ANALOG ELECTRONICS & LINEAR IC

PREPARED BY:SOMA DASH

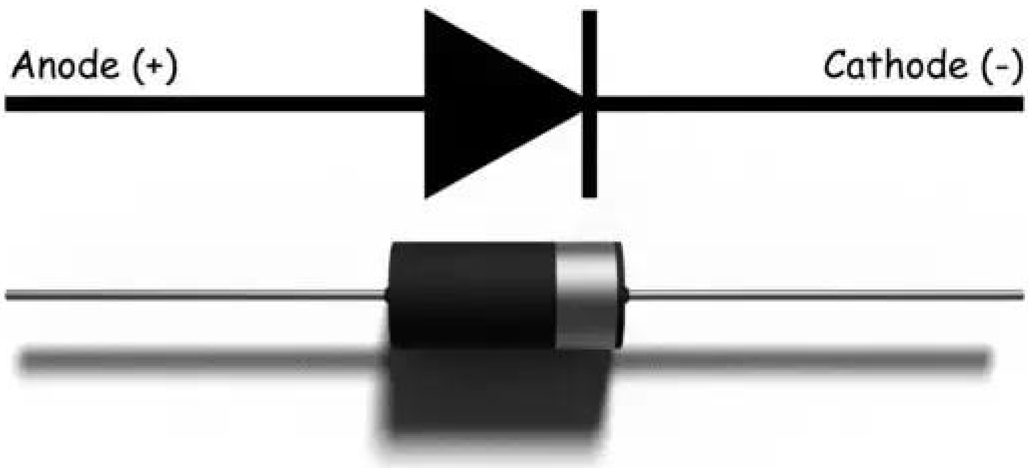
LECTURER IN ELECTRONICS & TELECOMMUNICATION ENGG.

GOVERNMENT POLYTECHNIC, BHUBANESWAR

Unit-1: DIODE, TRANSISTORS AND CIRCUITS

A **diode** is defined as a two-terminal electronic component that only conducts current in one direction (so long as it is operated within a specified voltage level). An ideal diode will have zero resistance in one direction, and infinite resistance in the reverse direction.

Semiconductor diodes are the most common type of diode. These diodes begin conducting electricity only if a certain threshold voltage is present in the forward direction (i.e. the "low resistance" direction). The diode is said to be "***forward biased***" when conducting current in this direction. When connected within a circuit in the reverse direction (i.e. the "high resistance" direction), the diode is said to be "***reverse biased***".
The diode is said to be "***forward biased***" when conducting current in this direction. When connected within a circuit in the reverse direction (i.e. the "high resistance" direction), the diode is said to be "***reverse biased***".



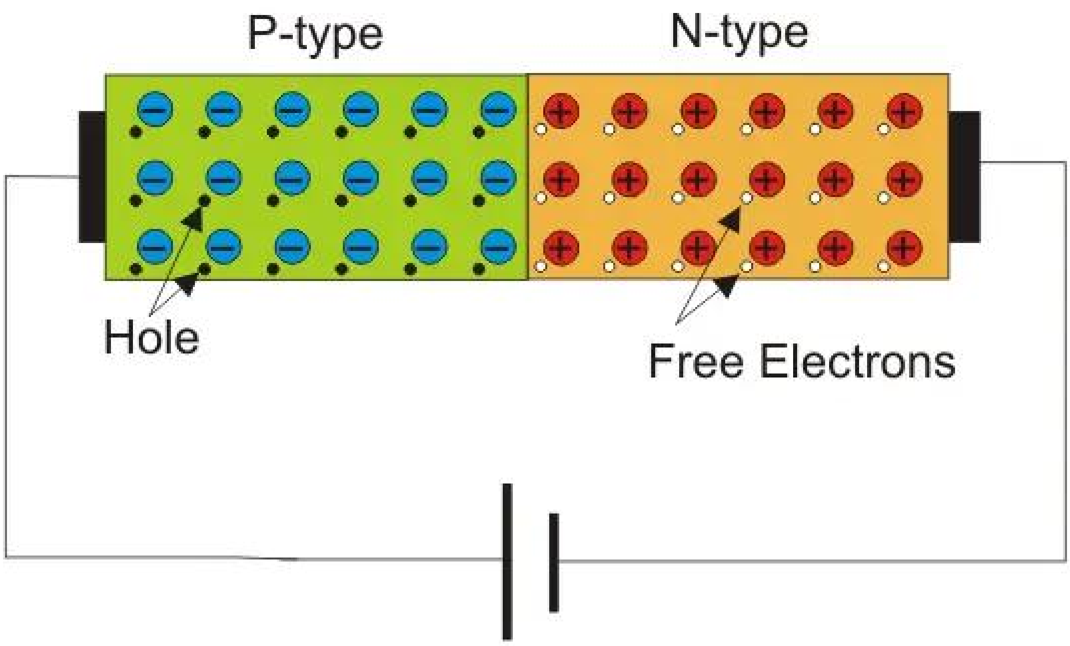
Forward Biased Diode

Now let us see what happens if a positive terminal of a source is connected to the p-type side and the negative terminal of the source is connected to the n-type side of the diode and if we increase the voltage of this source slowly from zero.

In the beginning, there is no current flowing through the diode. This is because although there is an external electrical field applied across the diode, the majority charge carriers still do not get sufficient influence of the external field to cross the depletion region. As it was told that the depletion region acts as a potential barrier against the majority charge carriers.

This potential barrier is called forward potential barrier. The majority charge carriers start crossing the forward potential barrier only when the value of externally applied voltage across the junction is more than the potential of the forward barrier. For silicon diodes, the forward barrier potential is 0.7 volt and for germanium diodes, it is 0.3 volt.

When the externally applied forward voltage across the diode becomes more than the forward barrier potential, the free majority charge carriers start crossing the barrier and contribute the forward diode current. In that situation, the diode would behave as a short-circuited path, and the forward current gets limited by only externally connected resistors to the diode.



Reverse Biased Diode

Now let us see what happens if we connect the negative terminal of the voltage source to the p-type side and the positive terminal of the voltage source to the n-type side of the diode. At that condition, due to electrostatic attraction of the negative potential of the source, the holes in the p-type region would be shifted more away from the junction leaving more uncovered negative ions at the junction.

In the same way, the free electrons in the n-type region would be shifted more away from the junction towards the positive terminal of the voltage source leaving more uncovered positive ions in the junction.

As a result of this phenomenon, the depletion region becomes wider. This condition of a diode is called the reverse biased condition. At that condition, no majority carriers cross the junction, and they instead move away from the junction. In this way, a diode blocks the flow of current when it is reverse biased.

