS} on X-axis and I_D on Y-axis at constant V_{GS}.
2. Plot the Transfer characteristics by taking V_{GS} on X-axis and I_D on Y-axis at constant V_{DS}.

Calculations from Graph:

Drain Resistance (rd):

It is given by the ration of small change in drain to source voltage (ΔV_{DS}) to the corresponding change in Drain current (ΔI_D) for a constant gate to source voltage (V_{GS}), when the JFET is operating in pinch-off or saturation region.

Trans-Conductance (gm):

The ratio of small change in drain current (ΔI_D) to the corresponding change in gate to source voltage (ΔV_{GS}) for a constant V_{DS} {gm = $\Delta I_D / \Delta V_{GS}$ at constant V_{DS} (from transfer characteristics)}.The value of gm is expressed in mhos or Siemens (s).

Amplification Factor (μ)**:**

It is given by the ratio of small change in drain to source voltage (ΔV_{DS}) to the corresponding change in gate to source voltage (ΔV_{GS}) for a constant drain current.

$\mu = \Delta V_{DS} / \Delta V_{GS}$.

$\mu = (\Delta V_{DS} / \Delta I_D) \text{ X } (\Delta I_D / \Delta V_{GS})$

$\mu = rd \text{ X } gm$.

Result:

Thus the drain and transfer characteristics of a JFET was studied and verified, the following parameters were calculated

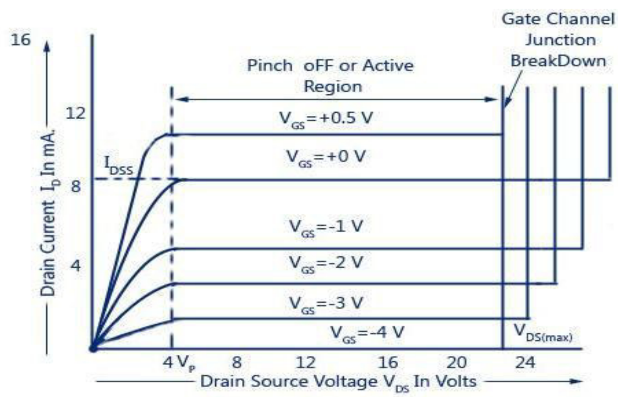
i. Drain Resistance (rd) = ………….

ii. Transconductance (gm) = ………….

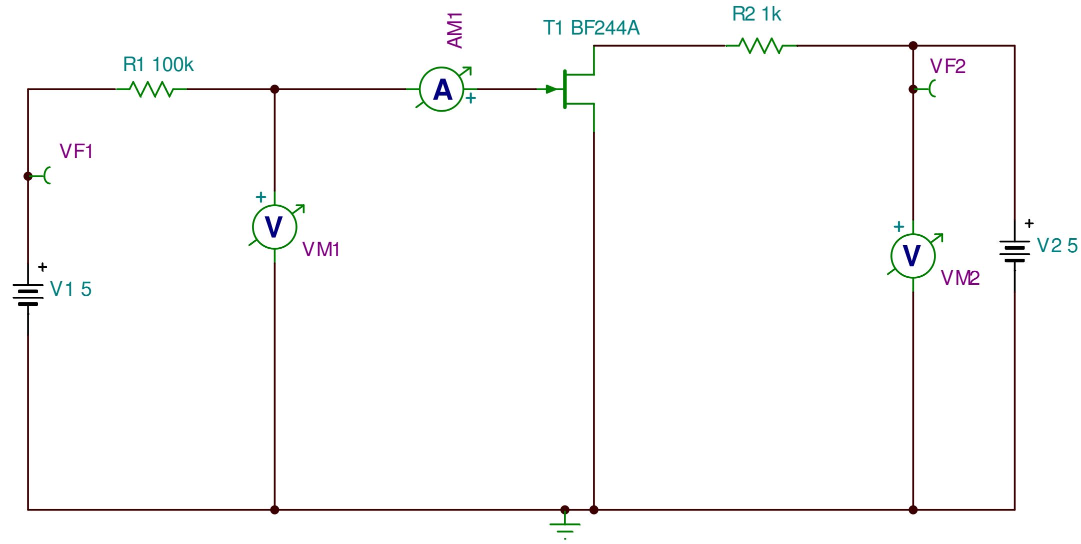
iii. Amplification factor (μ) = ……………

Questions:

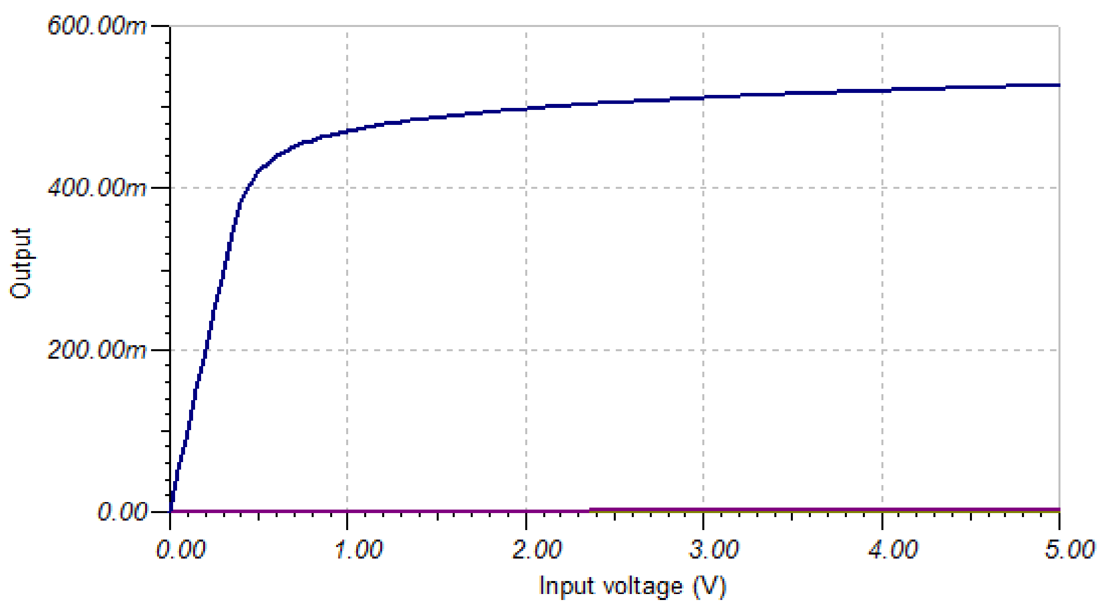
1. Why FET is called as a Unipolar transistor?
2. What are the advantages of FET over BJT?
3. State why FET is voltage controlled device?
4. JFET operates in either Enhancement mode or Depletion mode or both. Why?
5. How electron flows through P-channel JFET?
6. Why current gain is an important parameter in BJT where as conductance is important parameter in FET?
7. Why does drain current remains physically constant above pinch off voltage?
8. What are the conditions to be satisfied for the JFET to work in pinch off or active region?
9. State the reason for non symmetrical nature of depletion region around gate to source junction.
10. Interpret transfer characteristics curve from the following drain characteristics.



Simulation :



Output:



EXP. 4.A **HALF WAVE RECTIFIER**

Objective:

1. To plot Output waveform of the Half Wave Rectifier.
2. To find ripple factor for Half Wave Rectifier using the formulae.
3. To find the efficiency, Vr(pp), Vdc for Half Wave Rectifier.

Hardware Required:

S. No	Apparatus	Type	Range	Quantity
01	Transformer		6-0-6 V, 500mA, 1A Rating	1
02	Resistance		470 ohm, 10% tolerance, 1/2 watt rating	1
03	Capacitor		470µF	1
04	Diode	IN4001		1
05	Bread board and connecting wires			

Introduction:

A device is capable of converting a sinusoidal input waveform into a unidirectional waveform with non-zero average component is called a rectifier. A practical half wave rectifier with a resistive load is shown in the circuit diagram. In positive half cycle, Diode D is forward biased and conducts. Thus the output voltage is same as the input voltage. In the negative half cycle, Diode D is reverse biased, and therefore output voltage is zero. A smoothing filter is induced between the rectifier and load in order to attenuate the ripple component. The filter is simply a capacitor connected from the rectifier output to ground. The capacitor quickly charges at the beginning of a cycle and slowly discharges through R_L after the positive peak of the input voltage. The variation in the capacitor voltage due to charging and discharging is called ripple voltage. Generally, ripple is undesirable, thus the smaller the ripple, the better the filtering action.

Ripple factor is a measure of effectiveness of a rectifier circuit and defined as a ratio of RMS value of ac component to the dc component in the rectifier output.

Theoretical calculations for Ripple Factor:

Without Filter:

$V_{rms} = V_m / 2$

$$V_{dc} = \frac{V_m}{\pi}$$

Ripple factor (Theoretical) $= \sqrt{(\frac{V_{rms}}{V_{dc}})^2 - 1} = 1.21$

Ripple Factor(practical) $\gamma = \frac{V_{ac}}{V_{dc}}$ where $V_{ac} = \sqrt{(V_{rms}^2 - V_{dc}^2)}$

With Filter:

Ripple Factor (Theoretical) $r = \frac{1}{2\sqrt{3}fCR}$

Where f = 50Hz, R = 1K Ω, C = 1000 μF

$$V_{ac} = \frac{V_{r(p-p)}}{2\sqrt{3}}$$

$$V_{dc} = V_m - \frac{V_{r(p-p)}}{2}$$

Ripple Factor(practical) $\gamma = \frac{V_{ac}}{V_{dc}}$

Percentage Regulation $= \frac{V_{NL} - V_{FL}}{V_{FL}} * 100$ %

V_{NL} = DC voltage at the load without connecting the load (Minimum current).

V_{FL} = DC voltage at the load with load connected.

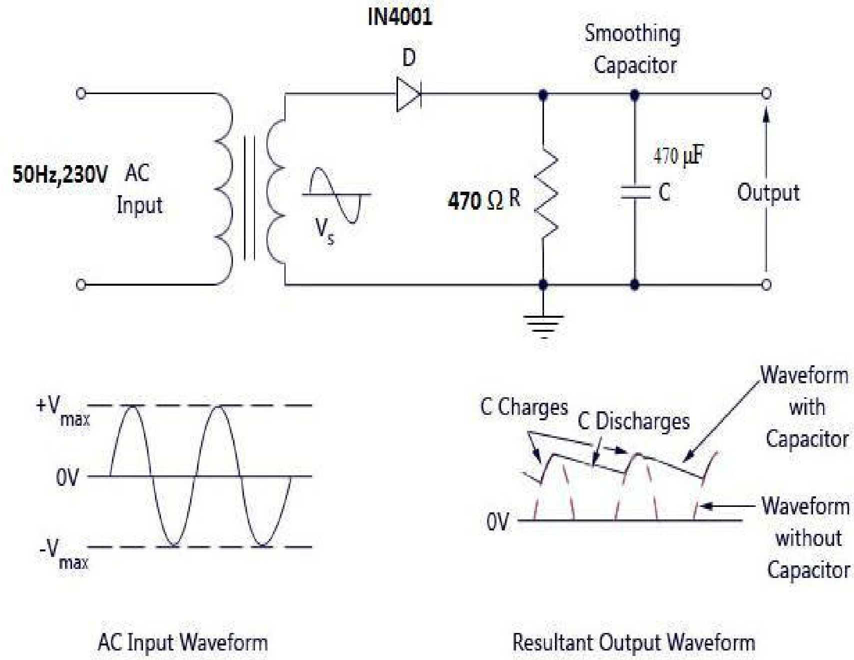
Efficiency $\eta = \frac{P_{DC}}{P_{AC}}$

$P_{AC} = V^2_{rms} / R_L$

$P_{DC} = V^2_{dc} / R_L$

The ripple factor can be lowered by increasing the value of the filter capacitor or increasing the load capacitance.

Circuit Diagram of Half Wave Rectifier



Characteristics of Half Wave Rectifier:

1. The output current in the load contains, in addition to dc component, ac components of basic frequency equal to that of the input voltage frequency. Ripple factor is high and an elaborate filtering is, therefore, required to give steady dc output.
2. The power output and, therefore, rectification efficiency is quite low. This is due to the fact that power is delivered only during one half cycle of the input alternating voltage.
3. Transformer utilization factor is low.
4. DC saturation of transformer core resulting in magnetizing current and hysteresis losses and generation of harmonics.

Precautions:

1. While doing the experiment do not exceed the ratings of the diode. This may lead to damage the diode.
2. Connect CRO using probes properly as shown in the circuit diagram.
3. Do not switch **ON** the power supply unless you have checked the circuit connections as per the circuit diagram.

Experiment:

1. Connections are given as per the circuit diagram without capacitor.
2. Apply AC main voltage to the primary of the transformer. Feed the rectified output voltage to the CRO and measure the time period and amplitude of the waveform.
3. Now connect the capacitor in parallel with load resistor and note down the amplitude and time period of the waveform.
4. Measure the amplitude and time period of the transformer secondary (input waveform) by connecting CRO.
5. Plot the input, output without filter and with filter waveform on a graph sheet.
6. Calculate the ripple factor.

Graph (instructions):

1. Take a graph sheet and divide it into 2 equal parts. Mark origin at the center of the graph sheet.
2. Now mark x-axis as Time and y-axis as Voltage
3. Mark the readings tabulated for Amplitude as Voltage and Time in graph sheet.

