

PRE-REQUISITES FOR BASIC UNDERSTANDING

S. No.	Title	Page no.
1	Basic Electronic Components (Resistors, Capacitors and Inductors)	4
2	Circuit Symbols	8
3	Study of Cathode Ray and Digital Storage Oscilloscope	16
4	Study of Function Generator	20
5	Study of Regulated Power Supply	22
6	Types of Circuit Boards	23

LIST OF EXPERIMENTS

PART A		
S. No.	Experiment	Page no.
1	Experimentation on Thevenin's and Norton's Theorems and Verification using PSPICE	26
2	Experimentation on Superposition Theorem and Maximum Power Transfer Theorem and Verification using PSPICE	32
3	Experimentation on Transient analysis of RL and RC Series Circuits and Verification using PSPICE	38
4	Series and Parallel Resonance	42
5	Determination of Self, Mutual Inductances and Coefficient of Coupling	49
PART B		
1	Forward and Reverse bias characteristics of PN Junction diode	55
2	Zener diode characteristics and Zener as Voltage Regulator.	60
3	Input and Output characteristics of Transistor in CB Configuration.	68
4	Input and Output characteristics of Transistor in CE Configuration.	73
5	Input and Output characteristics of Transistor in CC Configuration.	78
6	Half Wave Rectifier With and without filter.	83
7	Full wave Rectifier With and without filter.	89
8	JFET/MOSFET characteristics	93
9	Frequency response of CE Amplifier.	97
10	Frequency response of CC Amplifier.	101
11	Frequency response of Common Source FET Amplifier.	105
12	UJT Characteristics.	109

**PRE-REQUISITES
FOR BASIC UNDERSTANDING**

1. Basic Electronic Components (Resistors, Capacitors and Inductors)

RESISTOR

A Resistor is a passive two-terminal electrical component that implements electrical resistance as a circuit element. The current through a resistor is in direct proportion to the voltage across the resistor's terminals. This relationship is represented by Ohm's law:

$$V = I R$$

Where I is the current through the conductor in units of amperes, V is the potential difference measured across the conductor in units of volts, and R is the resistance of the conductor in units of ohms. The ratio of the voltage applied across a resistor's terminals to the intensity of current in the circuit is called its resistance, and this can be assumed to be a constant (independent of the voltage) for ordinary resistors working within their ratings.

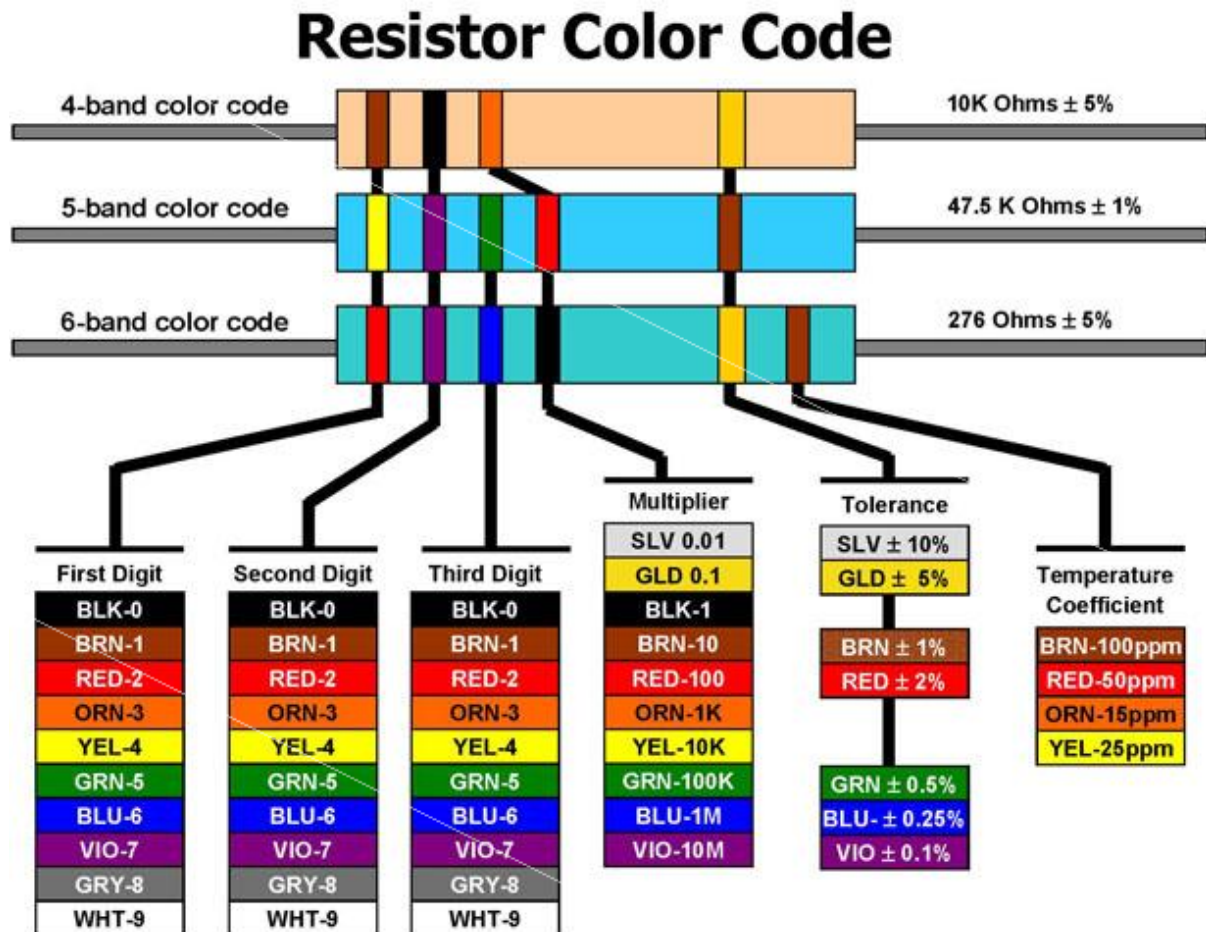
COLOUR CODING OF RESISTOR

Colour Codes are used to identify the value of resistor. The numbers to the Colour are identified in the following sequence which is remembered as BBROY GREAT BRITAIN VERY GOOD WIFE (BBROYGBVWG) and their assignment is listed in following table.

Black	Brown	Red	Orange	Yellow	Green	Blue	Violet	Grey	White
0	1	2	3	4	5	6	7	8	9

RESISTOR COLOR CODES

Resistors are devices that limit current flow and provide a voltage drop in electrical circuits. Because carbon resistors are physically small, they are color-coded to identify their resistance value in Ohms. The use of color bands on the body of a resistor is the most common system for indicating the value of a resistor. Color-coding is standardized by the Electronic Industries Association (EIA).



Use the Resistor Color Code Chart (above) to understand how to use the color code system. When looking at the chart, note the illustration of three round resistors with numerous color code bands. The first resistor in the chart (with 4 bands) tells you the minimum information you can learn from a resistor. The next (a 5-band code) provides a little more information about the resistor. The third resistor (a 6-band) provides even more information. Each color band is associated with a numerical value.

How to read a typical 4-band, 5-band and 6-band resistor:

4-Band: Reading the resistor from left to right, the first two color bands represent significant digits, the third band represents the decimal multiplier, and the fourth band represents the tolerance.

5- Band: The first three color bands represent significant digits, the fourth band represents the decimal multiplier, and the fifth band represents the tolerance.

6-Band: The first three color bands represent significant digits, the fourth band represents the decimal multiplier, the fifth band represents the tolerance, and the sixth band represents the temperature coefficient.

Definitions of color bands:

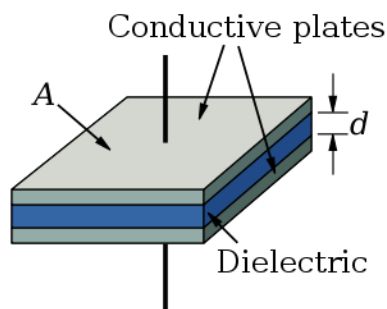
The color of the multiplier band represents multiples of 10, or the placement of the decimal point. For example: ORANGE (3) represents 10 to the third power or 1,000. The tolerance indicates, in a percentage, how much a resistor can vary above or below its value. A gold band stands for +/- 5%, a silver band stands for +/- 10%, and if there is no fourth band it is assumed to be +/- 20%. For example: A 100-Ohm 5% resistor can vary from 95 to 105 Ohms and still be considered within the manufactured tolerance. The temperature coefficient band specifies the maximum change in resistance with change in temperature, measured in parts per million per degree Centigrade (ppm/°C).

Example (from chart): Lets look at the first resistor on the chart. In this case, the first color band is BROWN. Following the line down the chart you can see that BROWN represents the number 1. This becomes our first significant digit. Next, look at the second band and you will see it is BLACK. Once again, follow the line down to the bar scale; it holds a value of 0, our second significant digit. Next, look at the third band, the multiplier, and you will see it is ORANGE. Once again, follow the line down to the bar scale; it holds a value of 3. This represents 3 multiples of 10 or 1000. With this information, the resistance is determined by taking the first two digits, 1 and 0 (10) and multiplying by 1,000. Example: $10 \times 1000 = 10,000$ or 10,000 Ohms. Using the chart, the fourth band (GOLD), indicates that this resistor has a tolerance of +/- 5%. Thus, the permissible range is: $10,000 \times .05 = +/- 500$ Ohms, or 9,500 to 10,500 Ohms.

CAPACITOR

A capacitor (originally known as a condenser) is a passive two-terminal electrical component used to store energy electro statically in an electric field. By contrast, batteries store energy via chemical reactions. The forms of practical capacitors vary widely, but all contain at least two electrical conductors separated by a dielectric (insulator); for example, one common construction consists of metal foils separated by a thin layer of insulating film. Capacitors are widely used as parts of electrical circuits in many common electrical devices.

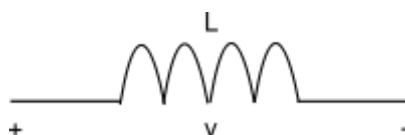
When there is a potential difference (voltage) across the conductors, a static electric field develops across the dielectric, causing positive charge to collect on one plate and negative charge on the other plate. Energy is stored in the electrostatic field. An ideal capacitor is characterized by a single constant value, capacitance. This is the ratio of the electric charge on each conductor to the potential difference between them. The SI unit of capacitance is the farad, which is equal to one coulomb per volt.




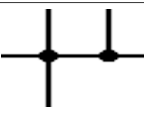
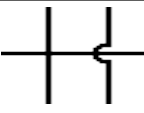
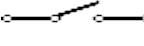
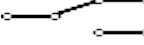
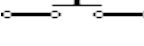
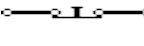



INDUCTOR

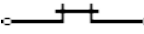





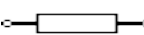

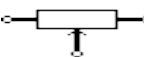
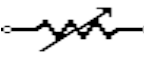
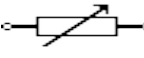
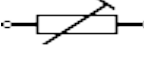
An inductor, also called a coil or reactor, is a passive two-terminal electrical component which resists changes in electric current passing through it. It consists of a conductor such as a wire, usually wound into a coil. When a current flows through it, energy is stored temporarily in a magnetic field in the coil. When the current flowing through an inductor changes, the time-varying magnetic field induces a voltage in the conductor, according to Faraday's Law of electromagnetic induction. According to Lenz's law the direction of induced e.m.f is always such that it opposes the change in current that created it. As a result, inductors always oppose a change in current.

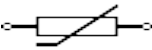
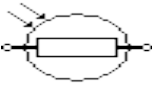
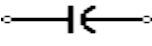
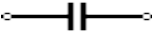
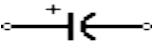
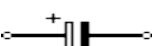
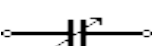
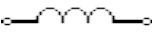
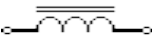
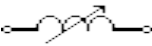

An inductor is characterized by its inductance, the ratio of the voltage to the rate of change of current, which has units of Henries (H). Many inductors have a magnetic core made of iron or ferrite inside the coil, which serves to increase the magnetic field and thus the inductance. Along with capacitors and resistors, inductors are one of the three passive linear circuit elements that make up electric circuits. Inductors are widely used in alternating current (AC) electronic equipment. They are used to block AC while allowing DC to pass; inductors designed for this purpose are called chokes. They are also used in electronic filters to separate signals of different frequencies, and in combination with capacitors to make tuned circuits, used to tune radio and TV receivers.




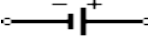












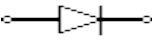
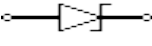
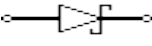
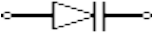
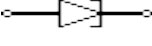
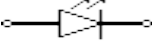
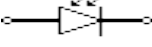


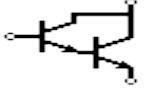
2. CIRCUIT SYMBOLS







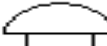
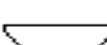
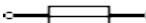




Symbol	Component name	Meaning
Wire Symbols		
	Electrical Wire	Conductor of electrical current
	Connected Wires	Connected crossing
	Not Connected Wires	Wires are not connected
Switch Symbols and Relay Symbols		
	SPST Toggle Switch	Disconnects current when open
	SPDT Toggle Switch	Selects between two connections
	Pushbutton Switch (N.O)	Momentary switch - normally open
	Pushbutton Switch (N.C)	Momentary switch - normally closed
	DIP Switch	DIP switch is used for onboard configuration
	SPST Relay	Relay open / close connection by an electromagnet
	SPDT Relay	

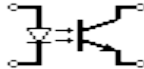
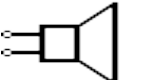



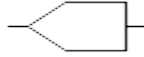

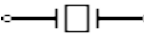



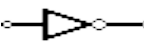
	Jumper	Close connection by jumper insertion on pins.
	Solder Bridge	Solder to close connection
Ground Symbols		
	Earth Ground	Used for zero potential reference and electrical shock protection.
	Chassis Ground	Connected to the chassis of the circuit
	Digital / Common Ground	
Resistor Symbols		
	Resistor (IEEE)	Resistor reduces the current flow.
	Resistor (IEC)	
	Potentiometer (IEEE)	Adjustable resistor - has 3 terminals.
	Potentiometer (IEC)	
	Variable Resistor / Rheostat (IEEE)	Adjustable resistor - has 2 terminals.
	Variable Resistor / Rheostat (IEC)	
	Trimmer Resistor	Preset resistor

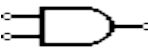




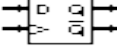



	Thermistor	Thermal resistor - change resistance when temperature changes
	Photoresistor / Light dependent resistor (LDR)	Photo-resistor - change resistance with light intensity change
Capacitor Symbols		
	Capacitor	Capacitor is used to store electric charge. It acts as short circuit with AC and open circuit with DC.
	Capacitor	
	Polarized Capacitor	Electrolytic capacitor
	Polarized Capacitor	Electrolytic capacitor
	Variable Capacitor	Adjustable capacitance
Inductor / Coil Symbols		
	Inductor	Coil / solenoid that generates magnetic field
	Iron Core Inductor	Includes iron
	Variable Inductor	
Power Supply Symbols		
	Voltage Source	Generates constant voltage

	Current Source	Generates constant current.
	AC Voltage Source	AC voltage source
	Generator	Electrical voltage is generated by mechanical rotation of the generator
	Battery Cell	Generates constant voltage
	Battery	Generates constant voltage
	Controlled Voltage Source	Generates voltage as a function of voltage or current of other circuit element.
	Controlled Current Source	Generates current as a function of voltage or current of other circuit element.
Meter Symbols		
	Voltmeter	Measures voltage. Has very high resistance. Connected in parallel.
	Ammeter	Measures electric current. Has near zero resistance. Connected serially.
	Ohmmeter	Measures resistance
	Wattmeter	Measures electric power
Lamp / Light Bulb Symbols		
	Lamp / light bulb	Generates light when current flows through

	Lamp / light bulb	
	Lamp / light bulb	
Diode / LED Symbols		
	Diode	Diode allows current flow in one direction only - left (anode) to right (cathode).
	Zener Diode	Allows current flow in one direction, but also can flow in the reverse direction when above breakdown voltage
	Schottky Diode	Schottky diode is a diode with low voltage drop
	Varactor / Varicap Diode	Variable capacitance diode
	Tunnel Diode	
	Light Emitting Diode (LED)	LED emits light when current flows through
	Photodiode	Photodiode allows current flow when exposed to light
Transistor Symbols		
	NPN Bipolar Transistor	Allows current flow when high potential at base (middle)
	PNP Bipolar Transistor	Allows current flow when low potential at base (middle)
	Darlington Transistor	Made from 2 bipolar transistors. Has total gain of the product of each gain.

	JFET-N Transistor	N-channel field effect transistor
	JFET-P Transistor	P-channel field effect transistor
	NMOS Transistor	N-channel MOSFET transistor
	PMOS Transistor	P-channel MOSFET transistor
Misc. Symbols		
	Motor	Electric motor
	Transformer	Change AC voltage from high to low or low to high.
	Electric bell	Rings when activated
	Buzzer	Produce buzzing sound
	Fuse	The fuse disconnects when current above threshold. Used to protect circuit from high currents.
	Fuse	
	Bus	Contains several wires. Usually for data / address.
	Bus	
	Bus	

	Optocoupler / Opto-isolator	Optocoupler isolates connection to other board
	Loudspeaker	Converts electrical signal to sound waves
	Microphone	Converts sound waves to electrical signal
	Operational Amplifier	Amplify input signal
	Schmitt Trigger	Operates with hysteresis to reduce noise.
	Analog-to-digital converter (ADC)	Converts analog signal to digital numbers
	Digital-to-Analog converter (DAC)	Converts digital numbers to analog signal
	Crystal Oscillator	Used to generate precise frequency clock signal
Antenna Symbols		
	Antenna / aerial	Transmits & receives radio waves
	Antenna / aerial	
	Dipole Antenna	Two wires simple antenna
Logic Gates Symbols		
	NOT Gate (Inverter)	Outputs 1 when input is 0

	AND Gate	Outputs 1 when both inputs are 1.
	NAND Gate	Outputs 0 when both inputs are 1. (NOT + AND)
	OR Gate	Outputs 1 when any input is 1.
	NOR Gate	Outputs 0 when any input is 1. (NOT + OR)
	XOR Gate	Outputs 1 when inputs are different. (Exclusive OR)
	D Flip-Flop	Stores one bit of data
	Multiplexer / Mux 2 to 1	Connects the output to selected input line.
	Multiplexer / Mux 4 to 1	
	Demultiplexer / Demux 1 to 4	Connects selected output to the input line.

3. STUDY OF CATHODE RAY AND DIGITAL STORAGE OSCILLOSCOPE

An oscilloscope is a test instrument which allows us to look at the 'shape' of electrical signals by displaying a graph of voltage against time on its screen. It is like a voltmeter with the valuable extra function of showing how the voltage varies with time. A graticule with a 1cm grid enables us to take measurements of voltage and time from the screen. The graph, usually called the trace, is drawn by a beam of electrons striking the phosphor coating of the screen making it emit light, usually green or blue. This is similar to the way a television picture is produced.

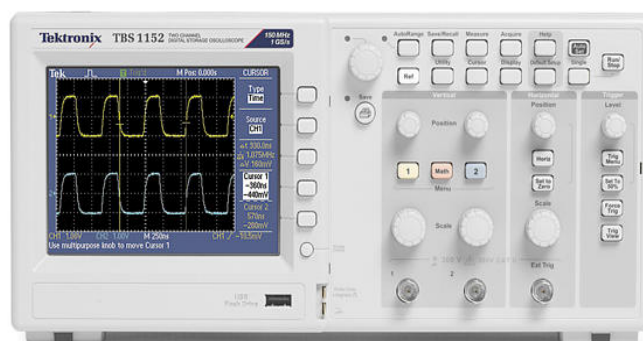
Oscilloscopes contain a vacuum tube with a cathode (negative electrode) at one end to emit electrons and an anode (positive electrode) to accelerate them so they move rapidly down the tube to the screen. This arrangement is called an electron gun. The tube also contains electrodes to deflect the electron beam up/down and left/right.

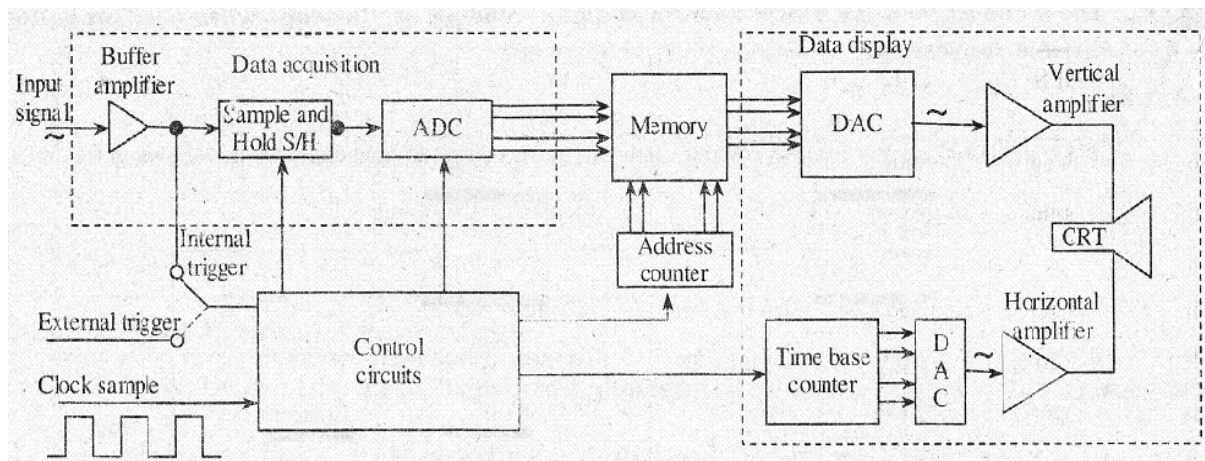
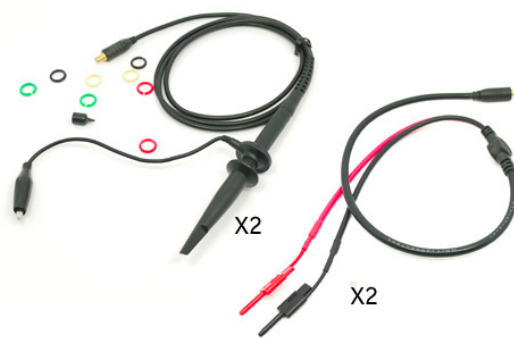
The electrons are called cathode rays because they are emitted by the cathode and this gives the oscilloscope its full name of cathode ray oscilloscope or CRO. A dual trace oscilloscope can display two traces on the screen, allowing us to easily compare the input and output of an amplifier for example. It is well worth paying the modest extra cost to have this facility.

A **digital storage oscilloscope** is an oscilloscope which stores and analyses the signal digitally rather than using analogue techniques. It is now the most common type of oscilloscope in use because of the advanced trigger, storage, display and measurement features which it typically provides.

The input analogue signal is sampled and then converted into a digital record of the amplitude of the signal at each sample time. The sampling frequency should be not less than the Nyquist rate to avoid aliasing. These digital values are then turned back into an analogue signal for display on a cathode ray tube (CRT), or transformed as needed for the various possible types of output—liquid crystal display, chart recorder, plotter or network interface.

