LABORATORY MANUAL

ANALOG ELECTRONICS & OP-AMP

IV SEMESTER

PREPARED BY

TILU BEHERA, LECTURER

DEPARTMENT OF ELECTRICAL ENGINEERING

GOVERNMENT POLYTECHNIC SONEPUR

LIST OF EXPERIMENTS

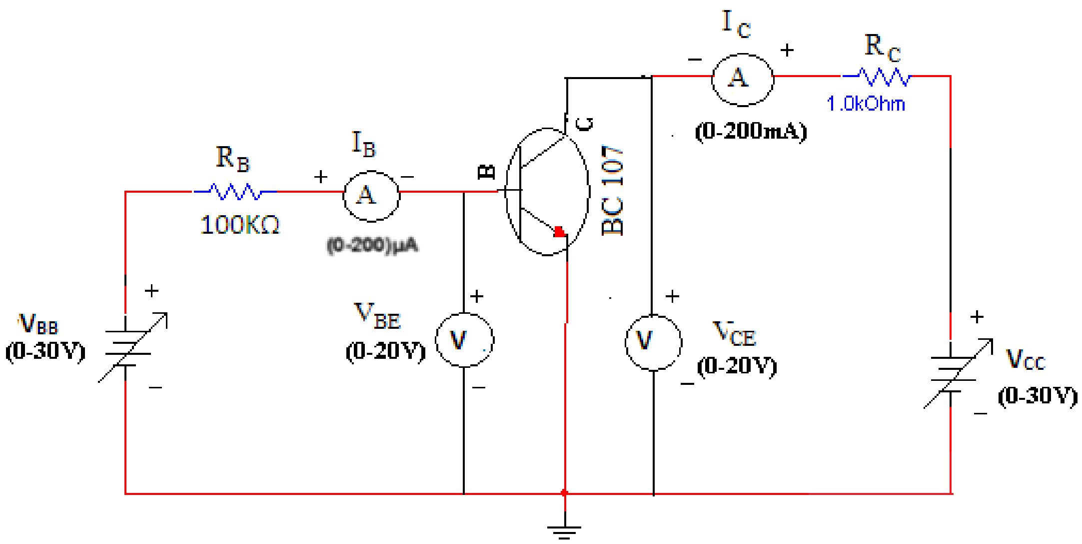
Experiment No. : 01	I/p & O/p characteristics of CE transistor

Aim: Determine the i/p & o/p characteristics of CE transistor configuration.

Equipment Required:

Sl. No.	Name	Quantity
1	Experimentation with Transistor trainer (NVIS-6502)	1(One) No.
2	Digital Multimeter	Four nos
3	Connecting wires (probes)	As per required

Circuit Diagram:

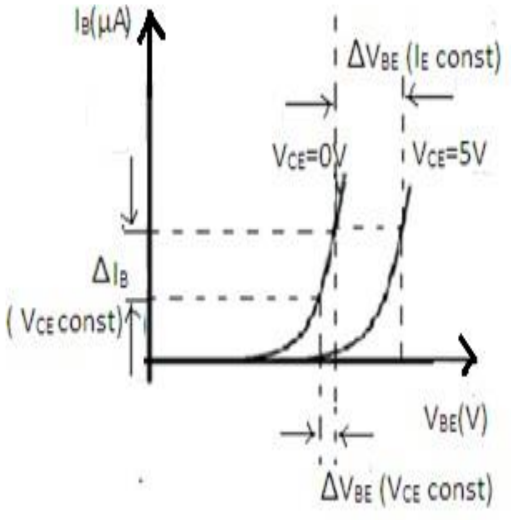


Observations:

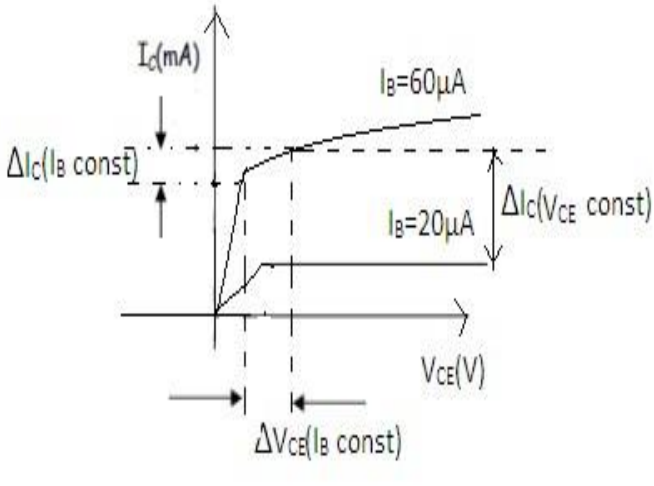
Input Characteristics				
V_{BB} (Volts)	V_{CE} = 0V		V_{CE} = 5V	
	V_{BE} (Volts)	I_B (µA)	V_{BE} (Volts)	I_B (µA)

Output Characteristics						
V_{CC} (Volts)	I_B = 0 µA		I_B = 20 µA		I_B = 40 µA	
	V_{CE} (Volts)	I_C (mA)	V_{CE} (Volts)	I_C (mA)	V_{CE} (Volts)	I_C (mA)

Graph:



Input Characteristics



Output Characteristics

Procedure:

Input Characteristics:

1. Connect the circuit as shown in the circuit diagram.
2. Keep output voltage V_{CE} = 0V by varying V_{CC}.
3. Varying V_{BB} gradually, note down base current I_B and base-emitter voltage V_{BE}.
4. Step size is not fixed because of non linear curve. Initially vary V_{BB} in steps of 0.1V. Once the current starts increasing vary V_{BB} in steps of 1V up to 12V.
5. Repeat above procedure (step 3) for V_{CE} = 5V.

Output Characteristics:

1. Connect the circuit as shown in the circuit diagram.
2. Keep emitter current I_B = 20 uA by varying V_{BB}.
3. Varying V_{CC} gradually in steps of 1V up to 12V and note down collector current I_C and Collector-Emitter Voltage(V_{CE}).
4. Repeat above procedure (step 3) for I_B = 60μA, 0μA.

To Plot Graph:

1. Plot the input characteristics by taking V_{BE} on X-axis and I_B on Y-axis at a constant V_{CE} as a constant parameter.
2. Plot the output characteristics by taking V_{CE} on X-axis and taking I_C on Y-axis taking I_B as a constant parameter.

Calculations from Graph:

1. Input Characteristics: To obtain input resistance find ΔV_{BE} and ΔI_B for a constant V_{CE}.
Input impedance = $h_{ie} = R_i = \Delta V_{BE} / \Delta I_B$ (V_{CE} is constant)
Reverse voltage gain = $h_{re} = \Delta V_{EB} / \Delta V_{CE}$ (I_B is constant)
2. Output Characteristics: To obtain output resistance find ΔI_C and ΔV_{CB} at a constant I_B.
Output admittance 1/hoe = $R_o = \Delta I_C / \Delta V_{CE}$ (I_B is constant)
Forward current gain = hfe = $\Delta I_C / \Delta I_B$ (V_{CE} is constant)

Conclusion: From the above experiment we are successfully plot the i/p & o/p characteristics of CE transistor configuration.

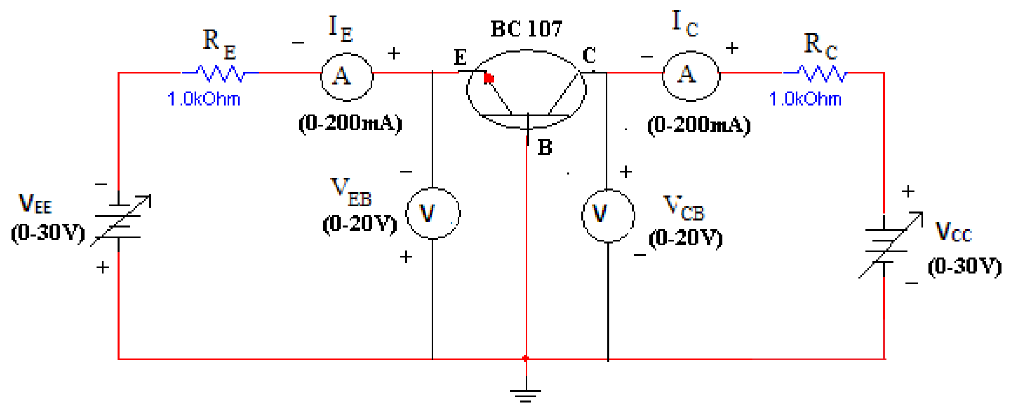
Experiment No. : 02	I/p & O/p characteristics of CB transistor

Aim: To study the input and output characteristics of a transistor in Common Base Configuration.

Equipment Required:

Sl. No.	Name	Quantity
1	Transistor BC 107	1(One) No.
2	Resistors (1KΩ)	2(Two) No.
3	Bread board	1(One) No.
4	Dual DC Regulated Power supply (0 – 30 V)	1(One) No.
5	Digital Multimeter	4(four) No.
6	Connecting wires (Single Strand)	As per required

Circuit Diagram:

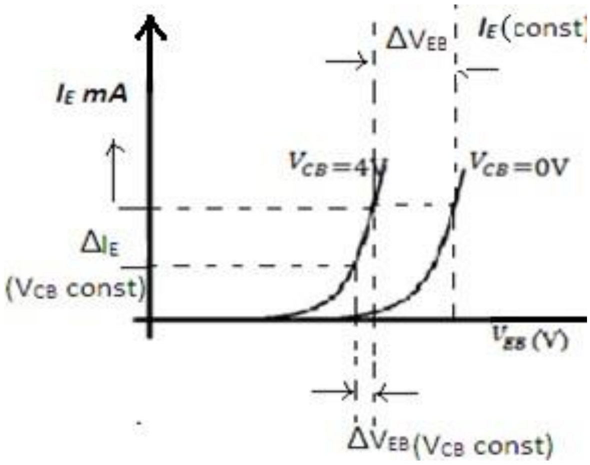


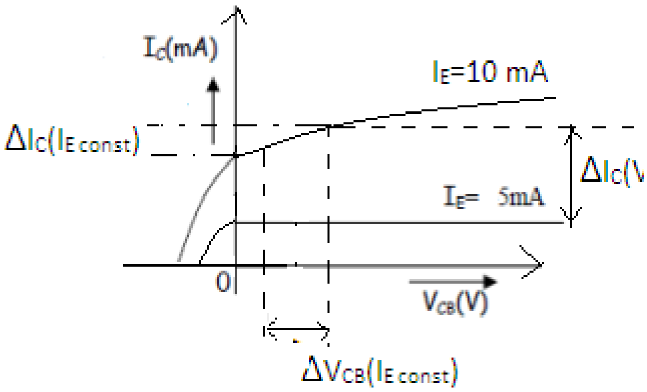
Observations:

Input Characteristics				
V_{EE} (Volts)	V_{CB} = 0V		V_{CB} = 4V	
	V_{EB} (Volts)	I_E (mA)	V_{EB} (Volts)	I_E (mA)

Output Characteristics						
V_{CC} (Volts)	I_E = 0mA		I_E = 5V		IE = 10mA	
	V_{CB} (Volts)	I_C (mA)	V_{CB} (Volts)	I_C (mA)	V_{CB} (Volts)	I_C (mA)

Graph:





To plot graph

1. Plot the input characteristics for different values of V_{CB} by taking V_{EE} on X-axis and I_E on Y-axis taking V_{CB} as constant parameter.
2. Plot the output characteristics by taking V_{CB} on X-axis and taking I_C on Y-axis taking I_E as a constant parameter.