Observations:

	Input Waveform	**Output Waveform** (without filter)	**Ripple Voltage** (with filter)
Amplitude			
Time Period			
Frequency			

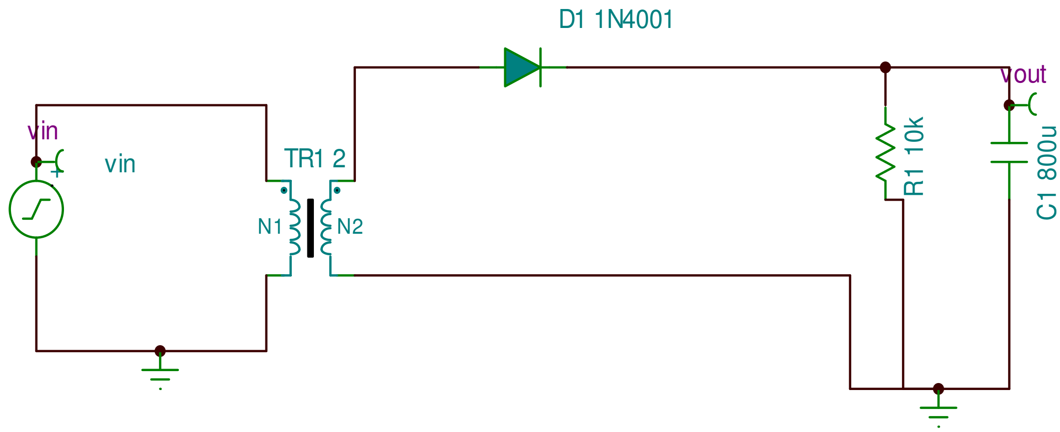
Result:

The Rectified output Voltage of Half Wave Rectifier Circuit is observed and the calculated value of ripple factor is ________________

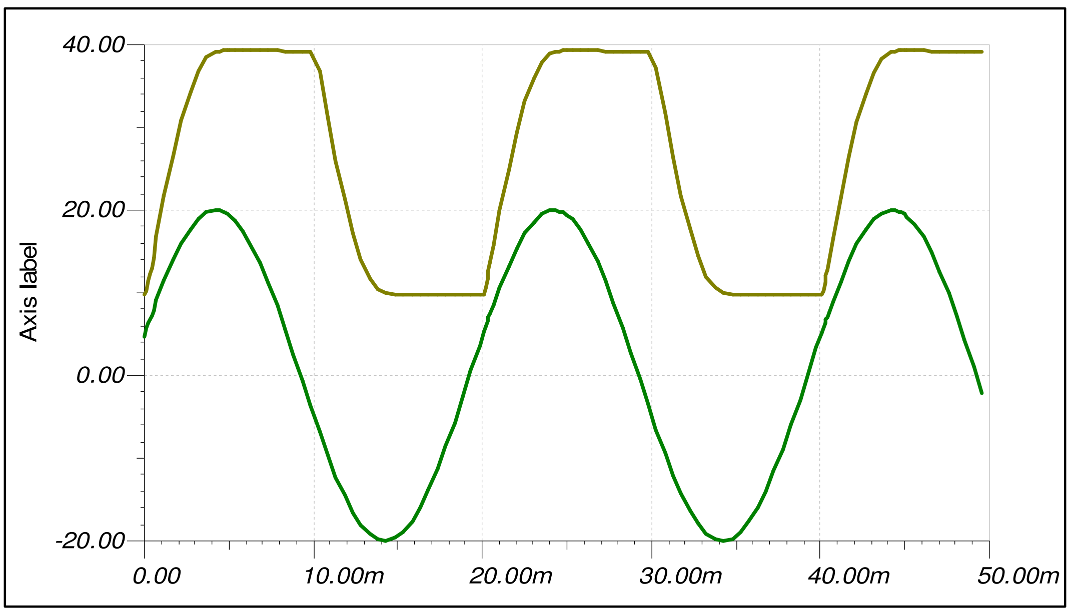
Pre Lab

1. What is the purpose of a rectifier?
2. Why filter is used in a rectifier?
3. What are the advantages of half wave rectifier?
4. Define Ripple factor and Efficiency. State the ideal values.
5. Define PIV. Give the PIV of Half wave rectifier.

Simulation :



HALF WAVE RECTIFIER SIMULATED IN CIRCUIT SIMULATOR IV SEMESTER EIE ELCTRON DEVICES LAB



EXP.4.B **FULL WAVE RECTIFIER**

Objective:

1. To plot Output waveform of the Full Wave Rectifier.
2. To find ripple factor for Full Wave Rectifier using the formulae.
3. To find the efficiency, Vp(rect), Vdc for Full Wave Rectifier.

Hardware Required:

S. No	Apparatus	Type	Range	Quantity
01	Transformer		6-0-6 V	1
02	Resistance		470 ohm, 10% tolerance, 1/2 watt	1
03	Capacitor		470µF	1
04	Diode	IN4001		2
05	Bread board and connecting wires			

Introduction:

A device is capable of converting a sinusoidal input waveform into a unidirectional waveform with non zero average component is called a rectifier.
A practical half wave rectifier with a resistive load is shown in the circuit diagram. It consists of two half wave rectifiers connected to a common load. One rectifies during positive half cycle of the input and the other rectifying the negative half cycle. The transformer supplies the two diodes (D1 and D2) with sinusoidal input voltages that are equal in magnitude but opposite in phase. During input positive half cycle, diode D1 is ON and diode D2 is OFF. During negative half cycle D1 is OFF and diode D2 is ON. Peak Inverse Voltage (PIV) is the maximum voltage that has to be withstand by a diode when it is reverse biased. Peak inverse voltage for Full Wave Rectifier is $2V_m$ because the entire secondary voltage appears across the non-conducting diode **.**

The output of the Full Wave Rectifier contains both ac and dc components. A majority of the applications, which cannot tolerate a high value ripple, necessitates further processing of the rectified output. The undesirable ac components i.e. the ripple, can be minimized using filters.

Ripple Factor:

Ripple factor is defined as the ratio of the effective value of AC components to the average DC value. It is denoted by the symbol ' γ'.

$$\gamma = \frac{V_{ac}}{V_{dc}}, (\gamma = 0.48)$$

Efficiency:

The ratio of output DC power to input AC power is defined as efficiency.

$$\eta = \frac{(V_{dc})^2}{(V_{ac})^2}$$

η= 81% (if R >> R_f, then R_f can be neglected).

The maximum efficiency of a Full Wave Rectifier is 81.2%.

Percentage of Regulation:

It is a measure of the variation of DC output voltage as a function of DC output current (i.e., variation in load).

Percentage of regulation = $(\frac{V_{NL} - V_{FL}}{V_{FL}}) * 100$ %

V_{NL} = Voltage across load resistance, when minimum current flows through it.

V_{FL} = Voltage across load resistance, when maximum current flows through.

For an ideal Full-wave rectifier, the percentage regulation is 0 percent. The percentage of regulation is very small for a practical full wave rectifier.

Peak- Inverse - Voltage (PIV):

It is the maximum voltage that the diode has to withstand when it is reverse biased. PIV = $2V_m$

Transformer Utilization Factor

Transformer utilization factor (TUF), which is defined as the ratio of power delivered to the load and ac rating of the transformer secondary, So TUF = dc power delivered to the load/ac rating of transformer secondary

Transformer Utilization Factor, TUF can be used to determine the rating of a transformer secondary. It is determined by considering the primary and the secondary winding separately and it gives a value of 0.693.

Theoretical Calculations:

Without filter:

$$V_{rms} = \frac{V_m}{\sqrt{2}}$$

$$V_{ac} = \sqrt{(V_{rms}^2 - V_{dc}^2)}$$

$$V_{dc} = \frac{2V_m}{\pi}$$

Ripple factor (Theoretical) = $\sqrt{(\frac{V_{rms}}{V_{dc}})^2 - 1} = 0.48$

Ripple Factor (Practical) $\gamma = \frac{V_{ac}}{V_{dc}}$

With filter:

Ripple factor (Theoretical) $\gamma = \frac{1}{4\sqrt{3}fCR}$

Where f = 50Hz, R =1KΩ, C = 1000 μF.

$$V_{ac} = \frac{V_{r(p-p)}}{2\sqrt{3}}$$

$$V_{dc} = V_m - \frac{V_{r(p-p)}}{2}$$

Ripple Factor $\gamma = \frac{V_{ac}}{V_{dc}}$

Percentage Regulation = $(\frac{V_{NL} - V_{FL}}{V_{FL}}) * 100$

V_{NL} = DC voltage at the load without connecting the load (Minimum current).

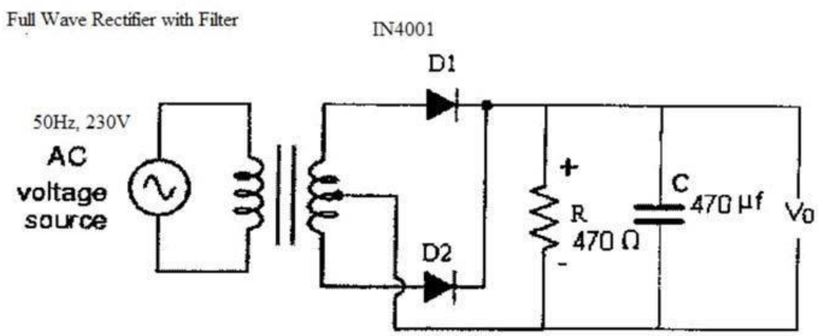
V_{FL} = DC voltage at the load with load connected.

Efficiency $\eta = \frac{P_{DC}}{P_{AC}}$

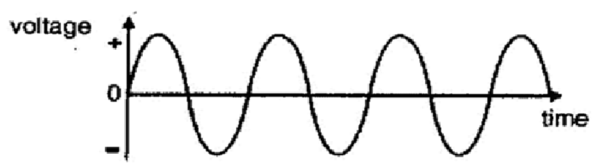
$P_{AC} = V^2_{rms} / R_L$

$P_{DC} = V^2_{dc} / R_L$

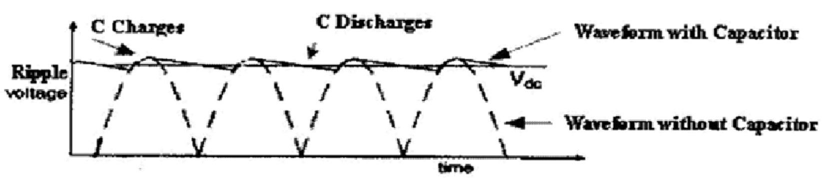
Circuit Diagram:



MODEL GRAPH:



Input Waveform



Resultant Output Waveform

Characteristics of Full Wave Rectifier:

1. The peak voltage in the full-wave rectifier is only half the peak voltage in the half-wave rectifier. This is because the secondary of the power transformer in the full-wave rectifier is center tapped; therefore, only half the source voltage goes to each diode.
2. A larger transformer for a given power output is required with two separate but identical secondary windings making this type of full wave rectifying circuit costly compared to the "Full Wave Bridge Rectifier" circuit equivalent.

Precautions:

1. While doing the experiment do not exceed the ratings of the diode. This may lead to damage the diode.
2. Connect CRO using probes properly as shown in the circuit diagram.
3. Do not switch **ON** the power supply unless you have checked the circuit connections as per the circuit diagram.

Procedure:

1. Connections are given as per the circuit diagram without capacitor.
2. Apply AC main voltage to the primary of the transformer. Feed the rectified output voltage to the CRO and measure the time period and amplitude of the waveform.
3. Now connect the capacitor in parallel with load resistor and note down the amplitude and time period of the waveform.
4. Measure the amplitude and time period of the transformer secondary(input waveform) by connecting CRO.
5. Plot the input, output without filter and with filter waveform on a graph sheet.
6. Calculate the ripple factor.

Graph (instructions)

1. Take a graph sheet and divide it into 2 equal parts. Mark origin at the center of the graph sheet.
2. Now mark x-axis as Time y-axis as Voltage
3. Mark the readings tabulated for/ Amplitude as Voltage and Time in graph sheet.

Observations:

	Input Waveform	**Output Waveform** (without filter)	**Ripple Voltage** (with filter)
Amplitude			
Time Period			
Frequency			

Result: The Rectified output Voltage of Full Wave Rectifier Circuit is observed and the calculated value of ripple factor is _______________

Questions

1. Each diode in a centre-tapped full-wave rectifier is _____ -biased and conducts for _____ of the input cycle.
   a) Forward, 90^0
   b) Reverse, 180^0
   c) Forward, 180^0
   d) Reverse, 90^0
2. What is the VRRM (PIV rating) for the 1N4001 rectifier diode? Give the PIV of Full Wave Rectifier.
3. Define Efficiency of rectifier. Compare the efficiency of Full Wave and Half Wave rectifier.