The figures shown below are Digital Storage Oscilloscope and it's Probes





The above figure shows the block diagram of DSO as consists of,

1. Data acquisition
2. Storage
3. Data display

Data acquisition is earned out with the help of both analog to digital and digital to analog converters, which is used for digitizing, storing and displaying analog waveforms. Overall operation is controlled by control circuit which is usually consists of microprocessor. Data acquisition portion of the system consist of a Sample-and-Hold (S/H) circuit and an analog to digital converter (ADC) which continuously samples and digitizes the input signal at a rate determined by the sample clock and transmit the digitized data to memory for storage. The control circuit determines whether the successive data points are stored in successive memory location or not, which is done by continuously updating the memories.

When the memory is full, the next data point from the ADC is stored in the first memory location writing over the old data. The data acquisition and the storage process is continues till the control circuit receive a trigger signal from either the input waveform or an external trigger source. When the triggering occurs, the system stops and enters into the display mode of operation in which all or some part of the memory data is repetitively displayed on the cathode ray tube. In display operation, two DACs are used which gives horizontal and

vertical deflection voltage for the CRT Data from the memory gives the vertical deflection of the electron beam, while the time base counter gives the horizontal deflection in the form of staircase sweep signal. The screen display consist of discrete dots representing the various data points but the number of dot is very large as 1000 or more that they tend to blend together and appear to be a smooth continuous waveform.

The display operation ends when the operator presses a front-panel button and commands the digital storage oscilloscope to begin a new data acquisition cycle.

Advantages:

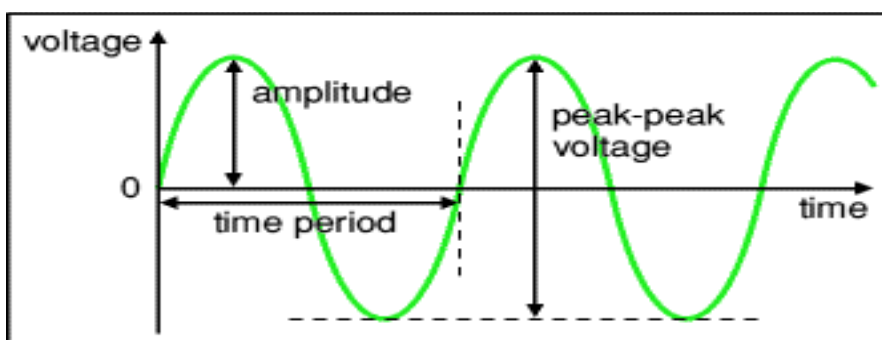
1. Allows for automation.
2. In this,slow traces like the temperature variation across a day can be recorded
3. With colour Bigger and brighter display, to distinguish multiple traces
4. peak detection

Disadvantages:

1. Digital Oscilloscope is limited by the refresh rate of the screen.

Measuring voltage and time period

The trace on an oscilloscope screen is a graph of voltage against time. The shape of this graph is determined by the nature of the input signal. In addition to the properties labeled on the graph, there is frequency which is the number of cycles per second. The diagram shows a sine wave but these properties apply to any signal with a constant shape.



1. Amplitude is the maximum voltage reached by the signal. It is measured in Volts.
2. Peak voltage is another name for amplitude.
3. Peak-peak voltage is twice the peak voltage (amplitude). When reading an oscilloscope trace it is usual to measure peak-peak voltage.

4. Time period is the time taken for the signal to complete one cycle. It is measured in seconds (s), but time periods tend to be short so milliseconds (ms) and microseconds (μs) are often used. $1\text{ms} = 0.001\text{s}$ and $1\mu\text{s} = 0.000001\text{s}$.
5. Frequency is the number of cycles per second. It is measured in hertz (Hz), but frequencies tend to be high so kilohertz (kHz) and megahertz (MHz) are often used.

$1\text{kHz} = 1000\text{Hz}$ and $1\text{MHz} = 1000000\text{Hz}$.

Frequency = $1 / \text{Time period}$

Voltage: Voltage is shown on the vertical y-axis and the scale is determined by the Y AMPLIFIER (VOLTS/CM) control. Usually peak-peak voltage is measured because it can be read correctly even if the position of 0V is not known. The amplitude is half the peak-peak voltage.

Time period: Time is shown on the horizontal x-axis and the scale is determined by the TIMEBASE (TIME/CM) control. The time period (often just called period) is the time for one cycle of the signal. The frequency is the number of cycles per second,

Frequency = $1/\text{Time period}$.

Time = distance in cm \times time/cm

4. STUDY OF FUNCTION GENERATOR

A function generator is a device that can produce various patterns of voltage at a variety of frequencies and amplitudes. It is used to test the response of circuits to common input signals. The electrical leads from the device are attached to the ground and signal input terminals of the device under test.



Function generator capabilities

Function generators are capable of producing a variety of repetitive waveforms, generally from the list below:

Sine wave: A function generator will normally have the capability to produce a standard sine wave output. This is the standard waveform that oscillates between two levels with a standard sinusoidal shape.



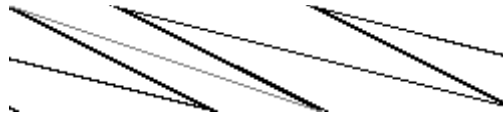
Square wave: A square wave is normally relatively easy for a function generator to produce. It consists of a signal moving directly between high and low levels.



Pulse: A pulse waveform is another type that can be produced by a function generator. It is effectively the same as a square wave, but with the mark space ratio very different to 1:1.



Triangular wave: This form of signal produced by the function generator linearly moves between a high and low point.



Sawtooth wave: Again, this is a triangular waveform, but with the rise edge of the waveform faster or slower than the fall, making a form of shape similar to a sawtooth.



Function generator controls

In addition to a selection of the basic waveforms that are available, other controls on the function generator may include:

Frequency: As would be expected, this control alters the basic frequency at which the waveform repeats. It is independent of the waveform type.

Waveform type : This enables the different basic waveform types to be selected:

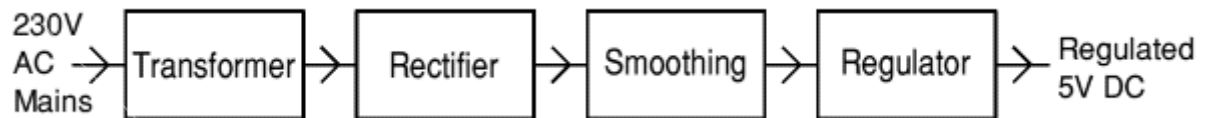
1. Sine Wave
2. Square Wave
3. Triangular Wave

DC offset: This alters the average voltage of a signal relative to 0V or ground.

Duty cycle: This control on the function generator changes the ratio of high voltage to low voltage time in a square wave signal, i.e. changing the waveform from a square wave with a 1:1 duty cycle to a pulse waveform, or a triangular waveform with equal rise and fall times to a sawtooth.

5. STUDY OF REGULATED POWER SUPPLY

There are many types of power supply. Most are designed to convert high voltage AC mains electricity to a suitable low voltage supply for electronic circuits and other devices. A power supply can be broken down into a series of blocks, each of which performs a particular function. For example a 5V regulated supply:



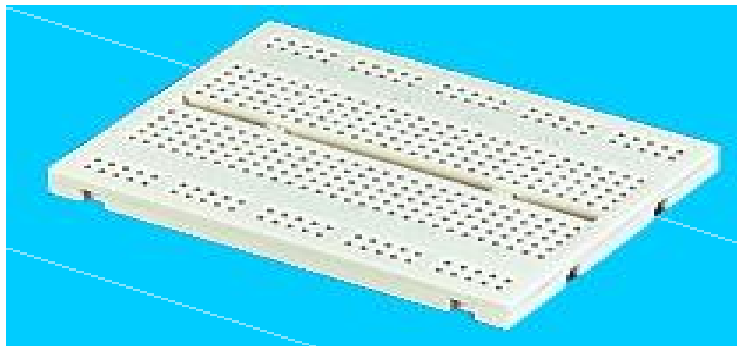
Each of the blocks is described in more detail below:

1. Transformer: Steps down high voltage AC mains to low voltage AC.
2. Rectifier: Converts AC to DC, but the DC output is varying.
3. Smoothing: Smooths the DC from varying greatly to a small ripple.
4. Regulator: Eliminates ripple by setting DC output to a fixed voltage.

6. TYPES OF CIRCUIT BOARD

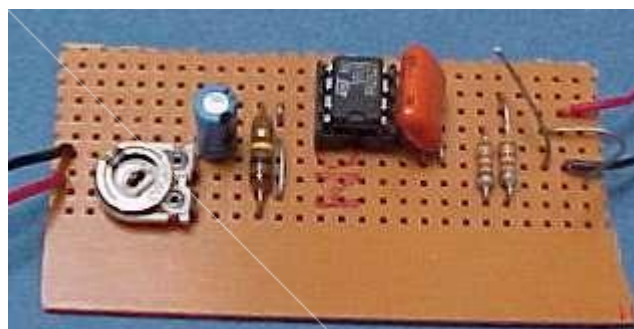
Breadboard

This is a way of making a temporary circuit, for testing purposes or to try out an idea. No soldering is required and all the components can be re-used afterwards. It is easy to change connections and replace components. Almost all the Electronics Club projects started life on a breadboard to check that the circuit worked as intended. The following figure depicts the appearance of Bread board in which the holes in top and bottom stripes are connected horizontally that are used for power supply and ground connection conventionally and holes on middle stripes connected vertically. And that are used for circuit connections conventionally.



Strip Board

Stripboard has parallel strips of copper track on one side. The strips are 0.1" (2.54mm) apart and there are holes every 0.1" (2.54mm). Stripboard requires no special preparation other than cutting to size. It can be cut with a junior hacksaw, or simply snap it along the lines of holes by putting it over the edge of a bench or table and pushing hard.



Printed Circuit Board

A printed circuit board, or PCB, is used to mechanically support and electrically connect electronic components using conductive pathways, tracks or traces etched from copper sheets laminated onto a non-conductive substrate. It is also referred to as printed wiring board (PWB) or etched wiring board. A PCB populated with electronic components is a printed circuit assembly (PCA), also known as a printed circuit board assembly (PCBA).

Printed circuit boards have copper tracks connecting the holes where the components are placed. They are designed specially for each circuit and make construction very easy. However, producing the PCB requires special equipment so this method is not recommended if you are a beginner unless the PCB is provided for you.



PCBs are inexpensive, and can be highly reliable. They require much more layout effort and higher initial cost than either wire-wrapped or point-to-point constructed circuits, but are much cheaper and faster for high-volume production. Much of the electronics industry's PCB design, assembly, and quality control needs are set by standards that are published by the IPC organization.

PART A
PSPICE EXPERIMENTS

1. Experimentation on Thevenin's and Norton's Theorems and Verification using PSPICE

Verification of Thevenin's Theorem

AIM: To construct a circuit and verify Thevenin's Theorem for the given circuit.

TOOLS USED: NI Multisim version 14

THEORY:

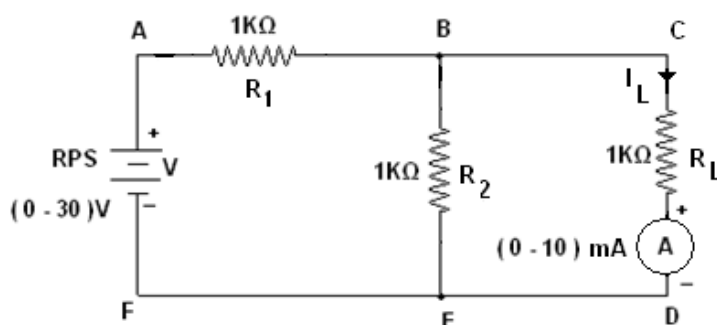
Thevenin's Theorem:

This theorem states that a network composed of lumped, linear circuit elements may, for the purposes of analysis of external circuit or terminal behaviour, be replaced by a voltage source $V(s)$ in series with a single impedance.

Thevenin's theorem simplifies the method of finding current through any specified branch. For this purpose we have to find two things:

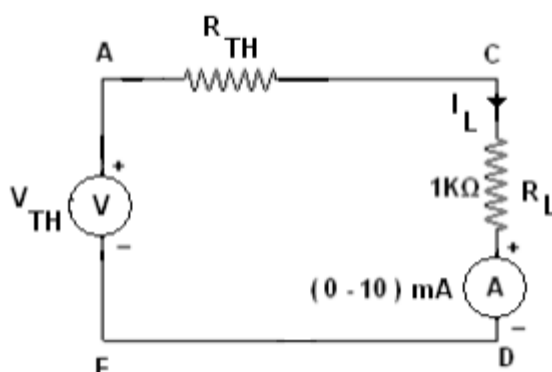
1. Thevenin's Resistance R_{TH}
2. Thevenin's Voltage V_{TH}

CIRCUIT DIAGRAM:



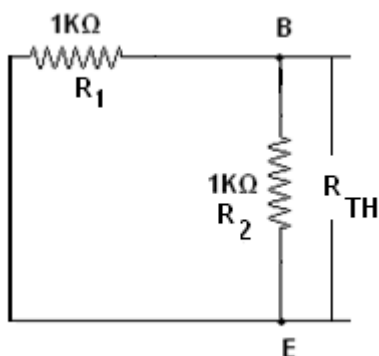
THEORETICAL CALCULATIONS:

The Thevenin's equivalent circuit is,



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

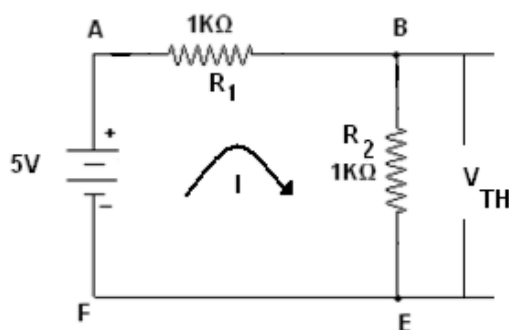
To find R_{TH}



$$R_1 = 1K\Omega, R_2 = 1K\Omega$$

$$R_{TH} = \frac{R_1 \cdot R_2}{R_1 + R_2}$$

To find V_{TH}



$$I = \frac{V}{R_L}$$

Say, $V = 5V, V_{TH} = V_{BE}$

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

PROCEDURE:

1. Find the Thevenin source voltage by removing the load resistor from the original circuit and calculating voltage across the open connection points where the load resistor used to be.
2. Find the Thevenin resistance by removing all power sources in the original circuit (voltage sources shorted and current sources open) and calculating total resistance between the open connection points.
3. Draw the Thevenin equivalent circuit, with the Thevenin voltage source in series with the Thevenin resistance. The load resistor re-attaches between the two open points of the equivalent circuit.
4. Analyze voltage and current for the load resistor following the rules for series circuits.

OBSERVATIONS:

Theoretical values			Practical values (Multisim)		
V_{TH}	R_{TH}	I_L	V_{TH}	R_{TH}	I_L
2.5V	500 ohm	4.167mA	2.5V	500 ohm	4.167mA

RESULT: Thus, the Thevenin’s theorem for the given circuit is verified successfully.

VIVA QUESTIONS:

1. To what type of circuit Thevenin's theorem is applicable ?
2. What is the use of Thevenin's theorem?
3. How RTH is connected with the ckt?
4. How is RTH connected with the load resistance?
5. What modification is done in galvanometer to convert it into a ammeter?
6. What modification is done in the galvanometer to convert it into a voltmeter?
7. Resistance is a n active element or the passive?
8. How will you calculate the RTH?
9. In place of current source, what is placed while calculating RTH?
10. In place of voltage source which electrical parameters is placed?

Verification of Norton's Theorem

AIM: To construct a circuit and verify Norton's Theorem for the given circuit.

TOOLS REQUIRED: NI Multisim version 14

THEORY:

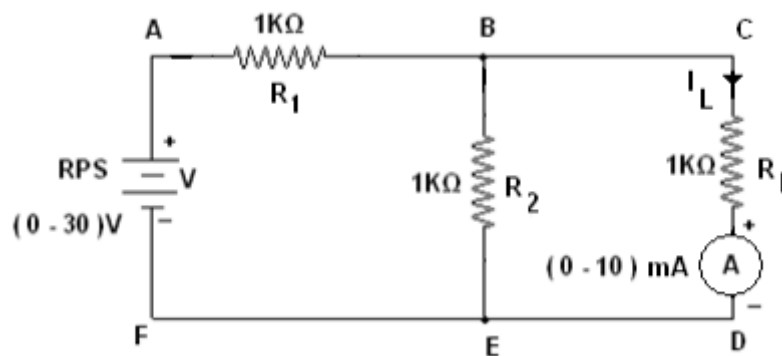
Norton's Theorem:

This theorem states that it is possible to simplify any linear circuit, no matter how complex, to an equivalent circuit with just a single current source and parallel resistance connected to a load i.e. the equivalent circuit composes of a single current source, parallel resistance, and parallel load.

Also similar to Thevenin's Theorem are the steps used in Norton's Theorem to calculate the

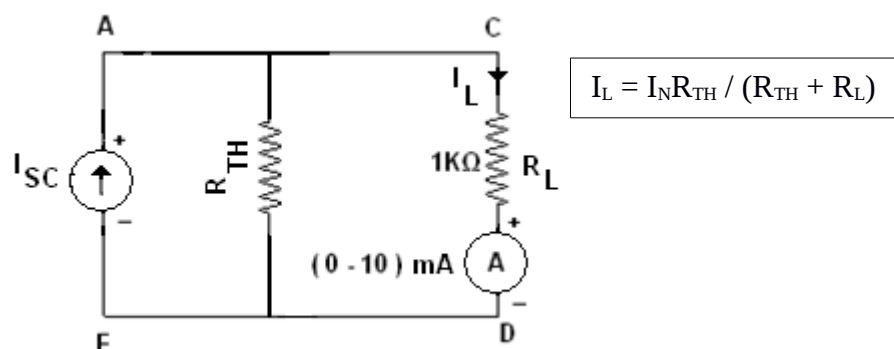
1. Norton source current (I_N)
2. Norton resistance (R_N).

CIRCUIT DIAGRAM:

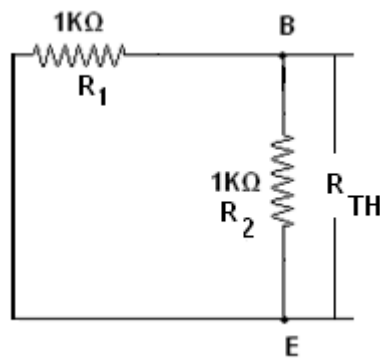


THEORETICAL CALCULATIONS:

The Norton's equivalent circuit is,



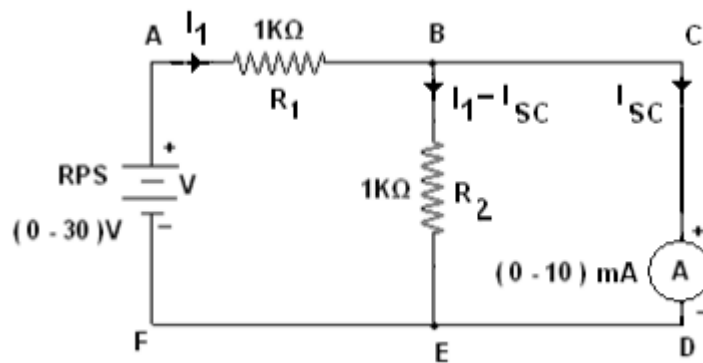
To find R_N



$$R_1 = 1k\Omega, R_2 = 1k\Omega$$

$$R_{TH} = \frac{R_1 \cdot R_2}{R_1 + R_2}$$

To find I_N



Say, $V = 5V$

In the loop ABEFA by applying KVL,

$$I_1 R_1 + (I_1 - I_{SC}) R_2 = 5$$

In the loop BCDEB by applying KVL,

$$(I_1 - I_{SC}) R_2 = 0$$

∴ From the above equations,

We get the value of I_{SC} which is also I_N i.e. Norton's Current

From I_N , we get the amount of current passing through the load by,

$$I_L = I_N R_{TH} / (R_{TH} + R_L)$$

PROCEDURE:

1. Find the Norton source current by removing the load resistor from the original circuit and calculating current through a short (wire) jumping across the open connection points where the load resistor used to be.
2. Find the Norton resistance by removing all power sources in the original circuit (voltage sources shorted and current sources open) and calculating total resistance between the open connection points.
3. Draw the Norton equivalent circuit, with the Norton current source in parallel with the Norton resistance. The load resistor re-attaches between the two open points of the equivalent circuit.
4. Analyze voltage and current for the load resistor following the rules for parallel circuits.

OBSERVATIONS:

Theoretical values			Practical values (Multisim)		
V_N	R_N	I_L	V_N	R_N	I_L
15V	500 ohm	30 mA	15V	500 ohm	30 mA

RESULT: Thus, the Norton’s theorem for the given circuit is verified successfully.

VIVA QUESTIONS:

1. To what type of network Norton’s theorem applicable?
2. How is R_N connected to I_N ?
3. What is placed in place of voltage sources while calculating the R_N ?
4. Give an example of unilateral ckt?
5. What is unilateral ckt?
6. Give one example of the bilateral n/w?
7. What is the limitation of Ohm’s law?
8. What is the reason that ground pin are made of greater diameter in the plugs?
9. Where is the voltage divider rule applicable?
10. Where is the current divider rule applicable?

2. Experimentation on Superposition Theorem and Maximum Power Transfer Theorem and Verification using MULTISIM

Verification of Superposition Theorem

AIM: To verify the superposition theorem for the given circuit.

TOOLS REQUIRED: NI Multisim version 14

THEORY:

Superposition Theorem:

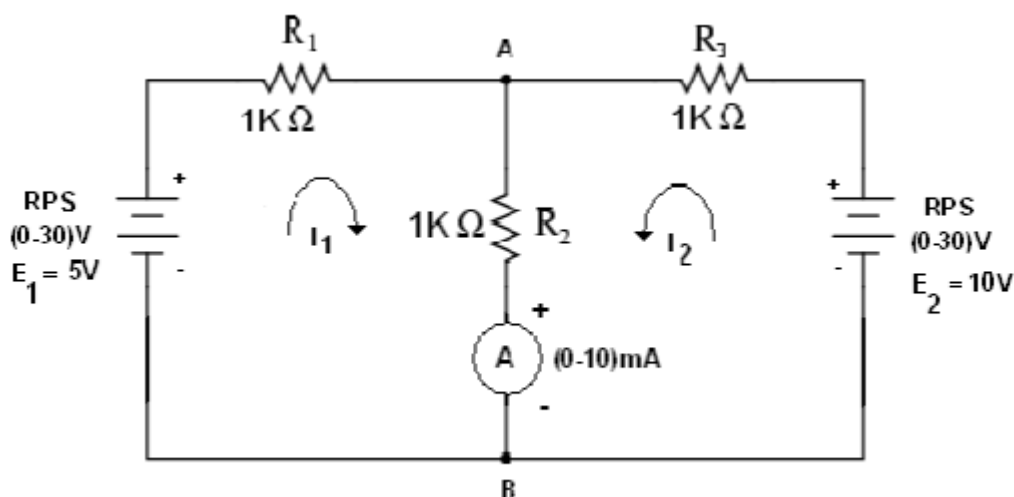
The superposition theorem for electrical circuits states that the total current in any branch of a bilateral linear circuit equals the algebraic sum of the currents produced by each source acting separately throughout the circuit.