Procedure:

Input Characteristics:

1) Connect the circuit as shown in the circuit diagram.
2) Keep output voltage V_{CB} = 0V by varying V_{CC}.
3) Varying V_{EE} gradually, note down emitter current I_E and emitter-base voltage(V_{EE}).
4) Step size is not fixed because of nonlinear curve. Initially vary V_{EE} in steps of 0.1 V. Once the current starts increasing vary V_{EE} in steps of 1V up to 12V.
5) Repeat above procedure (step 3) for V_{CB} = 4V.

Output Characteristics:

1. Connect the circuit as shown in the circuit diagram.
2. Keep emitter current I_E = 5mA by varying V_{EE}.
3. Varying V_{CC} gradually in steps of 1V up to 12V and note down collector current I_C and collector-base voltage (V_{CB}).
4. Repeat above procedure (step 3) for I_E = 10mA.
5. Repeat above procedure (step 3) for I_E = 10mA.

Conclusion: From the above experiment we are successfully plot the i/p & o/p characteristics of CB transistor configuration.

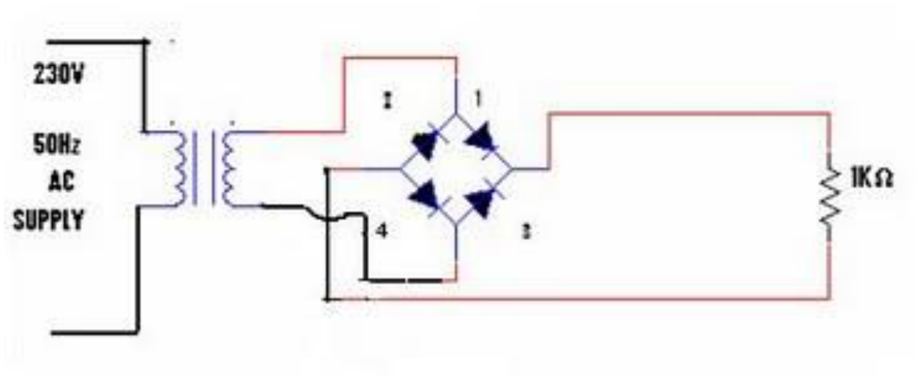
Experiment No. : 03	Study Bridge rectifier with & without filter

Aim: To calculate the ripple factor of a bridge rectifier, with and without filter.

Components Required:

Sl. No.	Name	Specification	Quantity
1	Experimental board		1 no.
2	Diodes	IN4007	4 nos
3	Resistor	1KΩ	1 no.
4	Capacitor	100µF	1 no.
5	Multi meter		2 nos
6	Transformer	6-0-6V	
7	Connecting wires		As per requirement

Circuit Diagram:



Theory:

The bridge rectifier is also a full-wave rectifier in which four p-n diodes are connected in the form of a bridge fashion. The Bridge rectifier has high efficiency when compared to half-wave rectifier. During every half cycle of the input, only two diodes will be conducting while other two diodes are in reverse bias.

Procedure:

1. Connections are made as per the circuit diagram.
2. Connect the ac main to the primary side of the transformer and secondary side to the bridge rectifier.
3. Measure the ac voltage at the input of the rectifier using the multi meter.
4. Measure both the ac and dc voltages at the output of the Bridge rectifier
5. Find the theoretical value of dc voltage by using the formula,

Calculation:

$V_{RMS} = V_M \div \sqrt{2}$

$V_M = V_{RMS} \times \sqrt{2}$

$V_{DC} = 2V_M \div \pi$

(i) Without filter:

Ripple factor, $\gamma = \sqrt{(V_{RMS} \div V_{DC})^2 - 1} = 0.482$

(ii) With filter:

Ripple factor, $\gamma = 1 \div (4\sqrt{3}fCR_L)$, where f = 50Hz

C = 100μF

R_L = 1KΩ

Observation:

Without filter:

V_{ac} (Volts)	V_{dc} (Volts)	$\gamma = V_{ac} \div V_{dc}$

With filter:

V_{ac} (Volts)	V_{dc} (Volts)	$\gamma = V_{ac} \div V_{dc}$

Precaution:

1. The voltage applied should not exceed in the ratings of the diode.
2. The diodes will be connected correctly.

Conclusion: The ripple factor of Bridge rectifier, with and without filter is calculated.

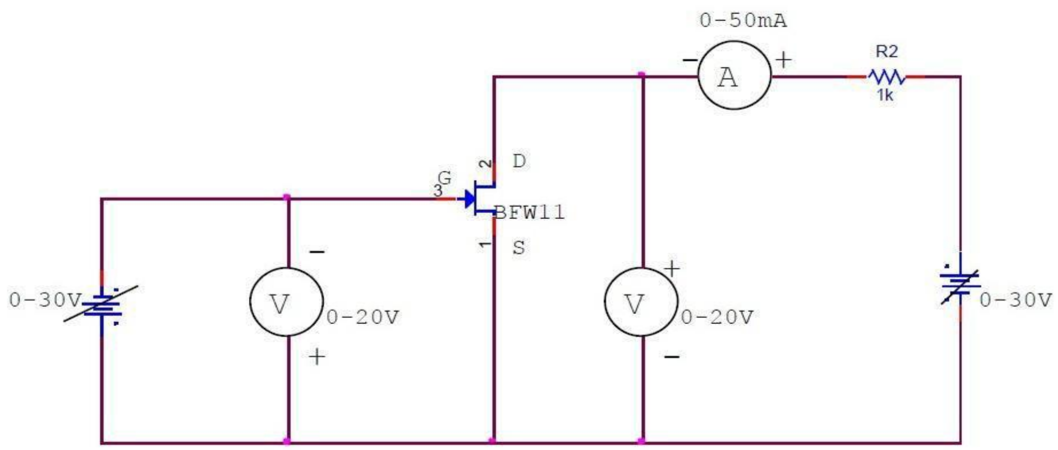
Experiment No. : 04	Transfer and drain characteristics of a JFET

Aim: To Plot the transfer and drain characteristics of a JFET and calculate its drain resistance, mutual conductance and amplification factor.

Equipment Required:

Sl..No.	Name	Quantity
1	Experimental trainer board FET(CL-500)	01 nos
2	Connecting wire or probes	As per required
3	Multimeter (DM-97)	03 nos

CIRCUIT DIAGRAM:



Specification:

For JFET BFW11: -

Gate Source Voltage VGS = - 30V

Forward Gain Current IGF = 10 mA

Maximum Power Dissipation PD = 300 mW.

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 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 Id the ohmic voltage drop between the and the channel region reverse biases the junction and the conducting position of the channel begins to remain In amplifier applications, the FET is always used in the region beyond the pinch off.

I_D= Idss (1-Vgs/ Vp)^2.

Procedure:

Drain characteristics:

1. Make the connections as per circuit diagram.
2. Keep VGS = 0V by varying VGG.
3. Varying VDD gradually, note down both drain current ID and drain to source voltage (VDS).
4. Step Size is not fixed because of non linear curve and vary the X-axis variable (i.e. if
5. Output variation is more, decrease input step size and vice versa).
6. Repeat above procedure (step 3) for VGS = -1V.

Transfer characteristics:

1. Keep VDS = 2V by varying VDD.
2. Varying VGG gradually from 0 – 5V, note down both drain current (ID) and gate to source Voltage (VGS).
3. Step Size is not fixed because of non linear curve and vary the X-axis variable (i.e. if
4. Output variation is more, decrease input step size and vice versa).
5. Repeat above procedure (step 2) for VDS = 4V.

Observations:

Drain characteristics:

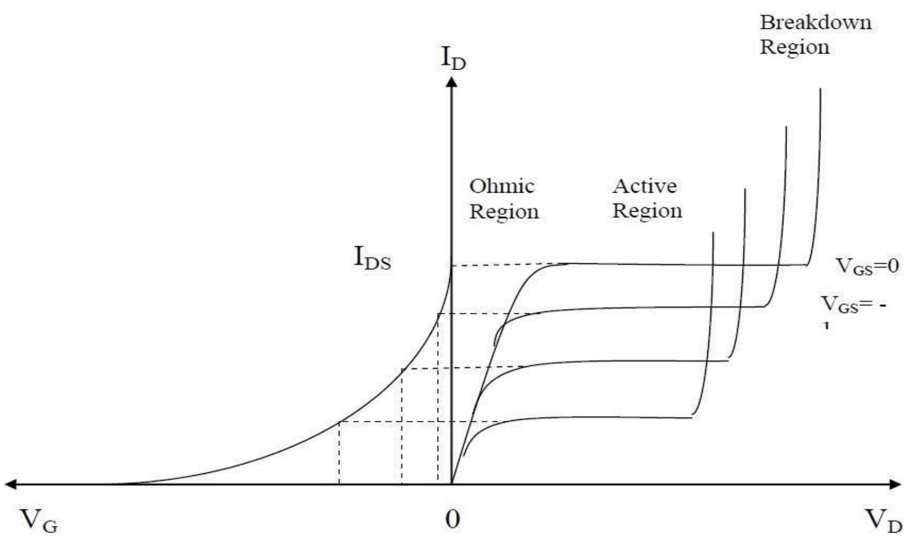
$V_{GS}(V) = 0$		$V_{GS}(V) = 0$	
$V_{DS}(V)$	I_D (mA)	$V_{DS}(V)$	I_D(mA)

Transfer characteristics:

$V_{DS}(V)= 1$		$V_{DS}(V)= 3$	
$V_{GS}(V)$	I_D (mA)	$V_{GS}(V)$	I_D(mA)

Model graph:

Transfer Characteristics **Drain Characteristics**



Calculations:

1. Drain resistance rd =ΔVDS/ΔID=
2. Trans conductance gm = ΔID/ ΔVGS=
3. Amplification factor μ= rd ×gm=

Result:

1. Drain Resistance (rd) =
2. Trans conductance (gm) =
3. Amplification factor (μ) =

Conclusion: From the above experiment we are successfully drawn the Drain & Transfer characteristics of JFET.