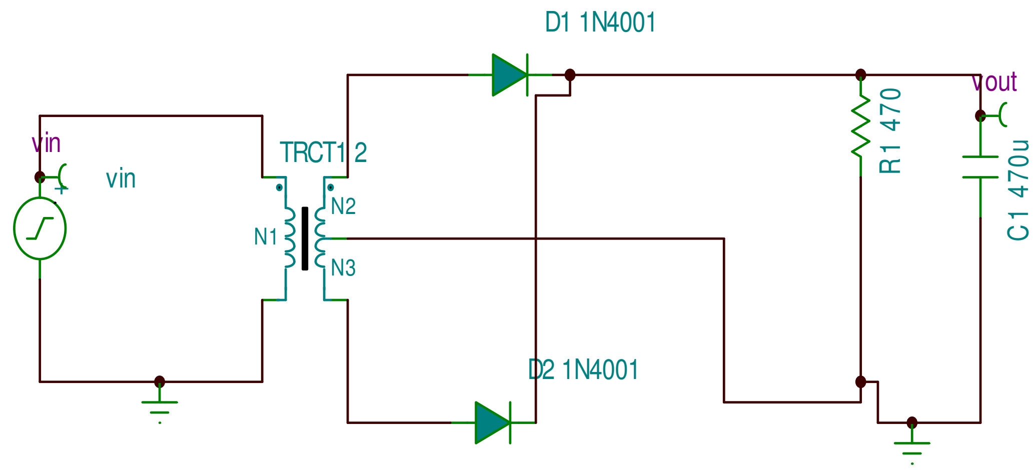
4. Calculate the ripple voltage of a full-wave rectifier with a 75-µF filter capacitor connected to a load drawing 40 mA.

   a. 1.20V b.1.28

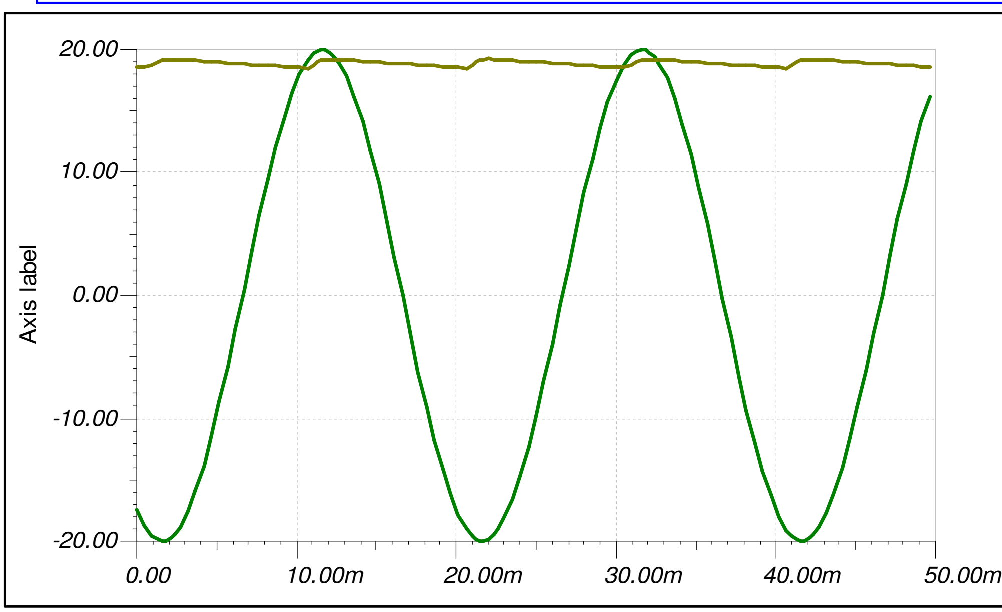
4. The output frequency of a full-wave rectifier is _____ the input frequency
5. In the full-wave rectifier circuit what will happen to the circuit if 1) D1 is disconnected, 2) D1's polarity is reversed.

6. Compare the ripple factors of half wave and full wave rectifier.

Simulation:



FULL WAVE RECTIFIER SIMULATED IN CIRCUIT SIMULATOR IV SEMESTER EIE ELCTRON DEVICES LAB



Expt.No: 5 Simulation of Passive filters

Objective :

To design and simulate passive filters such as Low pass filter , High pass filter and Band pass Filter

Theory :

Passive Low Pass Filter

A Low Pass Filter is a circuit that can be designed to modify, reshape or reject all unwanted high frequencies of an electrical signal and accept or pass only those signals wanted by the circuits designer.Filters are so named according to the frequency range of signals that they allow to pass through them, while blocking or "attenuating" the rest.

The most commonly used filter designs are the:

The Low Pass Filter – the low pass filter only allows low frequency signals from 0Hz to its cut-off frequency, fc point to pass while blocking those any higher.

The High Pass Filter – the high pass filter only allows high frequency signals from its cut-off frequency, fc point and higher to infinity to pass through while blocking those any lower.

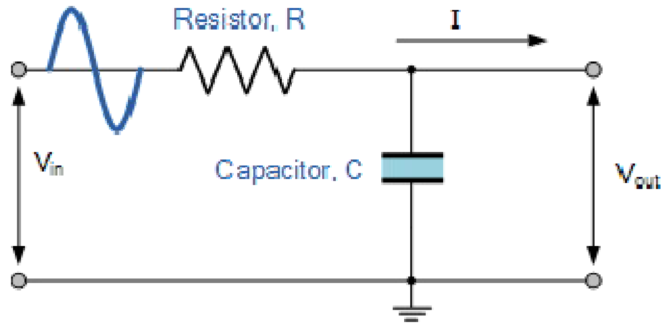
The Band Pass Filter – the band pass filter allows signals falling within a certain frequency band setup between two points to pass through while blocking both the lower and higher frequencies either side of this frequency band.

The Low Pass Filter

A simple passive **RC Low Pass Filter** or LPF, can be easily made by connecting together in series a single Resistor with a single Capacitor as shown below. In this type of filter arrangement the input signal (VIN) is applied to the series combination (both the Resistor and Capacitor together) but the output signal (VOUT) is taken across the capacitor only.

This type of filter is known generally as a "first-order filter" or "one-pole filter", why first-order or single-pole?, because it has only "one" reactive component, the capacitor, in the circuit.

RC Low Pass Filter Circuit



As mentioned previously in the Capacitive Reactance tutorial, the reactance of a capacitor varies inversely with frequency, while the value of the resistor remains constant as the frequency changes. At low frequencies the capacitive reactance, (XC) of the capacitor will be very large compared to the resistive value of the resistor, R.

This means that the voltage potential, VC across the capacitor will be much larger than the voltage drop, VR developed across the resistor. At high frequencies the reverse is true with VC being small and VR being large due to the change in the capacitive reactance value.

While the circuit above is that of an RC Low Pass Filter circuit, it can also be thought of as a frequency dependant variable potential divider circuit similar to the one we looked at in the Resistors tutorial. In that tutorial we used the following equation to calculate the output voltage for two single resistors connected in series.

$$V_{out} = V_{in} \times \frac{R_2}{R_1 + R_2}$$

where: $R_1 + R_2 = R_T$, the total resistance of the circuit

We also know that the capacitive reactance of a capacitor in an AC circuit is given as: $X_C = \frac{1}{2\pi f C}$ in Ohm's

Opposition to current flow in an AC circuit is called impedance, symbol Z and for a series circuit consisting of a single resistor in series with a single capacitor, the circuit impedance is calculated as:

$$Z = \sqrt{R^2 + X_C^2}$$

Then by substituting our equation for impedance above into the resistive potential divider equation gives us:

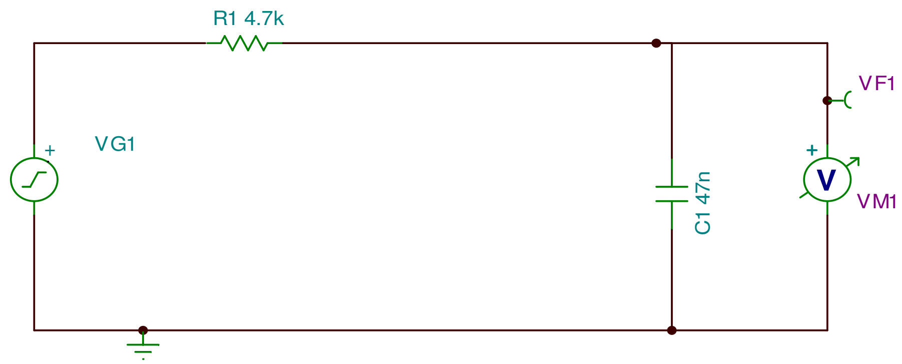
RC Potential Divider Equation

$$V_{out} = V_{in} \times \frac{X_C}{\sqrt{R^2 + X_C^2}} = V_{in} \frac{X_C}{Z}$$

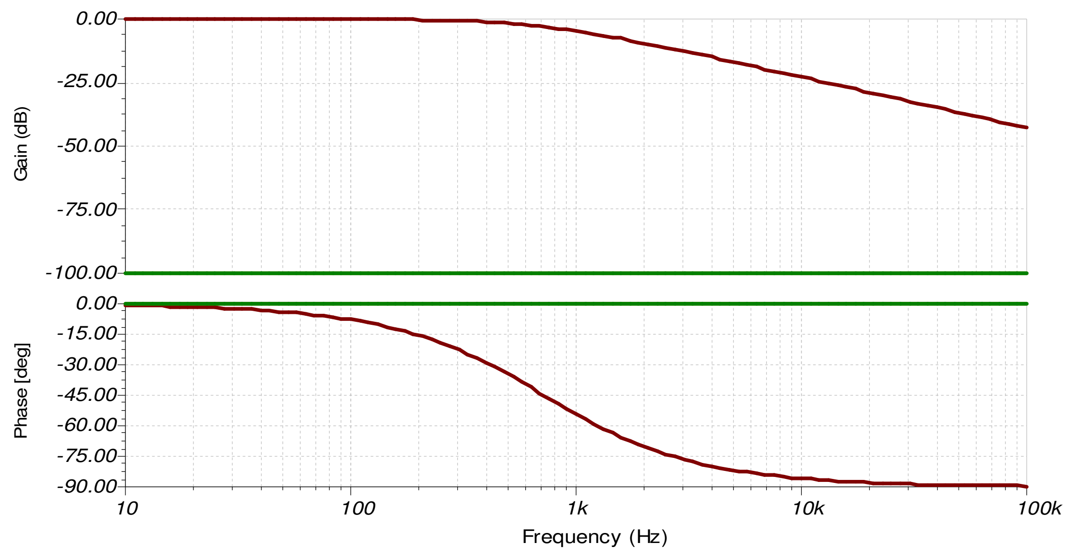
So, by using the potential divider equation of two resistors in series and substituting for impedance we can calculate the output voltage of an RC Filter for any given frequency.

Simulation circuit:

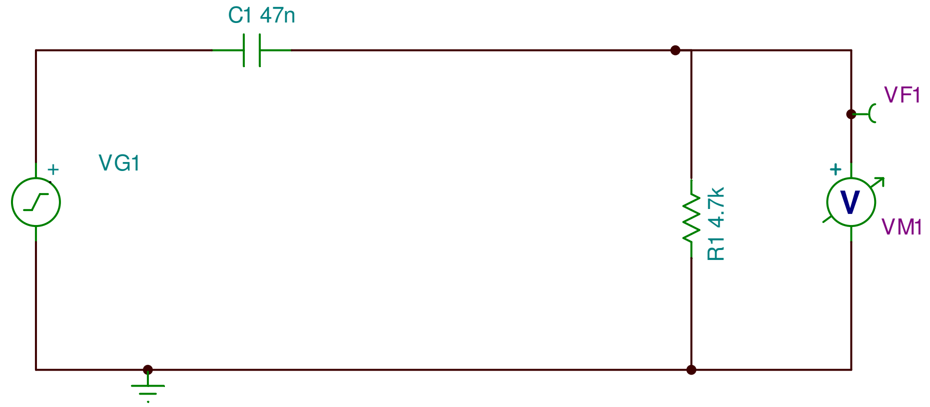
A **Low Pass Filter** circuit consisting of a resistor of 4k7Ω in series with a capacitor of 47nF



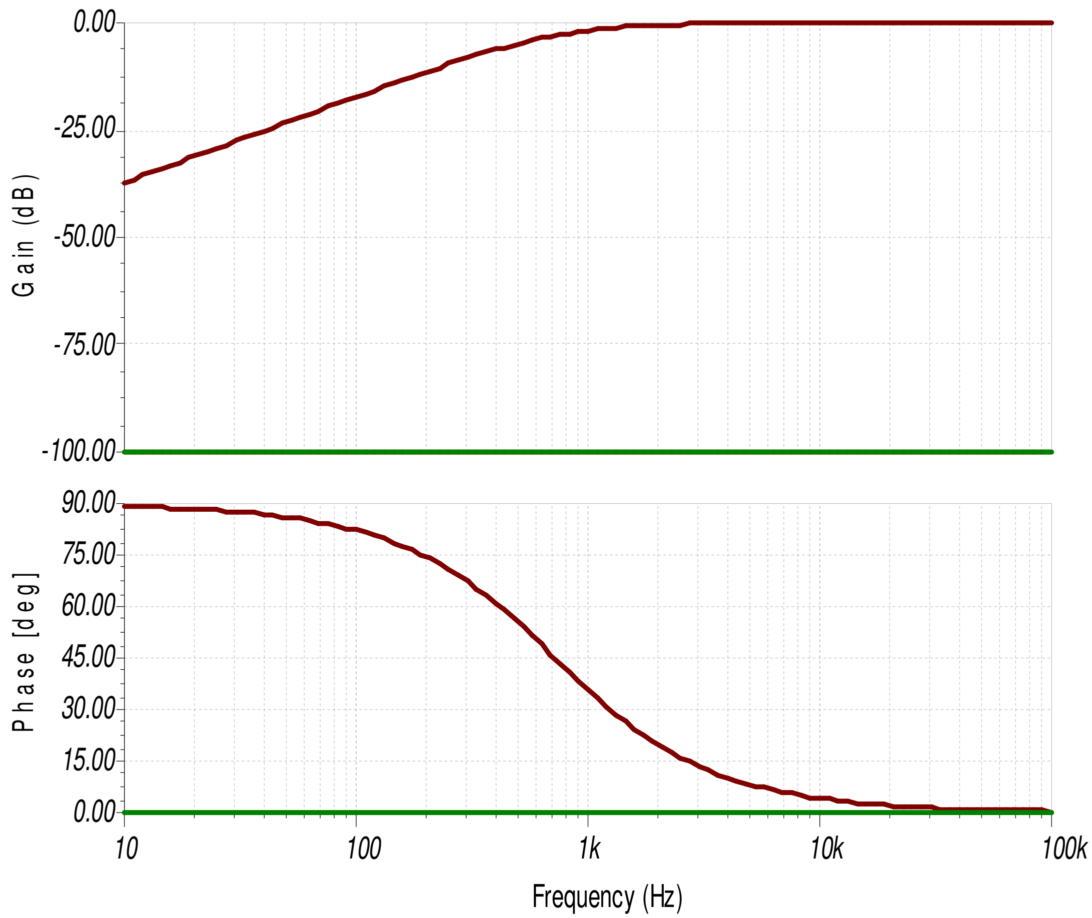
Output Frequency response:



High Pass filter:

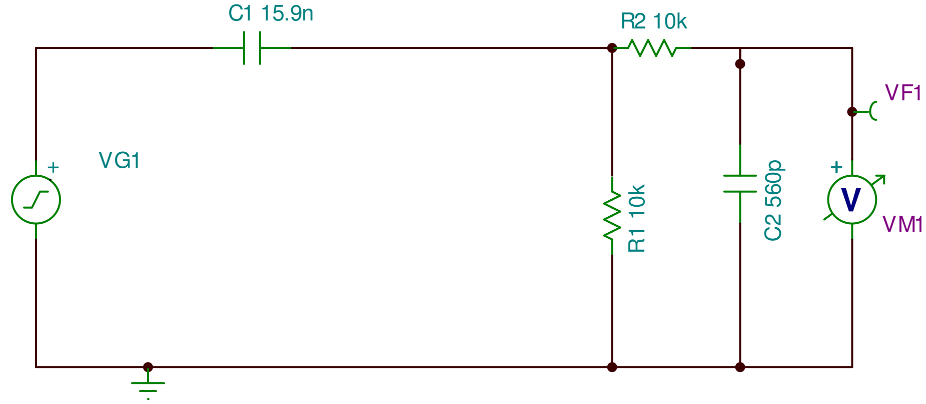


Output Frequency response:

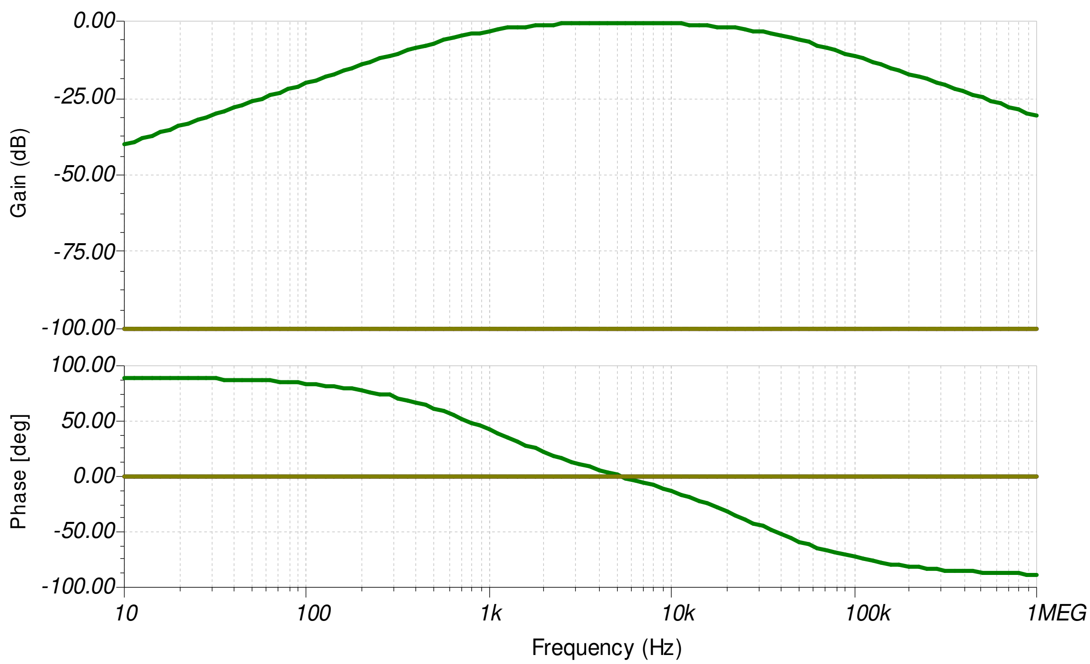


Band Pass Filter:

Fl= 1Khz   Fh= 30 Khz



Output Frequency response:



Result :

The filter circuits were designed and simulated. The AC characteristics were obtained.

Viva questions:

1.Explain what are the commonly used filters?

2. Explain what is the difference between active and passive filters?

3. Explain why are active filters preferred

4. Explain what is a low-pass filter?

5. Explain what is a high-pass filter?

Expt. 6.A. RC PHASE SHIFT OSCILLATOR

AIM:
To design a RC phase shift oscillator and to find the frequency of oscillation.

THEORY:

The low frequencies RC oscillators are more suitable. Tuned circuit is not an essential requirement for oscillation. The essential requirement is that there must be a 180^o phase shift around the feedback network and loop gain should be greater than unity. The 180^o phase shift in feedback signal can be achieved by suitable RC network.

APPARATUS REQUIRED :

S.No.	APPARATUS REQUIRED	RANGE	QUANTITY
1	Resistors	7.5kΩ 1.4 kΩ 4.8 kΩ 1.2 kΩ,19KΩ, 6.5KΩ	1 1 1 1 3
2	Power supply	(0-30)V	1
3	Transistor	BC107	1
4	Capacitors	1.3µf , 2.1µf,1.3µf 0.01µF	1 1 3
5	CRO	(0-30)MHz	1
6	Bread board	-	1

Design Example:

Specifications:

V_{CC} = 12V, I_Cq =1mA, β=100, Vceq = 5V, f=1 KHz, S=10, C=0.01 µf, h_{fe}= 330, A_V= 29

Design:

(i)To find R:

Assume f=1 KHz, C=0.01µf

$f=1/2\pi RC\sqrt{6}$

$R=1/2x3.14\sqrt{6}\ x1x10^3x0.01x10^{-6}$=6.5KΩ

Therefore **R=6.5KΩ**

(ii)To find R_E& R_C:

$V_{CE} = V_{CC}/2 = 6V$

r_e= 26mV / I_E= 26Ω

$h_{ie}= h_{fe}\ r_e= 330 \times 26= 8580\Omega$

On applying KVL to output loop,

$V_{CC}=I_C R_C + V_{CE} + I_E R_E$ ----- (1)

$V_E = I_E\ R_E$

$R_E = V_E / I_E =1.2/ 10^{-3} =1.2K\Omega$

From equation (1), $12= 10^{-3}(R_C+ 1200) +6= R_C= 4800\Omega=4.8K\Omega$

(iii)<u>To calculate R_1 & R_2:</u>

$V_{BB}= V_{CC}\ R_2 / R_1+ R_2$ ------ (2)

$V_B= V_{BE} +V_E = 0.7+12 =1.9V$

From equation (2), $1.9= 12\ R_2 / R_1+ R_2$

$R_2 / R_1+ R_2= 0.158$ -------- (3)

$S = 1+ R_B / R_E= R_B = 1.2K\Omega\ R_B =R_1 \parallel R_2$

$0.15R_1= 1.2x10^{-3} =7.5K\Omega$

$R_2 =0.158\ R_1 + 0.158\ R_2,\ R_2= 1.425K\Omega$

(iv)<u>To calculate Coupling capacitors</u>:

(i) $X_{Ci}= \{[h_{ie} + (1+h_{fe})\ R_E] \parallel R_B\ \}/ 10 = 0.12K\Omega$

$X_{Ci}= 1 / 2\Pi\ f\ C_i == 1.3\mu f$

(ii) $X_{CO}= R_{Leff}/10$ $[\ A_V = - h_{fe}R_{Leff}/h_{ie}]$

$R_{Leff} = 0.74K\Omega,\ X_{CO}=0.075\ K\Omega$

$X_{CO}= 1 / 2\Pi\ f\ C_O\ ,\ C_O = 2.1\mu f$

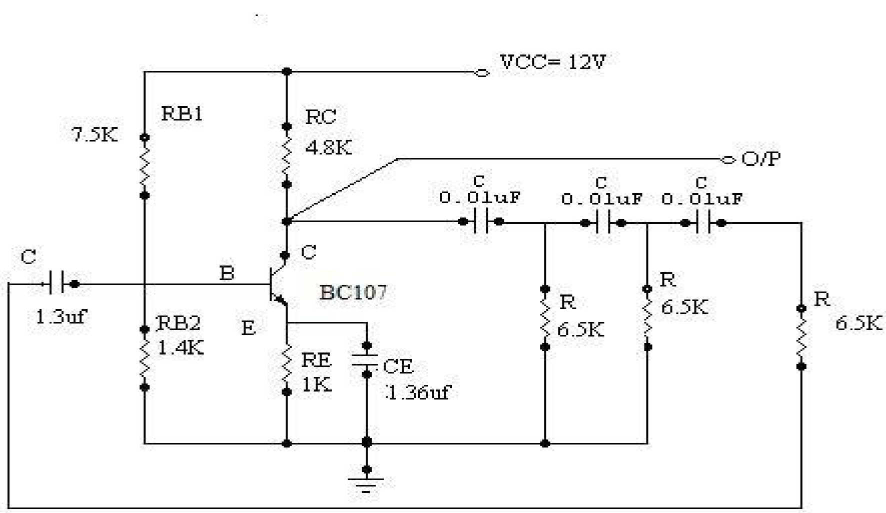
(iii) $X_{CE}= R_E/10 = 1.326\mu f\ X_{CE}$

$= 1 / 2\Pi\ f\ C_E=49.27\mu f$

(iv) Feed back capacitor, $X_{CF}= R_f/ 10$

$C_f = 0.636\mu f = 0.01\mu f$

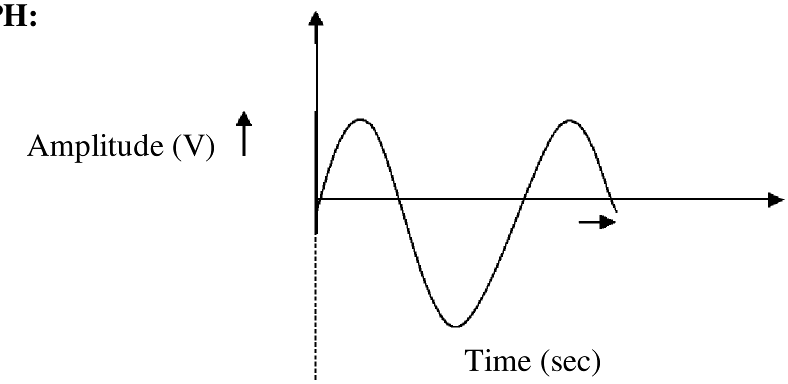
CIRCUIT DIAGRAM



PROCEDURE:

1. Connect the circuit as per the circuit diagram.
2. Set V_{CC} = 15V.
3. For the given supply the amplitude and time period is measured from CRO.
4. Frequency of oscillation is calculated by the formula f=1/T
5. Amplitude Vs time graph is drawn.