To ascertain the contribution of each individual source, all of the other sources first must be "killed" (set to zero) by:

1. Replacing all other voltage sources with a short circuit (thereby eliminating difference of potential. i.e. $V=0$)
2. Replacing all other current sources with an open circuit (thereby eliminating current. i.e. $I=0$)

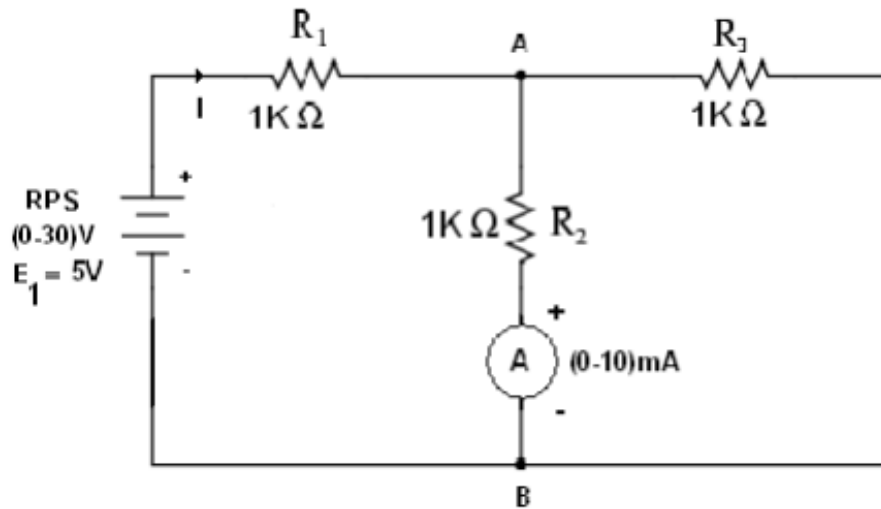
This procedure is followed for each source in turn, and then the resultant currents are added to determine the true operation of the circuit. The resultant circuit operation is the superposition of the various voltage and current sources.

CIRCUIT DIAGRAM:



THEORETICAL CALCULATIONS:

Source E2 is removed,

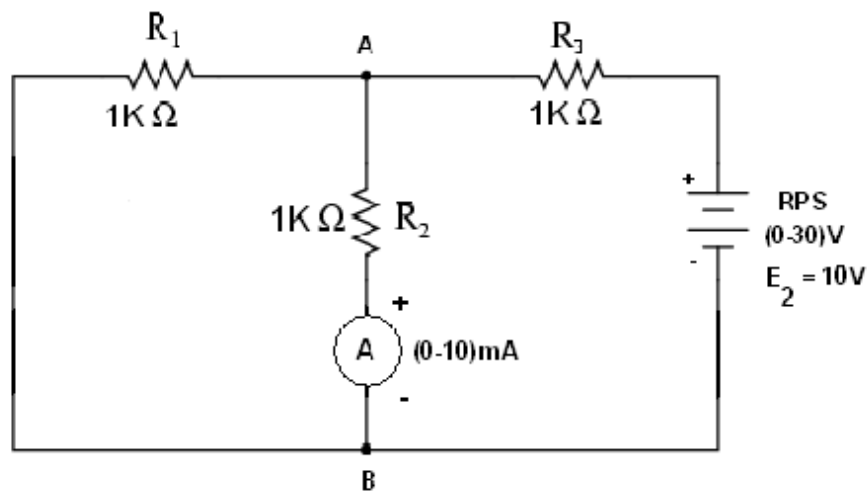


$$R_T = R_1 + [R_2.R_3 / (R_2 + R_3)]$$

$$I = V / R_T$$

$$I_{AB1} = I.R_3 / (R_2 + R_3)$$

Source E1 is removed,



$$R_T = R_3 + [R_2.R_1 / (R_2 + R_1)]$$

$$I = V / R_T$$

$$I_{AB2} = I.R_3 / (R_2 + R_3)$$

Both E1 and E2 sources are acting, then

$$I_{AB} = I_{AB1} + I_{AB2}$$

PROCEDURE:

1. For the given circuit, remove source E2 and by using KVL, determine the current flowing through R2 i.e. branch AB theoretically and designate it as I_{AB1}
2. Now, remove source E1 and reattach source E2 and by using KVL, determine the current flowing through R2 i.e. branch AB theoretically and designate it as I_{AB2}
3. Simulate the same in the Multisim tool and compare both theoretical and practical values
4. Apply KVL to the original circuit, without removing any sources, and determine the current flowing through branch AB both theoretically and practically.

OBSERVATIONS:

Source E2 is removed,

E1 Voltage (V)	Load Current in Branch AB (mA)	
	Theoretical	Practical
5	1.667	1.667

Source E1 is removed

E2 Voltage (V)	Load Current in Branch AB (mA)	
	Theoretical	Practical
10	3.33	3.33

Both E1 and E2 are acting,

E1 Voltage (V)	E2 Voltage (V)	Load Current in Branch AB (mA)	
		Theoretical	Practical
5	10	4.66	4.66

RESULT: Thus, the superposition theorem for the given circuit is verified.

Verification of Maximum Power Transfer Theorem

AIM: To verify the maximum power transfer theorem for the given circuit.

TOOLS REQUIRED: NI Multisim version 14

THEORY:

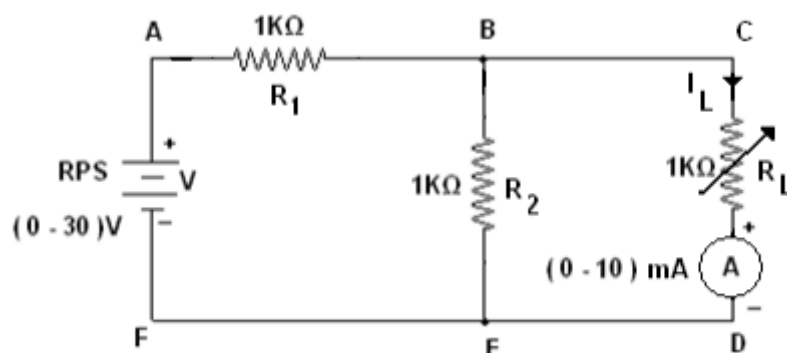
Maximum Power Transfer Theorem:

In electrical engineering, the maximum power (transfer) theorem states that, to obtain maximum external power from a source to a load with a finite internal resistance, the resistance of the load must be made the same as that of the source.

The theorem applies to maximum power, and not maximum efficiency. If the resistance of the load is made larger than the resistance of the source, then efficiency is higher, since most of the power is generated in the load, but the overall power is lower since the total circuit resistance goes up.

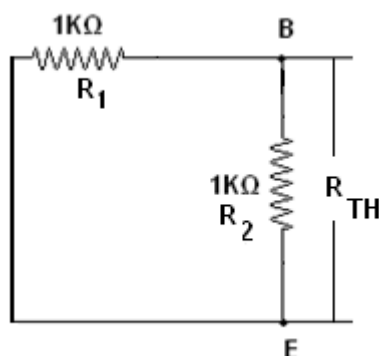
If the internal impedance is made larger than the load then most of the power ends up being dissipated in the source, and although the total power dissipated is higher, due to a lower circuit resistance, it turns out that the amount dissipated in the load is reduced.

CIRCUIT DIAGRAM:



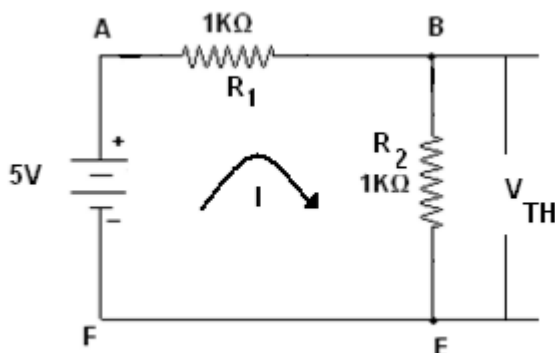
THEORETICAL CALCULATIONS:

To find R_{TH}



$R_1 = 1K\Omega, R_2 = 1K\Omega$ $R_{TH} = \frac{R_1 \cdot R_2}{R_1 + R_2}$
--

To find V_{TH}



$$I = V / R_L$$

Say, $V = 5V$, $V_{TH} = V_{BE}$

Now, we have values of both V_{TH} and R_{TH} , therefore, the load current,

$$I_L = V_{TH} / (R_{TH} + R_L)$$

$$\text{Power Transferred} = [V_{TH} / (R_{TH} + R_L)]^2 \cdot R_L$$

Maximum power is transferred when $R_{TH} = R_L$ i.e. $I_L = V_{TH} / 2R_L$

& Maximum power = $I_L^2 \cdot R_L$

$$= V_{TH}^2 / 4R_L$$

PROCEDURE:

1. As per the given circuit, calculate V_{TH} and R_{TH} theoretically for varying R_L
2. Note down the power delivered to the load and the power at the source for varying R_L
3. Verify the resistance (R_L) at which maximum power is transferred.

OBSERVATIONS:

Resistance (R_L)	V_{TH} (Volts)	R_{TH} (KΩ)	Load Current (I_L)	Power at Source ($V_{TH} \cdot I_L$)	Power Transferred ($(V_{TH} / R_{TH} + R_L)^2 \cdot R_L$)
500 ohm	2.5	0.5	2.5	6.25	0.0031

RESULT: Thus, the maximum power transfer theorem for the given circuit is verified successfully.

VIVA QUESTIONS:

1. State Superposition and Maximum Power Transfer Theorems and their advantages.

2. What is load matching?
3. What is max power transfer formula?
4. What is the field of application of this theorem?
5. What is electric network?
6. What is necessary to know the polarity of voltage drop across a resistance?
7. What is the reason that terminal voltage is less than emf?
8. What is the resistance of ideal voltage source? A. Zero
9. When will the power extracted from a ckt is maximum?
10. How is the ammeter connected in circuit?
11. To find the voltage drop across a resistance, where should the voltmeter be connected?

3. Experimentation on Transient analysis of RL and RC Series Circuits and Verification using MULTISIM

Transient Analysis of RC Circuit

AIM: To construct RC transient circuit and to obtain the transient curves.

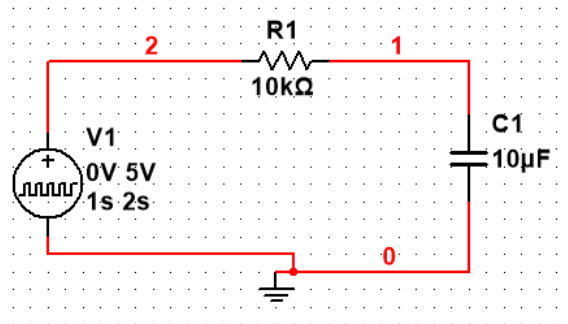
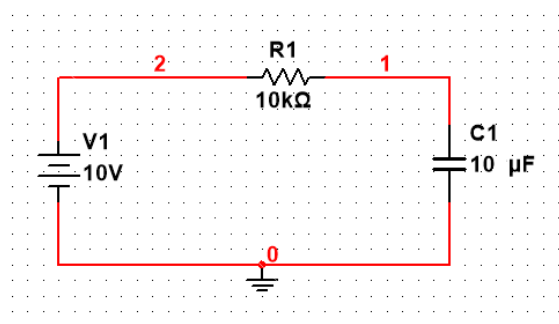
TOOLS REQUIRED: NI Multisim version 14

THEORY:

Electrical devices are controlled by switches which are closed to connect supply to the device, or opened in order to disconnect the supply to the device. The switching operation will change the current and voltage in the device. The purely resistive devices will allow instantaneous change in current and voltage.

An inductive device will not allow sudden change in current and capacitance device will not allow sudden change in voltage. Hence when switching operation is performed in inductive and capacitive devices, the current & voltage in device will take a certain time to change from pre switching value to steady state value after switching. This phenomenon is known as transient. The study of switching condition in the circuit is called transient analysis. The state of the circuit from instant of switching to attainment of steady state is called transient state. The time duration from the instant of switching till the steady state is called transient period. The current & voltage of circuit elements during transient period is called transient response.

CIRCUIT DIAGRAM:



THEORETICAL OBSERVATIONS:

Time constant = RC

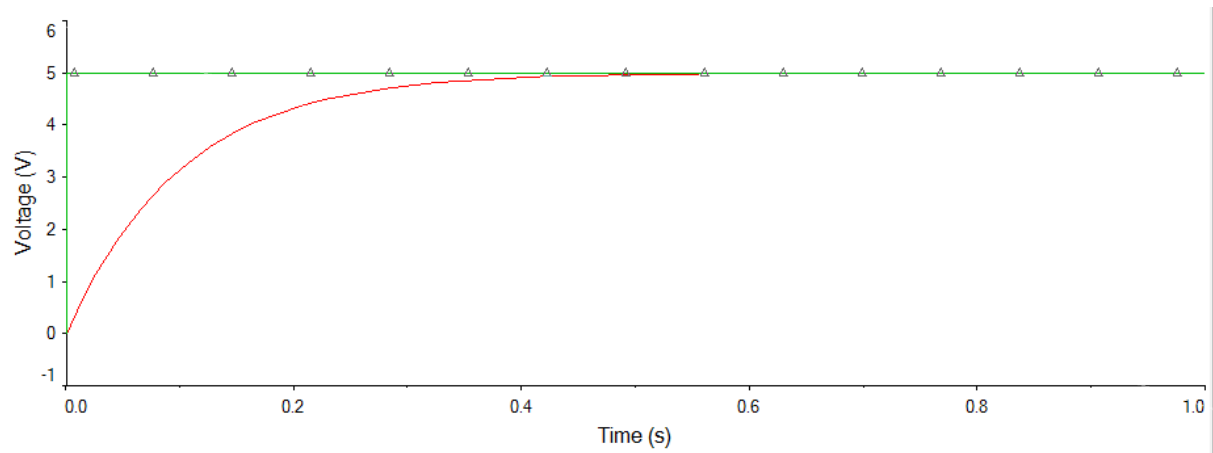
Note: The student must know how to derive the KVL equations (both charging and discharging situations) for a given RC circuit prior to the lab.

PROCEDURE:

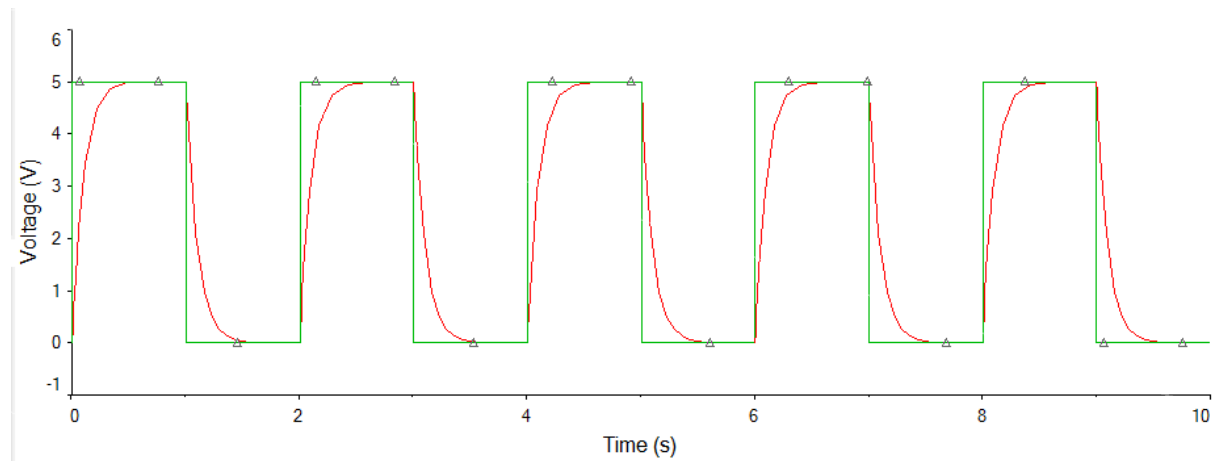
1. Simulate the above shown circuit in Multisim.s
2. Perform transient analysis for the given circuit and observe the voltage variations in the capacitor voltage.
3. Replace the DC voltage with a pulse voltage (shown above) and observe the capacitor voltage.

MODEL GRAPHS:

When DC source is applied, the capacitor voltage is, w.r.t input voltage,



When Pulse voltage is applied, the capacitor voltage is, w.r.t input waveform,



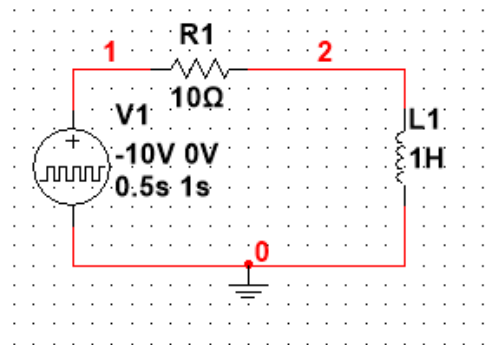
RESULT: Thus, the transient response of RC circuit for DC input and pulse input is verified.

Transient Analysis of RL Circuit

AIM: To construct RL transient circuit and to obtain the transient curves.

TOOLS REQUIRED: NI Multisim version 14

CIRCUIT DIAGRAM:



THEORETICAL OBSERVATIONS:

Time constant = L/R

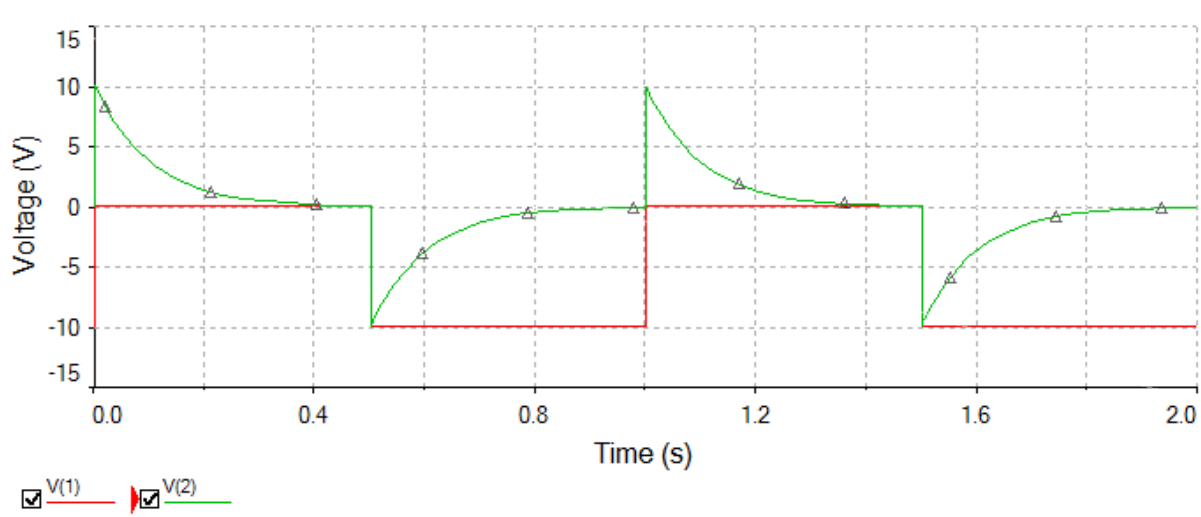
Note: The student must know learn how to derive the KVL equations (Both Charging and Discharging situations) for a given RL circuit prior to the lab.

PROCEDURE:

1. Simulate the above shown circuit in Multisim.
2. Perform transient analysis for the given circuit and observe the voltage variations in the inductor voltage.

MODEL GRAPHS:

When pulse source is applied, the inductor voltage is, w.r.t input voltage,



RESULT: Thus, the transient response of RL circuit for pulse input is verified.

VIVA QUESTIONS:

1. What is transient response?
2. What is transient state?
3. What is natural response?
4. Define time constant or RL Circuit
5. Define time constant of RC Circuit.
6. What is meant by natural frequency?
7. Define bandwidth.
8. Define selectivity.
9. Define quality factor.
10. Define half power frequencies ?
11. Write down the formula for inductive reactance and capacitive reactance?
12. What is impedance?
13. Define admittance.
14. Obtain the equivalent impedance and reactances for series and parallel connections.
15. Obtain the equivalent admittance for series and parallel connections.
16. Obtain the equivalent conductance and susceptance for series and parallel connections.

4. Experimentation on Frequency response of Series and Parallel Resonance using MULTISIM

Frequency Response of Series Resonance

AIM: To construct a Series Resonance circuit and study its frequency response.

TOOLS REQUIRED: NI Multisim version 14

THEORY:

Series RLC Circuit

When a resistor, inductor, and capacitor are connected in series with the voltage supply, the circuit so formed is called series RLC circuit. Since all these components are connected in series, the current in each element remains the same.

In a circuit containing inductor and capacitor, the energy is stored in two different ways.

- When a current flows in an inductor, energy is stored in magnetic field.
- When a capacitor is charged, energy is stored in static electric field.

The magnetic field in the inductor is built by the current, which gets provided by the discharging capacitor. Similarly, the capacitor is charged by the current produced by collapsing magnetic field of inductor and this process continues on and on, causing electrical energy to oscillate between the magnetic field and the electric field. In some cases, at certain frequency called resonant frequency, the inductive reactance of the circuit becomes equal to capacitive reactance which causes the electrical energy to oscillate between the electric field of the capacitor and magnetic field of the inductor. This forms a harmonic oscillator for current. In RLC circuit, the presence of resistor causes these oscillations to die out over period of time and it is called as the damping effect of resistor.

The frequency at which the reactance of the inductance and the capacitance cancel each other is the resonant frequency (or the unity power factor frequency) of this circuit. This occurs at

Input impedance is given by

$$Z = R + j\omega L + \frac{1}{j\omega C} = R + j\left(\omega L - \frac{1}{\omega C}\right)$$

For the circuit to be purely resistive, the imaginary part must be zero,

$$\omega_0 = \frac{1}{\sqrt{LC}} \quad (\text{rad/s})$$

$$f_0 = \frac{1}{2\pi\sqrt{LC}} \quad (\text{Hz})$$

Frequency response of current magnitude:

$$|I| = \frac{V_m}{\sqrt{R^2 + (\omega L - 1/\omega C)^2}}$$

The highest power:

$$P(\omega_0) = \frac{1}{2} \frac{V_m^2}{R}$$

Half-power frequencies (ω_1, ω_2):

$$P(\omega_1) = P(\omega_2) = \frac{1}{4} \frac{V_m^2}{R}$$

$$\omega_1 = -\frac{R}{2L} + \sqrt{\left(\frac{R}{2L}\right)^2 + \frac{1}{LC}}$$

$$\omega_2 = \frac{R}{2L} + \sqrt{\left(\frac{R}{2L}\right)^2 + \frac{1}{LC}}$$

$$\omega_0 = \sqrt{\omega_1 \omega_2}$$

Half-power bandwidth (B):

$$B = \omega_2 - \omega_1$$

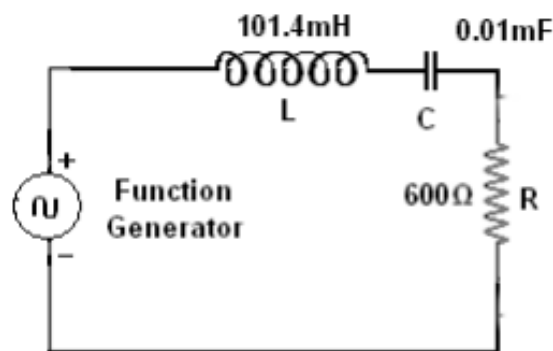
Quality factor (Q):

$Q = 2\pi$ (Peak energy stored)/(Energy dissipated in one period at resonance)

$$Q = \frac{2\pi fL}{R} = \frac{\omega_0 L}{R} = \frac{1}{\omega_0 CR}$$

$$B = \frac{R}{L} = \frac{\omega_0}{Q}$$

CIRCUIT DIAGRAM:



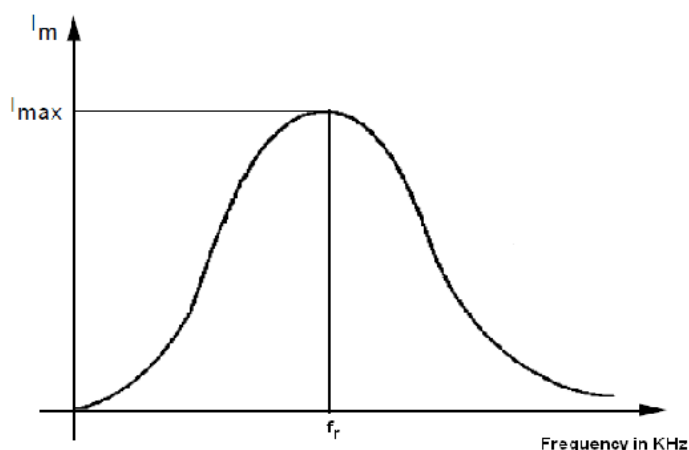
THEORETICAL OBSERVATIONS:

The student must calculate the resonant frequency (f_r), Bandwidth (BW), Half Power Frequencies (f_1 and f_2) and Quality factor (Q) using the above given equation with the values of R, L and C prior to the lab.