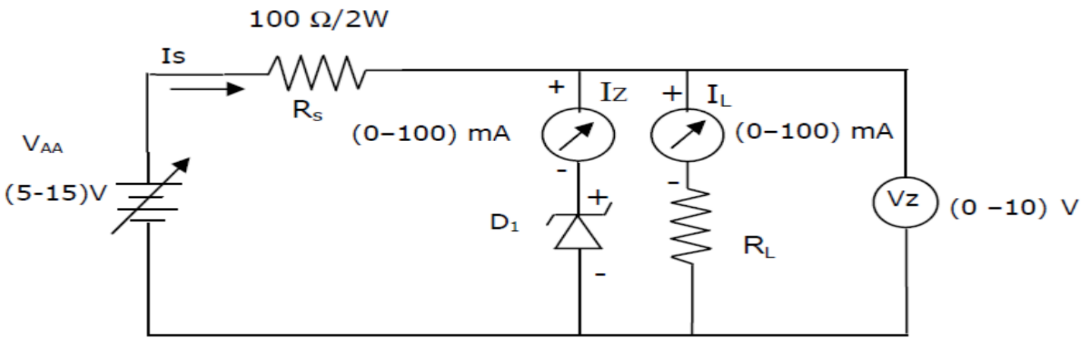
Experiment No. : 05	Voltage regulator using zener Diode

Aim: To study zener diode as voltage regulator, To calculate % line regulation, To calculate % load regulation.

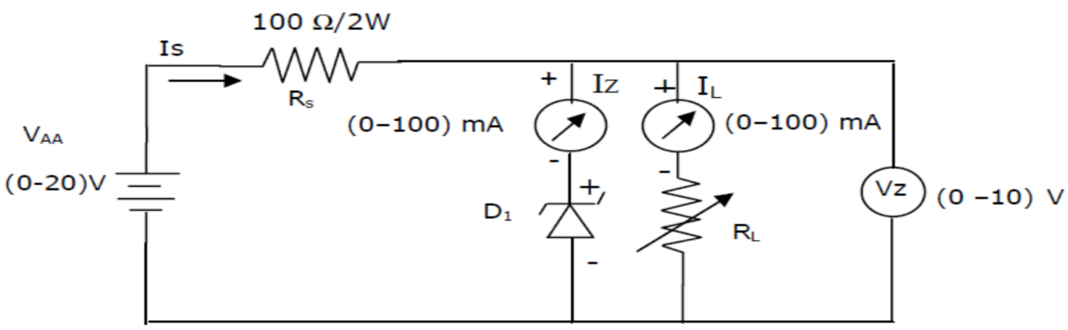
Apparatus: Zener diode voltage regulator trainer, Multi meter.

Sl. No.	Components Details	Specification	Qty
1.	Zener diode voltage regulator trainer	NVIS 6508	1 Nos.
2.	Multimeter	DM-97	2 Nos.
3.	Connecting probes	---	As per required

Circuit diagram:



Line Regulation: Zener Regulator



Load Regulation: Zener Regulator

Theory:

Zener diode is a P-N junction diode specially designed to operate in the reverse biased mode. It is acting as normal diode while forward biasing. It has a particular voltage known as break down voltage, at which the diode break downs while reverse biased. In the case of normal diodes the diode damages at the break down voltage. But Zener diode is specially designed to operate in the reverse breakdown region.

The basic principle of Zener diode is the Zener breakdown. When a diode is heavily doped, it's depletion region will be narrow. When a high reverse voltage is applied across the junction, there will be very strong electric field at the junction. And the electron hole pair generation takes place. Thus heavy current flows. This is known as Zener break down.

So a Zener diode, in a forward biased condition acts as a normal diode. In reverse biased mode, after the break down of junction current through diode increases sharply. But the voltage across it remains constant. This principle is used in voltage regulator using Zener diodes The figure shows the zener voltage regulator, it consists of a current limiting resistor R_S connected in series with the input voltage Vs and zener diode is connected in parallel with the load RL in reverse biased condition. The output voltage is always selected with a breakdown voltage Vz of the diode.

The input source current, $I_S = I_Z + I_L$………….. (1)
The drop across the series resistance, $Rs = V_{in} – V_Z$…….. (2)
And current flowing through it, $Is = (Vin – V_Z) / R_S$ ………….. (3)

From equation (1) and (2), we get, $(V_{in} – V_Z)/R_S = I_Z + I_L$ ………… (4)
Regulation with a varying input voltage (line regulation): It is defined as the change in regulated voltage with respect to variation in line voltage. It is denoted by „LR‟. In this, input voltage varies but load resistance remains constant hence, the load current remains constant. As the input voltage increases, form equation (3) Is also varies accordingly. Therefore, zener current Iz will increase. The extra voltage is dropped across the Rs. Since, increased Iz will still have a constant Vz and Vz is equal to Vout.
The output voltage will remain constant. If there is decrease in Vin, Iz decreases as load current remains constant and voltage drop across Rs is reduced. But even though Iz may change, Vz remains constant hence, output voltage remains constant.

Regulation with the varying load (load regulation): It is defined as change in load voltage with respect to variations in load current. To calculate this regulation, input voltage is constant and output voltage varies due to change in the load resistance value. Consider output voltage is increased due to increasing in the load current. The left side of the equation (4) is constant as input voltage Vin, IS and Rs is constant. Then as load current changes, the zener current Iz will also change but in opposite way such that the sum of Iz and IL will remain constant. Thus, the load current increases, the zener current decreases and sum remain constant. Form reverse bias characteristics even Iz changes, Vz remains same hence, and output voltage remains fairly constant.

Procedure:-

A) Line regulation:

1. Make the connections as shown in figure below.
2. Keep load resistance fixed value; vary DC input voltage from 5V to 15V.
3. Note down output voltage as a load voltage with high line voltage „V_{HL}" and as a load Voltage with low line voltage „V_{LL}".

4. Using formula, % Line Regulation = $(V_{HL}-V_{LL})/ V_{NOM}$ x100, where V_{NOM} = the nominal load voltage under the typical operating conditions. For ex. $V_{NOM} = 9.5 \pm 4.5$ V

B) Load Regulation:

1. For finding load regulation, make connections as shown in figure below.
2. Keep input voltage constant say 10V, vary load resistance value.
3. Note down no load voltage „V_{NL}" for maximum load resistance value and full load voltage „V_{FL}" for minimum load resistance value.

4. Calculate load regulation using, % load regulation = $(V_{NL}-V_{FL})/ V_{FL}$ x100.

Calculations:

% Line Regulation = $(V_{HL}-V_{LL}) / V_{NOM}$ x100 = ------------ %

% voltage regulation = $(V_{NL}-V_{FL})/V_{FL}$x100 =----------%

Conclusion: From the above experiment we are successfully Construct & test the regulator using zener Diode.

Experiment No. : 06	Study Fixed Bias of Transistor with and without emitter resistor

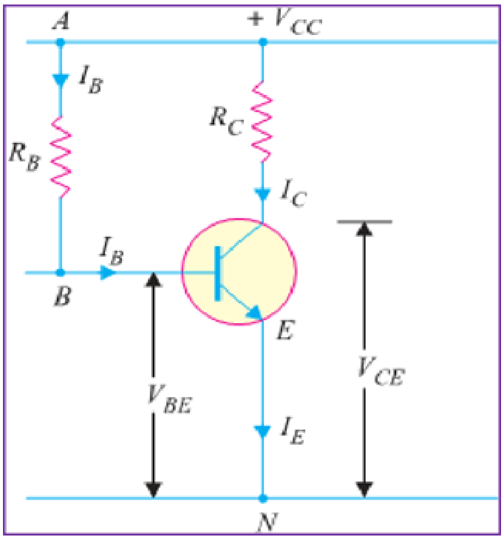
Aim: To construct and Study of the Fixed Bias of Transistor with and without emitter resistor

Component required:

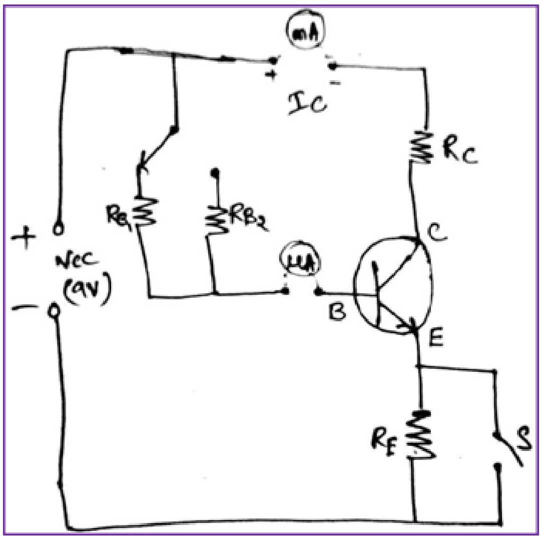
1. Experiment Board
2. Ammeter
3. Voltmeter

Theory:

- For the linear application of transistor it has to bias in active region.
- It can be done by providing a constant potential at bias. Generally the base power supply is same as collector resistor are return to positive supply. $I_B = (V_{CC} - V_{BE}) / R_B$
- Through simple, this is the worst possible way of biasing the transistor in linear region. AB 'β' changes widely temperature and the current 'Q' point is very unstable.
- Therefore fixed Biased Method is never used in linear circuit.
- It is generally used in digital circuit. Where transistor is operated in saturation and circuit up to compensate variation in 'β' increases.
- The ammeter and voltage which reduce the base current & affects for the increase in 'β'.



(Theoretical Circuit Diagram)



(Practical Circuit Diagram)

Procedure:

1. Do the connection as shown in the figure.
2. For both position of switch noted own current.
3. Compare observed I_b with calculated.

Observation:

R_B	Switch	Observed Value	Calculated Value
47K	Open		
	Closed		
100K	Open		
	Closed		

Precautions:

1. Select the multimeter mode correctly to measure voltage and current.
2. Connect voltmeter and Ammeter in correct polarities as shown in the circuit diagram.
3. Do not switch ON the power supply unless you have checked the circuit connections as per the circuit diagram.
4. While measuring output current, short circuit the base side.

Conclusion: The fixed bias circuit not provide stable 'Q' point, So cannot used in linear application. By connecting emitter resistor, the situation improves slightly and practical value of match closely.

Experiment No. : 07	Single Stage and two stage RC coupled common emitter transistor

Aim: To design a Single Stage and two stage RC coupled common emitter transistor (NPN) amplifier circuit and to study its frequency response curve and Voltage gain.