MODEL GRAPH:



TABULATION:

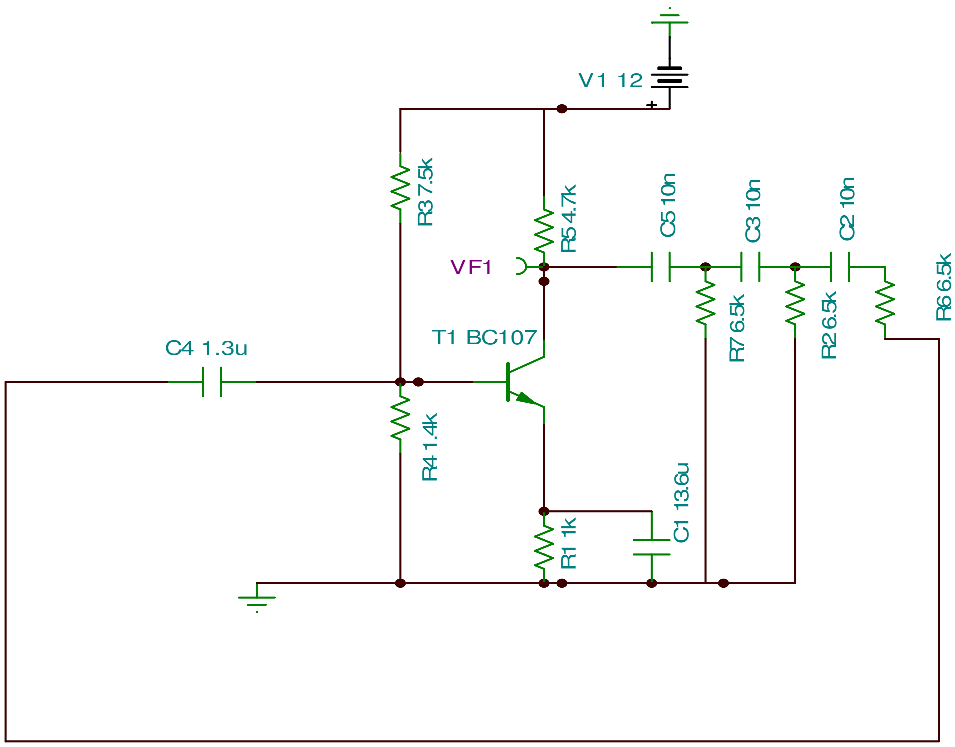
Amplitude (V)	Time period (msec)	Frequency (Hz)

REVIEW QUESTIONS:

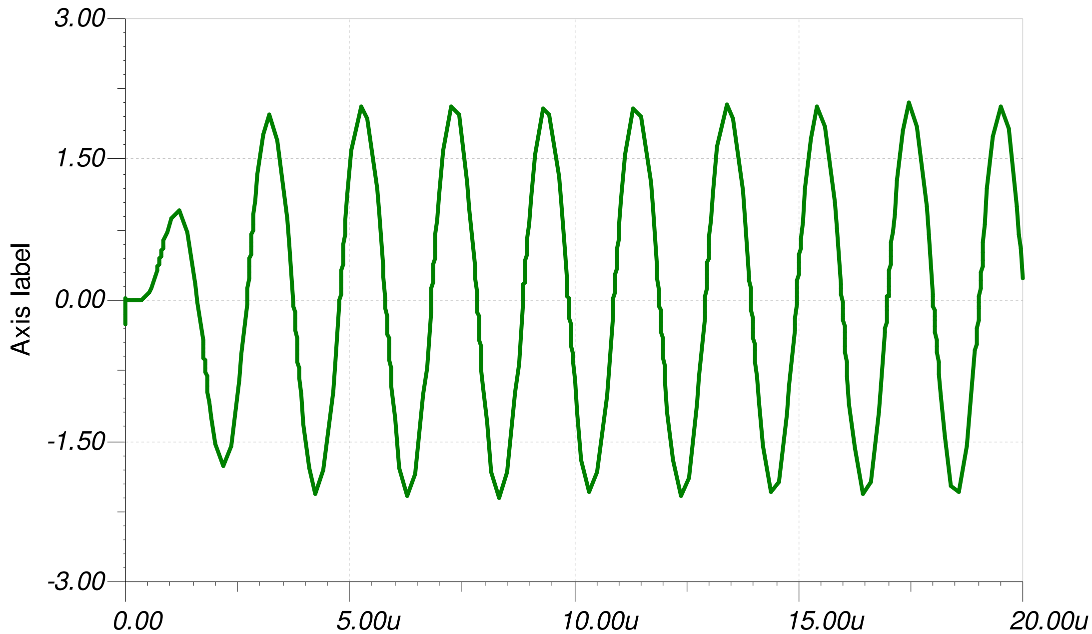
1. What is an oscillator?
2. What is barkhausen criterion for oscillation?
3. Which feedback is used in oscillators?
4. Give the frequency of oscillation for RC-phase shift oscillator?
5. Give the disadvantages of phase shift oscillator.

RESULT: Thus the RC-phase shift oscillator is designed and constructed for the given frequency.

Simulation :



Output:



Exp.6.B LC OSCILLATOR: HARTLEY OSCILLATOR

AIM:

To design and construct a Hartley oscillator at the given operating frequency

THEORY:

Hartley oscillator is very popular and is commonly used as local oscillator in radio receivers. The collector voltage is applied to the collector through inductor L whose reactance is high compared with X_2 and may therefore be omitted from equivalent circuit, at zero frequency, however capacitor C_b acts as an open circuit.

APPARATUS REQUIRED:

S.No	APPARATUS	RANGE	QUANTITY
1	Resistors	2kΩ 1KΩ 100 kΩ 22kΩ	1 1 1 1
2	RPS	(0-30)V	1
3	Transistor	BC107	1
4	Capacitors	3.2nf 0.1μF 0.01μF	1 1 2
5	Inductor	10mH	2
6	CRO	30MHz	1
7	Bread board	-	1

Design Example:

Design of feed back Network:

Given $L_1 = L_2 = 10mH$, f=20 KHz, $V_{CC}=12V$, $I_C=3mA$, S=12

$f = 1/2\Pi\sqrt{(L1 + L2)C}$

C= 3.2nf

Amplifier design:

(i) Selection of R_C: Gain formula is,

Av= - hfe RLeff / hie

Assume $V_{CE}=V_{CC}/2$ (Transistor active)

$V_{CE}= 12/2= 6V$

$V_E=I_ER_E= V_{CC}/10=1.2V$

$V_{CC}=I_CR_C + V_{CE} + I_ER_E$

$R_C = (V_{CC} - V_{CE} - I_E R_E) / I_C$

Therefore $R_C = 1.6K\Omega = 2\ K\Omega$

(ii) Selection of R_E: $I_C = I_E = 3mA$

$R_E = V_E / I_E$

$R_E = 1.2 / 3x10^{-3} = 400\ \Omega = 1K\Omega$

(iii) Selection of R_1 & R_2:

Stability factor S=12

$S = 1 + (R_B / R_E)$

$12 = 1 + (R_B / 1x10^3)$

$R_B = 11K$

Using potential divider rule,

$R_B = R_1 R_2 / R_1 + R_2$ & $V_B = (R2 / R_1 + R_2)\ V_{CC}$

$R_B / R_1 = R_2 / R_1 + R_2$

Therefore $R_B / R_1 = V_B / V_{CC}$

$V_B = V_{BE} + V_E = 0.7 + 1.2 = 1.9V = 2V$

$R_1 = (V_{CC} / V_B) R_B$

$R_1 = (12/2) x\ 11x10^3 = 66K\Omega = 100K\Omega$

$V_B / V_{CC} = R_2 / R_1 + R_2$

$2 / 12 = R_2 / 100x10^3 + R_2$

$(100x10^3) + R_2 = R_2 / 0.16 = 19K\Omega$

$R_2 = 19K\Omega = 22\ K\Omega$

(iv)Output capacitance (C_O):

$X_{CO} = R_C / 10 = 2x10^3 / 10 = 200$

$1/2\Pi f C_O = 200$

$C_O = 1/2x3.14x20x10^3 x200$

$C_O = 0.039 = 0.01\mu f$

(v) Input capacitance (C_i):

$X_{Cin} = R_B / 10 = 11x10^3 / 10 = 1.1x10^3$

$1/2\Pi f C_{in} = 1.1x10^3$

$C_{in} = 1/2x3.14x20x10^3 x1.1x10^3$

$C_{in} = 0.007 = 0.01\mu f$

(vi) By pass Capacitance (C_E):

$X_{CE} = R_E / 10 = 1x10^3 / 10 = 100$

$1/2\Pi f C_E = 100$

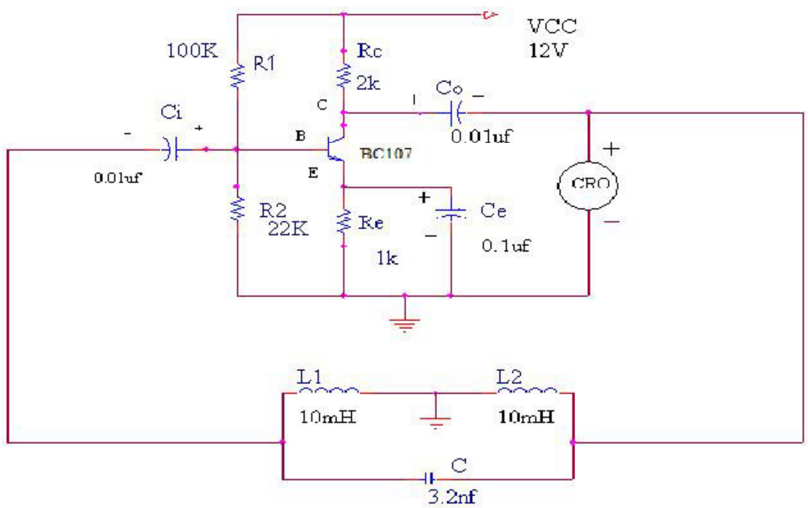
$C_E = 1/2x3.14x20x10^3 x100$

$C_E = 0.079\mu f = 0.1\mu f$

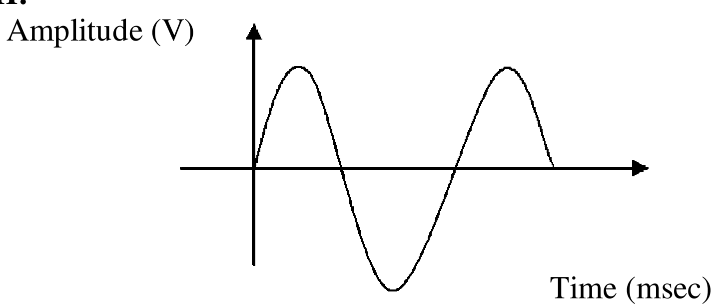
PROCEDURE:

1. Connect the circuit as per the circuit diagram.
2. Set V_{CC} = 12V.
3. For the given supply the amplitude and time period is measured from CRO.
4. Frequency of oscillation is calculated by the formula f=1/T
5. Verify it with theoretical frequency, $f = 1/2\Pi\ (\sqrt{(L1 + L2)C}\ )$
6. Amplitude Vs time graph is drawn.

CIRCUIT DIAGRAM:



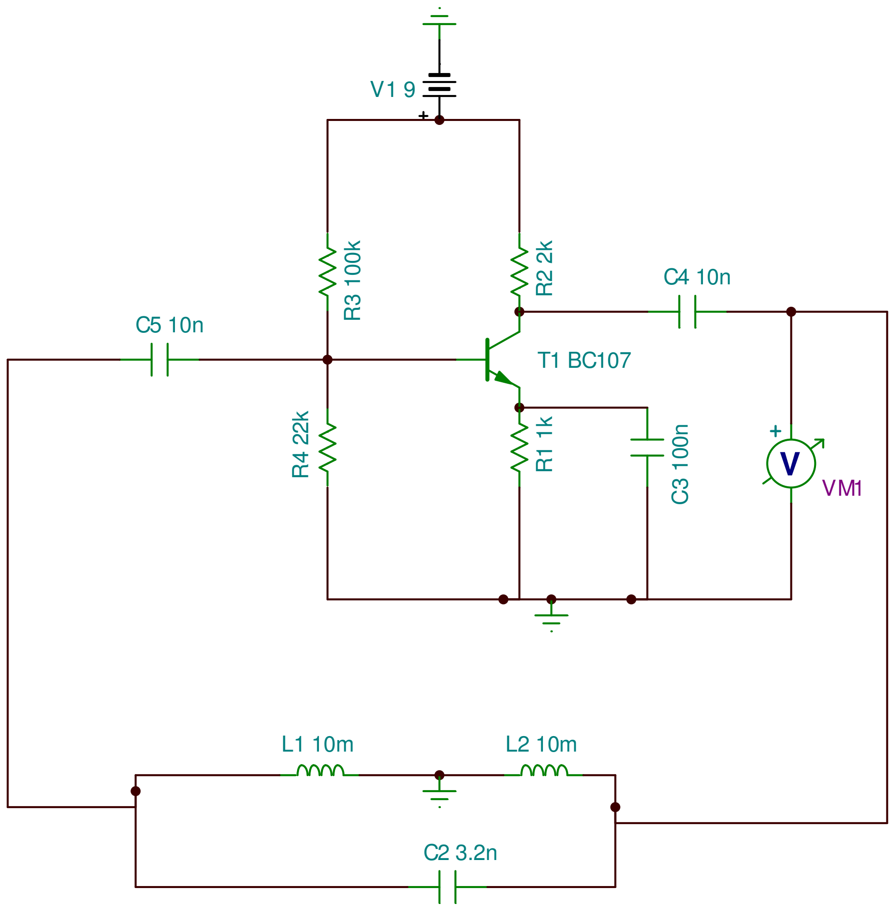
MODEL GRAPH:



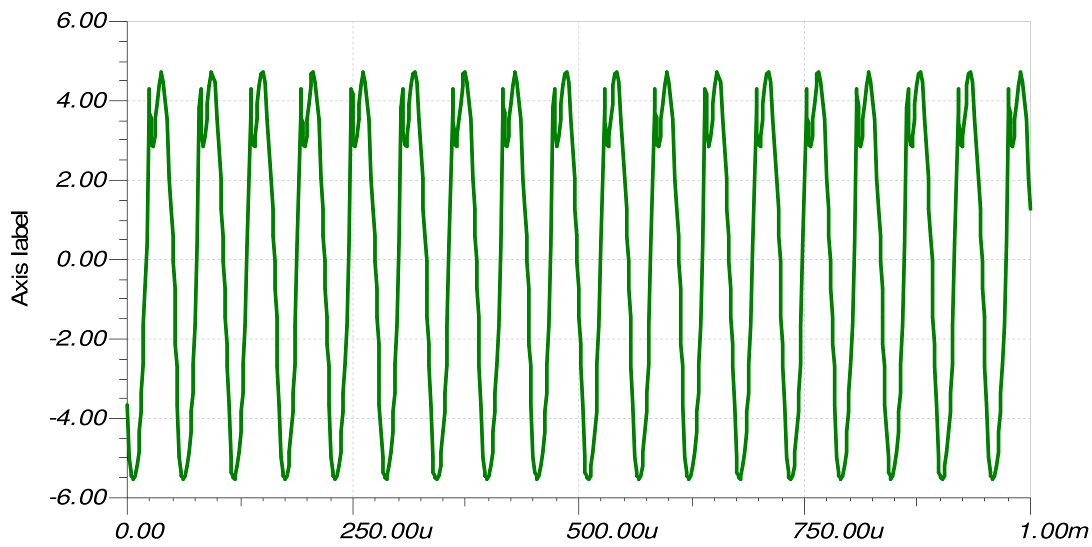
TABULATION:

Amplitude(V)	Time(msec)	Frequency(Hz)

Simulation circuit: HARTLEY OSCILLATOR



Simulation output



RESULT:

Thus the Hartley oscillator is designed and constructed for the given frequency.