PROCEDURE:

1. Simulate the above shown circuit in Multisim.
2. The input is given in the form of sine wave by function generator.
3. The current amplitude of the response across the resistor is noted for various frequency ranges is tabulated.
4. On the graph, Current vs Frequency, note down the maximum current. Calculate 70.7% of this current and draw a horizontal line corresponding to this value on the graph. Note down the values at which this horizontal line intersects the curve (f_1 and f_2).

MODEL GRAPHS:



OBSERVATIONS:

Frequency (Hz)	Current (A)
1	0.93
2	2.11
2.5	3
3	4.12
3.5	5.97
4.0	9.59
4.5	16.29
5.0	23.33
5.5	15.66
6.0	10.17
6.5	7.53
7	6.01
8	4.34
9	3.44
10	2.87

RESULT: Thus, the frequency response of series resonant circuits is analysed

VIVA QUESTIONS:

1. If frequency is 50 Hz, what is the angular frequency?
2. If time period is 1/50 sec, what is the frequency?
3. If $I=200\sin 100\pi t$, at which time it will have the value of 100A?
4. What is the average value of a square wave of peak value 200V?
5. What is the relation between the max value and the average value of the square wave?
6. What is the form factor?
7. What is the form factor for a sine wave?
8. What is the impedance for a series resonance circuit?
9. What is the condition for resonance in a series RLC ckt?
10. What is the quality factor?

Frequency Response of Parallel Resonance

AIM: To construct a Parallel Resonance circuit and study its frequency response.

TOOLS REQUIRED: NI Multisim version 14

THEORY:

Parallel RLC Circuit

A parallel resonant circuit consists of a resistor, a capacitor, and an inductor in parallel, typically driven by a current source. At some frequency the capacitive and inductive reactances will be of the same magnitude, and as they are 180 degrees in opposition, they effectively nullify each other. This leaves the circuit purely resistive, the source “seeing” only the resistive element. At any lower or higher frequency the inductive or capacitive reactance will shunt the resistance. The result is a maximum impedance magnitude at resonance, and thus, a minimum current voltage.

Admittance:

$$Y = \frac{I}{V} = \frac{1}{R} + j\omega C + \frac{1}{j\omega L} = \frac{1}{R} + j\left(\omega C - \frac{1}{\omega L}\right)$$

Resonant frequency:

$$\omega_0 = \frac{1}{\sqrt{LC}} \quad (\text{rad/s})$$

$$f_0 = \frac{1}{2\pi\sqrt{LC}} \quad (\text{Hz})$$

Frequency response of voltage amplitude:

$$|V| = \frac{I_m}{\sqrt{(1/R)^2 + (\omega C - 1/\omega L)^2}}$$

Half-power frequencies (ω_1, ω_2):

$$\omega_1 = -\frac{1}{2RC} + \sqrt{\left(\frac{1}{2RC}\right)^2 + \frac{1}{LC}}$$

$$\omega_2 = \frac{1}{2RC} + \sqrt{\left(\frac{1}{2RC}\right)^2 + \frac{1}{LC}}$$

$$\omega_0 = \sqrt{\omega_1\omega_2}$$

Half-power bandwidth (B):

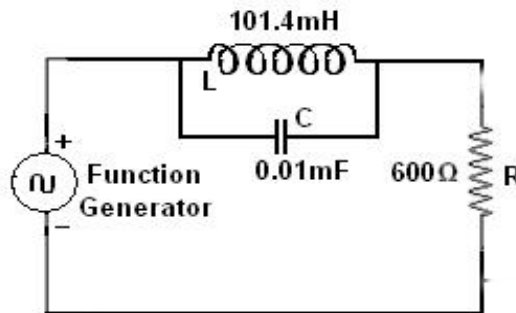
$$B = \omega_2 - \omega_1 = \frac{1}{RC}$$

Quality factor (Q):

$Q = 2 \pi$ (Peak energy stored)/(Energy dissipated in one period at resonance)

$$Q = \frac{\omega_0}{B} = \omega_0.$$

CIRCUIT DIAGRAM:



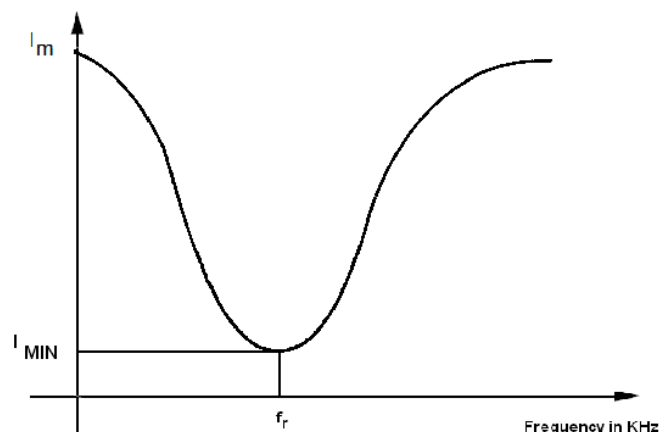
THEORETICAL OBSERVATIONS:

The student must calculate the resonant frequency (f_r), Bandwidth (BW), Half Power Frequencies (f_1 and f_2) and Quality factor (Q) using the above given equation with the values of R, L and C prior to the lab.

PROCEDURE:

1. Simulate the above shown circuit in Multisim.
2. The input is given in the form of sine wave by function generator.
3. The amplitude of the response across the resistor is noted for various frequency ranges.
4. On the graph, Current vs Frequency, note down the minimum current. Calculate 70.7% of this current and draw a horizontal line corresponding to this value on the graph. Note down the values at which this horizontal line intersects the curve (f_1 and f_2).

MODEL GRAPHS:



OBSERVATIONS:

Frequency (Hz)	Current (A)
1	15.80
1.5	11.68
2	8.65
2.5	6.38
3.0	4.61
3.5	3.16
4.0	1.94
4.5	0.87
5.0	0.76
5.5	0.96
6.0	1.73
6.5	2.46
7.0	3.14
7.5	3.78
8.0	4.38
8.5	4.96
9.0	5.51
9.5	6.03
10	6.00

RESULT: Thus, the frequency response of parallel resonant circuits is analysed

VIVA QUESTIONS:

1. Define resonant circuit.
2. When the circuit is said to be in resonance?
3. What is resonant frequency ?
4. Define bandwidth.
5. Define selectivity.
6. Define quality factor.
7. Define half power frequencies ?
8. Write down the formula for inductive reactance and capacitive reactance?
9. Define phasor.
10. Define phase angle.

5. Determination of Self, Mutual Inductances and Coefficient of Coupling using MULTISIM

Determination of Self Inductance

AIM: To construct a circuit and determine it's Self inductance.

TOOLS REQUIRED: NI Multisim version 14

THEORY:

Phasor Diagram for Complex Impedances

From the figure on the right, the resultant impedance is

$$Z = \sqrt{(R^2 + X_L^2)}$$

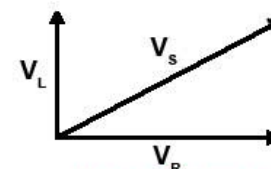
where, $X_L = 2\pi f L$

Assuming there is a phase angle, Φ , between Z and R then,

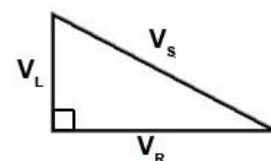
$$\tan \phi = \frac{X_L}{R} = \frac{2\pi f L}{R}$$

$$\sin \phi = \frac{2\pi f L}{Z}$$

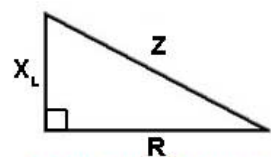
$$\text{Power Factor} = \cos \phi = \frac{R}{Z}$$



Phasor Diagram for LR Circuit



Phasors Re-drawn as a Right-angle Triangle



Impedance Triangle

Inductance is defined as the magnetic induction of a voltage in a current carrying wire when the current in a wire changes. This can occur in the same wire and additionally in another wire.

Self-inductance can be defined as the phenomenon in which a change in electric current in a circuit produces an induced electro-motive-force in the same circuit.

When current passes along a wire, and especially when it passes through a coil or inductor, a magnetic field is induced. This extends outwards from the wire or inductor and could couple with other circuits. However it also couples with the circuit from which it is set up.

The magnetic field can be envisaged as concentric loops of magnetic flux that surround the wire, and larger ones that join up with others from other loops of the coil enabling self-coupling within the coil.

When the current in the coil changes, this causes a voltage to be induced the different loops of the coil - the result of self-inductance.

When current passes along a wire, and especially when it passes through a coil or inductor, a magnetic field is induced. This extends outwards from the wire or inductor and could couple with other circuits. However it also couples with the circuit from which it is set up.

The magnetic field can be envisaged as concentric loops of magnetic flux that surround the wire, and larger ones that join up with others from other loops of the coil enabling self-coupling within the coil.

When the current in the coil changes, this causes a voltage to be induced the different loops of the coil - the result of self-inductance.

In terms of quantifying the effect of the inductance, the basic formula below quantifies the effect.

$$V_L = -N \frac{d\phi}{dt}$$

Where:

V_L = induced voltage in volts

N = number of turns in the coil

$d\phi/dt$ = rate of change of magnetic flux in webers / second

The induced voltage in an inductor may also be expressed in terms of the inductance (in henries) and the rate of change of current.

$$V_L = -L \frac{di}{dt}$$

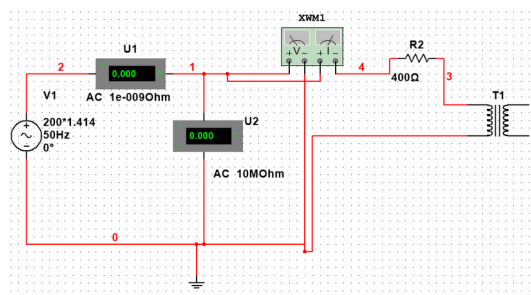
It can be seen from the formula that the voltage induced by a change in current is in the opposite sense to the change in current. Any current induced in a conductor will oppose the change in current that caused the change in flux.

This is effectively what Lenz's law states because an induced current has a direction such that its magnetic field opposes the change in magnetic field that induced the current.

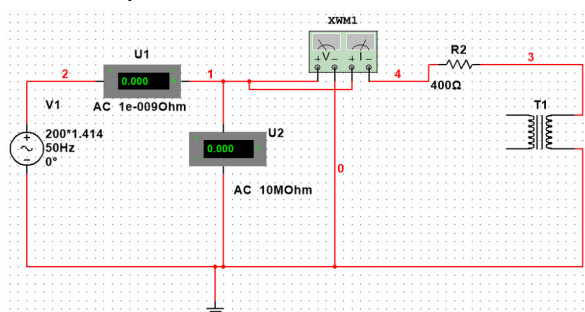
Lenz's law states that an induced electromotive force, EMF gives rise to a current whose magnetic field opposes the original change in magnetic flux.

CIRCUIT DIAGRAM:

Self Inductance for Primary Inductor



Self Inductance for Secondary Inductor



PROCEDURE:

1. Construct the circuit as shown above in Multisim.

Note: The Ammeters and Voltmeters should be configured in AC mode and T1 is a coupled inductor found in Basic > Transformer > Coupled_Inductors of the Place Component section

2. The input is given in the form of AC Voltage
3. Measure the Voltage, Current, Power and Power Factor from the indicators to calculate the value of inductance from the above given formulas.

Note: Inductance values for both Primary and Secondary coils must be calculated using the same observation tables below.

OBSERVATIONS:

Voltage (V)	Current (I)	Power (W)	$Z=V/I$	$\text{Cos}\Phi=W/VI$	Phase angle Φ $=\text{cos}^{-1}(W/VI)$	$L=Z\text{sin}\Phi/2\pi f$

RESULT: Thus, the self inductance of the given circuit is obtained successfully.

Determination of Mutual Inductance and Coefficient of Coupling

AIM: To construct a circuit and determine its Mutual inductance and Coefficient of Coupling.

TOOLS REQUIRED: NI Multisim version 14

THEORY:

When an emf is produced in a coil because of the change in current in a coupled coil, the effect is called mutual inductance. The emf is described by Faraday's law and its direction is always opposed to the change in the magnetic field produced in it by the coupled coil (Lenz's law). The induced emf in coil 1 is due to self inductance L .

The induced emf in coil #2 caused by the change in current I_1 can be expressed as

$$Emf_2 = -N_2 A \frac{\Delta B}{\Delta t} = -M \frac{\Delta I_1}{\Delta t}$$

The mutual inductance M can be defined as the proportionality between the emf generated in coil 2 to the change in current in coil 1 which produced it.

The most common application of mutual inductance is the transformer.

In case of Aiding condition,

$$L_{eff} = L_1 + L_2 + 2M$$

and for Opposing condition,

$$L_{eff} = L_1 + L_2 - 2M$$

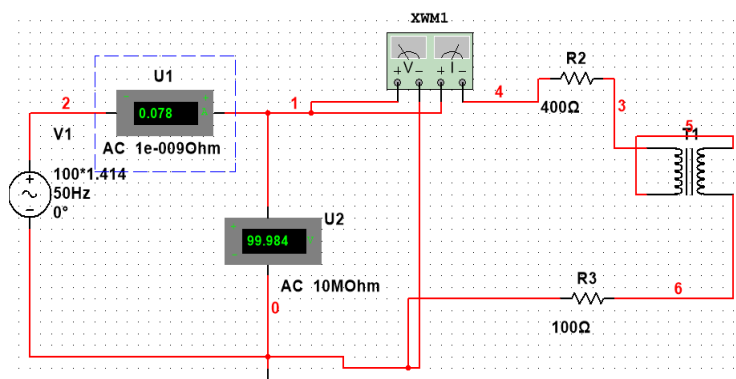
where M is the Mutual Inductance

The Coupling of Coefficient is given by

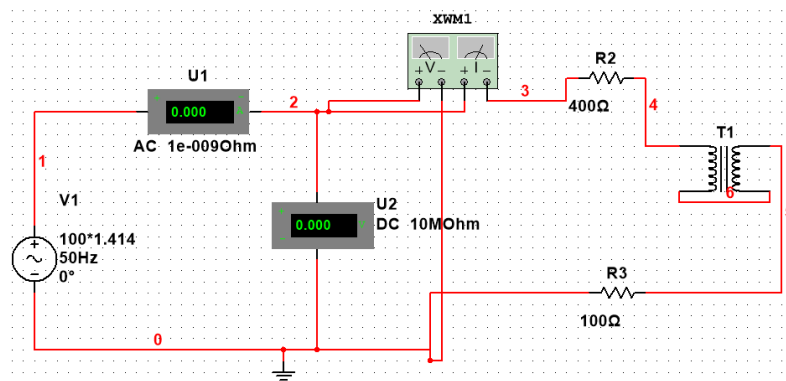
$$k = \frac{M}{\sqrt{L_1 L_2}}$$

CIRCUIT DIAGRAM:

Mutual Inductance for Aiding condition



Mutual Inductance for Opposing condition



PROCEDURE:

1. Construct the circuit as shown above in Multisim.

Note: The Ammeters and Voltmeters should be configured in AC mode and T1 is a coupled inductor found in Basic > Transformer > Coupled_Inductors of the Place Component section.

2. The input is given in the form of AC Voltage
3. Measure the Voltage, Current, Power and Power Factor from the indicators to calculate the value of effective inductance (L_{eff}) and obtain Mutual Inductance (M) and Coefficient of Coupling (k) from the above given formulas.

Note: Inductance values for both Aiding and Opposing must be calculated using the same observation tables below.

OBSERVATIONS:

Voltage (V)	Current (I)	Power (W)	$Z=V/I$	$\text{Cos } \Phi = W/VI$	Phase angle Φ $=\text{cos}^{-1}(W/VI)$	$L_{eff}=Z\text{sin}\Phi/2\pi f$

RESULT: Thus, the mutual inductance and coefficient of coupling of the given circuit is obtained successfully.

PART B
HARDWARE EXPERIMENTS

1. PN Junction Diode Characteristics

AIM: To observe and draw the Forward and Reverse bias V-I Characteristics of a P-N Junction diode..

COMPONENTS REQUIRED:

S.No	Name	Type / Range	Quantity
1	P-N Diode	1N4007	1
2	Regulated Power supply	(0-15V)	1
3	Resistor	1K Ω	1
4	Ammeter	(0-200mA, 0-200 μ A)	1
5	Voltmeter	(0-20V)	1
6	Breadboard		
7	Connecting wires		

THEORY:

PN Junction Diode

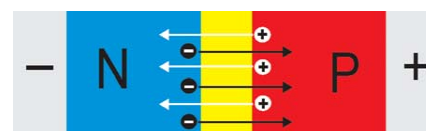
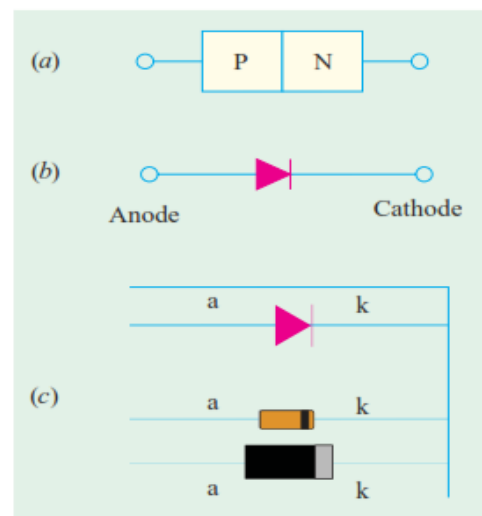
Construction

It is a two-terminal device consisting of a P-N junction formed either in Ge or Si crystal. Its circuit symbol is shown on the right. The P-and N-type regions are referred to as anode and cathode respectively. The arrowhead indicates the conventional direction of current flow when forward-biased. It is the same direction in which hole flow takes place.

Commercially available diodes usually have some means to indicate which lead is P and which lead is N. Standard notation consists of type numbers preceded by 'IN' such as IN 240 and IN 1250. Here, 240 and 1250 correspond to colour bands.

Working

A P-N junction diode is one-way device offering low resistance when forward-biased and behaving almost as an insulator when reverse-biased. Hence, such diodes are mostly used as rectifiers i.e. for converting alternating current into direct current.

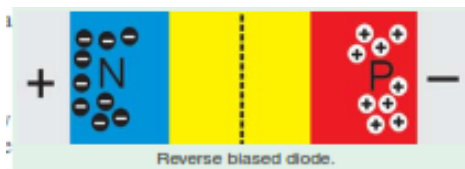


Forward Characteristic

When the diode is forward-biased and the applied voltage is increased from zero, hardly any current flows through the device in the beginning. It is so because the external voltage is being opposed by the internal barrier voltage V whose value is 0.7 V for Si and 0.3 V for Ge.

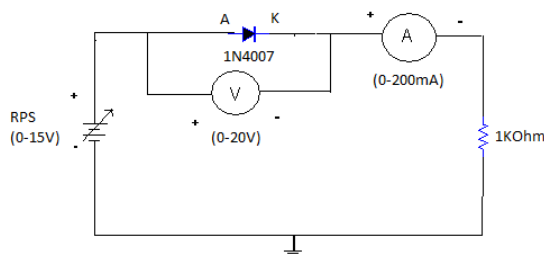
Reverse Characteristic

When the diode is reverse-biased, majority carriers are blocked and only a small current (due to minority carriers) flows through the diode. As the reverse voltage is increased from zero, the reverse current very quickly reaches its maximum or saturation value I which is also known as leakage current. It is of the order of nanoamperes (nA) for Si and microamperes (μA) for Ge.

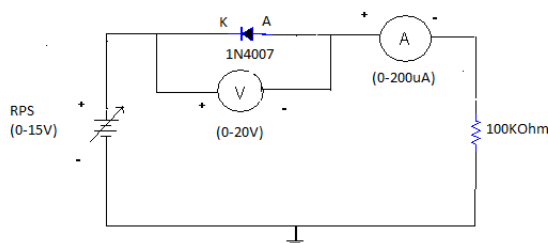


CIRCUIT DIAGRAM:

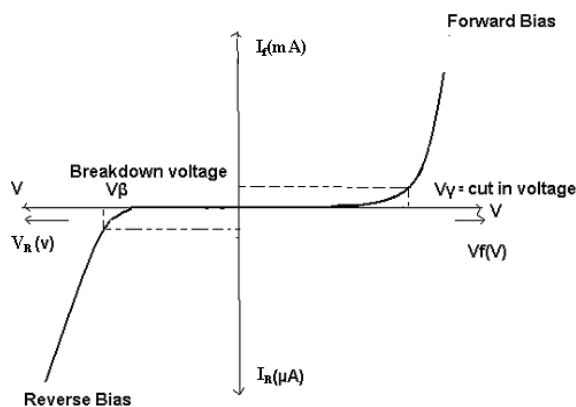
Forward Bias



Reverse Bias



Model Graph



PROCEDURE:

Forward Bias

1. Connections are made as per the circuit diagram.
2. For forward bias, the RPS +ve is connected to the anode of the diode and RPS –ve is connected to the cathode of the diode
3. Switch on the power supply and increase the input voltage (supply voltage) in steps of 0.1V
4. Note down the corresponding current flowing through the diode and voltage across the diode for each and every step of the input voltage.
5. The reading of voltage and current are tabulated.
6. Graph is plotted between voltage (V_f) on X-axis and current (I_f) on Y-axis.

Reverse Bias

1. Connections are made as per the circuit diagram
2. For reverse bias, the RPS +ve is connected to the cathode of the diode and RPS –ve is connected to the anode of the diode.
3. Switch on the power supply and increase the input voltage (supply voltage) in steps of 1V.
4. Note down the corresponding current flowing through the diode voltage across the diode for each and every step of the input voltage.
5. The readings of voltage and current are tabulated
6. Graph is plotted between voltage (V_R) on X-axis and current (I_R) on Y-axis.

Precautions:

1. All the connections should be correct.
2. Parallax error should be avoided while taking the readings from the Analog meters.