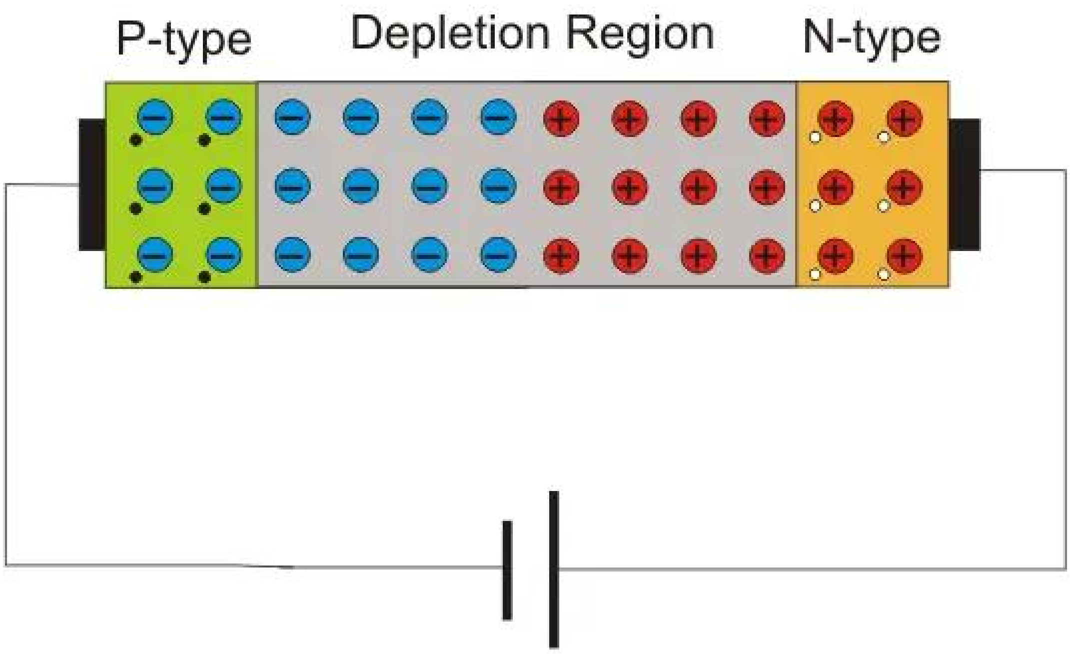
As we already told at the beginning of this article that there are always some free electrons in the p-type semiconductor and some holes in the n-type semiconductor. These opposite charge carriers in a semiconductor are called minority charge carriers.

In the reverse biased condition, the holes find themselves in the n-type side would easily cross the reverse-biased depletion region as the field across the depletion region does not present rather it helps minority charge carriers to cross the depletion region.

As a result, there is a tiny current flowing through the diode from positive to the negative side. The amplitude of this current is very small as the number of minority charge carriers in the diode is very small. This current is called reverse saturation current.

If the reverse voltage across a diode gets increased beyond a safe value, due to higher electrostatic force and due to higher kinetic energy of minority charge carriers colliding with atoms, a number of covalent bonds get broken to contribute a huge number of free electron-hole pairs in the diode and the process is cumulative.

The huge number of such generated charge carriers would contribute a huge reverse current in the diode. If this current is not limited by an external resistance connected to the diode circuit, the diode may permanently be destroyed.



BREAKDOWN IN DIODE

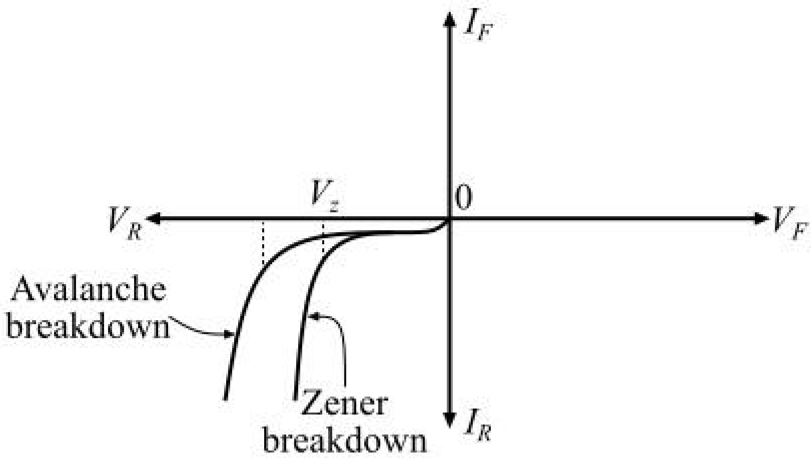
In a semiconductor diode, the term breakdown implies the short circuit of the diode. As we know, the diode allows the flow of electric current in only one direction (forward direction) and blocks the flow of current in the reverse direction. But, when the applied voltage in the reverse direction exceeds a limit (called breakdown voltage), the diode starts conducting in the reverse direction as well. This stage is called the breakdown in the diode.

The following two types of breakdowns take place in a PN-junction semiconductor diode −

1. **Zener Breakdown**

2. **Avalanche Breakdown**

In this , we shall study all the significant differences between Zener breakdown and avalanche breakdown. But, before learning about the differences, we will learn that the Zener breakdown and avalanche breakdown are, which makes it easier to understand the differences between them.



What is Zener Breakdown?

The breakdown of PN junction in a semiconductor diode which occurs due to the flow of free electrons across the junction is called Zener breakdown. The Zener breakdown mainly occurs in the heavy doped diodes that have a thin depletion region.

When a high electric field is applied across the PN junction in reverse direction, the charge carriers start flowing across the junction. As a result of it, a heavy current flows in the reverse direction through the diode. The Zener breakdown is temporary breakdown of the PN junction in the diode, which does not destroy the diode. Therefore, once the reverse voltage is removed, the PN junction regains its original state.

The value of reverse voltage at which the Zener breakdown in the PN junction diode takes place is called the Zener voltage. The Zener breakdown takes place in highly doped PN junction diodes.

What is Avalanche Breakdown?

The type of PN junction breakdown in which the applied electric field in the reverse direction across the diode increases the velocity of charge carriers and these charge carriers create a large number of hole-electron pairs by colliding with the atoms of the semiconductor materials is called the avalanche breakdown.

In case avalanche breakdown, the production of hole-electron pairs is continuous which causes an avalanche of free charge carries. The flow of free charge carriers across the junction results a high reverse current in the diode which permanently destroys the PN junction.

The applied reverse voltage at which the avalanche breakdown occurs is called avalanche breakdown voltage. The avalanche breakdown mainly occurs in such PN junction diodes that have thick depletion region. The avalanche breakdown in the diode is permanent, i.e. it cannot regain its original state.