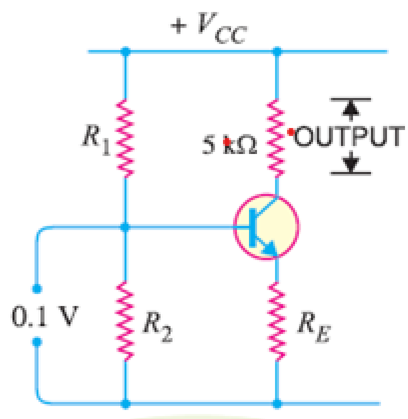
Components Required:

Sl. No.	Name	Specification	Qty.
1	Transistor	CL100	2 nos.
2	Resistor	27KΩ, 4.7+0.22 KΩ, 3.9 KΩ, 1 KΩ, 470 Ω, 560 Ω	2 each
3	Capacitor	1 μF	3 nos.
4	Capacitor	100 μF	2 nos.

Equipment Required:

Name	Range	Qty
Bread Board	-	1
Function Generator	1Hz to 1 MHz	1
Oscilloscope	Dual channel, 0-20 MHz	1
Power Supply	12 V	-
Connecting Wires	-	As required

Theory:



➢ An amplifier is a device that is used to increase the amplitude of signal without changing other parameters of the waveform like frequency or wave shape.

➢ In a transistor amplifier transistor takes the major role in amplification of signal but a transistor can't do this alone. In order to perform the process of amplification the transistor has to be connected with some auxiliary components like resistor, capacitor, battery etc in proper way that is called transistor biasing.

➢ The term biasing refers to the proper flow of zero signal collectors current and the maintenance of proper collector emitter voltage during the passage of signal.

➢ When only one transistor with associated circuitry is used for amplifying a signal, the circuit is known as single stage transistor amplifier.

➢ If there is use of more than one transistor then the circuit is known as multistage transistor amplifier.

- A multistage transistor amplifier is nothing but a combination of a number of single stage amplifiers.
- In a multistage amplifier the output of one stage is feed as input to the next stage and show on.
- The way in which output of one stage is feed as input to the next stage is called coupling.
- The name of the multistage amplifier is given on the basis of coupling it uses.
- There are three types of coupling such as

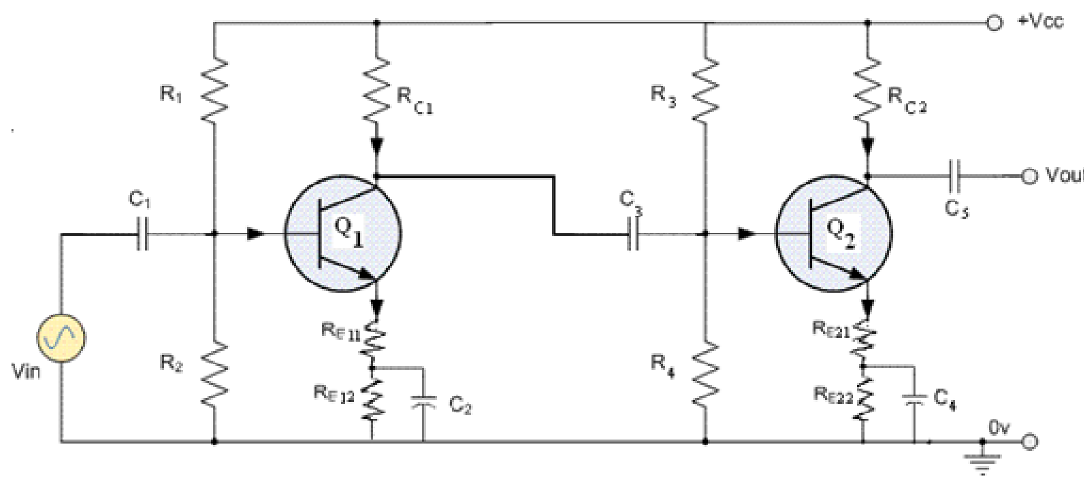
i. Direct coupling (Direct Coupled Amplifier)
ii. RC coupling (RC Coupled Amplifier)
iii. Transformer coupling (Transformer Coupled Amplifier)

Working of single stage transistor amplifier:

- A single stage amplifier has one transistor, bias circuit and other auxiliary components.
- The circuit diagram of single stage amplifier is given below.
- When a weak input signal is given to the base of the transistor as shown in the figure, a small amount of base current flows. Due to the transistor action, a larger current flows in the collector of the transistor.(As the collector current is β times of the base current which means $IC = \beta I_B$).
- Now, as the collector current increases, the voltage drop across the resistor RC also increases because of R_c is quite high (usually 4-10k ohm), which is collected as the output.
- Hence a small input at the base gets amplified as the signal of larger magnitude and strength at the collector output. In this way transistor act as an amplifier.

Working of R-C coupled multistage amplifier:

The resistance-capacitance coupling is, in short termed as RC coupling. This is mostly used coupling technique in amplifiers. The circuit diagram of a two stage R-C coupled amplifier is given below.



➢ The two stage amplifier circuit has two transistors, connected in CE configuration and a common power supply V_{cc} is used. The potential divider network R1 and R2 and the resistor Re form the biasing and stabilization network. The emitter by-pass capacitor Ce offers a low reactance path to the signal.

➢ The resistor RL is used as a load impedance. The input capacitor Cin present at the initial stage of the amplifier couples AC signal to the base of the transistor.

➢ The capacitor CC is the coupling capacitor that connects two stages and prevents DC interference between the stages and controls the shift of operating point.

➢ When an AC input signal is applied to the base of first transistor, it gets amplified and appears at the collector load RL which is then passed through the coupling capacitor CC to the next stage.

➢ This becomes the input of the next stage, whose amplified output again appears across its collector load. Thus the signal is amplified in stage by stage action.

➢ The important point that has to be noted here is that the total gain is less than the product of the gains of individual stages. This is because when a second stage is made to follow the first stage, the effective load resistance of the first stage is reduced due to the shunting effect of the input resistance of the second stage. Hence, in a multistage amplifier, only the gain of the last stage remains unchanged.

Procedure:

- Connect the circuit as per the given circuit diagram.
- In single stage amplifier the input is from function generator is given to the base of the transistor and output is obtained in collector circuit in amplified form.
- Measure the input and input voltage.
- The gain of a single stage amplifier is given by $G = V_{o/p}/V_{i/p}$
- In multistage amplifier the input is given to the base of the transistor, which is the output of the first stage, through a suitable coupling device (capacitor, transformer)
- The output is obtained in the collector circuit of the second stage transistor.
- The gain of the multistage amplifier is given by $G_e=G_1*G_2$
- As the output of first stage is same as input of second stage then G_e=output/input.
- To study the frequency response of the two stage amplifier, vary the input signal frequency in the range 20 Hz – 2 MHz, keeping the input signal amplitude always constant.
- Observe measure and record the output voltage, Vo at the second stage. (You may have to measure Vi and take the ratio Vo/Vi each time in case input fluctuation is too large to hold constant.) Calculate voltage gain for each frequency.
- Plot frequency response curve, i.e. voltage gain in dB versus frequency on a semi-log graph-sheet.
- Estimate the mid-frequency gain and also the lower and higher cut off frequencies and hence the bandwidth.

Tabulation:

Sl No	Type of Amplifier	I/P VOLTAGE	O/P VOLTAGE	Gain = Output / Input
1	Single stage	2	4	2
2	Multistage	2	8	4

Here gain of first stage G_1=4/2=2 ; gain of second stage G_2=8/4=2; Total gain G=G_1*G_2=2*2=4

Sl. No	Frequency, f (kHz)	V_0(pp) (Volt)	Gain, $A_V = V_o\,(pp)/\,V_i\,(pp)$	Gain (dB)
1				
2				
3				
4				

- Plot the frequency response curve and determine the cut-off frequencies, bandwidth & mid-band gain

Precautions:

- Do not switch ON the power supply unless you have checked the circuit connections as per diagram.
- Vary input signal frequency very slowly.
- Connect the electrolytic capacitors carefully.

Conclusion: From the above experiment we have constructed and studied about the single stage and multistage Transistor amplifier. Here we have verified the Product of Individual gain is equal to overall gain also studied and draw the frequency response curve and observed the mid frequency range.

Experiment No. : 08	Study the push pull amplifier

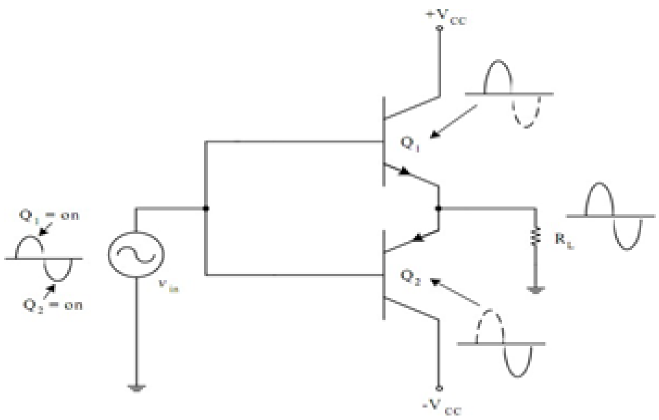
Aim: To design and study the push pull amplifier.

Components required:

Sl. No.	NAME	Qty.
1.	Function Generator having Frequency =1 kHz, Amplitude = 2V, Duty cycle = 50%	1 no.
2.	CRO having Dual channel, 0-20 MHz	1 no.
3.	Regulated Power supply, +12v and -12v	-
4.	PNP and NPN Transistors	1 nos.
5.	Resistance	1 nos.
6.	Connecting wires.	As required

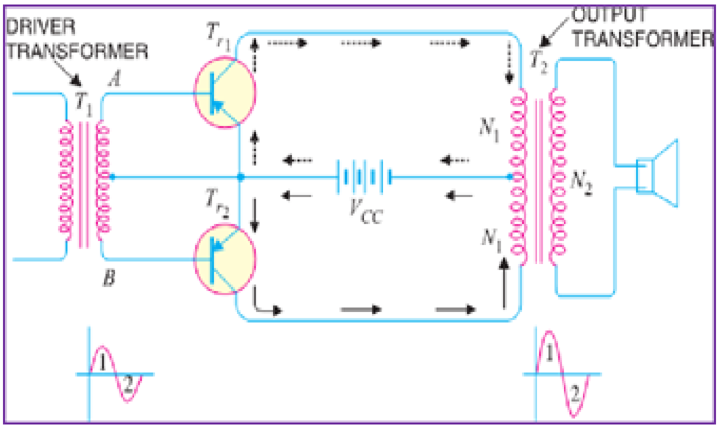
Theory:

A push-pull output is a type of electronic circuit that can drive either a positive or a negative current into a load. Push-pull outputs are present in TTL and CMOS digital logic circuits and in some types of amplifiers, and are usually realized as a complementary pair of transistors, one dissipating or sinking current from the load to ground or a negative power supply, and the other supplying or sourcing current to the load from a positive power supply.