OBSERVATIONS:

Forward Bias

S.No.	Forward Voltage(V)	Forward Current(mA)
1	0	0
2	0.15	0
3	0.47	0.1
4	0.52	0.3
5	0.56	0.7
6	0.59	1.5

7	0.62	2
8	0.65	5.1
9	0.68	11.6
10	0.7	17.9

Reverse Bias

S.No.	Reverse Voltage(V)	Reverse Current(mA)
1	0	0
2	0.1	0.1
3	1.04	1
4	2.2	2
5	5.22	5
6	7.3	7
7	9.4	9
8	11.4	11
9	13.5	13
10	15.5	14.9

Calculations

Calculation of Static and Dynamic Resistance for a given diode.

In forward bias condition:

Static Resistance , $R_s = V_f/I_f = 130 \text{ ohm}$

Dynamic Resistance, $R_D = \Delta V_f / \Delta I_f = 9.6 \text{ ohm}$

In Reverse bias condition:

Static Resistance , $R_s = V_R/I_R = 1.1K \text{ ohm}$

Dynamic Resistance, $R_D = \Delta V_R / \Delta I_R = 1K \text{ ohm}$

RESULT: Forward and Reverse Bias characteristics for a p-n diode are observed and cut-in voltage in forward bias condition is obtained. Cut-in voltage = 0.47 V

VIVA QUESTIONS:

1. Define depletion region of a diode.
2. What is meant by transition & space charge capacitance of a diode?
3. Is the V-I relationship of a diode Linear or Exponential?
4. Define cut-in voltage of a diode and specify the values for Si and Ge diodes.
5. What are the applications of a p-n diode?
6. Draw the ideal characteristics of P-N junction diode.
7. What is the diode equation?
8. What is PIV?
9. What is the break down voltage?
10. What is the effect of temperature on PN junction diodes?

2a. Zener Diode Characteristics

AIM:

1. To plot volt-ampere characteristics of a given zener diode.
2. To find the zener breakdown voltage, forward bias resistance and reverse bias resistances.

COMPONENTS REQUIRED:

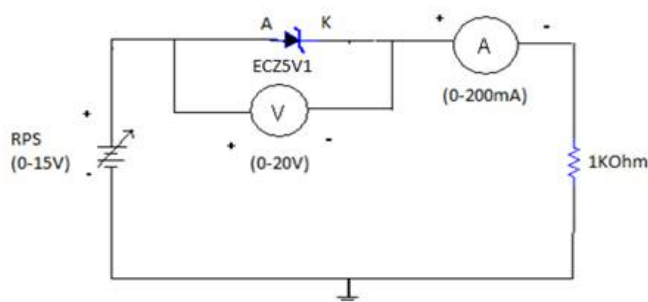
S.No	Name	Type / Range	Quantity
1	Zener Diode	ECZ5V1	1
2	Regulated Power supply	(0-15V)	1
3	Resistor	1K Ω ,100 Ω	1
4	Ammeter	(0-200mA)	1
5	Voltmeter	(0-20V)	1
6	Breadboard		
7	Connecting wires		

THEORY:

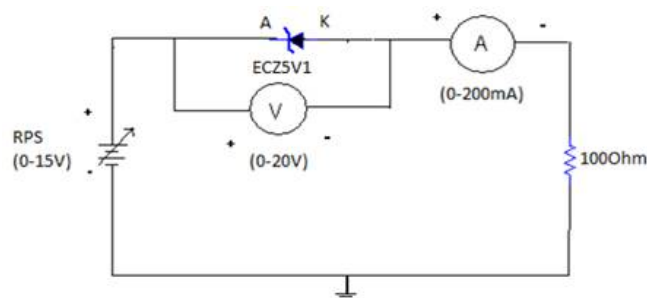
A zener diode is heavily doped p-n junction diode, specially made to operate in the break down region. A p-n junction diode normally does not conduct when reverse biased. But if the reverse bias is increased, at a particular voltage it starts conducting heavily. This voltage is called Break down Voltage. High current through the diode can permanently damage the device. To avoid high current, we connect a resistor in series with zener diode. Once the diode starts conducting it maintains almost constant voltage across the terminals whatever may be the current through it, i.e., it has very low dynamic resistance. It is used in voltage regulators.

CIRCUIT DIAGRAM:

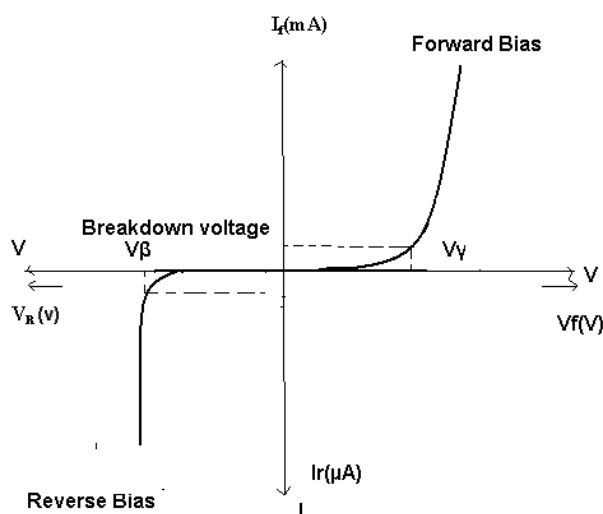
Forward Bias



Reverse Bias



Model graph



PROCEDURE:

Forward Bias

1. Connections are made as per the circuit diagram.
2. For forward bias, the RPS +ve is connected to the anode of the diode and RPS -ve is connected to the cathode of the diode,
3. Switch on the power supply and increase the input voltage (supply voltage) in steps. Note down the corresponding forward current flowing through the diode and forward voltage across the diode for each step of the input voltage.
4. The reading of forward voltage and current are tabulated.
5. Graph is plotted between forward voltage and current.

Reverse Bias

1. Connections are made as per the circuit diagram.
2. For reverse bias, the RPS +ve is connected to the cathode of the diode and RPS -ve is connected to the anode of the diode.
3. Switch on the power supply and increase the input voltage (supply voltage) in steps. Note down the corresponding reverse current flowing through the diode and

reverse voltage across the diode for each step of the input voltage.

4. The readings of reverse voltage and current are tabulated
5. Graph is plotted between reverse voltage and current.

OBSERVATIONS:

Forward Bias

S.No.	Forward Voltage(V)	Forward Current(mA)
1	0	0
2	0.51	0
3	0.62	0.1
4	0.66	0.3
5	0.67	0.5
6	0.69	0.7
7	0.7	1.3
8	0.73	4
9	0.75	8.7
10	0.76	16

Reverse Bias

S.No.	Reverse Voltage(V)	Reverse Current(mA)
1	0	0
2	4.85	0.1
3	5.1	0.3
4	5.35	1
5	5.45	2
6	5.55	5
7	5.6	11
8	5.62	13

9	5.66	22
10	5.68	34

RESULT: Forward and Reverse Bias characteristics for a zener diode are observed and the breakdown voltage of zener diode is obtained. Breakdown voltage = 4.85 V

2b. Zener Diode as Voltage Regulator

AIM: To observe and draw the regulator characteristics of a zener diode at supply and load side.

COMPONENTS REQUIRED:

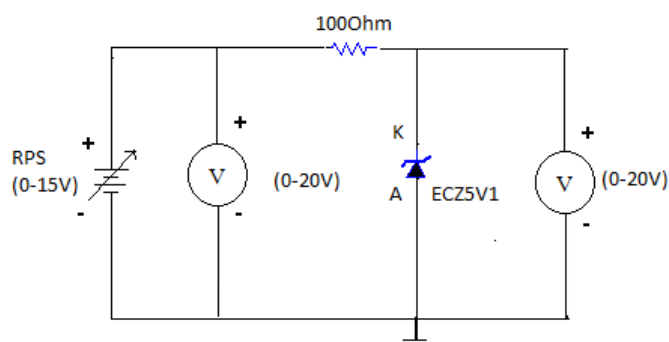
S.No	Name	Type / Range	Quantity
1	Zener Diode	ECZ5V1	1
2	Regulated Power supply	(0-15V)	1
3	Resistor	100Ω	1
4	Ammeter	(0-200mA)	1
5	Voltmeter	(0-20V)	1
6	Breadboard		
7	Connecting wires		

THEORY:

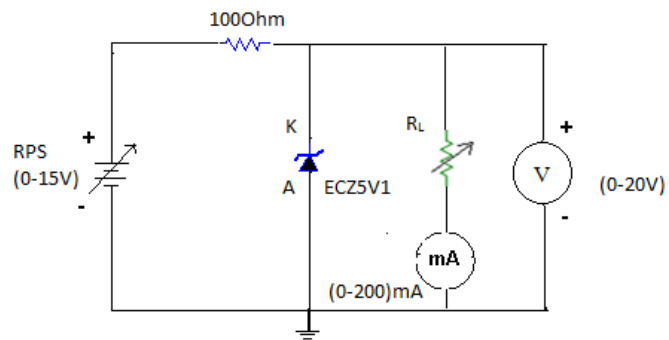
A zener diode is heavily doped p-n junction diode, specially made to operate in the break down region. A p-n junction diode normally does not conduct when reverse biased. But if the reverse bias is increased, at a particular voltage it starts conducting heavily. This voltage is called Break down Voltage. High current through the diode can permanently damage the device. To avoid high current, we connect a resistor in series with zener diode. Once the diode starts conducting it maintains almost constant voltage across the terminals whatever may be the current through it, i.e., it has very low dynamic resistance. It is used in voltage regulators.

CIRCUIT DIAGRAM:

Supply Side

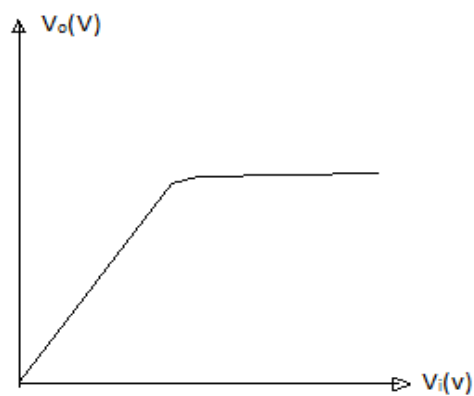


Load Side

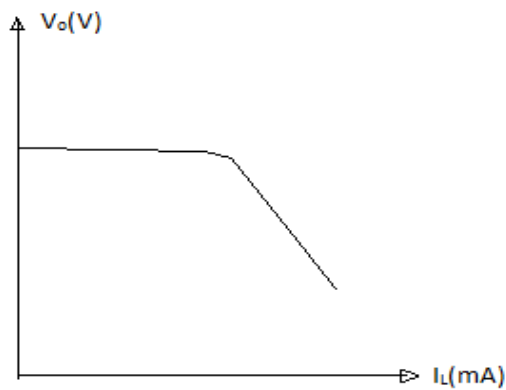


Model Graphs

Supply Side



Load Side



PROCEDURE:

Supply Side

1. Connections are made as per the circuit diagram.
2. The Regulated power supply voltage is increased in steps.
3. For different input voltages (V_i) corresponding output voltages (V_o) are observed and then noted in the tabular form.

4. A graph is plotted between input voltage (V_i) and output voltage (V_o).

Load Side

1. Connection are made as per the circuit diagram
2. The load is placed in full load condition and the output voltage (V_o), load current (I_L) are measured.
3. The above step is repeated by decreasing the value of the load in steps.
4. All the readings are tabulated and a graph is plotted between load current (I_L) and the output voltage (V_o).

OBSERVATIONS:

Supply Side

S.No.	V_i (V)	V_o (V)
1	0	0
2	2	2.01
3	4	4.2
4	6	5.17
5	8	5.18
6	10	5.2
7	12	5.2
8	13	5.2
9	15	5.2

Load Side

S.No.	I_L (mA)	V_o (V)
1	0	0
2	15	5.3
3	30	5.3
4	45	5.3
5	55	5.29
6	90	5.29

7	105	4.8
8	120	3.3
9	130	2.6

RESULT: Regulator characteristics of zener diode are obtained and graphs are plotted for load and supply side.

VIVA QUESTIONS:

1. What type of temp coefficient does the zener diode have?
2. If the impurity concentration is increased, how does the depletion width get
3. effected?
4. Does the dynamic impedance of a zener diode vary?
5. Explain briefly about avalanche and zener breakdowns.
6. Draw the zener equivalent circuit.
7. Differentiate between line regulation & load regulation.
8. Which region zener diode can be used as a regulator?
9. How the breakdown voltage of a particular diode can be controlled?
10. What type of temperature coefficient does the Avalanche breakdown has?
11. By what type of charge carriers the current flows in zener and avalanche breakdown diodes?

3. Input and output Characteristics of Transistor in CB Configuration

AIM:

1. To observe and draw the input and output characteristics of a transistor connected in common base configuration.
2. To find α of the given transistor and also its input and output Resistances.

COMPONENTS REQUIRED:

S.No	Name	Type / Range	Quantity
1	NPN-Transistor	BC107	1
2	Regulated Power supply	(0-15V)	2
3	Resistor	1K Ω	2
4	Ammeter	(0-200mA)	1
5	Voltmeter	(0-20V)	1
6	Breadboard		
7	Connecting wires		

THEORY:

A transistor is a three terminal active device. The terminals are emitter, base, collector. In CB configuration, the base is common to both input (emitter) and output (collector). For normal operation, the E-B junction is forward biased and C-B junction is reverse biased.

In CB configuration, I_E is +ve, I_C is -ve and I_B is -ve.

$$\text{So, } V_{EB} = f_1(V_{CB}, I_E) \text{ and } I_C = f_2(V_{CB}, I_E)$$

With an increasing the reverse collector voltage, the space-charge width at the output junction increases and the effective base width decreases. This phenomenon is known as “Early effect”. Then, there will be less chance for recombination within the base region. With increase of charge gradient with in the base region, the current of minority carriers injected across the emitter junction increases. The current amplification factor of CB configuration is given by,

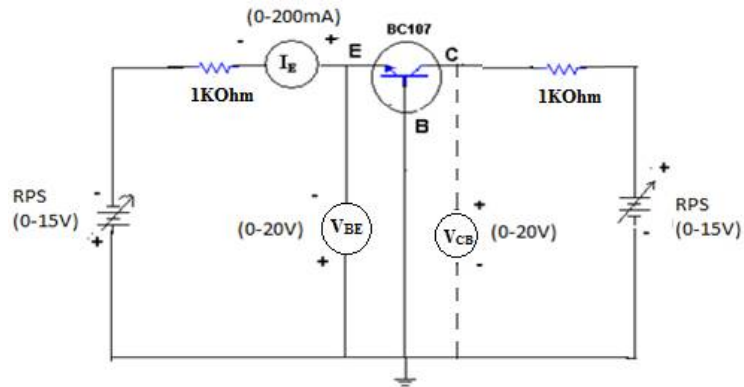
$$\alpha = \Delta I_C / \Delta I_E$$

$$\text{Input Resistance, } r_i = \Delta V_{BE} / \Delta I_E \quad \text{at Constant } V_{CB}$$

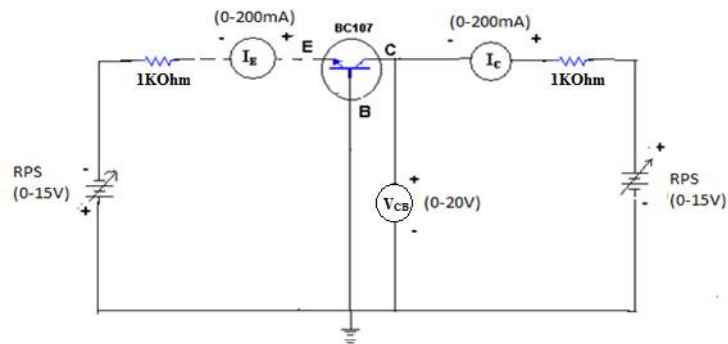
$$\text{Output Resistance, } r_o = \Delta V_{CB} / \Delta I_C \quad \text{at Constant } I_E$$

CIRCUIT DIAGRAM:

Input Characteristics

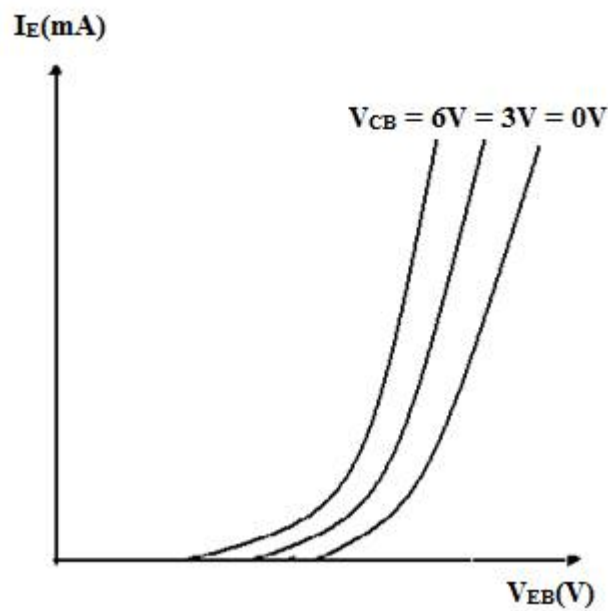


Output Characteristics

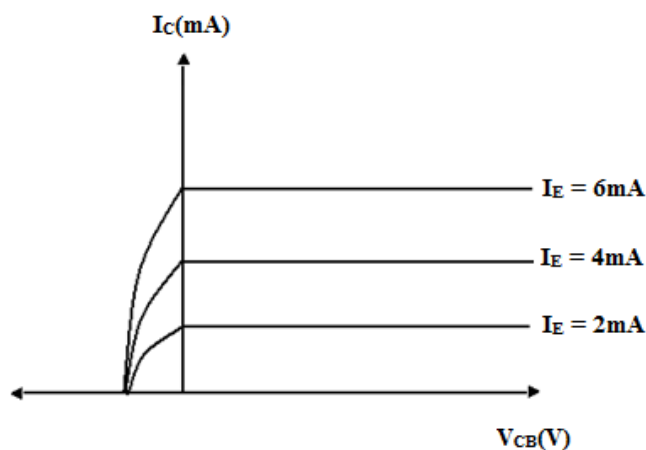


Model Graphs

Input Characteristics



Output Characteristics



PROCEDURE:

Input Characteristics

1. Connections are made as per the circuit diagram.
2. For plotting the input characteristics, the output voltage V_{CB} is kept constant at 0V and for different values of V_{BE} note down the values of I_E .
3. Repeat the above step keeping V_{CB} at 5V, 10V. All the readings are tabulated.
4. A graph is drawn between V_{BE} and I_E for constant V_{CB} .

Output Characteristics:

1. Connections are made as per the circuit diagram.
2. For plotting the output characteristics, the input I_E is kept constant at 1mA and for different values of V_{CB} , note down the values of I_C .
3. Repeat the above step for the values of I_E at 2 mA, and 3 mA, all the readings are tabulated.
4. A graph is drawn between V_{CB} and I_C for constant I_E

OBSERVATIONS:

Input Characteristics:

S. No.	$V_{CB}=0V$		$V_{CB}=3V$		$V_{CB}=6V$	
	$V_{EB}(V)$	$I_E(mA)$	$V_{EB}(V)$	$I_E(mA)$	$V_{EB}(V)$	$I_E(mA)$
1	0.02	0	0.02	0	0.02	0
2	0.59	0.1	0.58	0.1	0.6	0.1
3	0.69	$0.5I_{Cmax}$	0.63	0.5	0.63	0.5
4	0.72	1	0.65	1	0.64	1

5	0.74	1.5	0.68	1.5	0.65	1.5
6	0.75	2	0.72	2	0.66	2
7	0.76	2.5	0.74	2.5	0.66	2.5
8	0.77	3	0.75	3	0.67	3
9	0.78	3.5	0.77	3.5	0.68	3.5
10	0.78	4	0.77	4	0.7	4
11	0.79	4.5	0.78	4.5	0.73	4.5
12	0.8	5.1	0.79	5	0.77	5
13	0.8	5.4	0.79	5.4	0.78	5.4

Output Characteristics:

S. No.	IE=2mA		IE=4mA		IE=6mA	
	VCB(V)	IC(mA)	VCB(V)	IC(mA)	VCB(V)	IC(mA)
1	-0.7	0	-0.72	0.9	-0.75	0.8
2	-0.67	0.4	-0.71	1.3	-0.74	1.4
3	-0.64	0.7	-0.7	1.5	-0.74	2
4	-0.6	0.9	-0.68	2.2	-0.73	2.8
5	-0.53	0.9	-0.67	2.4	-0.72	3.1
6	-0.41	1	-0.66	2.6	-0.71	4
7	-0.38	1	-0.61	2.9	-0.68	5.1
8	1.02	1	0.23	3	-0.66	5.5
9	2.02	1	1.12	3	0.39	5.9
10	3.33	1	4.76	3	2.48	5.9
11	5.82	1	6.64	3	4.84	5.9
12	8.41	1	8.73	3	6.4	5.9
13	9.4	1	11.41	3	8.72	5.9
14	13.88	1	12.1	3	9.52	5.9

RESULT: The input and output characteristics of the transistor connected in common base configuration are obtained.

VIVA QUESTIONS:

1. What is the range of α for the transistor?
2. Draw the input and output characteristics of the transistor in CB configuration.
3. Identify various regions in output characteristics.
4. What is the relation between α and β ?
5. What are the applications of CB configuration?
6. What are the input and output impedances of CB configuration?
7. Define α (alpha).
8. What is early effect?
9. Draw Circuit diagram of CB configuration for PNP transistor.
10. What is the power gain of CB configuration?

4. Input and output Characteristics of Transistor in CE Configuration

AIM: To obtain the input and output characteristics of transistor connected in common emitter configuration

COMPONENTS REQUIRED:

S.No	Name	Type / Range	Quantity
1	NPN-Transistor	BC107	1
2	Regulated Power supply	(0-15V)	2
3	Resistors	10KΩ, 1.5KΩ	1
4	Ammeter	(0-200mA), (0-200μA)	1
5	Voltmeter	(0-20V)	1
6	Breadboard		
7	Connecting wires		

THEORY:

A transistor is a three terminal device. The terminals are emitter, base, collector. In common emitter configuration, input voltage is applied between base and emitter terminals and output is taken across the collector and emitter terminals. Therefore the emitter terminal is common to both input and output.

The input characteristics resemble that of a forward biased diode curve. This is expected since the Base-Emitter junction of the transistor is forward biased. As compared to CB arrangement I_B increases less rapidly with V_{BE} . Therefore input resistance of CE circuit is higher than that of CB circuit.

The output characteristics are drawn between I_C and V_{CE} at constant I_B . the collector current varies with V_{CE} upto few volts only. After this the collector current becomes almost constant, and independent of V_{CE} . The value of V_{CE} up to which the collector current changes with V_{CE} is known as Knee voltage. The transistor always operated in the region above Knee voltage, I_C is always constant and is approximately equal to I_B .