Difference between Zener Breakdown and Avalanche Breakdown

The following table highlights all the major differences between Zener breakdown and Avalanche breakdown −

Basis of Difference	Zener Breakdown	Avalanche Breakdown
Definition	The breakdown in a PN junction diode occurs due to the tunneling effect (or field ionization) is known as Zener breakdown.	The breakdown in a PN junction diode that occurs due to impact ionization, i.e. increase in the number of free electrons flowing in the reverse direction is called avalanche breakdown.
Breakdown voltage (In reverse direction)	The reverse voltage at which the Zener breakdown occurs is called Zener voltage. It is denoted by VZ and its value typically varies from 5 to 8 volts.	The reverse voltage at which avalanche breakdown takes place is called avalanche breakdown voltage. The value of avalanche breakdown voltage is usually more than 8 volts, which is always greater than Zener voltage.
Depletion region	The Zener breakdown occurs in the PN junction diodes having comparatively thin depletion region.	Avalanche breakdown occurs in such PN junction diodes that have thick depletion region.
Destruction of junction	The Zener breakdown does destroy the PN junction of the diode.	The avalanche breakdown permanently destroys the PN junction of the diode.
Electric field	In the Zener breakdown, the electric field across the junction is strong.	In case of avalanche breakdown, the electric field across the junction is

		relatively weak.
Doping level	Zener breakdown occurs in heavily doped PN junction diodes.	Avalanche breakdown can occur in the diode having any level of doping.
Reverse voltage	Zener breakdown takes place at relatively lower reverse voltage.	Avalanche breakdown occurs at a reverse voltage more than Zener voltage.
Ionization	In case of Zener breakdown, the ionization is due to electric field.	In case of avalanche breakdown, the ionization is due to collision between charge carries and atoms.
Relation between breakdown voltage & temperature	The Zener breakdown voltage is inversely proportional to the temperature.	The avalanche breakdown voltage is directly proportional to the temperature.
Change in voltage after breakdown	In case of Zener breakdown, once the breakdown occurs, the reverse voltage across the diode becomes constant.	The voltage across the diode may vary even after the occurrence of avalanche breakdown.
Temperature coefficient of voltage	Zener breakdown has a negative temperature coefficient of voltage, i.e. Zener voltage decreases with the increase in temperature.	Avalanche breakdown has a positive temperature coefficient of voltage, which means the avalanche breakdown voltage increases when the temperature decreases.

Charge carriers	Zener breakdown generates electrons.	Avalanche breakdown generates holeelectrons pairs.
Effect on junction	In case of Zener breakdown, the PN junction regains its original state.	In case of avalanche breakdown, the PN junction does not regain its original state.

What is Rectifier?

A rectifier is an electronic device that converts an alternating current into a direct current by using one or more P-N junction diodes. A diode behaves as a one-way valve that allows current to flow in a single direction. This process is known as rectification.

A rectifier can take the shape of several physical forms such as solid-state diodes, vacuum tube diodes, mercury-arc valves, silicon-controlled rectifiers, and various other silicon-based semiconductors switches.

Different Types of Rectifier

Rectifiers are mainly classified into two types as:

1. **Uncontrolled Rectifier**
2. **Controlled Rectifier**

Uncontrolled Rectifiers

The type of rectifier whose voltage cannot be controlled is known as an uncontrolled rectifier. Uncontrolled rectifiers are further divided as follows:

- **Half Wave Rectifier**
- **Full Wave Rectifier**

The type of rectifier that converts only the half cycle of the alternating current into the direct current is known as a half-wave rectifier. Likewise, a full-wave rectifier converts both positive and negative half cycles of the AC. An example of this is a bridge rectifier. A bridge rectifier uses 4 diodes that are connected in the form of a Wheatstone bridge.

Controlled Rectifiers

A type of rectifier whose voltage can be varied is known as the controlled rectifier. We use SCRs, MOSFETs and IGBTs to make an uncontrolled rectifier a controlled one. These rectifiers are preferred over their uncontrolled counterparts. There are two types of controlled rectifiers, and they are Half Wave Controlled Rectifier and Full Wave Controlled Rectifier. Half-wave controlled rectifier has the same design as the half-wave uncontrolled rectifier except we replace the diode with an SCR.
Related Links

- **Full Wave Rectifier**
- **Half Wave Rectifier**

What Are Some Applications of Rectifiers?

Some common applications of rectifiers are:

- Rectifiers are used in electric welding to provide polarized voltage
- Half-wave rectifiers are used as a mosquito repellent
- Half-wave rectifiers are used as a signal peak detector in AM radio
- Rectifiers are used in modulation, demodulation and voltage multipliers

What is a Half Wave Rectifier?

A half wave rectifier is the rectifier circuit which converts only half cycle of the alternating current into direct current. The circuit of a typical half-wave rectifier consists of a semiconductor diode, the circuit of half-wave rectifier and the output waveform are shown in Figure-1.

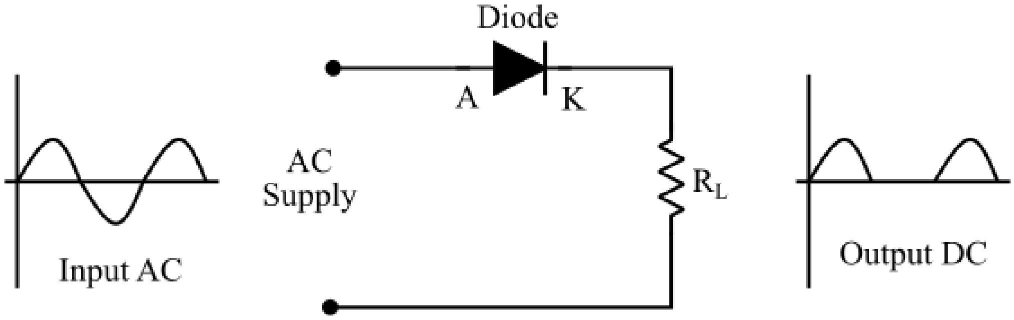


Figure 1 - Half Wave Rectifier

The alternating current is stepped-up or stepped-down to the desired voltage using a transformer and then the transformed AC is fed to the diode. The rectification process of the half wave rectifier can be understood as follows −

During positive half cycle of AC, the diode being forward biased acts as a short circuit and allows the electric current to pass through it. While, during the negative half cycle of AC, the diode gets reverse biased and acts as an open circuit and hence do not conduct. In this way, the electric supply (voltage) at the load terminals will appear only for the positive half cycle of AC. And, during the negative half cycle, there is no voltage across the load terminals. Thus, the alternating current is converted into direct current, which flows only in one direction, but only half cycle of AC.

What is a Full Wave Rectifier?

A full wave rectifier is the one which converts the complete cycle of alternating current into direct current. The full-wave rectifier circuit consists of a center-tapped type step-down transformer and two semiconductor diodes. The anode terminals of the diodes are connected to the secondary winding terminals of the transformer and the cathode terminals of the diodes are connected to a common point. The load resistor is connected between the common terminal and the center-tapping point of the transformer.