QUESTIONS:

1. How does an oscillator differ from an amplifier?
2. What is the approximate value of h_{fe} in a Hartley oscillator using BJT?
3. Mention the expression for frequency of oscillation?
4. Mention the reasons why LC oscillator is preferred over RC oscillator at radio frequency?
5. How the Hartley oscillator satisfy the barkhausen criterion

Exp.7.A. ASTABLE MULTIVIBRATOR

AIM:

To design an Emitter coupled Astable multivibrator and study the output waveform.

APPARATUS REQUIRED:

S.No	APPARATUS	RANGE	QUANTITY
1	Resistors	4.7kΩ 470kΩ	2 2
2	RPS	(0-30)V	1
3	Transistor	BC107	2
4	Capacitors	0.01μF	2
5	CRO	(0-30)MHz	1
6	Bread board	-	1

THEORY:

The astable multivibrator generates square wave without any external triggering pulse. It has no stable state, i.e., it has two quasi-stable states. It switches back and forth from one stable state to other, remaining in each state for a time depending upon the discharging of a capacitive circuit. When supply voltage + Vcc is applied, one transistor will conduct more than the other due to some circuit imbalance.

Design example:

Given specifications:

V_{CC}= 10V; hfe = 100; f=1 KHz; I c = 2mA; Vce (sat) = 0.2v;

To design R_C:

$R \leq h_{FE} R_C$

$R_C = V_{CC} - V_{C2\,(Sat)} / I_C = 4.9\ k\Omega$

Since $R \leq h_{FE}\, R_C$

Therefore $R \leq 100 \times 4.7 \times 10^3 = 490\ k\Omega \approx 470\ k\Omega$

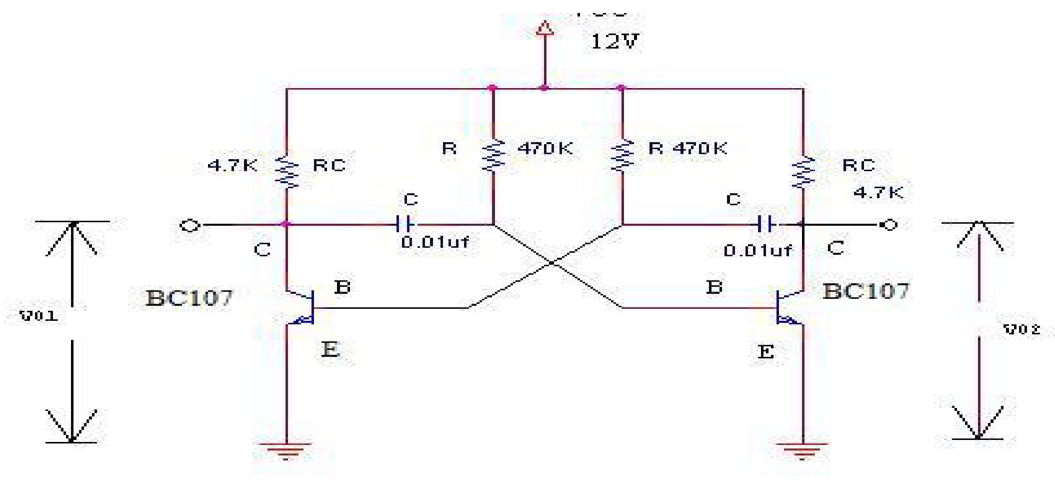
To Design C:

Since T= 1.38RC

$1\times10^{-3} = 1.38 \times 490\times10^{3} \times C$

Therefore C=0.01μF

CIRCUIT DIAGRAM



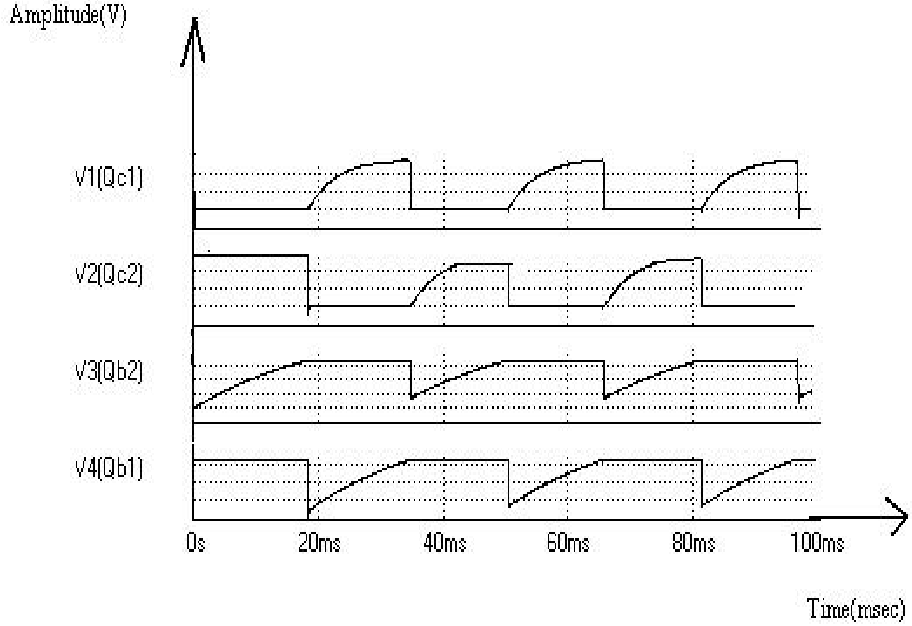
TABULATION:

Types of Waveform	Amplitude(V)	period(msec)

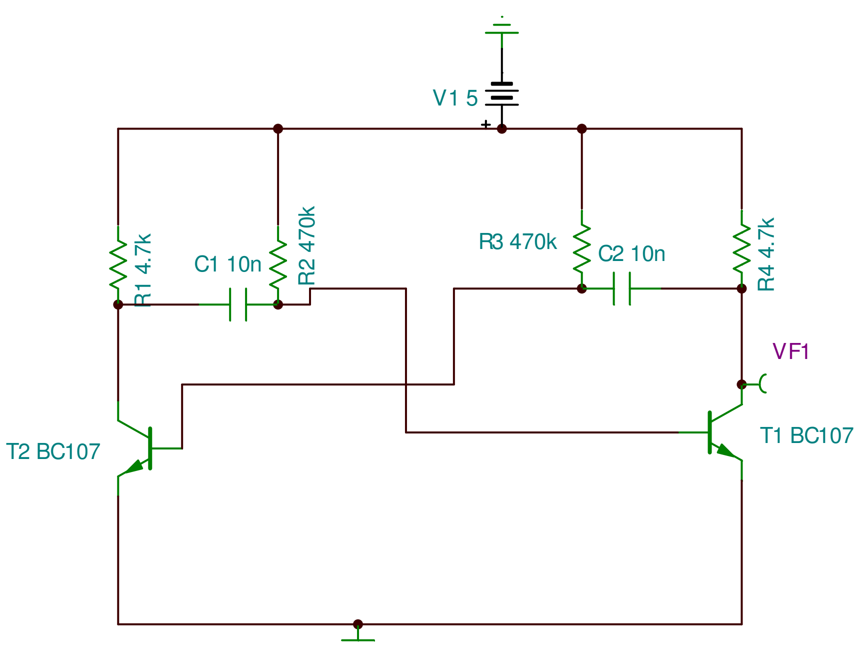
PROCEDURE:

1. Connect the circuit as per the circuit diagram.
2. Set V_{CC} = 5V.
3. For the given supply the amplitude and time period is measured from CRO.
4. Frequency of oscillation is calculated by the formula f=1/T
5. Amplitude Vs time graph is drawn.

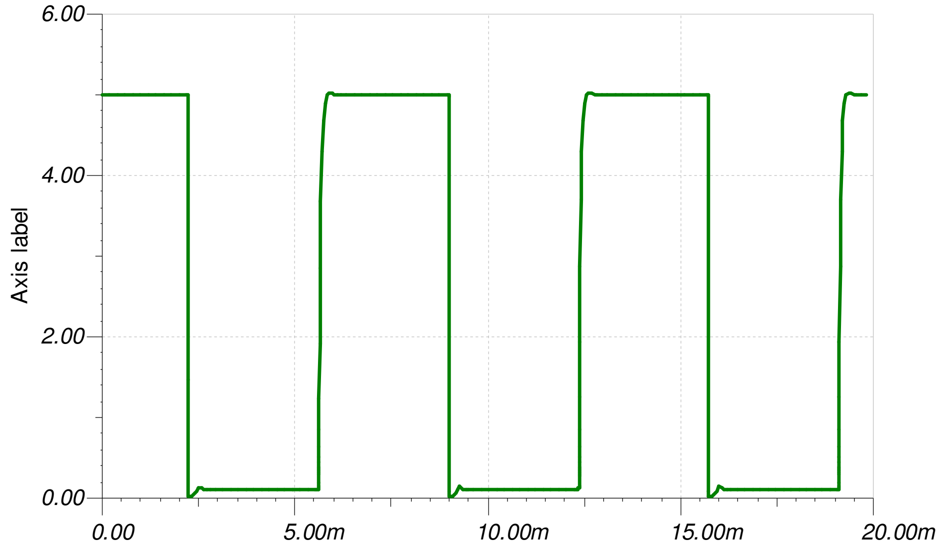
MODEL GRAPH:



SIMULATION CIRCUIT:



SIMULATION OUTPUT:



RESULT:

Thus the astable multivibrator is designed and output waveform is plotted.

QUESTIONS:

1. What is meant by multivibrator?
2. What is the frequency of oscillation of astable multivibrator?
3. Distinguish oscillator and multivibrator?
4. List the applications of astable multivibrator.
5. Why it is called as free running multivibrator.

EXP.7.B. # MONOSTABLE MULTIVIBRATOR

AIM:

To design and test the performance of Monostable multivibrator for the given frequency.

APPARATUS REQUIRED:

S.No	APPARATUS	RANGE	QUANTITY
1	Resistors	5.9 kΩ 452 kΩ, 100 kΩ 10KΩ	2 1 1 1
2	RPS	(0-30)V	1
3	Transistor	BC107	1
4	CRO	(0-30)MHz	1
5	Capacitor	3.2nf 25pf	1 1
6	Bread board	-	1

THEORY:

The monostable multivibrator has one stable state when an external trigger input is applied the circuit changes its state from stable to quasi stable state. And then automatically after some time interval the circuit returns back to the original normal stable state. The time T is dependent on circuit components.

The capacitor C_1 is a speed-up capacitor coupled to base of Q2 through C.Thus DC coupling in bistable multivibrator is replaced by a capacitor coupling. The resistor R at input of Q2 is returned to V_{CC}.

The value of R2 , V_{BB} are chosen such that transistor Q1 is off by reverse biasing it. Q2 is on. This is possible by forward biasing Q2 with the help of VCC and resistance R. Thus Q2-ON and Q1-OFF is normal stable state of circuit.

Design Example:

Given specifications:

V_{CC}= 12V; hfe = 200; f=1 KHz; I_c = 2mA; Vce (sat) = 0.2v;

V_{BB}= - 2V,

(i)**To calculate R_C:**

$R_C = V_{CC} - Vce(sat) / I_C$

$R_C = 12 - 0.2 / 2x10^{-3} = 5.9K\Omega$

(ii) **To calculate R:**

$I_{B2(min)} = I_{C2} / h_{fe} = 2x10^{-3} / 200 = 10\mu A$

Select $I_{B2} > I_{B1(min)}$ (say 25μ A)

Then $R = V_{CC} - V_{BE\,(sat)} / I_{B2}$

Therefore $R = 12-0.7/25\times10^{-6} = 452K\Omega$

(iii) **To calculate C:** $T=0.69RC$

$1\times10^{-3} = 0.69\times452\times10^{3}\times C$ $C=3.2nf$

To calculate R1 & R2:

$V_{B1} = \{(V_{BB}\ R1/\ R1 + R2) + (V_{CE\,(sat)}\ R2 / R1+R2)\}$ Since Q1 is in off state ,$V_{B1} \le 0$