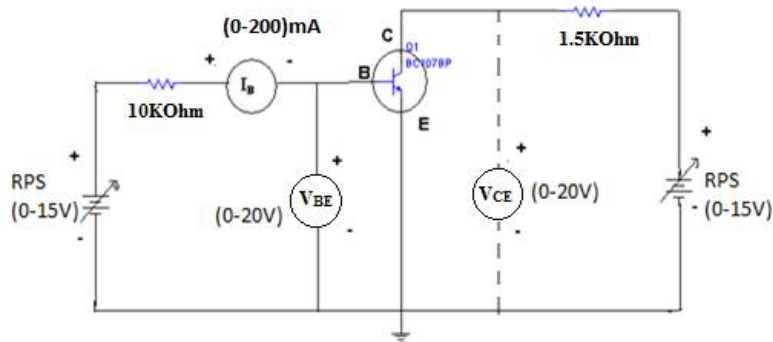
The current amplification factor of CE configuration is given by

$$\beta = \Delta I_C / \Delta I_B$$

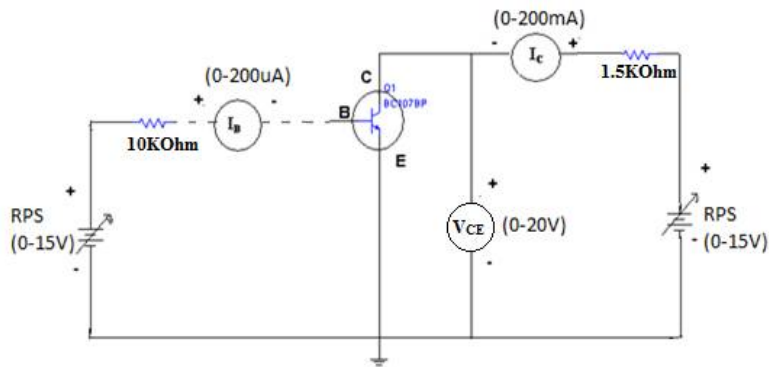
$$\begin{aligned} \text{Input Resistance, } r_i &= \Delta V_{BE} / \Delta I_B (\mu A) && \text{at Constant } V_{CE} \\ \text{Output Résistance, } r_o &= \Delta V_{CE} / \Delta I_C && \text{at Constant } I_B (\mu A) \end{aligned}$$

CIRCUIT DIAGRAM:

Input Characteristics

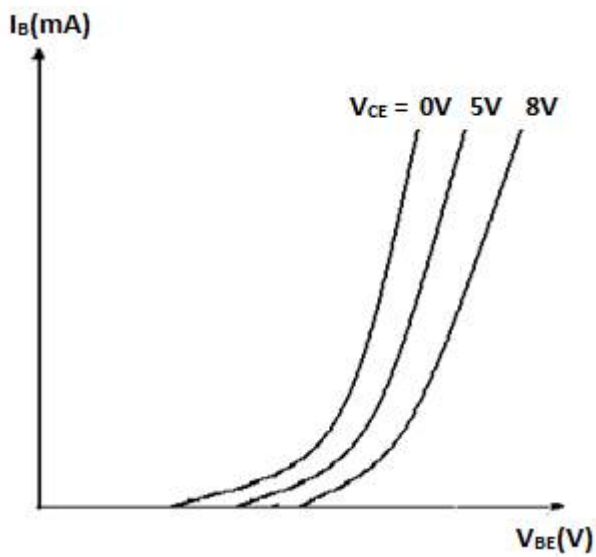


Output Characteristics

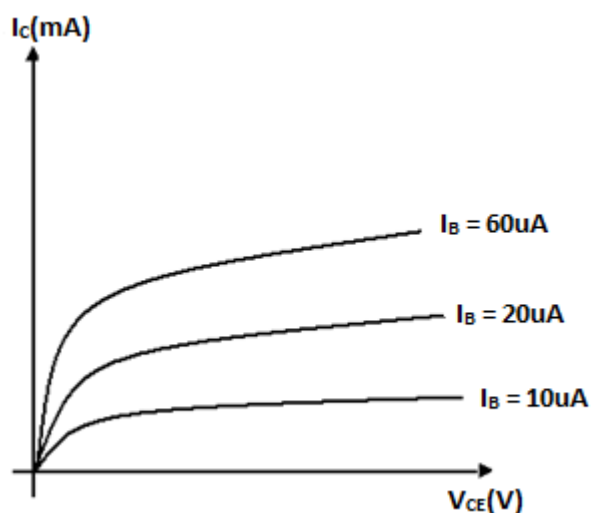


Model Graphs

Input Characteristics



Output Characteristics



PROCEDURE:

Input Characteristics

1. Connect the circuit as per the circuit diagram.
2. For plotting the input characteristics the output voltage V_{CE} is kept constant at 0V and for different values of V_{BE} . Note down the values of I_C
3. Repeat the above step by keeping V_{CE} at 5V and 8V.
4. Tabulate all the readings and plot the graph between V_{BE} and I_B for constant V_{CE}

Output Characteristics

1. Connect the circuit as per the circuit diagram
2. For plotting the output characteristics the input current I_B is kept constant at $10\mu A$ and for different values of V_{CE} note down the values of I_C
3. Repeat the above step by keeping I_B at $20\mu A$, $60\mu A$
4. Tabulate the all the readings and plot the graph between V_{CE} and I_C for constant I_B

OBSERVATIONS:

Input Characteristics

S. No.	VCE=0V		VCE=5V		VCE=8V	
	VBE(V)	IB(mA)	VBE(V)	IB(mA)	VBE(V)	IB(mA)
1	0	0	0	0	0	0
2	0.6	0.1	0.67	0.1	0.68	0.1
3	0.63	0.2	0.68	0.2	0.7	0.2

4	0.66	0.3	0.69	0.3	0.71	0.3
5	0.68	0.5	0.7	0.6	0.72	0.8
6	0.69	0.6	0.71	0.7	0.73	1
7	0.7	0.7	0.72	1.2	0.74	1.2
8	0.71	1.2	0.73	1.4	0.75	1.4
9	0.71	1.4	0.73	1.7	0.75	1.7
10	0.71	1.8	0.73	1.8	0.75	1.8

Output Characteristics

S. No.	IB=10uA		IB=20uA		IB=60uA	
	VCE(V)	IC(mA)	VCE(V)	IC(mA)	VCE(V)	IC(mA)
1	0	0	0	0	0	0
2	0.03	0.1	0.03	0.1	0.01	0.1
3	0.06	0.2	0.08	0.4	0.05	1
4	0.1	0.4	0.1	0.5	0.4	8
5	0.12	0.5	0.5	1	0.9	8
6	0.2	0.6	1	1.1	1	8.2
7	5	0.7	4	1.1	2	8.5
8	8	0.7	8	1.2	4	9
9	11	0.7	9.5	1.2	6	9.5
10	13	0.7	10	1.2	6.6	10.3
11	15	0.7	11.7	1.3	7.4	10.8

RESULT: The input and output characteristics of a transistor in common emitter configuration are obtained.

VIVA QUESTIONS:

1. What is the range of β for the transistor?
2. What are the input and output impedances of CE configuration?
3. Identify various regions in the output characteristics.
4. What is the relation between α and β ?

5. Define current gain in CE configuration.
6. Why CE configuration is preferred for amplification?
7. What is the phase relation between input and output?
8. Draw diagram of CE configuration for PNP transistor.
9. What is the power gain of CE configuration?
10. What are the applications of CE configuration?

5. Input and output Characteristics of Transistor in CC Configuration

AIM:

- To observe and draw the input and output characteristics of a transistor connected in common collector configuration.
- To find α of the given transistor and also its input and output Resistances.

COMPONENTS REQUIRED:

S.No	Name	Type / Range	Quantity
1	NPN-Transistor	BC107	1
2	Regulated Power supply	(0-15V)	2
3	Resistor	1K Ω	2
4	Ammeter	(0-200mA)	1
5	Voltmeter	(0-20V)	1
6	Breadboard		
7	Connecting wires		

THEORY:

A transistor is a three terminal active device. The terminals are emitter, base, collector. In CB configuration, the base is common to both input (emitter) and output (collector). For normal operation, the E-B junction is forward biased and C-B junction is reverse biased.

In CB configuration, I_E is +ve, I_C is -ve and I_B is -ve.

$$\text{So, } V_{EB} = f_1(V_{CB}, I_E) \text{ and } I_C = f_2(V_{CB}, I_E)$$

With an increasing the reverse collector voltage, the space-charge width at the output junction increases and the effective base width decreases. This phenomenon is known as "Early effect". Then, there will be less chance for recombination within the base region. With increase of charge gradient with in the base region, the current of minority carriers injected across the emitter junction increases. The current amplification factor of CB configuration is given by,

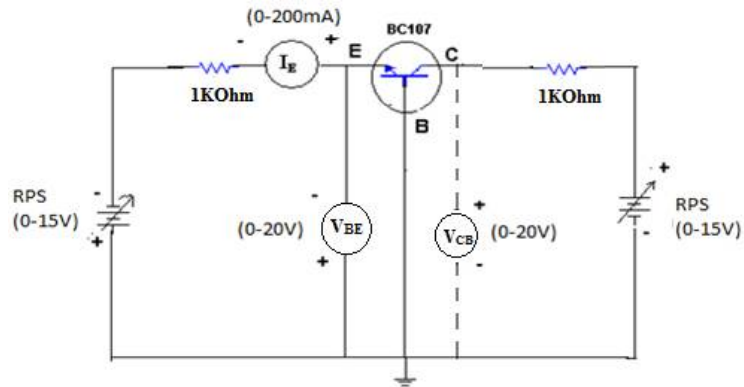
$$\alpha = \Delta I_C / \Delta I_E$$

$$\text{Input Resistance, } r_i = \Delta V_{BE} / \Delta I_E \quad \text{at Constant } V_{CB}$$

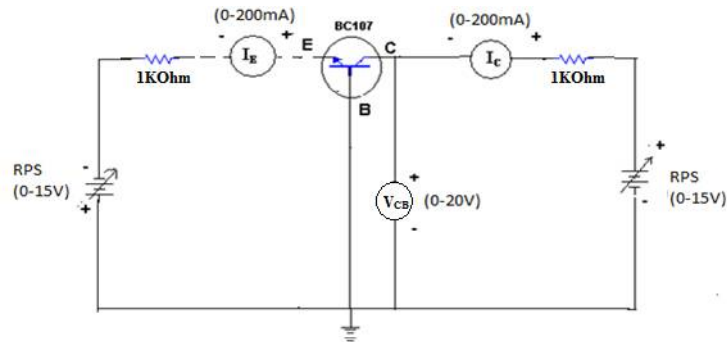
$$\text{Output Resistance, } r_o = \Delta V_{CB} / \Delta I_C \quad \text{at Constant } I_E$$

CIRCUIT DIAGRAM:

Input Characteristics

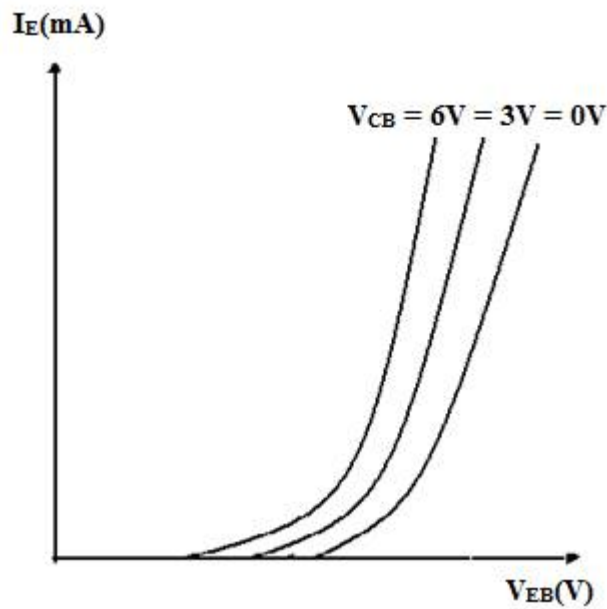


Output Characteristics

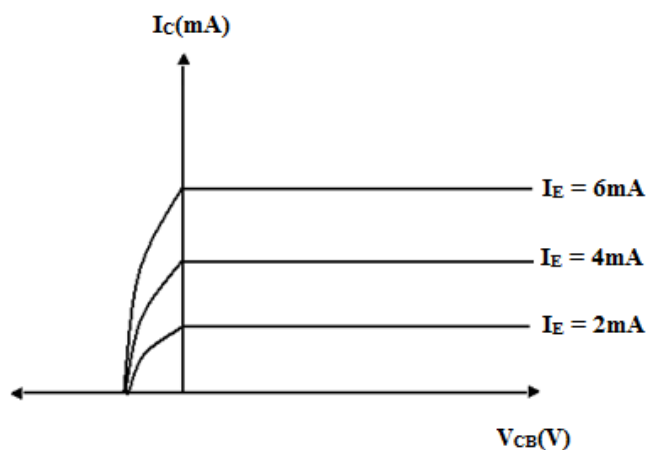


Model Graphs

Input Characteristics



Output Characteristics



PROCEDURE:

Input Characteristics

5. Connections are made as per the circuit diagram.
6. For plotting the input characteristics, the output voltage V_{CB} is kept constant at 0V and for different values of V_{BE} note down the values of I_E .
7. Repeat the above step keeping V_{CB} at 5V, 10V. All the readings are tabulated.
8. A graph is drawn between V_{BE} and I_E for constant V_{CB} .

Output Characteristics:

5. Connections are made as per the circuit diagram.
6. For plotting the output characteristics, the input I_E is kept constant at 1mA and for different values of V_{CB} , note down the values of I_C .
7. Repeat the above step for the values of I_E at 2 mA, and 3 mA, all the readings are tabulated.
8. A graph is drawn between V_{CB} and I_C for constant I_E

OBSERVATIONS:

Input Characteristics:

S. No.	$V_{CB}=0V$		$V_{CB}=3V$		$V_{CB}=6V$	
	$V_{EB}(V)$	$I_E(mA)$	$V_{EB}(V)$	$I_E(mA)$	$V_{EB}(V)$	$I_E(mA)$
1	0.02	0	0.02	0	0.02	0
2	0.59	0.1	0.58	0.1	0.6	0.1
3	0.69	$0.5I_{Cmax}$	0.63	0.5	0.63	0.5
4	0.72	1	0.65	1	0.64	1

5	0.74	1.5	0.68	1.5	0.65	1.5
6	0.75	2	0.72	2	0.66	2
7	0.76	2.5	0.74	2.5	0.66	2.5
8	0.77	3	0.75	3	0.67	3
9	0.78	3.5	0.77	3.5	0.68	3.5
10	0.78	4	0.77	4	0.7	4
11	0.79	4.5	0.78	4.5	0.73	4.5
12	0.8	5.1	0.79	5	0.77	5
13	0.8	5.4	0.79	5.4	0.78	5.4

Output Characteristics:

S. No.	IB=10uA		IB=20uA		IB=60uA	
	VCE(V)	IC(mA)	VCE(V)	IC(mA)	VCE(V)	IC(mA)
1	0	0	0	0	0	0
2	0.03	0.1	0.03	0.1	0.01	0.1
3	0.06	0.2	0.08	0.4	0.05	1
4	0.1	0.4	0.1	0.5	0.4	8
5	0.12	0.5	0.5	1	0.9	8
6	0.2	0.6	1	1.1	1	8.2
7	5	0.7	4	1.1	2	8.5
8	8	0.7	8	1.2	4	9
9	11	0.7	9.5	1.2	6	9.5
10	13	0.7	10	1.2	6.6	10.3
11	15	0.7	11.7	1.3	7.4	10.8

RESULT: The input and output characteristics of the transistor connected in common base configuration are obtained.

VIVA QUESTIONS:

11. What is the range of α for the transistor?
12. Draw the input and output characteristics of the transistor in CC configuration.
13. Identify various regions in output characteristics.
14. What is the relation between α and β ?
15. What are the applications of CC configuration?
16. What are the input and output impedances of CC configuration?
17. Define α (alpha).
18. What is early effect?
19. Draw Circuit diagram of CC configuration for PNP transistor.
20. What is the power gain of CC configuration?

6. Half Wave Rectifier with and without Filter

AIM: To examine the input and output waveforms of half wave Rectifier and also calculate its % load regulation and ripple factor.

1. With Filter
2. Without Filter

COMPONENTS REQUIRED:

S.No	Name	Type / Range	Quantity
1	PN Diode	1N4007	1
2	AC Power supply	12V	1
3	Variable Resistor	(0-10K Ω)	1
4	Capacitor	470 μ F	1
5	Multimeter		
6	Breadboard		
7	Connecting wires		

THEORY:

In Half Wave Rectification, When AC supply is applied at the input, only Positive Half Cycle appears across the load whereas, the negative Half Cycle is suppressed. How this can be explained as follows:

During positive half-cycle of the input voltage, the diode D1 is in forward bias and conducts through the load resistor R1. Hence the current produces an output voltage across the load resistor R1, which has the same shape as the +ve half cycle of the input voltage. During the negative half-cycle of the input voltage, the diode is reverse biased and there is no current through the circuit. i.e., the voltage across R1 is zero.

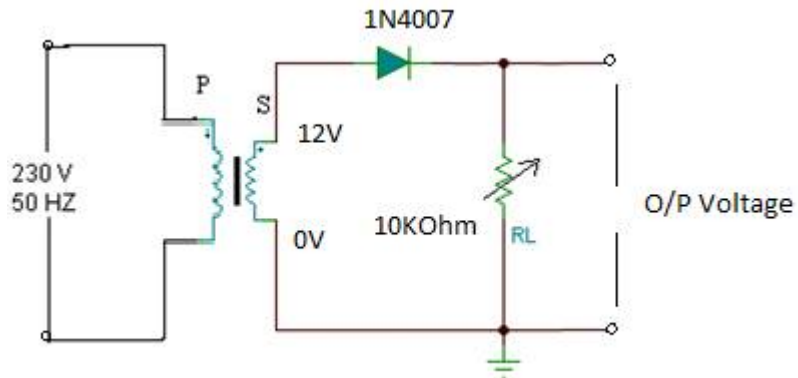
The net result is that only the +ve half cycle of the input voltage appears across the load. The average value of the half wave rectified o/p voltage is the value measured on dc voltmeter.

For practical circuits, transformer coupling is usually provided for two reasons.

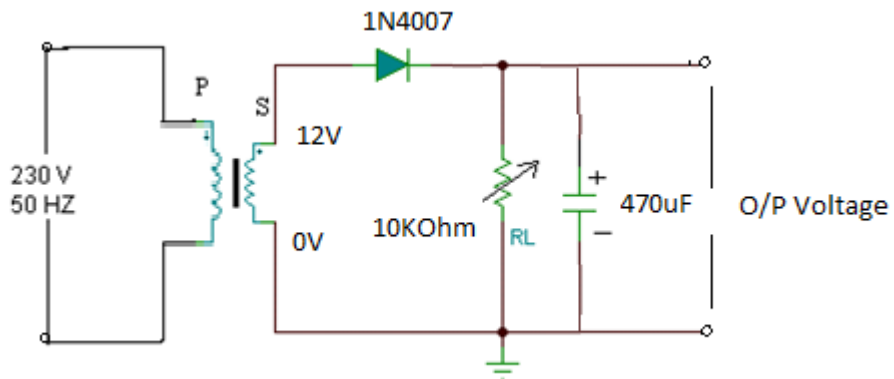
1. The voltage can be stepped-up or stepped-down, as needed.
2. The ac source is electrically isolated from the rectifier. Thus preventing shock hazards in the secondary circuit.

CIRCUIT DIAGRAMS:

Without Filter

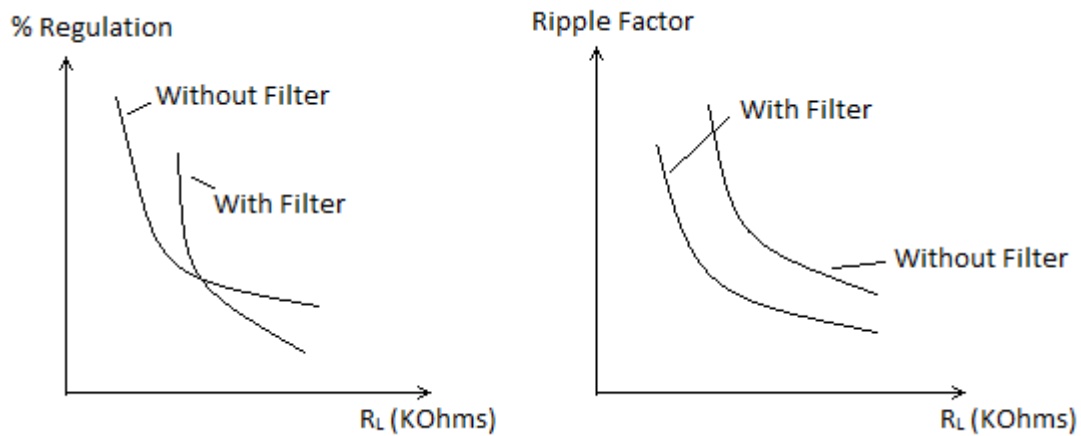


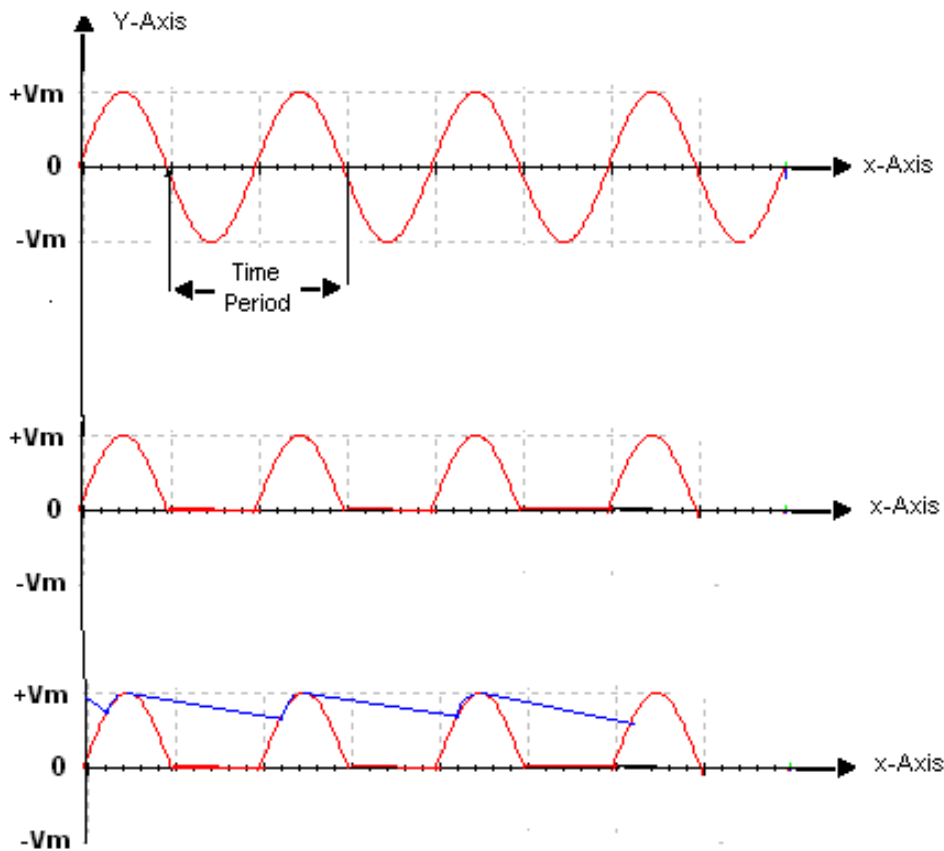
With Filter



Model graphs

% Regulation & Ripple Factor (With & Without Filter)





PROCEDURE:

1. Connections are made as per the circuit diagram.
2. Connect the primary side of the transformer to ac mains and the secondary side to the rectifier input.
3. By using the multimeter, measure the ac input voltage of the rectifier and, ac and dc voltage at the output of the rectifier.
4. Find the theoretical value of dc voltage by using the formula,

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

Where, $V_m = 2V_{rms}$, (V_{rms} =output ac voltage.)

The Ripple factor is calculated by using the formula
 $r = \text{ac output voltage} / \text{dc output voltage}$.