A special configuration of push-pull, though in fact an exception, are the outputs of TTL and related families. The upper transistor is functioning as an active pull-up, in linear mode, while the lower transistor works digitally. For this reason they aren't capable of supplying as much current as they can sink (typically 20 times less). Because of the way these circuits are drawn schematically, with two transistors stacked vertically, normally with a protection diode in between, they are called "totem pole" outputs.

Class B amplifiers only amplify half of the input wave cycle, thus creating a large amount of distortion, but their efficiency is greatly improved and is much better than Class A. Class B has a maximum theoretical efficiency of 78.5% (i.e., π/4). This is because the amplifying element is switched off altogether half of the time, and so cannot dissipate power. A single Class B element is rarely found in



practice, though it has been used for driving the loudspeaker in the early IBM Personal Computers with beeps, and it can be used in RF power amplifier where the distortion levels are less important. However Class C is more commonly used for this.

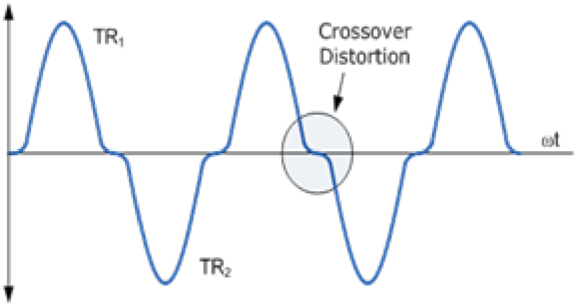
A practical circuit using Class B elements is the push-pull stage, such as the very simplified complementary pair arrangement shown below. Here, complementary or quasi-complementary devices are each used for amplifying the opposite halves of the input signal, which is then recombined at the output.

This arrangement gives excellent efficiency, but can suffer from the drawback that there is a small mismatch in the cross-over region - at the "joins" between the two halves of the signal, as one output device has to take over supplying power exactly as the other finishes. This is called crossover distortion. An improvement is to bias the devices so they are not completely off when they're not in use. This approach is called Class AB operation.

The simple circuit configuration of push pull amplifier is shown in figure 1. Which uses complementary transistors, one of the transistors is a npn and the other is a pnp. The two transistors in a class-B amplifier conduct on alternating half-cycles of the input. The combined half-cycles then provide an output for a full 360^0 of operation.

No Input:

When the transistor is in its quiescent state (no input), both transistors are biased at cutoff.



Positive Input:

During the positive half-cycle of the input signal, Q1 is biased above cutoff, and conduction results through the transistor R_L. During this time, Q2 is still biased at cutoff.

Negative Input:

During the negative half-cycle of the input signal, Q1 is returned to the cutoff state, and Q2 is biased above cutoff. As a result, conduction of Q2 start to built while Q1 remains off.

The combined half-cycles then provide an output for a full 360^0 of operation.

Crossover distortion:

When the signal changes or "crosses-over" from one transistor to the other at the zero voltage point it produces an amount of "distortion" to the output wave shape. This result in a condition that is commonly called Crossover Distortion.

Procedure:

- Connect the circuit as shown in the circuit diagram.
- Give the input signal as specified.
- Switch on the power supply.
- Note down the outputs from the CRO.

Observations:

- Observe the output waveform from CRO.
- Measure the frequency and the voltage of the output waveform in the CRO. Rectified output can be observed.
- Observe the cross over distortion.

Precautions:

- Connections should be verified before clicking run button.
- Crossover distortion should be observed carefully.

Conclusions: From the above experiment we have constructed and studied the Class B Push-pull power amplifier.

Experiment No. : 09	Study Astable, Bistable & Monstable Multivibrator

Aim: Study Multivibrator (Astable, Bistable, Monstable) circuits & draw itswave forms.

Apparatus required:

Sl. No	Name of the Equipment	Specification	Quantity
1	Multvibrator Trainer Kit	-	1no
2	CRO	2 KHz	1 no
3	Function Generator	2 KHz	1 no
4	Connecting Probes	-	As Per Required

Theory:

Multivibrator:

A Multivibrator is a two-stage resistance coupled amplifier with positive feedback from theoutput of one amplifier to the input of the other.

Two transistors are connected in feedback so that one controls the state of the other. Hence the ON and OFF states of the whole circuit, and the time periods for which the transistors are driven into saturation or cut off are controlled by the conditions of the circuit.

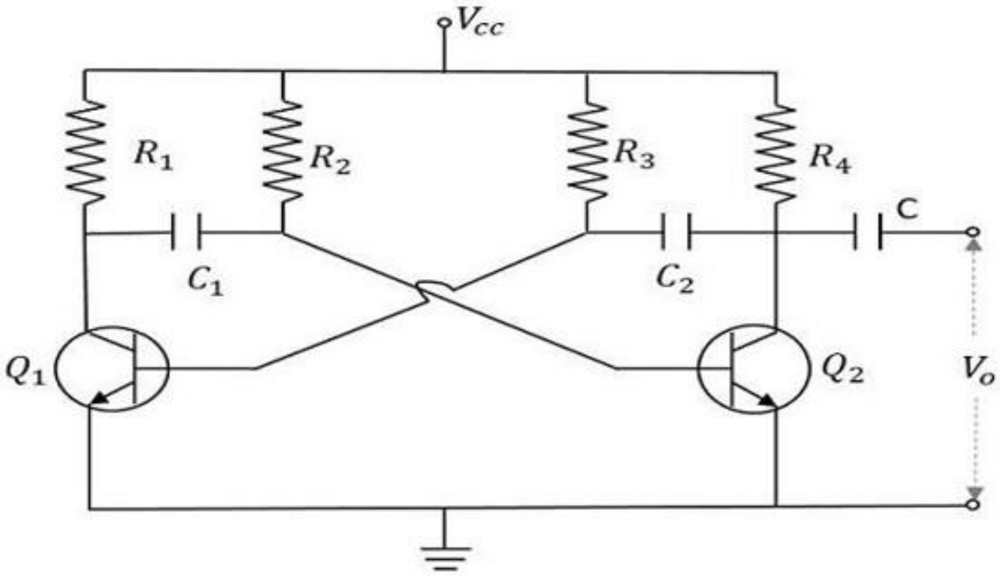
Types of Multivibrators:

1. Astable Multivibrator
2. Bistable Multivibrator
3. Monstable Multivibrator

Astable Multivibrator:

An astable multivibrator has no stable states. Once the multivibrator is ON, it just changes its states on its own after a certain time period which is determined by the R_C time constants. A dc power supply or V_{cc} is given to the circuit for its operation.

Two transistors named Q_1 and Q_2 are connected in feedback to one another. The collector of transistor Q_1 is connected to the base of transistor Q_2 through the capacitor C_1 and vice versa. The emitters of both the transistors are connected to the ground. The collector load resistors R_1 and R_4 and the biasing resistors R_2 and R_3 are of equal values. The capacitors C_1 and C_2 are ofequal values.

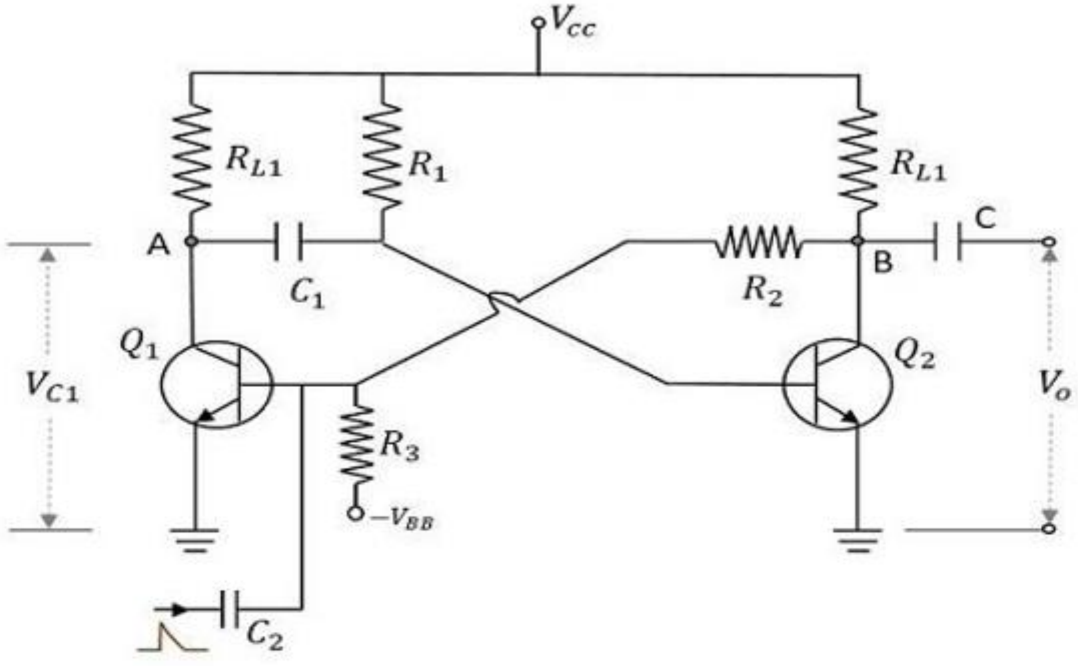


[Circuit Diagram of Astable Multivibrator]

Monostable Multivibrator:

A monostable multivibrator, as the name implies, has only one stable state. When the transistorconducts, the other remains in non-conducting state. A stable state is such a state where the transistor remains without being altered, unless disturbed by some external trigger pulse. As Monostable works on the same principle, it has another name called as One-shot Multivibrator.

Two transistors Q_1 and Q_2 are connected in feedback to one another. The collector of transistor Q_1 is connected to the base of transistor Q_2 through the capacitor C_1. The base Q_1 is connected to the collector of Q_2 through the resistor R_2 and capacitor C. Another dc supply voltage $-V_{BB}$ is given to the base of transistor Q_1 through the resistor R_3. The trigger pulse is given to the base of Q_1 through the capacitor C_2 to change its state. R_{L1} and R_{L2} are the load resistors of Q_1 and Q_2.
One of the transistors, when gets into a stable state, an external trigger pulse is given to change its state. After changing its state, the transistor remains in this quasi-stable state or Meta-stable state for a specific time period, which is determined by the values of RC time constants and gets back to the previous stable state.