Then $(V_{BB}\ R1/\ R1 + R2) = (V_{CE\,(sat)}\ R2 / R1+R2)$

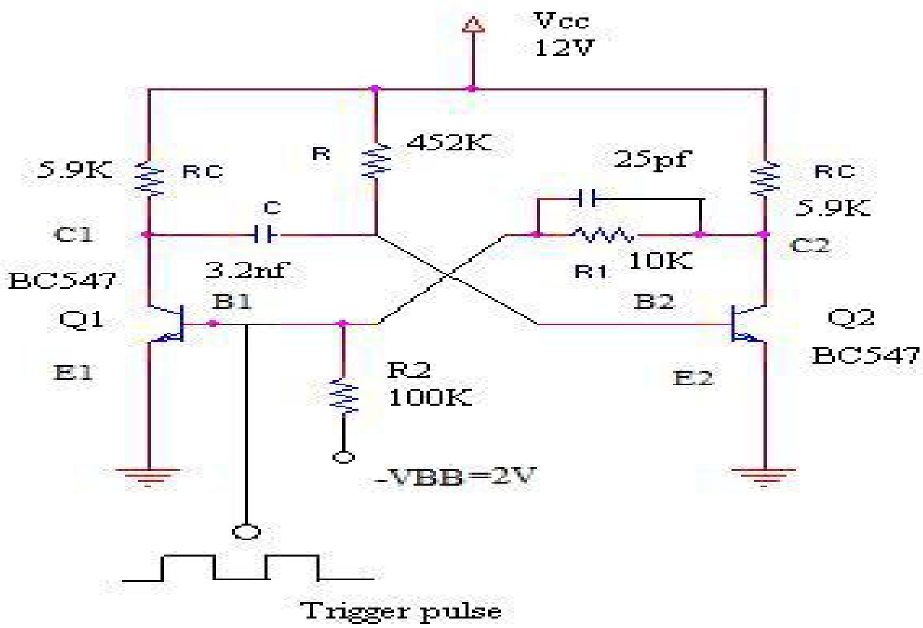
$V_{BB}\ R1 = V_{CE\,(sat)}\ R2$

$2\ R1 = 0.2\ R2$

Assume R1=10KΩ, then R2=100 KΩ

Consider, C_1= 25pf (commutative capacitor)

CIRCUIT DIAGRAM:



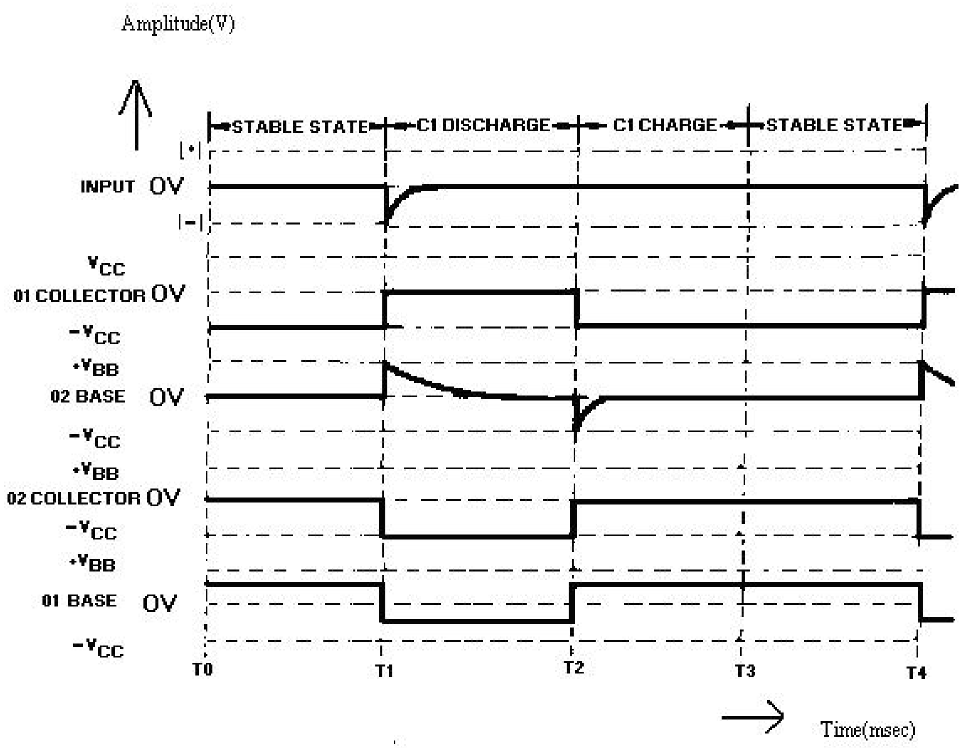
PROCEDURE:

1. Connect the circuit as per the circuit diagram.
2. Give a negative trigger input to Q2.
3. Note the output of transistor Q2 and Q1.
4. Find the value of Ton and Toff.

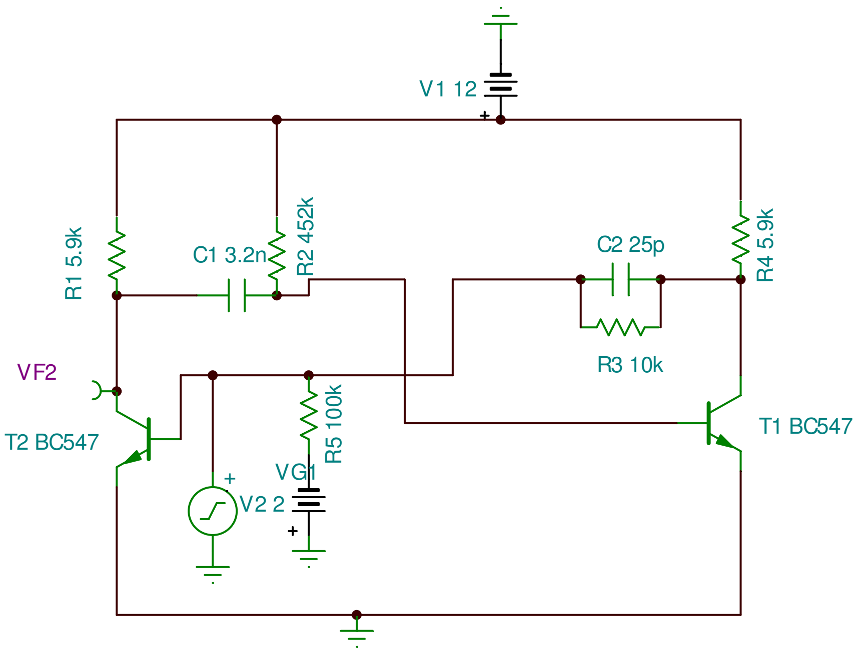
TABULATION:

Amplitude(V)	Time period(msec)	
	T_{ON}	T_{OFF}

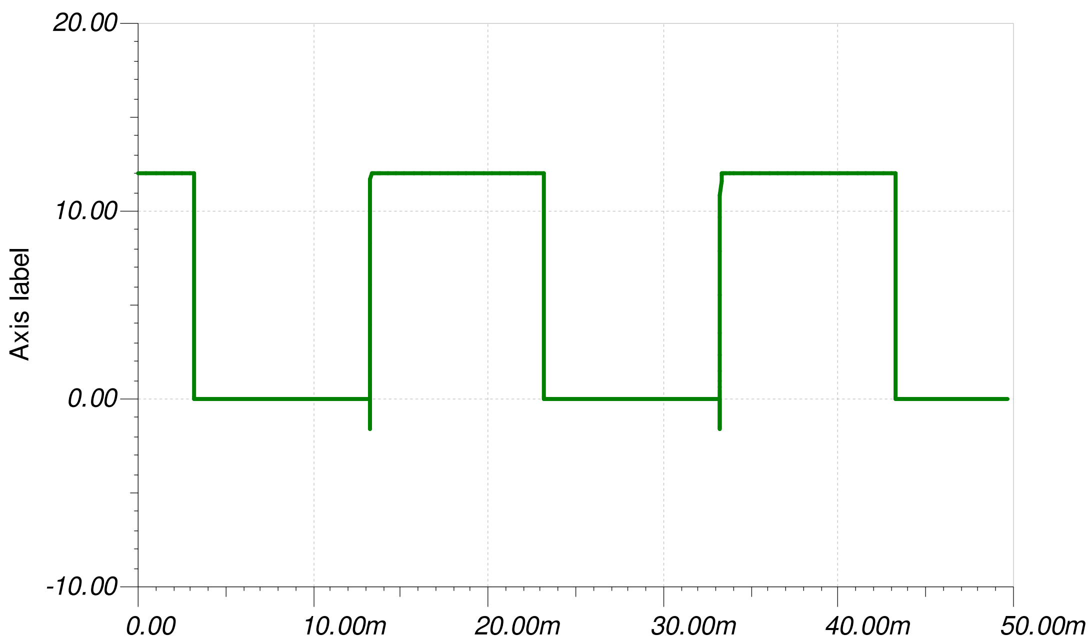
MODEL GRAPH:



Simulation circuit:



Simulation output :



RESULT:

Thus the monostable multivibrator is designed and the performance is tested.

QUESTIONS:

1. Give the other names of monostable multivibrator?
2. What is the use of commutating capacitor?
3. What is the frequency of oscillation of monostable multivibrator?
4. Why it is called as one-shot multivibrator?
5. List the applications of mono stable multivibrator.

Ex.No.8

CHARACTERISTICS OF SCR AND CONTROLLED RECTIFIER USING SCR

Aim:

To obtain the V-I characteristics of SCR and find the break over voltage and holding current.

APPARATUS REQUIRED:

S.No.	Name	Range	Type	Qty
1	R.P.S	(0-30)V		2
2	Ammeter	(0–10)mA (0–100)mA	MC	1
3	Voltmeter	(0–30)V	MC	1

COMPONENTS REQUIRED:

S.No.	Name	Range	Type	Qty
1	SCR	TYN604		1
2	Resistor	330Ω		1 1
3	Bread Board			1
4	Wires			
5	Diode	IN4001		1

Theory:

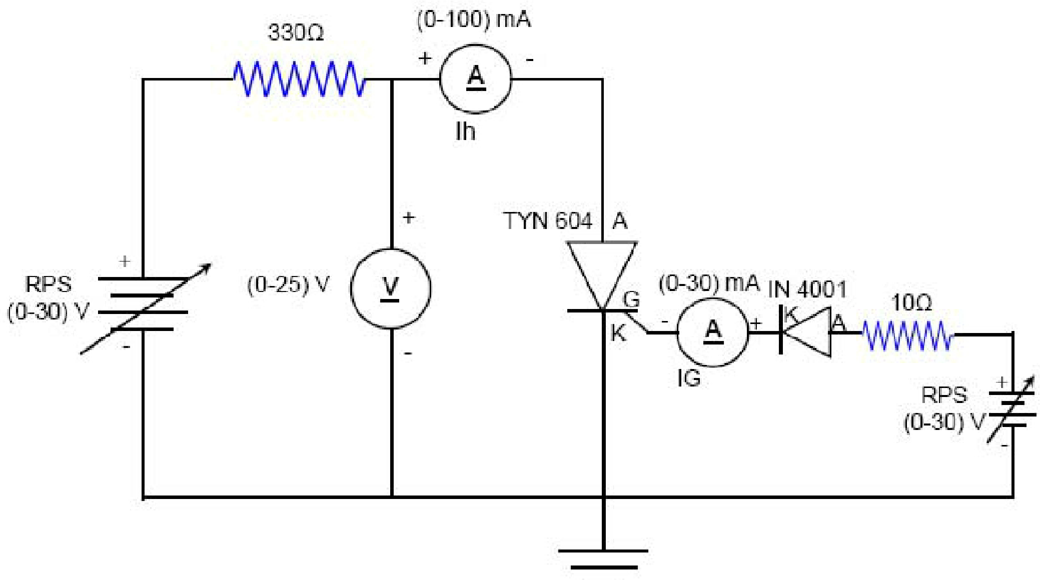
A silicon controlled rectifier (SCR) is a semiconductor device that acts as a true electronic switch.

It can change the alternating current in to directcurrent. It can control the amount of power fed to the load. Thus the SCR combines the features of rectifier and a transistor.If the supply voltage is less than the break over voltage, the gate willopen (IG = 0). Then increase the supply voltage from zero, a point is reached when the SCR starts conductingUnder this condition, the voltage across the SCR suddenly drop and most of the

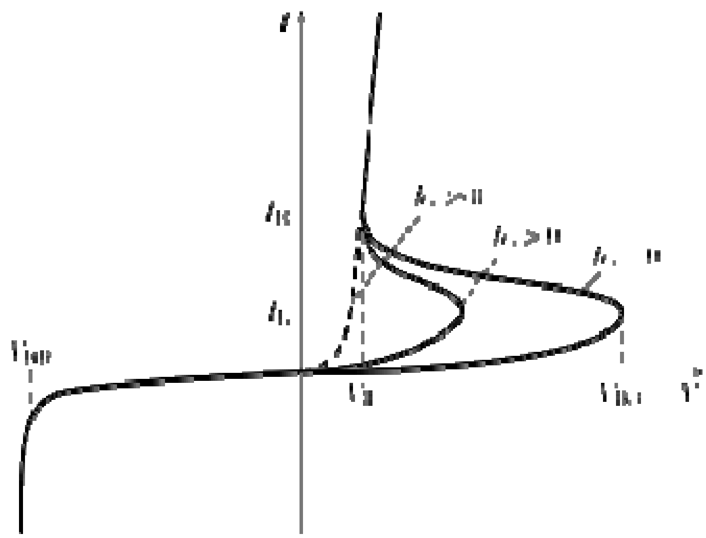
supply voltage appears across the loadresistance RL.If proper gate current is made to flow the SCR can close at much smaller supply voltage.