Regulation Characteristics

1. Connections are made as per the circuit diagram.
2. Obtain the no load voltage V_{NL} (V_{dc}) before connecting the load resistance.
3. By increasing the value of the variable resistance, the voltage across the load and

current flowing through the load are measured.

4. The reading is tabulated.
5. Draw a graph for load resistance (RL) versus Ripple factor and load resistance (RL) versus % regulation for without and with filter.
6. Connect the o/p terminals to CRO and observe the waveform for without and with filter.

OBSERVATIONS:

Without Filter

VNL = 6.65 V

RL (K Ohms)	Vac (Volts)	Vdc (Volts)	Ripple Factor = Vac/ Vdc	% Regulation (VNL-VFL)/VFL *100
1.1	6.15	4.95	1.242	14.141
2.33	6.17	4.98	1.239	13.454
3.02	6.18	4.99	1.238	13.226
4.09	6.18	4.99	1.238	13.226
5.13	6.19	5	1.237	13
6.21	6.2	5.01	1.236	12.774
7.09	6.32	5.1	1.239	10.784
8.12	6.34	5.1	1.243	10.784
9.49	6.35	5.11	1.243	10.568

With Filter

VNL = 15.19 V

RL (K Ohms)	Vac (Volts)	Vdc (Volts)	Ripple Factor = Vac/ Vdc	% Regulation (VNL-VFL)/VFL *100
1.1	14.13	0.22	0.0156	7.502
2.33	14.48	0.07	0.0048	4.903
3.02	14.63	0.07	0.0047	3.228
4.09	14.64	0.04	0.003	3.757
5.13	14.66	0.03	0.002	3.615
6.21	14.68	0.02	0.001	3.474
7.09	14.75	0.02	0.001	2.983
8.12	14.92	0.01	0.0006	1.809
9.49	15.05	0.01	0.0006	0.9302

RESULT: The Ripple factor and % Regulation for the Half-wave Rectifier with and without filters are measured and plotted.

VIVA QUESTIONS:

1. What is the PIV of Half wave rectifier?
2. What is the efficiency of half wave rectifier?
3. What is the rectifier?
4. What is the difference between the half wave rectifier and full wave Rectifier?
5. What is the o/p frequency of Bridge Rectifier?
6. What are the ripples?
7. What is the function of the filters?
8. What is TUF?
9. What is the average value of o/p voltage for HWR?
10. What is the peak factor?

6. Full Wave Rectifier with and without Filter

AIM: To examine the input and output waveforms of full wave Rectifier and also calculate its % load regulation and ripple factor.

COMPONENTS REQUIRED:

S.No	Name	Type / Range	Quantity
1	PN Diode	1N4007	2
2	AC Power supply	(12V-0-12V)	1
3	Variable Resistor	(0-10KΩ)	1
4	Capacitor	470μF	1
5	Multimeter		1
6	Breadboard		
7	Connecting wires		
8	CRO & CRO Probes		

THEORY:

The circuit of a center-tapped full wave rectifier uses two diodes D1&D2. During positive half cycle of secondary voltage (input voltage), the diode D1 is forward biased and D2 is reverse biased. The diode D1 conducts and current flows through load resistor RL. During negative half cycle, diode D2 becomes forward biased and D1 reverse biased. Now, D2 conducts and current flows through the load resistor RL in the same direction. There is a continuous current flow through the load resistor RL, during both the half cycles and will get unidirectional current as show in the model graph. The difference between full wave and half wave rectification is that a full wave rectifier allows unidirectional (one way) current to the load during the entire 360 degrees of the input signal and half-wave rectifier allows this only during one half cycle (180 degree).

Theoretical Calculations

$$V_{rms} = V_m / \sqrt{2} \Rightarrow V_m = V_{rms} \sqrt{2}$$

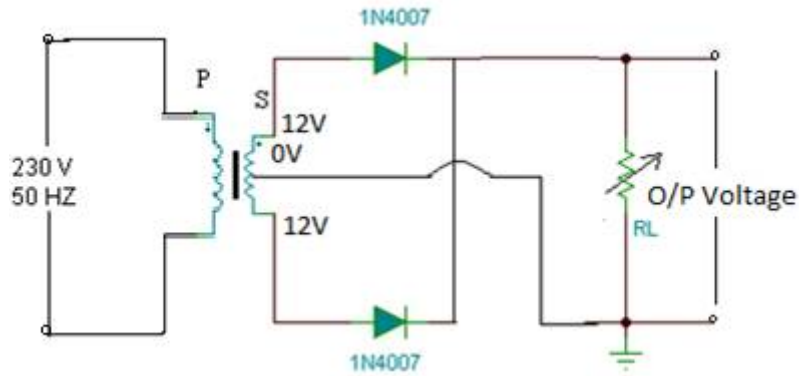
$$V_{dc} = 2V_m / \pi$$

(i) Without filter: Ripple factor, $r = \sqrt{(V_{rms} / V_{dc})^2 - 1} = 0.812$

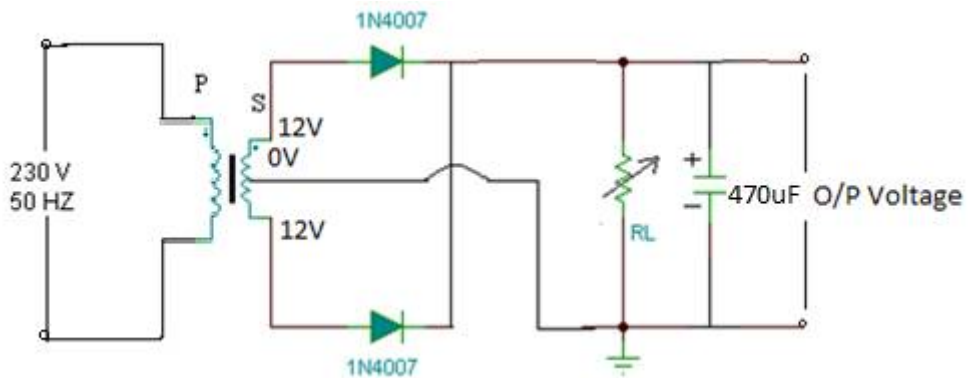
(ii) With filter: Ripple factor, $r = 1 / (4\sqrt{3} f C R_L)$

CIRCUIT DIAGRAM:

Without Filter

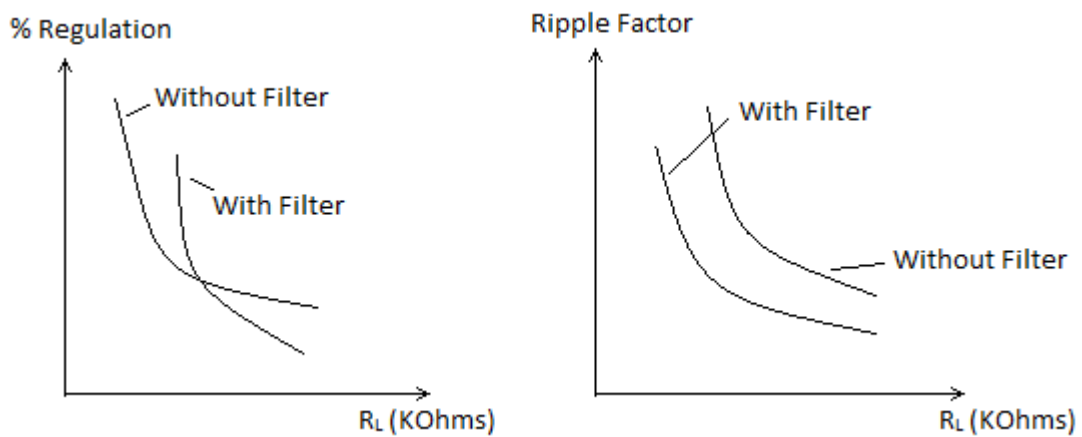


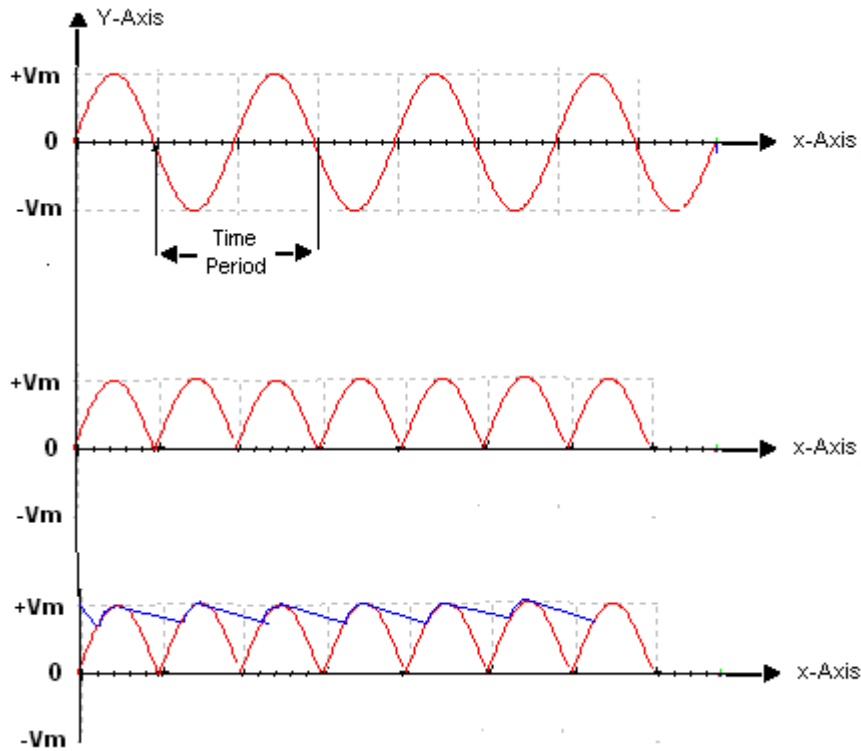
With Filter



Model Graphs

% Regulation & Ripple Factor (With & Without Filter)





PROCEDURE:

1. Connections are made as per the circuit diagram.
2. Obtain the no load voltage V_{NL} (Vdc) before connecting the load resistance.
3. By increasing the value of the variable resistance, the voltage across the load and current flowing through the load are measured.
4. The reading is tabulated.
5. Draw a graph for load resistance (RL) versus Ripple factor and load resistance (RL) versus % regulation for without and with filter.
6. Connect the o/p terminals to CRO and observe the waveform for without and with filter.

OBSERVATIONS:

Without Filter

VNL = 10.6 V

RL (Ohms)	Vac (Volts)	Vdc (Volts)	Ripple Factor = Vac/ Vdc	% Regulation (VNL-VFL)/VFL *100
1.07	4.55	9.89	0.46	7.179
2.02	4.57	9.91	0.461	6.963
3.04	4.58	9.94	0.46	6.639
4.08	4.6	9.97	0.461	6.319
5.1	4.62	9.97	0.463	6.319
6.44	4.62	9.98	0.462	6.212
7.31	4.63	10.04	0.462	5.578
8.41	4.64	10.04	0.462	5.578
9.65	4.66	10.08	0.462	5.159
10.04	4.68	10.09	0.463	5.055

With Filter

VNL = 15.54 V

RL (Ohms)	Vac (Volts)	Vdc (Volts)	Ripple Factor = Vac/ Vdc	% Regulation (VNL-VFL)/VFL *100
1.07	0.21	14.67	0.014	5.93
2.02	0.14	14.83	0.009	4.788
3.04	0.1	14.93	0.006	4.086
4.08	0.09	15.05	0.005	3.256
5.1	0.07	15.12	0.004	2.778
6.44	0.05	15.12	0.003	2.778
7.31	0.04	15.22	0.002	2.103
8.41	0.04	15.23	0.002	2.035
9.65	0.04	15.24	0.002	1.969
10.04	0.03	15.27	0.001	1.768

RESULT: The ripple factor and % regulation of the Full-wave rectifier (with filter and without filter) are calculated and plotted.

VIVA QUESTIONS:

1. Define regulation of the full wave rectifier?
2. Define peak inverse voltage (PIV)? And write its value for Full-wave rectifier?
3. If one of the diode is changed in its polarities what wave form would you get?
4. Does the process of rectification alter the frequency of the waveform?
5. What is ripple factor of the Full-wave rectifier?

6. What is the necessity of the transformer in the rectifier circuit?
7. What are the applications of a rectifier?
8. What is meant by ripple and define Ripple factor?
9. Explain how capacitor helps to improve the ripple factor?
10. Can a rectifier made in INDIA ($V=230v$, $f=50Hz$) be used in USA ($V=110v$, $f=60Hz$)?

7. FET Characteristics

AIM: To obtain the drain and transfer characteristics of a given JFET.

COMPONENTS REQUIRED:

S.No	Name	Type / Range	Quantity
1	JFET	BFW11	1
2	Regulated Power Supply	(0-15V)	1
3	Resistors	100Ω, 560 Ω	1
4	Voltmeter	(0-20V)	1
5	Ammeter	(0-200mA)	1
6	Breadboard		
7	Connecting wires		

THEORY:

A FET is a three terminal device, having the characteristics of high input impedance and less noise, the Gate to Source junction of the FET s always reverse biased. In response to small applied voltage from drain to source, the n-type bar acts as sample resistor, and the drain current increases linearly with V_{DS} . With increase in I_D the ohmic voltage drop between the source and the channel region reverse biases the junction and the conducting position of the channel begins to remain constant. The V_{DS} at this instant is called “pinch of voltage”.

If the gate to source voltage (V_{GS}) is applied in the direction to provide additional reverse bias, the pinch off voltage ill is decreased. In amplifier application, the FET is always used in the region beyond the pinch-off.

FET Parameters

AC Drain Resistance, $r_d = \frac{\Delta V_{DS}}{\Delta I_D}$ at constant V_{GS}

Transconductance, $g_m = \frac{\Delta I_D}{\Delta V_{GS}}$ at constant VDS

Amplification, $\mu = \frac{\Delta V_{DS}}{\Delta V_{GS}}$ at constant ID

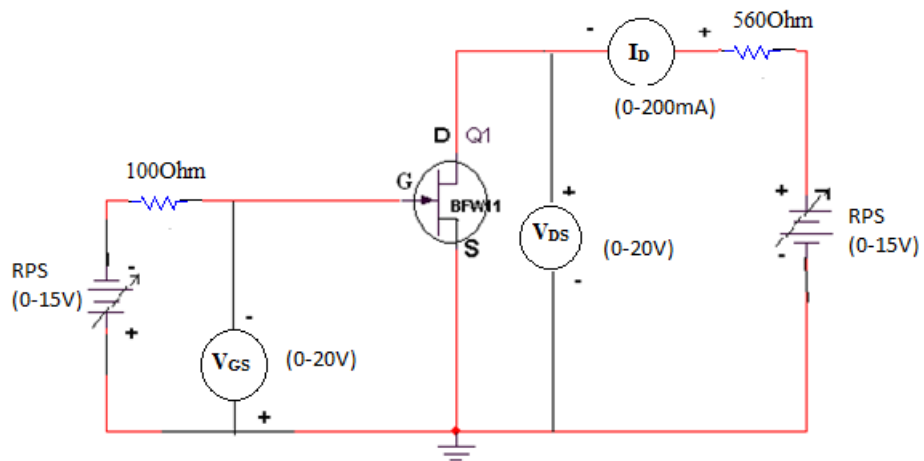
Relation between above parameters

$$\mu = r_d * g_m$$

The drain current is given by

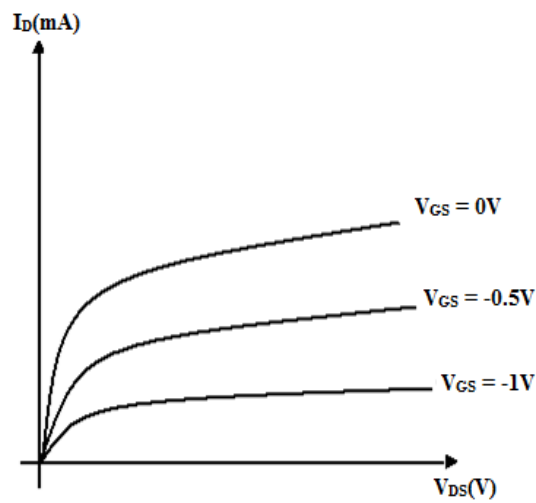
$$I_D = I_{DSS} \left(1 - \frac{V_{GS}}{V_P} \right)^2$$

CIRCUIT DIAGRAM:

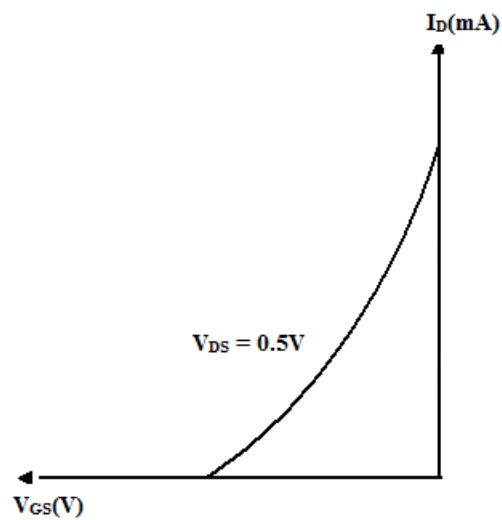


Model graphs

Drain Characteristics



Transfer Characteristics



PROCEDURE:

1. All the connections are made as per the circuit diagram.
2. To plot the drain characteristics, keep VGS constant at 0V.
3. Vary the VDD and observe the values of VDS and ID.
4. Repeat the above steps 2, 3 for different values of VGS.
5. All the readings are tabulated.
6. To plot the transfer characteristics, keep VDS constant at some convenient voltage.
7. Vary VGG and observe the values of VGS and ID.
8. The readings are tabulated.

OBSERVATIONS:

Drain Characteristics

S.NO	VGS = 0V		VGS = -0.5V		VGS = -1V	
	VDS(V)	ID(mA)	VDS(V)	ID(mA)	VDS(V)	ID(mA)
1	0	0	0	0	0	0
2	0.01	0.1	0.01	0.1	0.03	0.1
3	0.5	2.5	0.25	1	0.1	0.2
4	1	4	0.6	2	0.5	0.8
5	1.6	5	1	2.5	0.75	1
6	3	5.5	1.2	2.8	2.2	1.2
7	5	5.6	2.6	3.1	6	1.3
8	7	5.6	4.8	3.3	8	1.3
9	10	5.6	8	3.3	11	1.3
10	11	5.6	11	3.3	12	1.3
11	12.3	5.6	13.8	3.3	14.8	1.3

Transfer Characteristics

S.NO	VDS = 0.5V	
	VGS (V)	ID(mA)
1	0	4.2
2	0.1	4
3	0.2	3.8
4	0.4	3.8
5	0.6	2.6
6	0.8	1.8
7	0.9	1.5
8	1	1.2
9	1.2	0.6
10	1.3	0.2
11	1.5	0

RESULT: The drain and transfer characteristics of a JFET are obtained.

VIVA QUESTIONS:

1. What are the advantages of FET?
2. What is the difference between FET and BJT?
3. Explain different regions of V-I characteristics of FET.
4. What are the applications of FET?
5. What are the types of FET?
6. Draw the symbol of FET.
7. What are the disadvantages of FET?
8. What are the parameters of FET?

8. Frequency Response of CE Amplifier

AIM:

1. To measure the voltage gain of CE amplifier
2. To draw the frequency response curve of CE amplifier

COMPONENTS REQUIRED:

S.No	Name	Type / Range	Quantity
1	Transistor	BC107	1
2	Regulated Power Supply	(0-15V)	1
3	Resistors	10K Ω , 1K Ω , 4.7K Ω	1, 2, 1
4	Variable Resistor	0-100K Ω	1
5	Capacitors	22 μ F, 47 μ F	2,1
6	Voltmeter	(0-20V)	1
7	Function Generator		
8	CRO Breadboard		
9	Connecting wires		

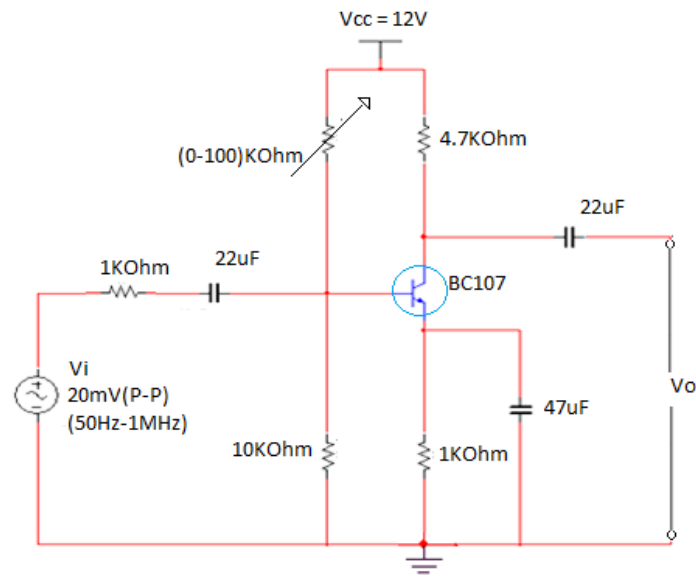
THEORY:

The CE amplifier provides high gain & wide frequency response. The emitter lead is common to both input & output circuits and is grounded. The emitter-base circuit is forward biased. The collector current is controlled by the base current rather than emitter current. The input signal is applied to base terminal of the transistor and amplifier output is taken across collector terminal. A very small change in base current produces a much larger change in collector current.

When +VE half-cycle is fed to the input circuit, it opposes the forward bias of the circuit which causes the collector current to decrease, it decreases the voltage more -VE. Thus when input cycle varies through a -VE half-cycle, increases the forward bias of the circuit, which causes the collector current to increase thus the output signal is common emitter amplifier is in out of phase with the input signal. An amplified output signal is obtained when this fluctuating collector current flows through a collector resistor, R_c .