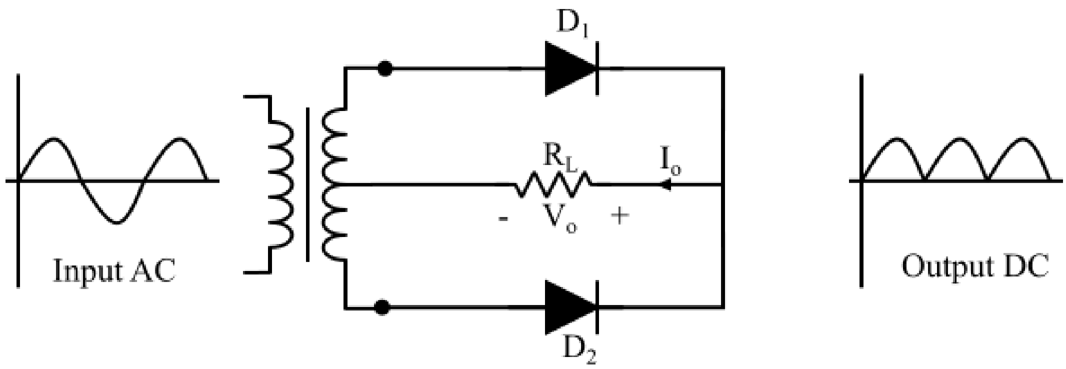
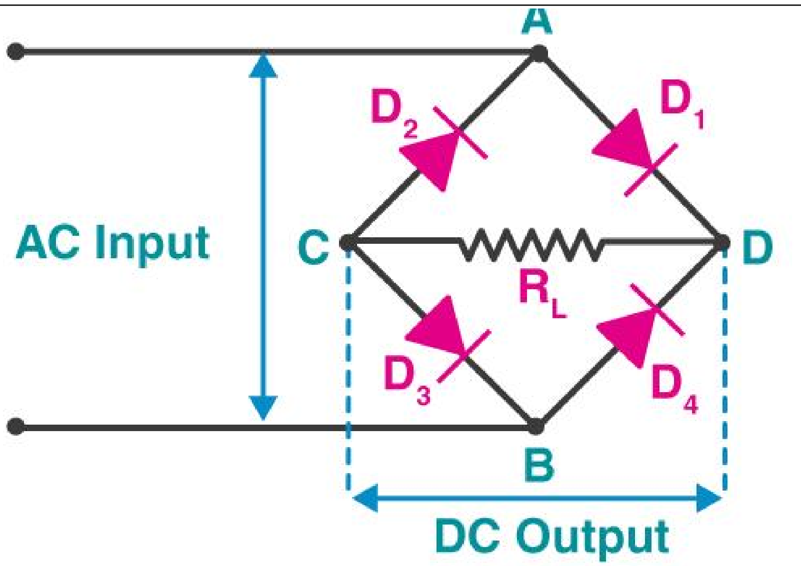


Figure 2 - Full Wave Rectifier

During the positive half cycle of AC, the diode D1 is forward biased and diode D2 is reverse biased. Hence, for the positive half cycle, the diode D1 conducts and current flows through the diode D1 and the load resistor RL. Now, during the negative half cycle of AC, the diode D1 is reverse biased and diode D2 is forward biased, thus only diode D2 conducts for the negative half cycle of AC and the current will flow through the diode D2 and the load resistor RL. The circuit of the full wave rectifier and the output voltage waveform are shown in Figure-2. In this way, a full wave rectifier converts the complete cycle of AC into DC.

Based on circuit configuration, the full-wave rectifier is further classified into two types viz. center-tapped FWR and bridge type FWR.

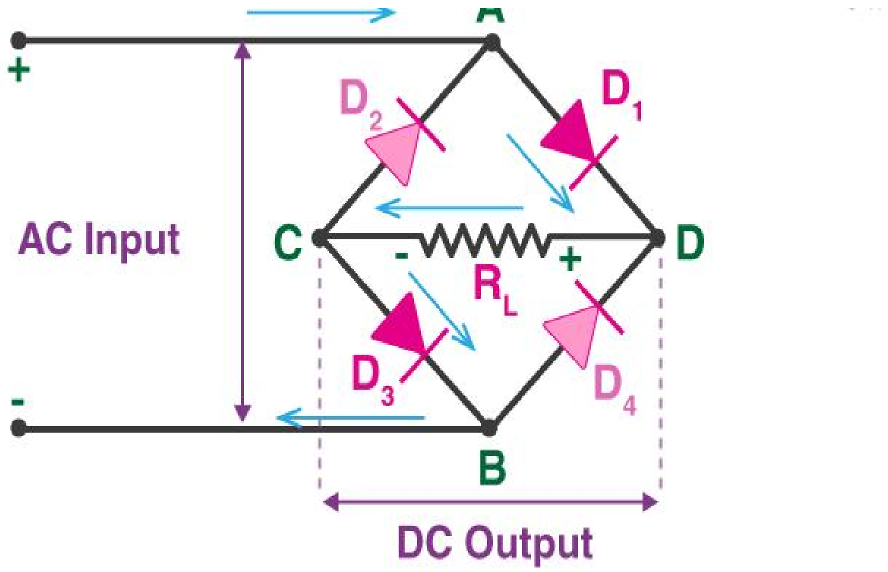
The construction of a bridge rectifier is shown in the figure below. The bridge rectifier circuit is made of four diodes D1, D2, D3, D4, and a load resistor RL. The four diodes are connected in a closed-loop configuration to efficiently convert the alternating current (AC) into Direct Current (DC). The main advantage of this configuration is the absence of the expensive centre-tapped transformer. Therefore, the size and cost are reduced.



Working

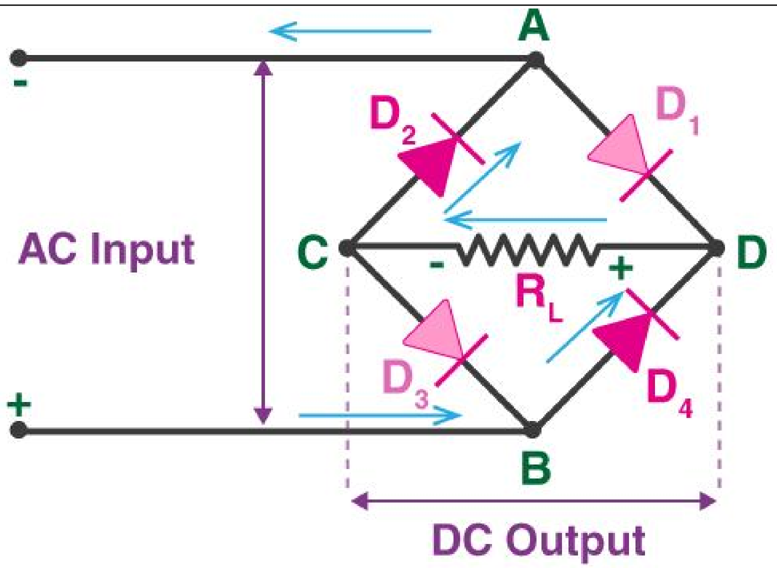
When an AC signal is applied across the bridge rectifier, terminal A becomes positive during the positive half cycle while terminal B becomes negative. This results in diodes D1 and D3 becoming forward biased while D2 and D4 becoming reverse biased.

The current flow during the positive half-cycle is shown in the figure below:



During the negative half-cycle, terminal B becomes positive while terminal A becomes negative. This causes diodes D2 and D4 to become forward biased and diode D1 and D3 to be reverse biased.