Characteristics of SCR:

Circuit Diagram:



Model Graph



Procedure:

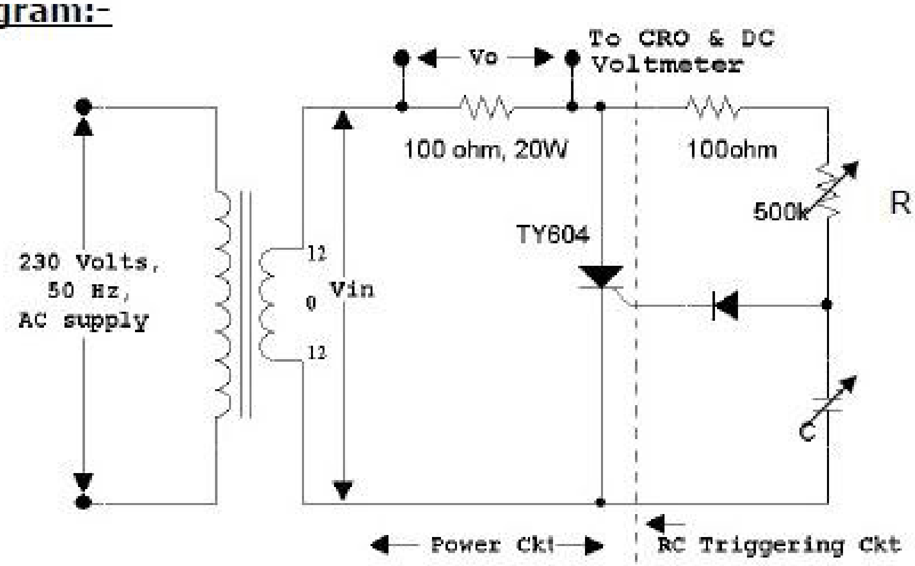
- The connections are made as per in the circuit diagram.
- First by varying RPS 2 then gate current (IG) is kept constant.
- The voltage between anode and cathode is increased in step by step by varying the RPS 1.
- The corresponding anode current (IA) is noted.
- The process is repeated for two more constant value of IG, the readings are tabulated.

Tabulation:

S. No	Anode-Cathode Voltage Vak (Volts)	Gate Current Ig (mA)	Anode Current Ia (mA)	Anode – Cathode volt when SCR in ON (volts)

CONTROLLED RECTIFIER USING SCR:

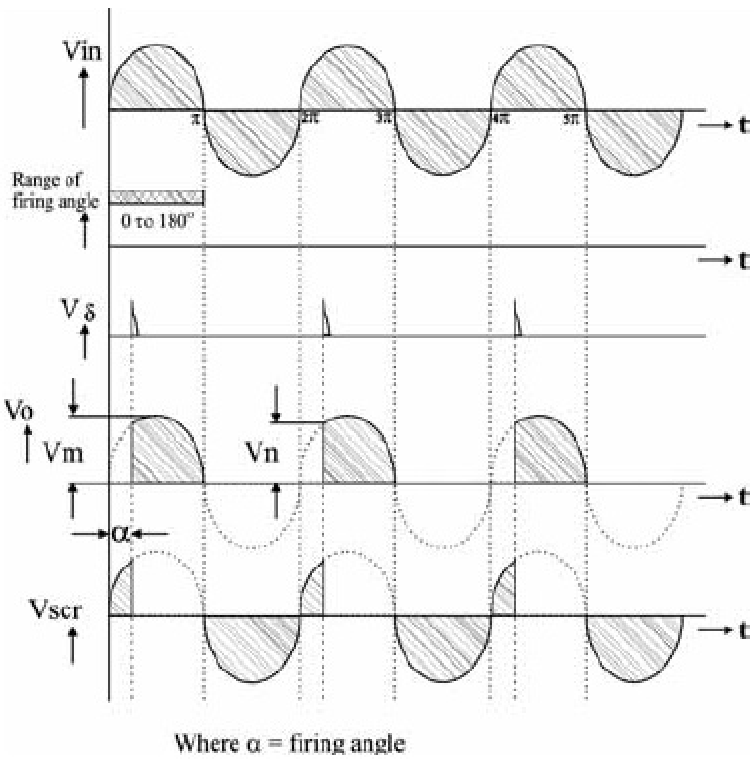
Circuit diagram:-



Tabulation:

Waveform	Time	Amplitude

Model Graph:



Result:

Thus the V-I characteristics of SCR was obtained and graph was drawn also Controlled Rectifier using SCR is implemented .

Viva questions :

1.Define holding current.

2.What are the specifications of the SCR ?

3.What is UJT, explain the input characteristics of it?.

4.What is the symbol and principle of operation of Tunnel Diode?

5.Explain the V-I characteristics of Tunnel diode?

6.What is the symbol and principle of operation of Varactor Diode?

7.What is the symbol and principle of operation of SchottkyBarrier Diode?

8.What is the symbol and principle of operation of photo Diode?

Ex.No: 9

CHARACTERISTICS OF UNIJUNCTION TRANSISTOR &SAWTOOTH WAVEFORM GENERATOR

AIM:

To Plot and study the characteristics of UJT & determine it's intrinsic standoff Ratio.

APPARATUS REQUIRED:

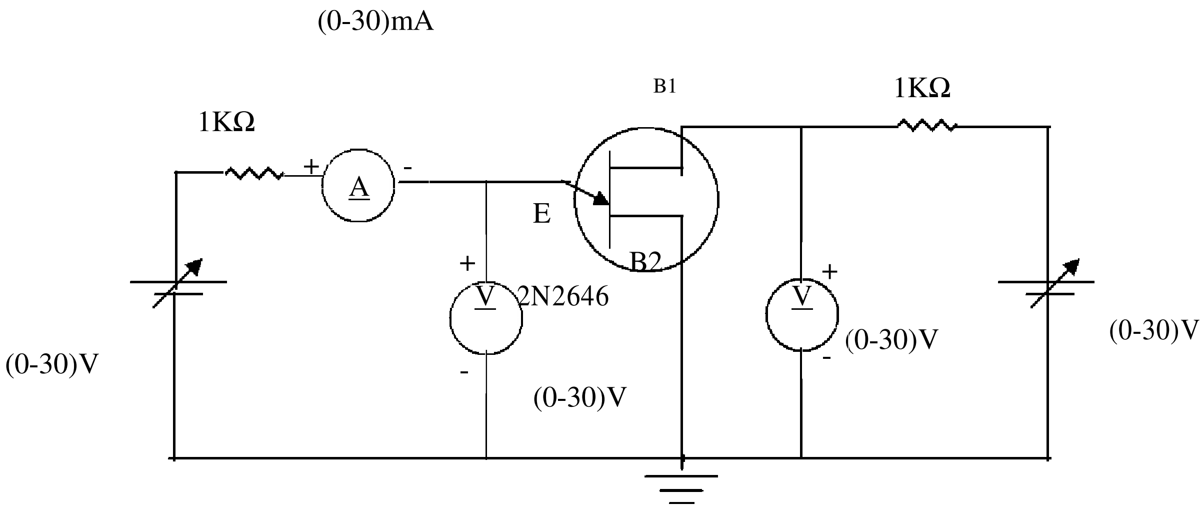
S. No.	Name	Range	Type	Qty
1	R.P.S	(0-30)V		2
2	Ammeter	(0–30)mA	MC	1
3	Voltmeter	(0–30)V	MC	1
		(0–10)V	MC	1

S. No.	Name	Range	Type	Qty
1	UJT	2N2646		1
2	Resistor	1KΩ		2
3	Bread Board			1

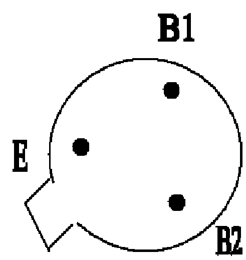
Theory :

UJT(Double base diode) consists of a bar of lightly doped n-type silicon with a small piece of heavily doped P type material joined to one side. It has got three terminals. They are Emitter(E), Base1(B1),Base2(B2).Since the silicon bar is lightly doped, it has a high resistance & can be represented as two resistors, r_{B1}& r_{B2}. When $V_{B1B2} = 0$, a small increase in V_E forward biases the emitter junction. The resultant plot of V_E& I_E is simply the characteristics of forward biased diode with resistance. Increasing V_{EB1} reduces the emitter junction reverse bias. When $V_{EB1} = Vr_{B1}$ there is no forward or reverse bias. & $I_E = 0$. Increasing V_{EB1} beyond this point begins to forward bias the emitter junction. At the peak point, a small forward emitter current is flowing. This current is termed as peak current(I_P). Until this point UJT is said to be operating in cutoff region. When I_E increases beyond peak current the device enters the negative resistance region. In which the resistance r_{B1} falls rapidly & V_E falls to the valley voltage.Vv. At this point I_E = Iv. A further increase of I_E causes the device to enter the saturation region.

CIRCUIT DIAGRAM:



PIN DIAGRAM2N2646:



SPECIFICATION FOR 2N2646:

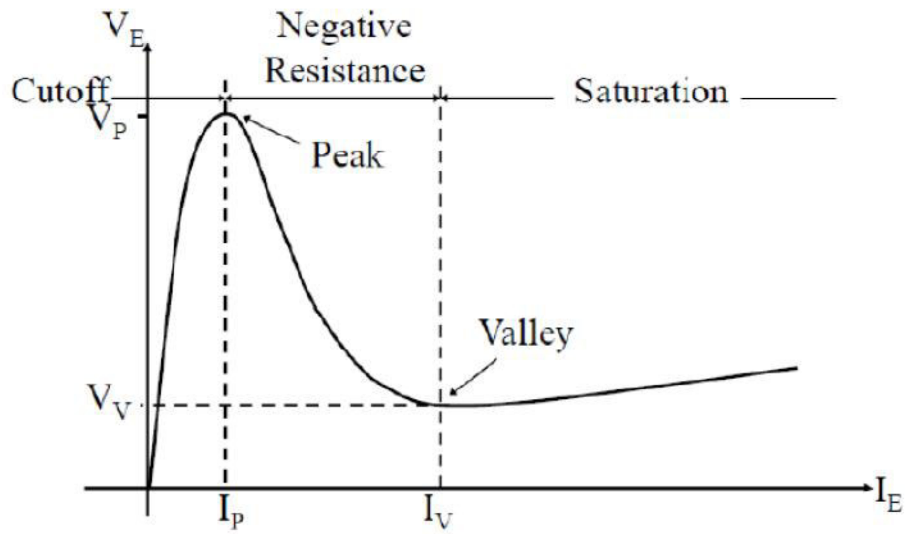
* Inter base resistance R_{BB} = 4.7 to 9.1 KΩ

* Minimum Valley current = 4 mA

* Maximun Peak point emitter current 5 µA

*Maximum emitter reverse current 12 µA.

MODEL GRAPH:



TABULAR COLUMN:

V_{B1B2} = 0V		V_{B1B2} = 10V	
V_{EB1} (V)	I_E (mA)	V_{EB1} (V)	I_E (mA)

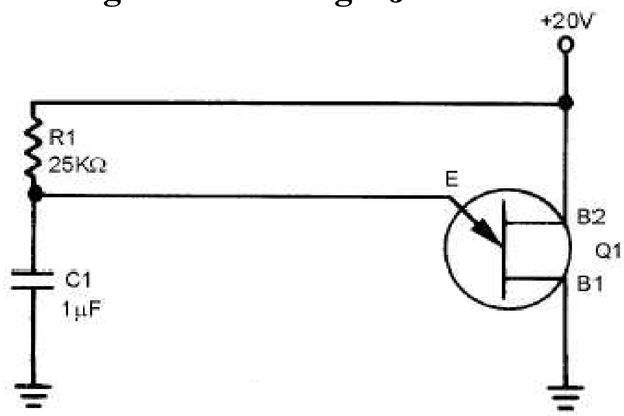
PROCEDURE:

1. Connect the circuit as per the circuit diagram.
2. Set V_{B1B2} = 0V, vary V_{EB1} , & note down the readings of I_E& V_{EB1}
3. Set V_{B1B2} = 10V , vary V_{EB1} , & note down the readings of I_E& V_{EB1}
4. Plot the graph : I_E Versus V_{EB1} for constant V_{B1B2}.
5. Find the intrinsic standoff ratio.

FORMULA FOR INTRINSIC STANDOFF RATIO:

$\eta = V_P - V_D / V_{B1B2}$., where $V_D = 0.7V$.

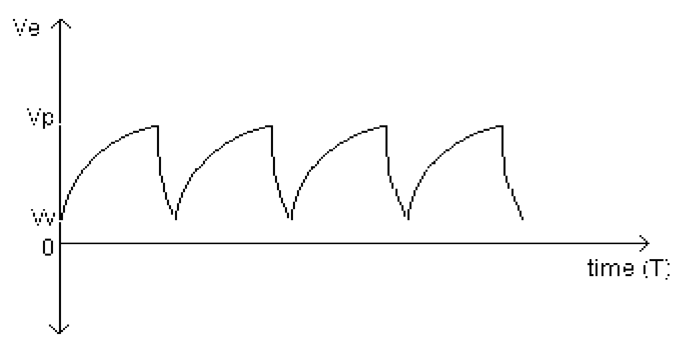
Saw tooth waveform generator using UJT



PROCEDURE:

1. Give the circuit connections as per the circuit diagram.
2. The dc input voltage is set to 20 V in RPS.
3. The output sweep waveform is measured using CRO.
4. The graph of output sweep waveform is plotted

Model Graph:



Wave form across the capacitor in a UJT relaxation oscillator

Tabulation

Waveform	Time	Amplitude

RESULT: Thus the characteristics of given UJT was Plotted & its intrinsic standoff Ratio = ----.

Questions :

1. Write the features of UJT.
2. What is the difference between UJT and FET?
3. What is a UJT?
4. What is relaxation oscillator?
5. What are the applications of UJT?
6. What is the importance of intrinsic stand-off ratio?
7. Why does negative resistance region appears in UJT?
8. What is the doping profile of UJT?
9. Is there any break down region in UJT?

1907305 DEVICES LABORATORY

CYCLE -1

CIRCUITS LAB