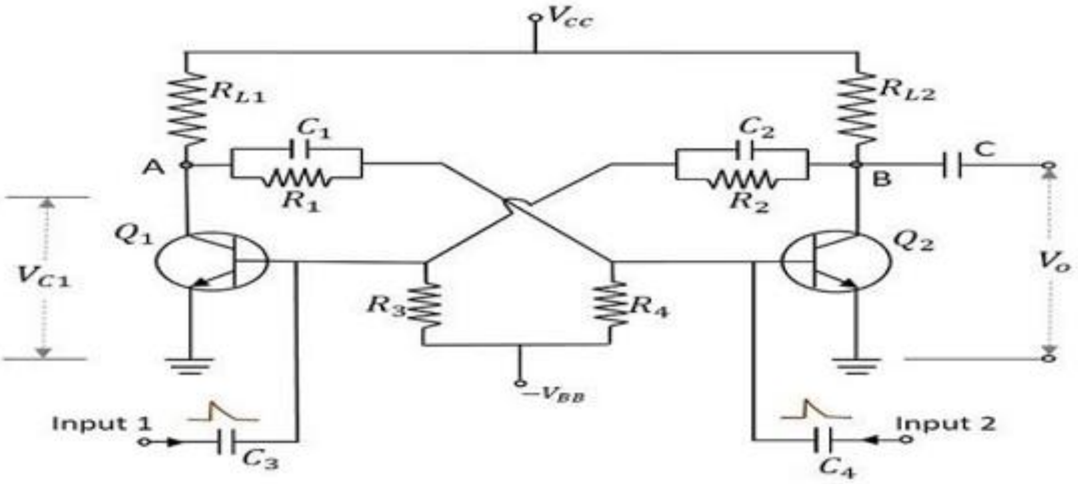
[Circuit Diagram of Monostable Multivibrator]

Bistable Multivibrator:

A Bistable Multivibrator has two stable states. The circuit stays in any one of the two stable states. It continues in that state, unless an external trigger pulse is given. This Multivibrator is also known as Flip-flop. This circuit is simply called as Binary.

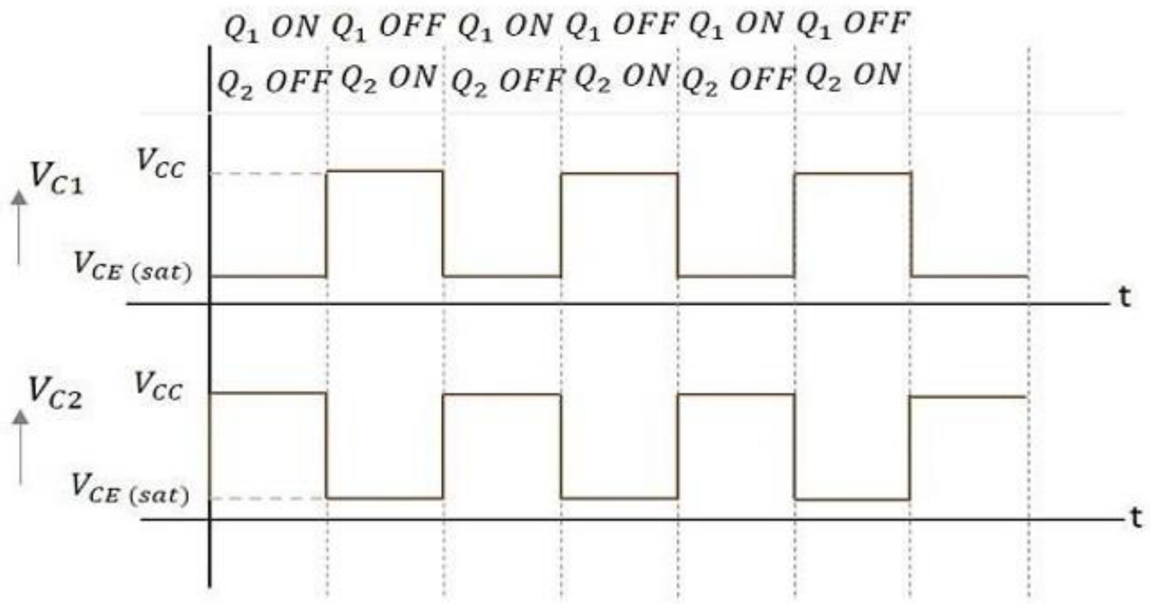
Two similar transistors Q_1 and Q_2 with load resistors R_{L1} and R_{L2} are connected in feedback to one another. The base resistors R_3 and R_4 are joined to a common source $-V_{BB}$. The feedback resistors R_1 and R_2 are shunted by capacitors C_1 and C_2 known as Commutating Capacitors. The transistor Q_1 is given a trigger input at the base through the capacitor C_3 and the transistor Q_2 is given a trigger input at its base through the capacitor C_4.

The capacitors C_1 and C_2 are also known as Speed-up Capacitors, as they reduce the transition time, which means the time taken for the transfer of conduction from one transistor to the other.

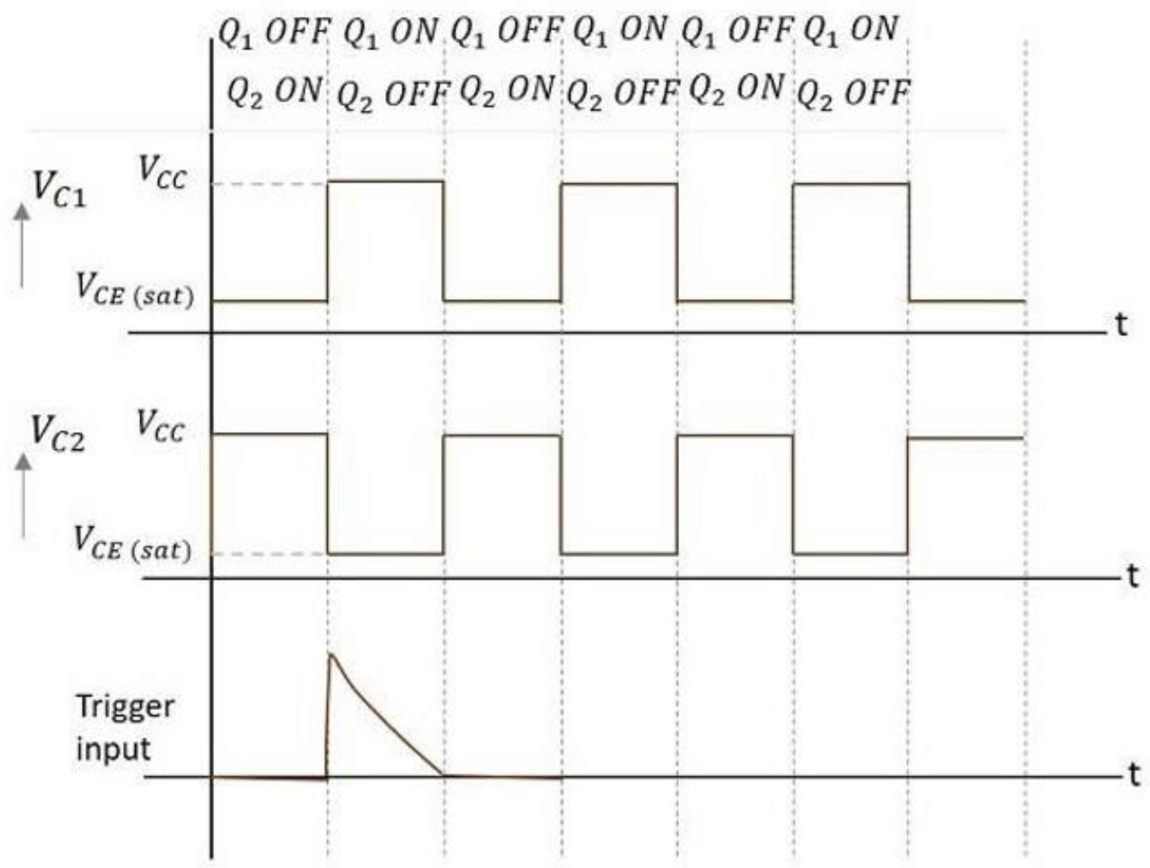


[Circuit Diagram of Bistable Multivibrator]

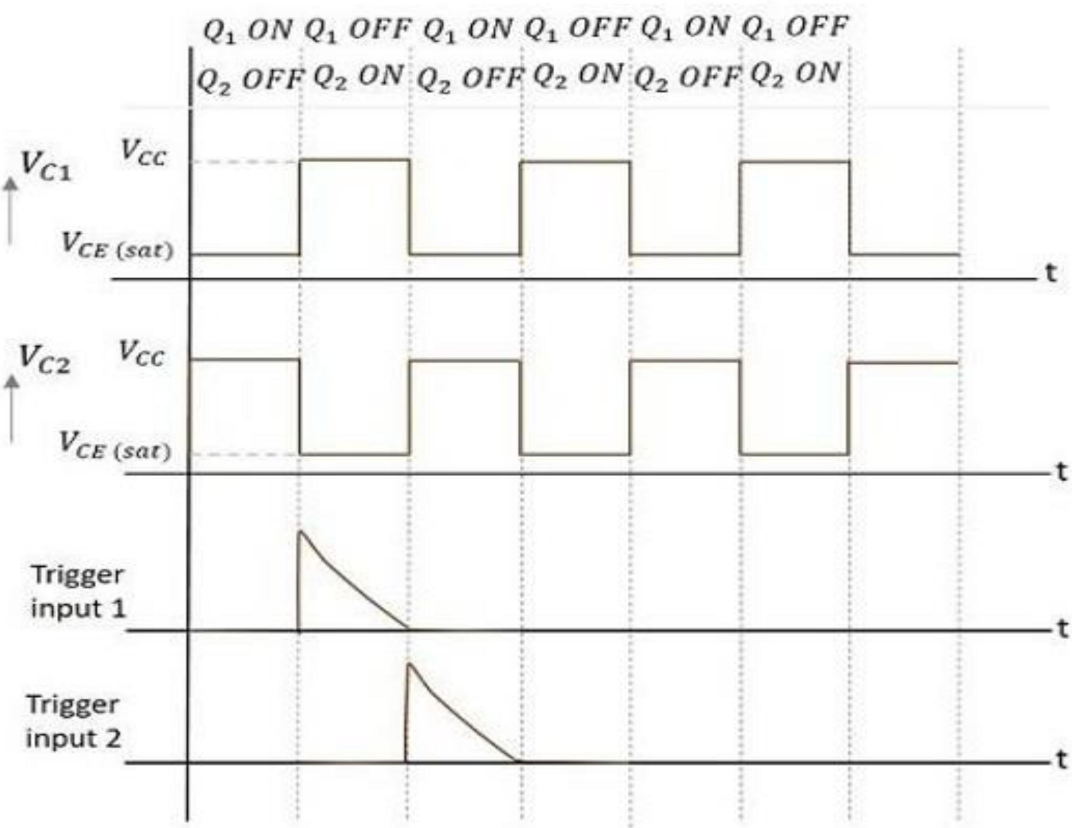
Wave Forms:



[Wave form of Astable Multivibrator]



[Wave form of Bistable Multivibrator]



[Wave form of Monstable Multivibrator]

Procedure:

1. Connect the circuit as per the circuit diagrams shown in connection diagram.
2. Verify the stable states.
3. Apply the square wave of 2v p-p, 1 KHz signal to the trigger circuit from externalfunction generator.
4. Observe the wave forms at base of each transistor simultaneously.
5. Observe the wave forms at collectors of each transistor simultaneously.
6. Note down the parameters carefully.
7. Note down the time period and compare it with theoretical value
8. Plot the wave forms.