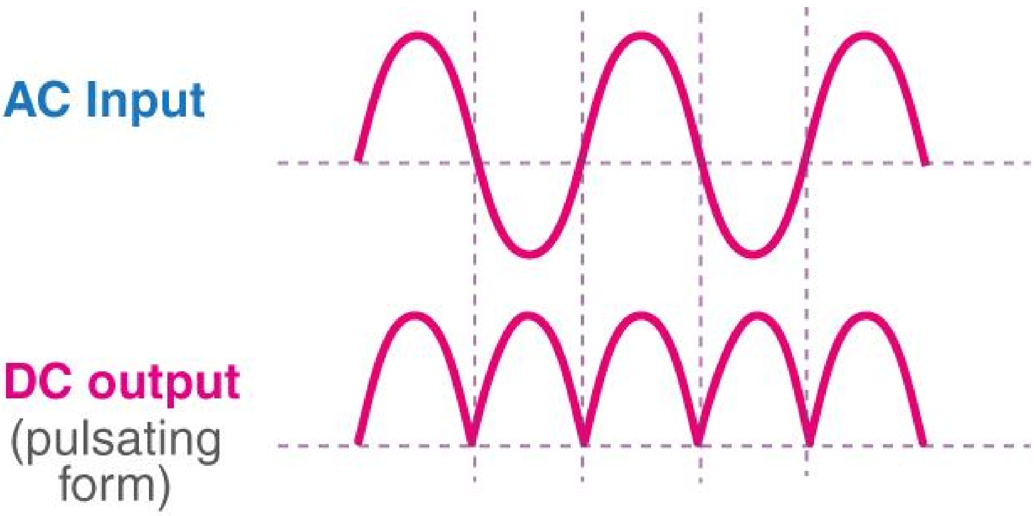
The current flow during the negative half cycle is shown in the figure below:



From the figures given above, we notice that the current flow across load resistor RL is the same during the positive and negative half-cycles. The output DC signal polarity may be either completely positive or negative. In our case, it is completely positive. If the diodes' direction is reversed, we get a complete negative DC voltage.

Thus, a bridge rectifier allows electric current during both positive and negative half cycles of the input AC signal.

The output waveforms of the bridge rectifier are shown in the below figure.



What is a Transistor?

A transistor is a semiconductor device that transfers a weak signal from a low resistance circuit to a high resistance circuit. In simple words, what it means is that it regulates and amplifies electrical signals such as voltage or current.

Transistors are special because they allow you to control how much current flows through a circuit. This can be achieved by controlling the voltage across two of the transistor leads. Each transistor has three leads.

In this article, let us know in detail about PNP transistors and NPN transistors.

What are Bipolar Junction Transistors?

Bipolar Junction Transistors are also known as junction transistors. These were the first type of transistors that were mass-produced in 1947 by Bell Labs. These transistors are a combination of two junction diodes.

The three-layer structure of the junction transistors can contain either of the combinations:

- An n-type semiconductor layer sandwiched between p-type layers forming a p-n-p configuration

or

- A p-type layer between n-type layers forming an n-p-n configuration.

It has two junctions between p-type and n-type semiconductors. BJTs are current-controlled devices which means that a small amount of current flowing through the base of a Bipolar Junction Transistor results in a large current that flows from emitter to collector.

What are Field Effect Transistors (FET)?

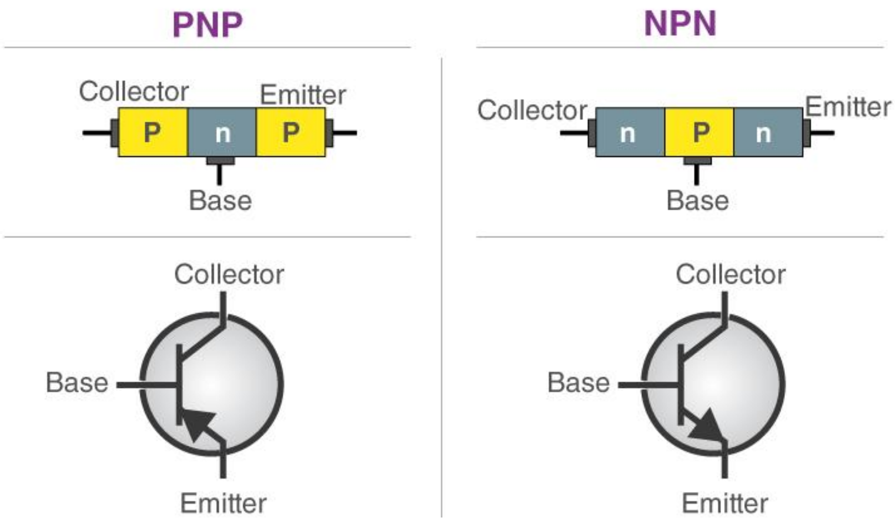
A field-effect transistor uses an electric field to control the flow of current. They have three terminals which are named source, gate, and drain. FETs control the flow of current by the application of a voltage to the gate, which in turn alters the conductivity between the drain and source. FETs are classified into three types as Junction type FETs, MOS (Metal-Oxide-Semiconductor) type FETs, and MES (Metal-Semiconductor) type FETs.

Junction type FETs are used in analog circuits such as those in audio equipment. MOS type FETs are used in digital ICs such as those used in microcomputers. MES type FETs are used for the amplification of microwaves.

Construction of Bipolar Junction Transistor

A transistor is a three-layer semiconductor device in which one type of semiconductor (either P-type or N-type) is sandwiched between two other similar types of semiconductors.

A bipolar junction transistor is formed by three layers of semiconductor materials, if it is a p-n-p transistor, it will have two p-type regions and one n-type region, likewise, if it is an n-p-n transistor, it will have two n-type regions and one p-type region.



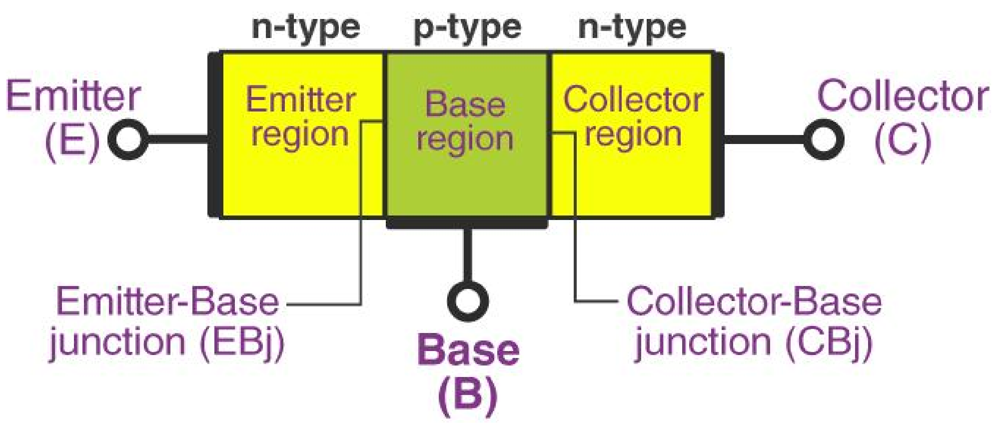
Transistors have three terminals namely emitter, collector and base. We have explained the functionalities of each of these terminals below:

Emitter – In a transistor, the emitter supplies a large section of majority charge carriers. The emitter is always forward biased with respect to the base so that it supplies the majority charge carrier to the base. The emitter of a transistor is heavily doped and moderate in size.