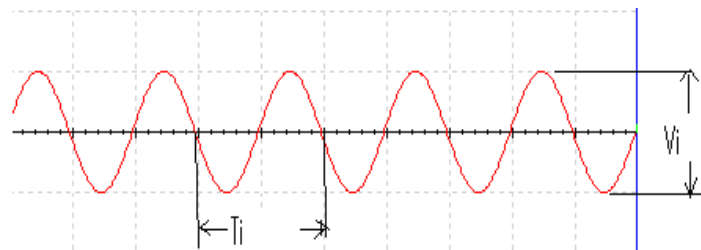
The capacitor across the collector resistor R_c will act as a bypass capacitor. This will improve high frequency response of amplifier

CIRCUIT DIAGRAM:

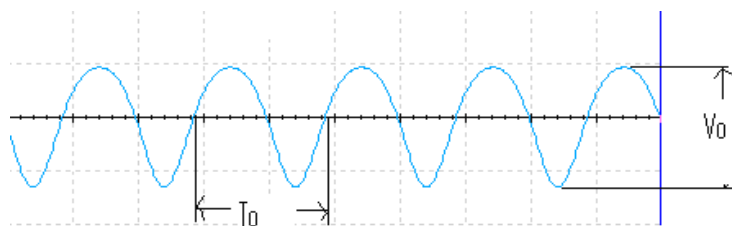


Model Waveforms

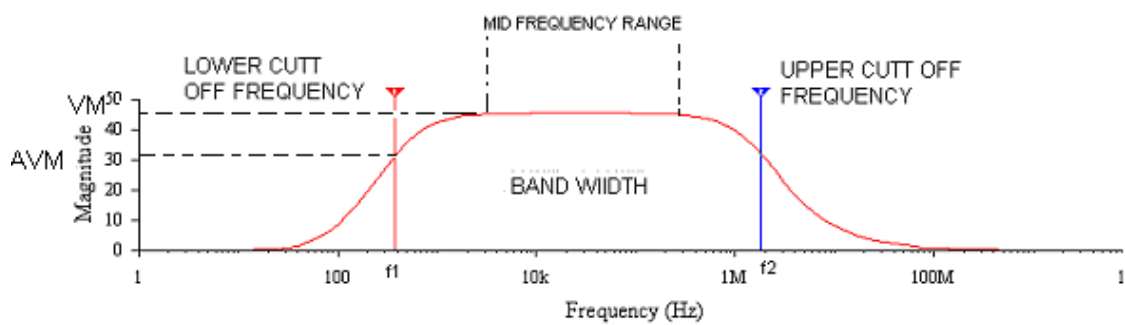
Input Waveform



Output Waveform



Frequency Response



PROCEDURE:

1. Connect the circuit as shown in circuit diagram
2. Apply the input of 20mV peak-to-peak and 50Hz frequency using function generator.
3. Measure the Output Voltage V_O (p-p).
4. Tabulate the readings in the tabular form.
5. The voltage gain can be calculated by using the expression $A_v = (V_O/V_i)$
6. For plotting the frequency response the input voltage is kept Constant at 20mV peak-to- peak and the frequency is varied from 50Hz to 1MHz Using function generator.
7. All the readings are tabulated and voltage gain in dB is calculated by using the expression $A_v = 20 \log_{10} (V_O/V_i)$
8. A graph is drawn by taking frequency on x-axis and gain in dB on y-axis on Semi-log graph.
9. The band width of the amplifier is calculated from the graph using the expression,

Bandwidth, $BW = f_2 - f_1$

Where f_1 is the lower cut-off frequency of CE amplifier, and

Where f_2 is the upper cut-off frequency of CE amplifier

The bandwidth product of the amplifier is calculated using the expression

Gain Bandwidth product = (3dB mid-band gain) X (Bandwidth)

OBSERVATIONS:

Frequency Response

Frequency (Hz)	Input voltage (v_i)	Output voltage (v_o)	A_v	Gain in dB $A_v = 20 \log_{10} (v_o/v_i)$
50	20	0.9	45	33.06
300	20	2	100	40
700	20	2.35	117.5	41.4
1K	20	2.4	120	41.6
2K	20	2.4	120	41.6
5K	20	2.4	120	41.6
10K	20	2.4	120	41.6
20K	20	2.4	120	41.6
40K	20	2.4	120	41.6
64K	20	2.4	120	41.6
65K	20	2.35	117.5	41.4
100K	20	2	100	40
500K	20	0.65	32.5	3.24
1M	20	0.4	20	26.02

RESULT: The voltage gain and frequency response of the CE amplifier are obtained.

VIVA QUESTIONS:

1. What is phase difference between input and output waveforms of CE amplifier?
2. What type of biasing is used in the given circuit?
3. If the given transistor is replaced by a p-n-p, can we get output or not?
4. What is effect of emitter-bypass capacitor on frequency response?
5. What is the effect of coupling capacitor?
6. What is region of the transistor so that it is operated as an amplifier?
7. How does transistor act as an amplifier?
8. Draw the h-parameter model of CE amplifier.
9. What type of transistor configuration is used in intermediate stages of a multistage amplifier?
10. What is early effect?

9. Frequency Response of CC Amplifier

AIM:

1. To measure the voltage gain of CC amplifier
2. To draw the frequency response curve of CC amplifier

COMPONENTS REQUIRED:

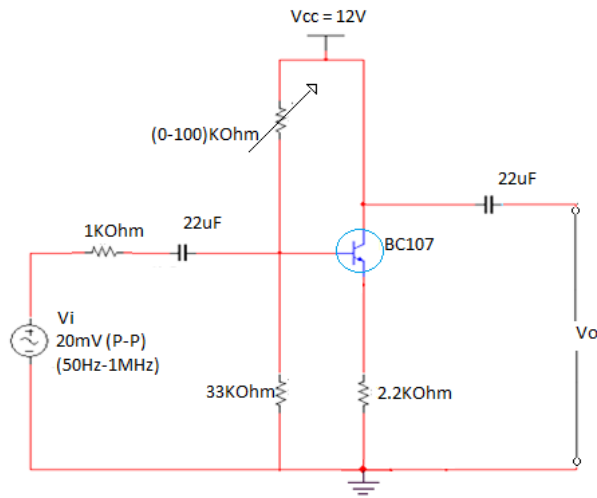
S.No	Name	Type / Range	Quantity
1	Transistor	BC107	1
2	Regulated Power Supply	(0-15V)	1
3	Resistors	1K Ω , 33K Ω , 2.2K Ω	1
4	Variable Resistor	0-100K Ω	1
5	Capacitors	22 μ F (0-	2
6	Voltmeter	20V)	1
7	Function Generator		
8	CRO Breadboard		
9	Connecting wires		

THEORY:

In common-collector amplifier the input is given at the base and the output is taken at the emitter. In this amplifier, there is no phase inversion between input and output. The input impedance of the CC amplifier is very high and output impedance is low. The voltage gain is less than unity. Here the collector is at ac ground and the capacitors used must have a negligible reactance at the frequency of operation.

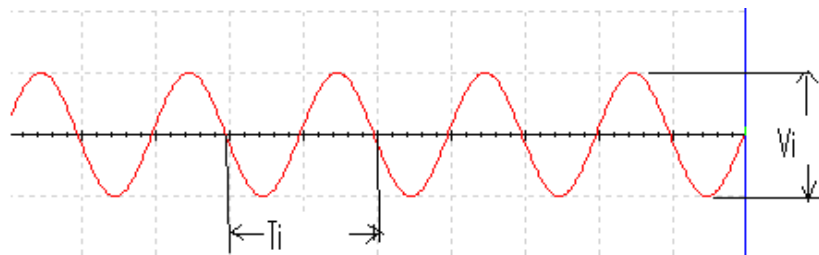
This amplifier is used for impedance matching and as a buffer amplifier. This circuit is also known as emitter follower. The most common use of the emitter follower is as a circuit, which performs the function of impedance transformation over a wide range of frequencies.

CIRCUIT DIAGRAM:

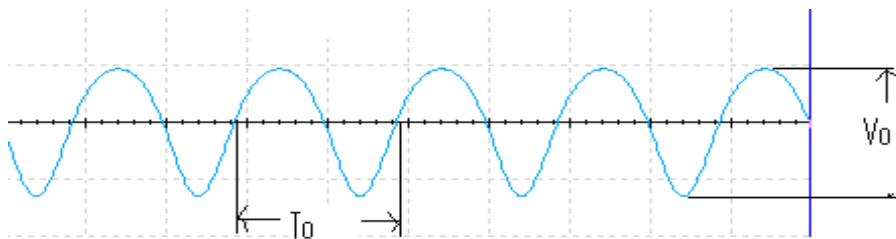


Model Waveform

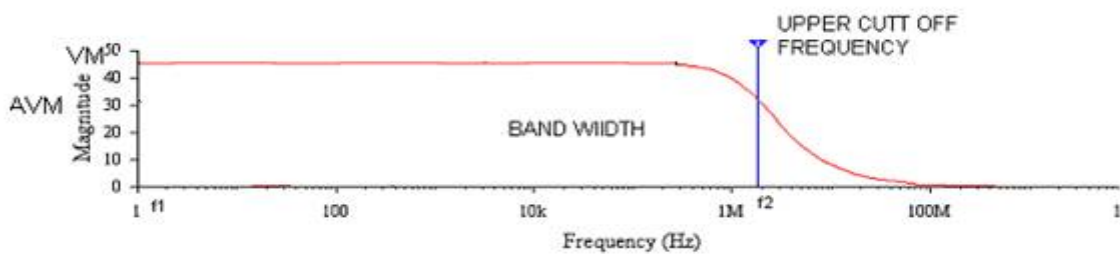
Input Waveform



Output Waveform



Frequency response



PROCEDURE:

1. Connect the circuit as shown in circuit diagram
2. Apply the input of 20mV peak-to-peak and 50Hz frequency using function generator.
3. Measure the Output Voltage V_O (p-p)
4. Tabulate the readings in the tabular form.
5. The voltage gain can be calculated by using the expression $A_v = (V_O/V_i)$
6. For plotting the frequency response the input voltage is kept Constant at 20mV peak-to- peak and the frequency is varied from 50Hz to 1MHz Using function generator.
7. All the readings are tabulated and voltage gain in dB is calculated by using the expression $A_v = 20 \log_{10} (V_O/V_i)$
8. A graph is drawn by taking frequency on x-axis and gain in dB on y-axis on Semi-log graph.
9. The band width of the amplifier is calculated from the graph using the expression,

Bandwidth, $BW = f_2 - f_1$

Where f_1 is the lower cut-off frequency of CE amplifier, and

Where f_2 is the upper cut-off frequency of CE amplifier

The bandwidth product of the amplifier is calculated using the expression

Gain Bandwidth product = (3dB mid-band gain) X (Bandwidth)

OBSERVATIONS:

Frequency Response

Frequency (Hz)	Input voltage (v_i)	Output voltage (v_o)	A_v	Gain in dB $A_v = 20 \log_{10} (v_o/v_i)$
50	20	0.02	1	0
300	20	0.02	1	0
1K	20	0.02	1	0
6K	20	0.02	1	0
30K	20	0.02	1	0
60K	20	0.02	1	0
100K	20	0.02	1	0
250K	20	0.02	1	0
300K	20	0.019	0.95	-0.445
800K	20	0.019	0.95	-0.445
1M	20	0.018	0.9	-0.915

RESULT: The voltage gain and frequency response of the CC amplifier are obtained.

VIVA QUESTIONS:

1. What are the applications of CC amplifier?
2. What is the voltage gain of CC amplifier?
3. What are the values of input and output impedances of the CC amplifier?
4. To which ground the collector terminal is connected in the circuit?
5. Identify the type of biasing used in the circuit?
6. Give the relation between α , β and γ .
7. Write the other name of CC amplifier?
8. What are the differences between CE, CB and CC?
9. When compared to CE, CC is not used for amplification. Justify your answer?
10. What is the phase relationship between input and output in CC?

10. Frequency Response of Common Source FET Amplifier

AIM: To find the frequency response and bandwidth of a given single stage FET amplifier.

COMPONENTS REQUIRED:

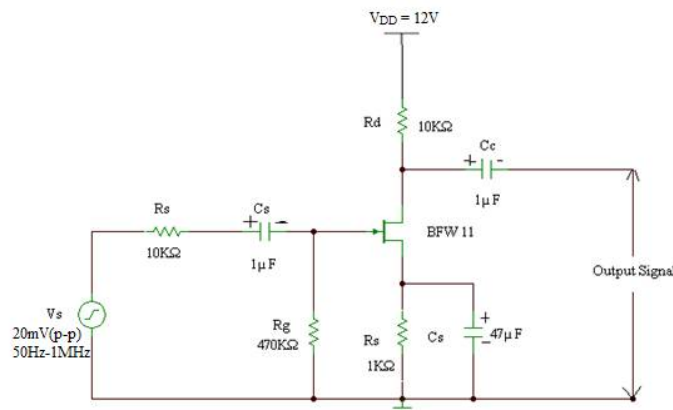
S.No	Name	Type / Range	Quantity
1	JFET	BFW11	1
2	Regulated Power Supply	(0-15V)	1
3	Resistors	1 K Ω , 10 K Ω , 470 K Ω	1, 2, 1
4	Capacitors	1 μ F, 47 μ F	2, 1
5	Voltmeter	F (0-20V)	1
6	Function Generator		
7	CRO Breadboard		
8	Connecting wires		

THEORY:

A field-effect transistor (FET) is a type of transistor commonly used for weak-signal amplification (for example, for amplifying wireless (signals)). The device can amplify analog or digital signals. It can also switch DC or function as an oscillator. In the FET, current flows along a semiconductor path called the channel. At one end of the channel, there is an electrode called the source. At the other end of the channel, there is an electrode called the drain. The physical diameter of the channel is fixed, but its effective electrical diameter can be varied by the application of a voltage to a control electrode called the gate. Field-effect transistors exist in two major classifications. These are known as the junction FET (JFET) and the metal-oxide- semiconductor FET (MOSFET). The junction FET has a channel consisting of N-type semiconductor (N-channel) or P-type semiconductor (P-channel) material; the gate is made of the opposite semiconductor type. In P-type material, electric charges are carried mainly in the form of electron deficiencies called holes. In N-type material, the charge carriers are primarily electrons. In a JFET, the junction is the boundary between the channel and the gate. Normally, this P-N junction is reverse-biased (a DC voltage is applied to it) so that no current flows between the channel and the gate. However, under some conditions there is a small current through the junction during part of the input signal cycle. The FET has some advantages and some disadvantages relative to the bipolar transistor. Field-effect transistors are preferred

for weak-signal work, for example in wireless, communications and broadcast receivers. They are also preferred in circuits and systems requiring high impedance. The FET is not, in general, used for high-power amplification, such as is required in large wireless communications and broadcast transmitters.

CIRCUIT DIAGRAM:

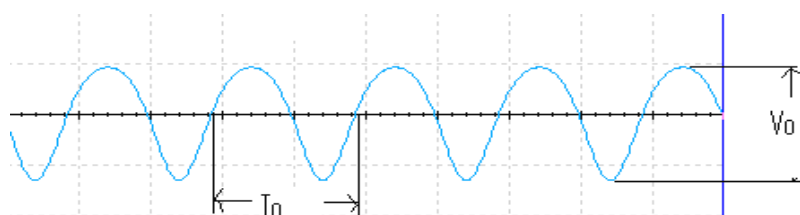


Model Waveforms

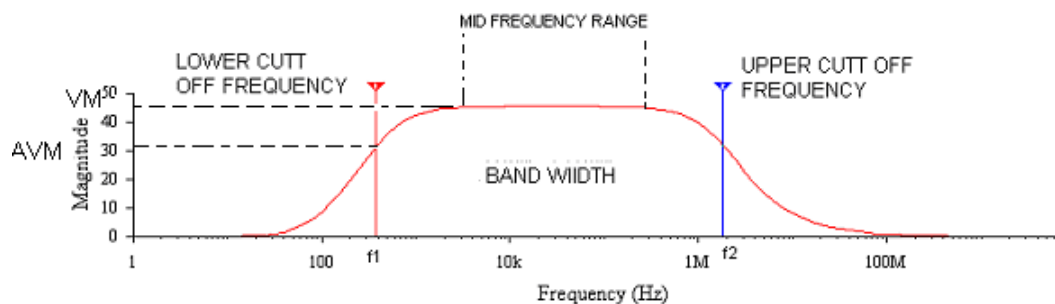
Input Waveform



Output Waveform



Frequency Response



PROCEDURE:

1. As per the design specifications, connect the circuit as shown.
2. Set the frequency of I/P signal at 5 KHz and increase the amplitude, till O/P gets distorted. The value of I/P signal is maximum signal handling capacity.
3. Set I/P signal at a constant value, less than the maximum signal handling capacity, vary frequency in the range 50Hz to 1MHz and find O/P voltage for each and every frequency.
4. Calculate voltage gain at each and every frequency.
5. Plot the frequency versus gain and determine fH and fL.
6. Calculate bandwidth fH - fL.
7. Procedure for measuring input impedance: Set the signal generator frequency at 2KHz and measure Vs and Vi. Then $I_i = \frac{V_s - V_i}{R_S}$. I/P impedance = V_i / I_i
8. Procedure for measuring O/P impedance: Open the O/P circuit and measure voltage (Vopen) across O/P using CRO. After connecting variable resistor at O/P terminals, vary the resistance to make the O/P (V open) become to half of its value. Then existing resistance is its O/P resistance.

OBSERVATIONS:

S. No	Frequency (Hz)	O/P voltage (V0)	Gain (V0/ VI)	Gain in dB $20 \cdot \log_{10}(V_0 / V_i)$
1	30	20	0.18	9
2	50	20	0.19	9.5
3	100	20	0.19	9.5
4	1K	20	0.19	9.5
5	3K	20	0.19	9.5
6	10K	20	0.19	9.5
7	30K	20	0.19	9.5
8	40K	20	0.19	9.5
9	50K	20	0.15	7.5
10	100K	20	0.11	5.5
11	500K	20	0.03	1.5
12	1M	20	0.02	1

RESULT: The frequency response of a common source FET Amplifier is obtained.

VIVA QUESTIONS:

1. What is the difference between FET and BJT?
2. FET is unipolar or bipolar?
3. Draw the symbol of FET?
4. What are the applications of FET?
5. FET is voltage controlled or current controlled?

6. Draw the equivalent circuit of common source FET amplifier?
7. What is the voltage gain of the FET amplifier?
8. What is the input impedance of FET amplifier?
9. What is the output impedance of FET amplifier?
10. What are the FET parameters?
11. What are the FET applications

11. Silicon UJT Characteristics

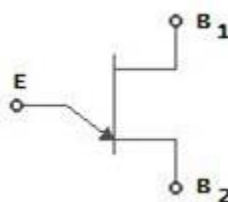
AIM: To observe the characteristics of UJT and to find the negative resistance region.

COMPONENTS REQUIRED:

S.No	Name	Type / Range	Quantity
1	UJT	2N2646	1
2	Regulated Power Supply	(0-15V)	2
3	Resistors	10k Ω , 47 Ω , 330 Ω	1
4	Voltmeter	(0-20V) (0-	1
5	Ammeter	200) mA	1
6	Breadboard		
7	Connecting wires		

THEORY:

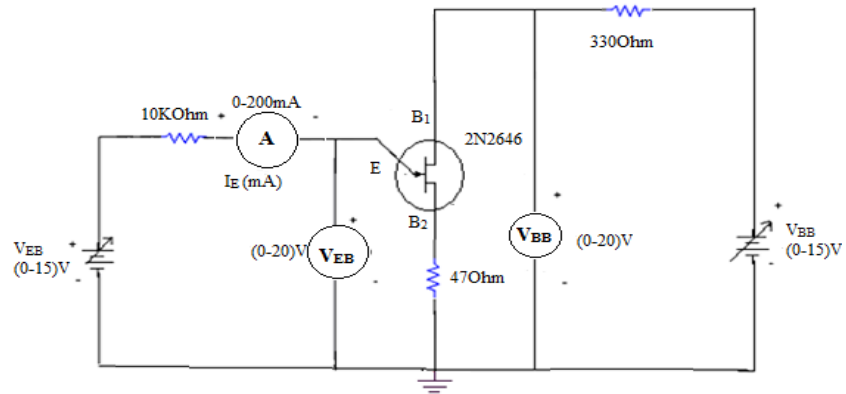
A Uni-junction Transistor (UJT) is an electronic semiconductor device that has only one junction. The UJT Uni-junction Transistor (UJT) has three terminals, an emitter (E) and two bases (B1 and B2). The base is formed by lightly doped n-type bar of silicon. Two ohmic contacts B1 and B2 are attached at its ends. The emitter is of p-type and it is heavily doped. The resistance between B1 and B2, when the emitter is open-circuit is called inter-base resistance. The original uni-junction transistor, or UJT, is a simple device that is essentially a bar of N type semiconductor material into which P type material has been diffused somewhere along its length. The 2N2646 is the most commonly used version of the UJT.



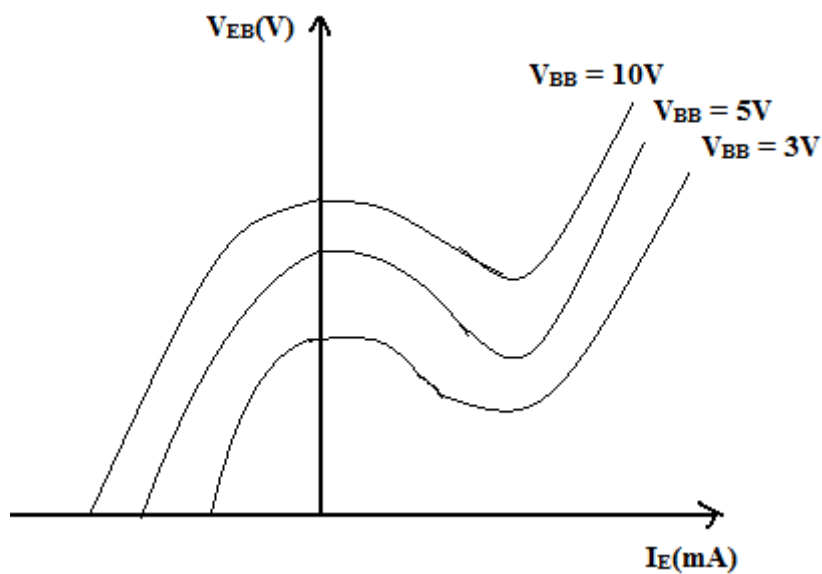
The UJT is biased with a positive voltage between the two bases. This causes a potential drop along the length of the device. When the emitter voltage is driven approximately one diode voltage above the voltage at the point where the P diffusion (emitter) is, current will begin to flow from the emitter into the base region. Because the base region is very lightly doped, the additional current (actually charges in the base region) causes (conductivity modulation) which reduces the resistance of the portion of the base between the emitter junction and the B2 terminal. This reduction in resistance means that the emitter junction is more forward biased, and so even more current is injected. Overall, the effect is a negative resistance at the emitter terminal. This is what makes the UJT useful, especially in simple oscillator circuits. When the emitter voltage reaches V_p , the current starts to increase and the emitter voltage starts to decrease. This is

represented by negative slope of the characteristics which is referred to as the negative resistance region, beyond the valley point; R_{B1} reaches minimum value and this region, V_{EB} proportional to I_E .

CIRCUIT DIAGRAM:



Model Graph



PROCEDURE:

1. Connection is made as per circuit diagram.
2. Output voltage V_{BB} is fixed at a constant level and by varying input voltage V_{EB}
3. corresponding emitter current I_E values are noted down.
4. This procedure is repeated for different values of output voltages V_{BB} .
5. All the readings are tabulated and a graph is plotted between V_{EB} and I_E for different values of V_{BB} .

OBSERVATIONS:

S.No	VBB = 3V		VBB = 5V		VBB = 10V	
	VEB(V)	IE(mA)	VEB(V)	IE(mA)	VEB(V)	IE(mA)
1	2.38	0	3.67	0	6.4	0
2	1.76	0.1	2.61	0.1	4.95	0.1
3	1.33	0.2	2.42	0.2	4.57	0.2
4	1.27	0.3	1.51	0.3	4.28	0.3
5	1.25	0.4	1.45	0.4	4.09	0.4
6	1.24	0.5	1.42	0.5	1.85	0.5
7	1.25	0.6	1.4	0.6	1.81	0.6
8	1.26	0.7	1.4	0.7	1.77	0.7
9	1.27	0.8	1.4	0.8	1.72	0.8
10	1.27	0.9	1.4	0.9	1.7	0.9

RESULT: The characteristics of UJT are observed for different values of VBB and the negative resistance region is obtained.

VIVA QUESTIONS:

1. What is the symbol of UJT?
2. Draw the equivalent circuit of UJT?
3. What are the applications of UJT?
4. Formula for the intrinsic stand off ratio?
5. What does it indicates the direction of arrow in the UJT?
6. What is the difference between FET and UJT?
7. Is UJT is used an oscillator? Why?
8. What is the Resistance between B 1 and B 2 is called as?
9. What is its value of resistance between B 1 and B 2?
10. Draw the characteristics of UJT?