Conclusion: From the above experiment, we learnt about the circuit diagram and wave forms of multivibrators.

Experiment No. : 10	Differentiator and Integrator using R-C

Aim: Construct & Test Differentiator and Integrator using R-C circuit.

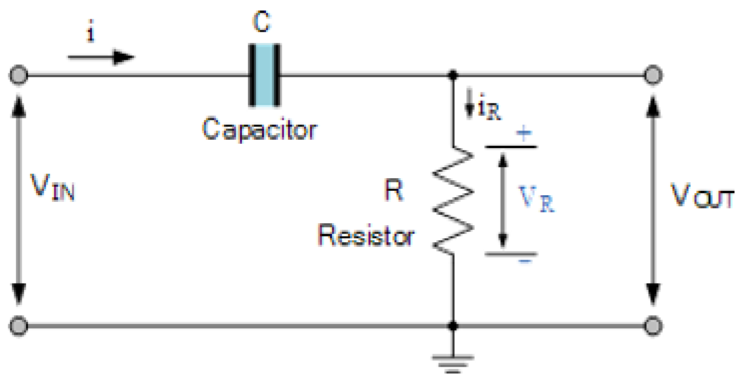
Apparatus required:

Sl. No	Name of the Equipment	Specification	Quantity
1	Capacitor	220 p F / 2.2 µF	1no
2	Resistor	5.6 KΩ	1no
3	Signal Generator	1 Khz	1no
4	CRO	2 Khz	1no
5	Connecting Wires	-	As Per Required

Theory:-

1. Differentiator

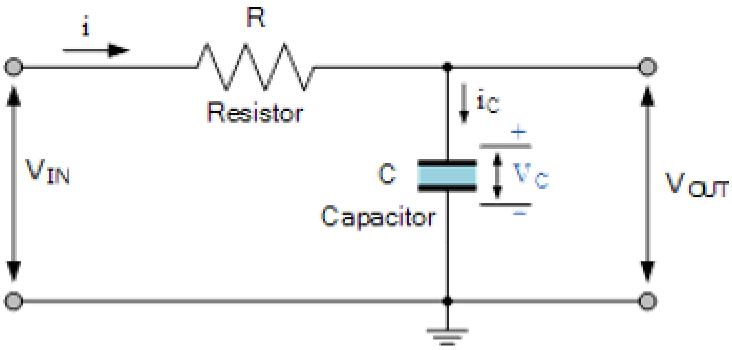
Input voltage is 20 v pp.



Design:

RC<0.0016T; Take R= 5.6 K (To avoid the loading R should be more than ten times the resistance ofsignal generator) So C= 220 p F (T=I ms because the input frequency is I kHz)

2. Integrator



Design

RC>16 T ; T= 1 ms, Take R=5.6 K to avoid loading effect of signal generator.So C=2.2 uF (approximately)

Theory:

A differentiator gives the derivative of input voltage as output. A differentiator using passive components resistors and capacitors is a high pass filter. The circuit is shown .It acts as a differentiatoronly when the time constant is too small. The voltage at output is proportional to the current through the capacitor. The current through the capacitor can be expressed as C dv/dt. The output is taking across theresistor. So output will be RC dv/dt. Thus differentiation of input takes place.

When a square wave is applied at the input, during the positive half cycle capacitor charges. So initially the voltage across the resistor will be the applied voltage. As the capacitor charges, the voltage across resistor decreases.

Now consider the case of integrator. It is a low pass filter. Here the time constant of the circuit should be very large. Here output is taking across the capacitor. As the input square wave is applied, during the positive half cycle the voltage across capacitor increases from zero, to the maximum (peak value of applied voltage). During the negative half cycle, the capacitor starts to discharge and comes to zero. This process repeats for the remaining cycles and a triangular wave is obtained.

Procedure:

1. Set up the differentiator circuit.

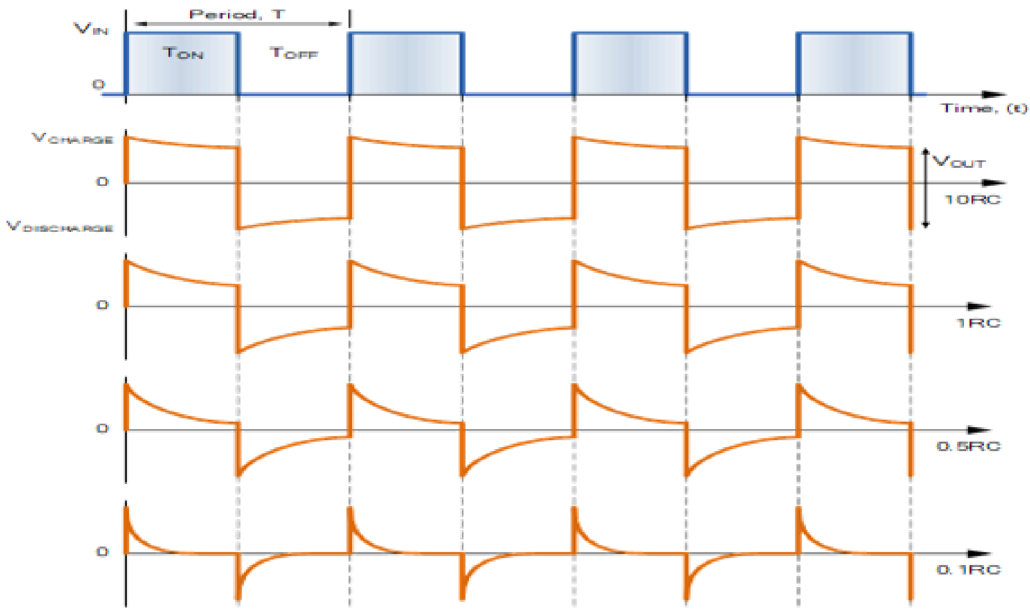
2. Apply the square wave of 5V pp at 1 KHz.

3. Observe the output and plot it.

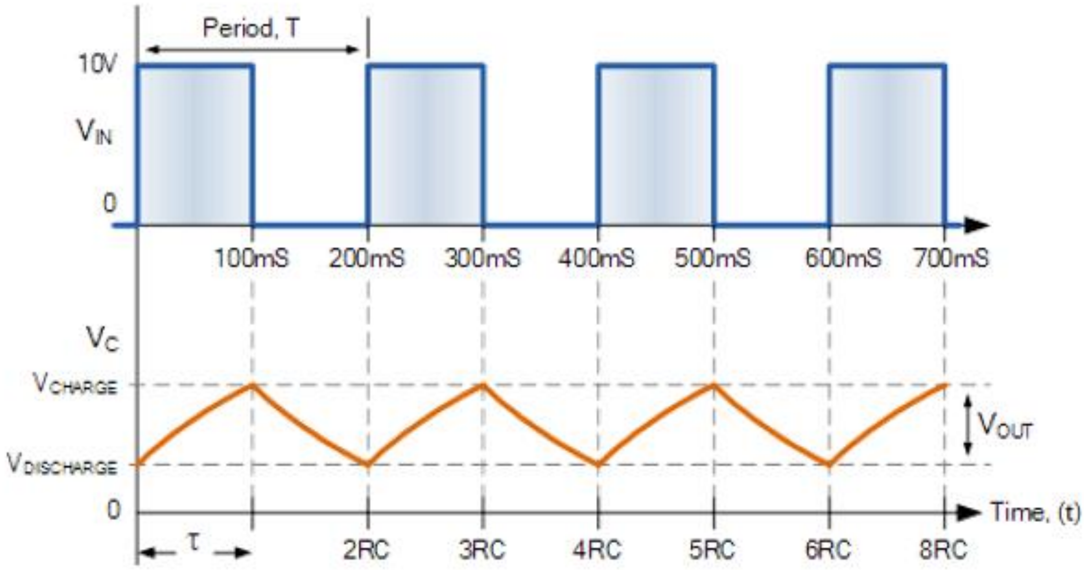
4. Do the above steps for differentiator also.

Wave forms:

1. Differentiator



2. Integrator



Conclusion: From the above experiment, we learnt about the construct & test ofdifferentiator and integrator using R-C circuit.

Experiment No. : 11	Calculate frequency of LC & RC Oscillators

Aim- Construct & calculate the frequency of (i) Hartley Oscillator (ii) Colpitts Oscillator (iii) Wein Bridge Oscillator (iv) R-C Phase shift oscillator and draw wave form & calculate the frequency.

Theory:

The oscillator works on the principle of the oscillation and it is a mechanical or electronic device.The periodic variation between the two things is based on the changes in the energy. The oscillations are used in the watches, radios, metal detectors and in many other devices use the oscillators.

Principle of Oscillators:

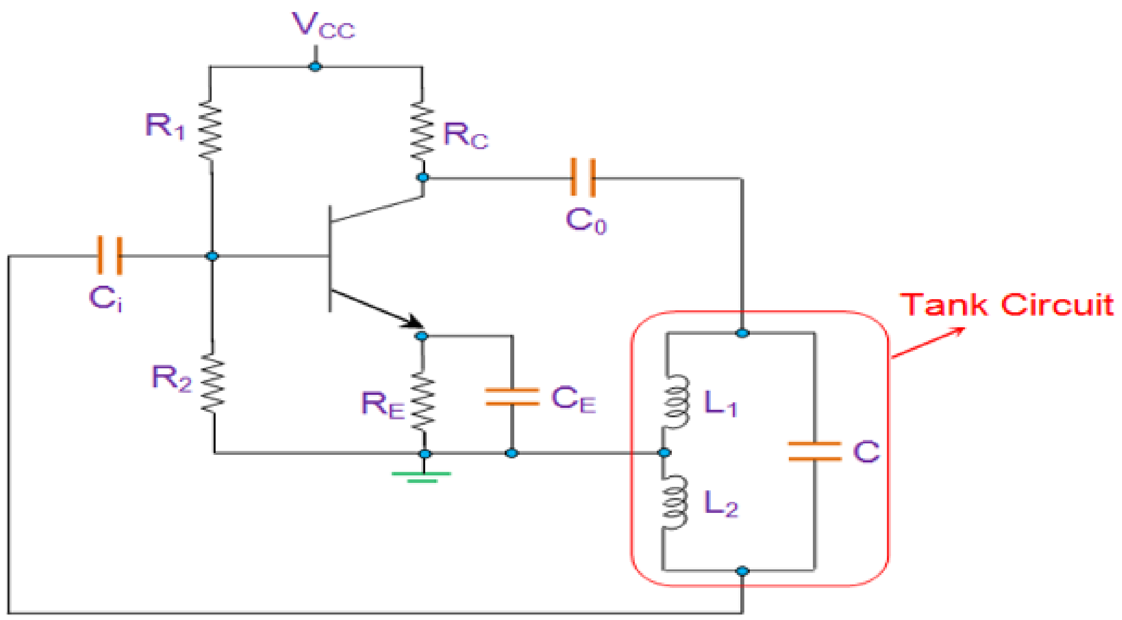
The oscillator converts the direct current from the power supply to an alternating current and they are used in many of the electronic devices. The signals used in the oscillators are a sine wave and the square wave. The some of the examples are the signals are broadcasted by the radio and television transmitter, clocks which are used in the computers and in the video games.