Collector – In a transistor, the section that collects the majority of the charge carrier supplied by the emitter is called a collector. The collector-base junction is always reverse biased. The collector section of the transistor is moderately doped, but larger in size so that it can collect most of the charge carrier supplied by the emitter.

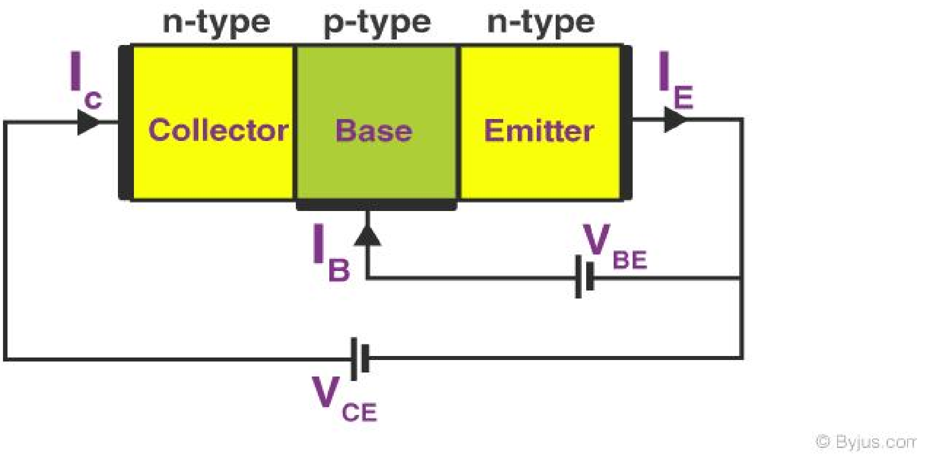
Base – The middle section of the transistor is known as the base. The base forms two circuits, the input circuit with the emitter and the output circuit with the collector. The emitter-base is forward-biased and offers low resistance to the circuit. The collector-base junction is in reverse bias and offers higher resistance to the circuit. The base of a transistor is lightly doped and very thin due to which it offers the majority charge carrier to the base.

The Action of n-p-n Transistor



The n-p-n transistor consists of two n-type semiconductors that sandwich a p-type semiconductor. Here, electrons are the majority charge carriers while holes are the minority charge carriers.

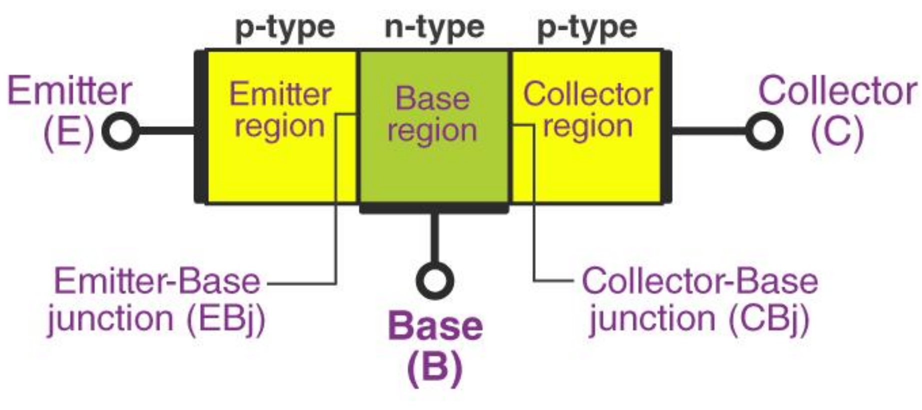
In an n-p-n transistor, the majority of the charge carriers are electrons and holes are the minority charge carriers. A small amount of current at the base terminal causes a large amount of current to flow from emitter to collector. The figure below represents the circuit diagram of the n-p-n transistor:



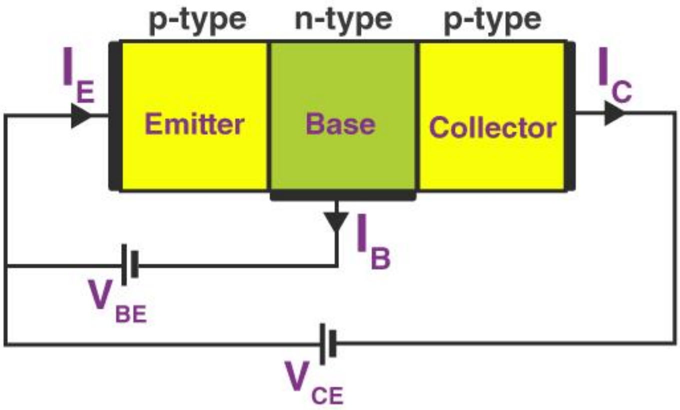
From the circuit diagram of the n-p-n transistor, it is seen that the emitter-base circuit is forward biased while the collector-emitter circuit is reverse biased.

Due to the forward bias, the majority of charge carriers in the emitter are repelled towards the base. The electron-hole recombination is very small in the base region because the base is lightly doped. Most of the electrons cross into the collector region.

The Action of p-n-p Transistor



The p-n-p transistor consists of two p-type semiconductors that sandwich an n-type semiconductor. Here, holes are the majority charge carriers while electrons are the minority charge carriers.
The figure below represents the circuit diagram of the p-n-p transistor:



The emitter-base (VBE) battery connects the p-type emitter which is forward biased whereas the collector-base (VBC) battery connects the p-type collector which is reverse biased.
In this case, the majority charge carriers in emitter are holes which are repelled towards the base. As the base layer is thin, thus only little interaction occurs when electrons and holes combine. Most of the holes reach the collector. The current is carried by holes in p-n-p transistors.

Transistor Modes of Operation

A transistor consists of two junctions which can be biased in different ways. The different working modes of the transistor based on different junction biasing are given in the table below:

Condition	Emitter Junction (EB)	Collector Junction (CB)	Region of Operation
FR	Forward-biased	Reversed-biased	Active
FF	Forward-biased	Forward-biased	Saturation
RR	Reversed-biased	Reversed-biased	Cut-off
RF	Reversed-biased	Forward-biased	Inverted

FR – In this case, the emitter-base junction is forward biased and the collector-base junction is reverse biased. The transistor is in the active region and the collector current depends on the emitter current. The transistor which operates in this region is used for amplification.
FF – In this case, both the junctions are forward biased. The transistor is in saturation and the collector current becomes independent of the base current. The transistors act like a closed switch.
RR – In this case, both the junctions are reverse biased. The emitter does not supply the majority charge carriers to the base because of which the carrier current is not collected by the collector. Thus, transistors, in this case, act like an open switch.

RF – In this case, the emitter-base junction is reverse biased and the collector-base junction forward biased. As the collector is lightly doped compared to the emitter junction it does not supply the majority charge carrier to the base. Hence poor transistor action is achieved.

Characteristics of Transistor

Any two-port network which is analogous to transistor configuration circuits can be analyzed using three types of characteristic curves. They are

- **Input Characteristics**: The curve describes the changes in the values of input current with respect to the values of input voltage, keeping the output voltage constant.
- **Output Characteristics**: The curve is obtained by plotting the output current against output voltage, keeping the input current constant.
- **Current Transfer Characteristics**: This characteristic curve describes the variation of output current in accordance with the input current, keeping the output voltage constant.

Configuration of Transistor

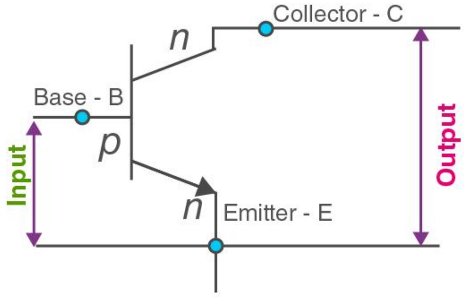
Any transistor circuit can be designed using three types of configuration. Three configurations of the transistor are based on the connection of the transistor terminal. The three types of transistor circuit configurations are:

- **Common Emitter Transistor**
- **Common Base Transistor**
- **Common Collector Transistor(emitter follower).**

Each of these three circuit configurations has its own characteristics curve. Based on the requirement the type will be chosen for the circuit.

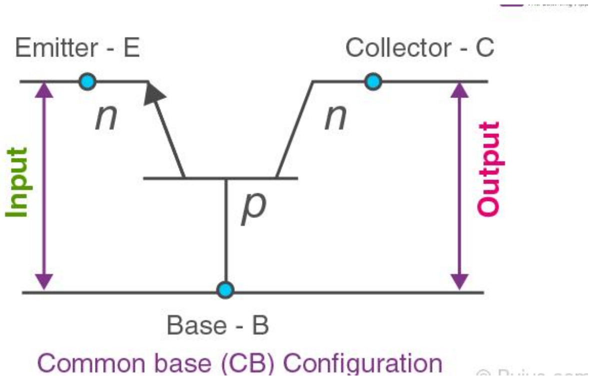
Common Emitter (CE) Configuration of Transistor

The configuration in which the emitter is connected between the collector and base is known as a common emitter configuration.



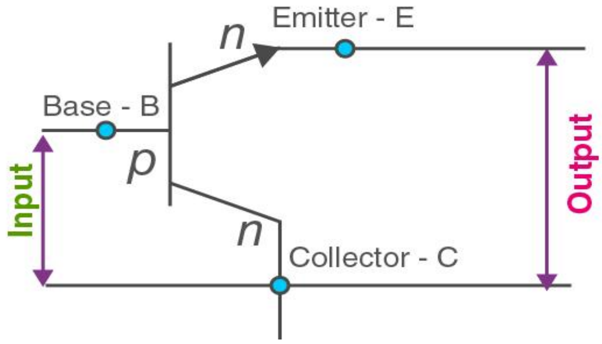
Common Base (CB) Configuration of Transistor

In CB Configuration, the base terminal of the transistor will be connected common between the output and the input terminals.



Common Collector (CC) Configuration of Transistor

In CE Configuration, the Collector terminal of the transistor will be connected common between the output and the input terminals.



Current Amplification Factor (α)

The ratio of change in collector current (ΔIC) to the change in emitter current (ΔIE) when collector voltage VCB is kept constant, is called as Current amplification factor. It is denoted by α.

$$\alpha=\Delta I_C/\Delta I_E \text{ at constant } V_{CB}$$

Base Current Amplification factor (β)

The ratio of change in collector current (ΔIC) to the change in base current (ΔIB) is known as Base Current Amplification Factor. It is denoted by β.

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

Relation between β and α

Let us try to derive the relation between base current amplification factor and emitter current amplification factor.

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

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

$$I_E=I_B+I_C$$

$$\Delta I_E=\Delta I_B+\Delta I_C$$

$$\Delta I_B=\Delta I_E-\Delta I_C$$

We can write

$\beta = \Delta I_C / \Delta I_E - \Delta I_C$

Dividing by ΔI_E

$\beta = (\Delta I_C / \Delta I_E) / (\Delta I_E / \Delta I_E) - (\Delta I_C / \Delta I_E)$

We have

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

Therefore,

$\boldsymbol{\beta = \alpha / 1 - \alpha}$

Current Amplification Factor (γ)

The ratio of change in emitter current (ΔIE) to the change in base current (ΔIB) is known as Current Amplification factor in common collector (CC) configuration. It is denoted by γ.

$\gamma = \Delta I_E / \Delta I_B$

- The current gain in CC configuration is same as in CE configuration.
- The voltage gain in CC configuration is always less than 1.

Relation between γ and α

Let us try to draw some relation between γ and α

$\gamma = \Delta I_E / \Delta I_B$

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

$I_E = I_B + I_C$
$\Delta I_E = \Delta I_B + \Delta I_C$
$\Delta I_B = \Delta I_E - \Delta I_C$

Substituting the value of I_B, we get

$$\gamma = \Delta I_E / \Delta I_E - \Delta I_C$$

Dividing by ΔI_E

$$\gamma = \Delta I_E / \Delta I_E / \ \Delta I_E / \Delta I_E - \Delta I_C / \Delta I_E$$

$$= 1 / 1 - \alpha$$

Introduction - Biasing

The analysis or design of a transistor amplifier requires knowledge of both the dc and ac response of the system.In fact, the amplifier increases the strength of a weak signal by transferring the energy from the applied DC source to the weak input ac signal

The analysis or design of any electronic amplifier therefore has two components:

- The dc portion and
- The ac portion

During the design stage, the choice of parameters for the required dc levels will affect the ac response.

What is biasing circuit?

Once the desired dc current and voltage levels have been identified, a network must be constructed that will establish the desired values of IB, IC and VCE, Such a network is known as biasing circuit. A biasing network has to preferably make use of one power supply to bias both the junctions of the transistor. Purpose of the DC biasing circuit

- To turn the device "ON"
- To place it in operation in the region of its characteristic where the device operates most linearly, i.e. to set up the initial dc values of I_B, I_C, and V_{CE}

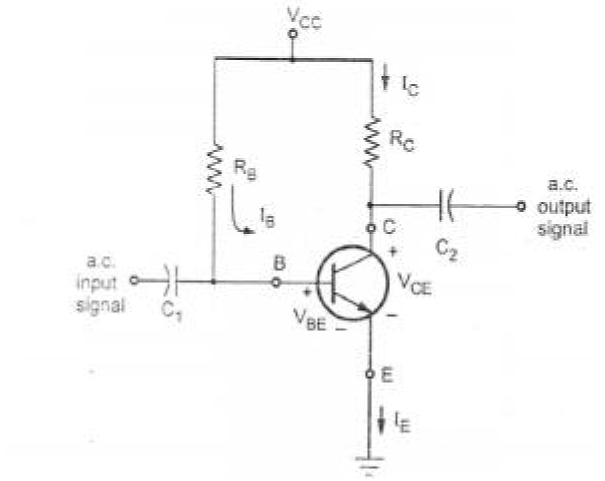
Important basic relationship

- $V_{BE} = 0.7V$
- $I_E = (\beta + 1) I_B = I_C$
- $I_C = \beta I_B$

Biasing circuits:

- Fixed – bias circuit
- Emitter bias
- Voltage divider bias
- DC bias with voltage feedback

Fixed bias



- The simplest transistor dc bias configuration. For dc analysis, open all the capacitance.