Hartley oscillator:

The Hartley oscillator is an electronic oscillator. The frequency of this oscillation is determined by the tuned circuit. The tuned circuit consists of the capacitor and inductor; hence it is an LC oscillator. In 1915 by American engineer Ralph Hartley has invented this oscillator. The featuresof the Hartley circuit are the tuned circuit consists of a single capacitor in parallel with the two inductors which are in series. From the center connection of the two inductors for oscillation purpose, the feedback signal is taken.

The Hartley oscillator is parallel to the Colpitts apart from that it uses a pair of tapping coils asan alternate of two tapped capacitors. From the below circuit the output voltage is developed across the inductor L1 and the feedback voltages are across the inductor L2. The feedback network is given in the mathematical expression which is given below

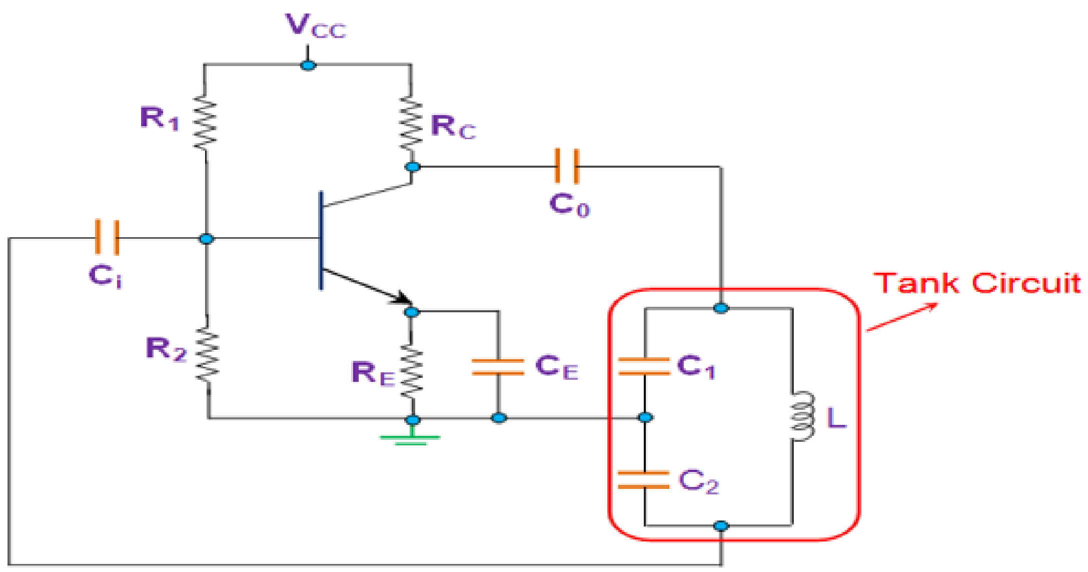
Feedback network = XL2 / XL1 = L 2 / L 1



[Hartley Oscillator]

<u>Colpitts Oscillator:</u>

The Colpitts Oscillator was by American engineering by Edwin H. Colpitts in the year of 1918.This oscillator is a combination of both inductors and capacitor. The features of the Colpitts Oscillator are the feedback for the active devices and they are taken from the voltage divider and made up of two capacitors which are in series across the inductor.



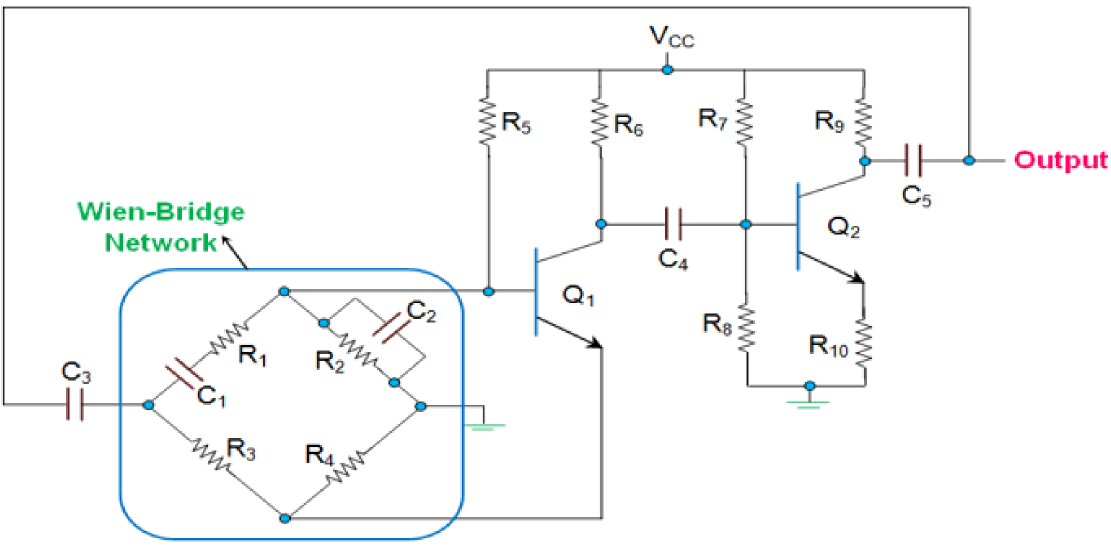
[Colpitts Oscillator]

The Colpitts circuits consist of gain devices such as the bipolar junction, field effect transistor, operational amplifier and vacuum tubes. The output is connected to an input in a feedback loop it has a parallel tuned circuit and it functioned as a band-pass filter is used as a frequency of the oscillator. This oscillator is an electrically dual of the Hartley oscillator hence the feedback signal is taken from the inductive voltage divider it has two coils in the series.

The circuit diagram shows the common base Colpitts circuit. The inductor L and the both the capacitors C1 & C2 are in series with the parallel resonant tank circuit and it gives the frequency of the oscillator. The voltage across the C2 terminal is applied to the base-emitter junction of the transistor to create the feedback oscillations.

Wien Bridge Oscillator:

A Wien-Bridge Oscillator is a type of phase-shift oscillator which is based upon a Wien-Bridge network (Figure 1a) comprising of four arms connected in a bridge fashion. Here two arms are purely resistive while the other two arms are a combination of resistors and capacitors. In particular, one arm has resistor and capacitor connected in series (R_1 and C_1) while the other hasthem in parallel (R_2 and C_2). This indicates that these two arms of the network behave identical to that of high pass filter or low pass filter.



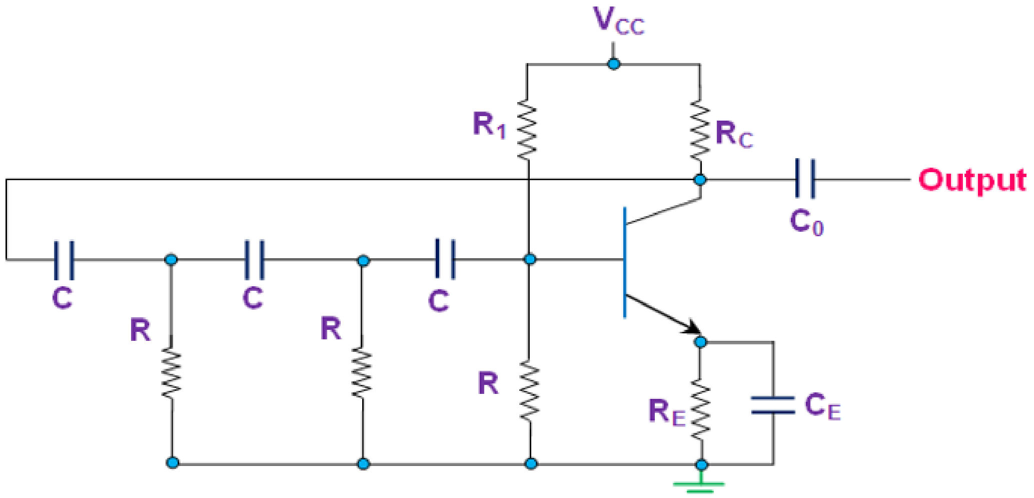
[Wein Bridge Oscillator using BJT]

In this circuit, at high frequencies, the reactance of the capacitors C_1 and C_2 will be much less due to which the voltage V_0 will become zero as R_2 will be shorted. Next, at low frequencies, thereactance of the capacitors C_1 and C_2 will become very high.

However even in this case, the output voltage V_0 will remain at zero only, as the capacitor C_1 would be acting as an open circuit. This kind of behavior exhibited by the Wien-Bridge networkmakes it a lead-lag circuit in the case of low and high frequencies, respectively.

RC phase shift oscillator:

RC phase-shift oscillators use resistor-capacitor (RC) network (Figure 1) to provide the phase-shift required by the feedback signal. They have excellent frequency stability and can yield a pure sine wave for a wide range of loads.



RC Phase-Shift Oscillator using BJT

Ideally a simple RC network is expected to have an output which leads the input by 90^o. However, in reality, the phase-difference will be less than this as the capacitor used in the circuit cannot be ideal. Mathematically the phase angle of the RC network is expressed as

$$\varphi = tan^{-1}\frac{X_C}{R}$$

Where, $X_C = 1/(2\pi fC)$ is the reactance of the capacitor C and R is the resistor. In oscillators, thesekind of RC phase-shift networks, each offering a definite phase-shift can be cascaded so as to satisfy the phase-shift condition led by the Barkhausen Criterion.

Here the collector resistor RC limits the collector current of the transistor, resistors R_1 and R (nearest to the transistor) form the voltage divider network while the emitter resistor R_E improves the stability. Next, the capacitors C_E and C_o are the emitter by-pass capacitor and the output DC decoupling capacitor, respectively. Further, the circuit also shows three RC networks employed in the feedback path.

Procedure:

1. We should take all the components for this experiment.
2. Make the connection as per circuit diagram.
3. Switch ON the kit using ON/OFF toggle switch
4. The input signal is applied with the function generator.
5. Then observe the wave form.
6. Then trace the waveform
7. Calculate the frequency

Conclusion: From the above experiment, we learnt about the different types of oscillator.