DC Analysis

• Applying KVL to the input loop: $V_{CC} = I_B R_B + V_{BE}$

• From the above equation, deriving for I_B, we get, $I_B = [V_{CC} – V_{BE}] / R_B$

• The selection of R_B sets the level of base current for the operating point.

• Applying KVL for the output loop: $V_{CC} = I_C R_C + V_{CE}$

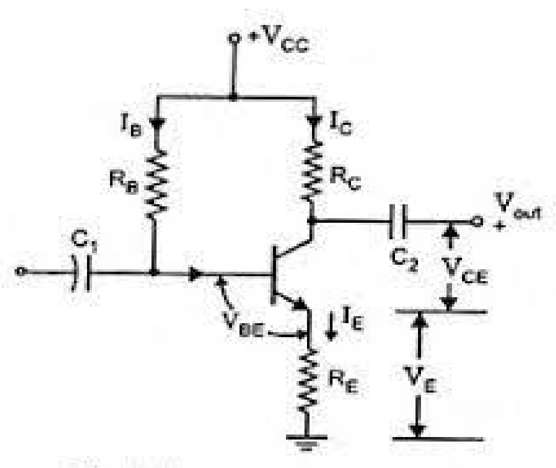
• Thus, $V_{CE} = V_{CC} – I_C R_C$

In circuits where emitter is grounded, $V_{CE} = V_E$ & $V_{BE} = V_B$

Emitter Bias

• It can be shown that, including an emitter resistor in the fixed bias circuit improves the stability of Q point.

• Thus emitter bias is a biasing circuit very similar to fixed bias circuit with an emitter resistor added to it.



Writing KVL around the input loop we get,

$V_{CC} = I_B R_B + V_{BE} + I_E R_E$ ………………………………..(1)

We know that, $I_E = (\beta+1)I_B$ …………………………..(2)

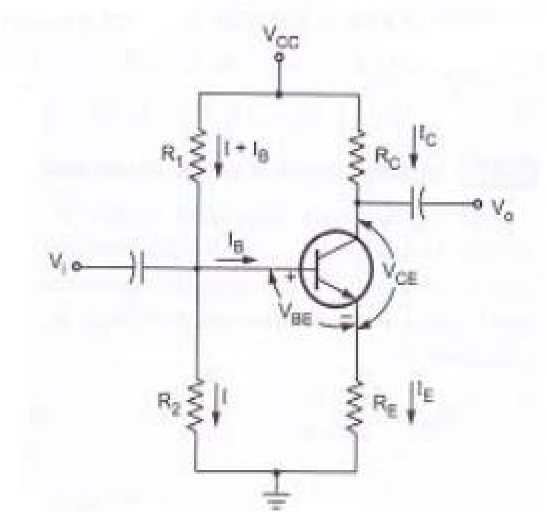
Substituting this in (1), we get,

$V_{CC} = I_BR_B + V_{BE} + (\beta+1)I_BR_E$

$V_{CC} - V_{BE} = I_B(R_B + (\beta+1) R_E)$

Solving for I_B: $I_B = (V_{CC} - V_{BE}) /[(R_B + (\beta+1) R_E)]$

Voltage divider bias



This is the biasing circuit wherein, I_{CQ} and V_{CEQ} are almost independent of β. The level of I_{BQ} will change with β so as to maintain the values of I_{CQ} and V_{CEQ} almost same, thus maintaining the stability of Q point.

Two methods of analyzing a voltage divider bias circuit are:

Exact method – can be applied to any voltage divider circuit Approximate method – direct method, saves time and energy, can be applied in most of the circuits.

Exact method

In this method, the Thevenin equivalent network for the network to the left of the base terminal to be found.

$Rth = R_1 || R_2 = R_1.R_2 / (R_1 + R_2)$

$Eth = V_{R2} = R_2VCC / (R1 + R2)$

Applying KVL

$(E_{th} - V_{BE}) = I_B [Rth +(\beta+1) R_E]I_B = (E_{th} - V_{BE}) / [R_{th} +(\beta + 1) R_E]$

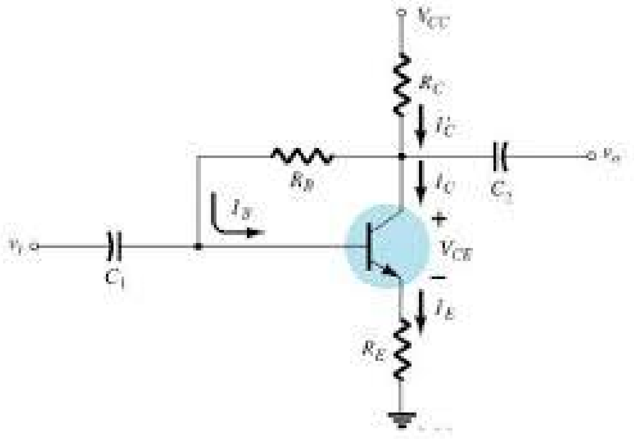
KVL to the output loop:

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

$I_E = I_C$

Thus, $V_{CE} = V_{CC} - I_C (R_C + R_E)$

DC bias with voltage feedback



Applying KVL for Input Loop:

$V_{CC} = I_{C1}R_C + I_BR_B + V_{BE} + I_ER_E$

Substituting for IE as $(\beta+1)I_B$ and solving for I_B, $I_B = (V_{CC} - V_{BE}) / [R_B + \quad (R_C + R_E)]$

Output loop

Neglecting the base current, KVL to the output loop results in,

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

In practical applications, the output of a single state amplifier is usually insufficient, though it is a voltage or power amplifier. Hence they are replaced by Multi-stage transistor amplifiers.

In Multi-stage amplifiers, the output of first stage is coupled to the input of next stage using a coupling device. These coupling devices can usually be a capacitor or a transformer. This process of joining two amplifier stages using a coupling device can be called as Cascading.

The following figure shows a two-stage amplifier connected in cascade.

The overall gain is the product of voltage gain of individual stages.

Purpose of coupling device

The basic purposes of a coupling device are

To transfer the AC from the output of one stage to the input of next stage.

To block the DC to pass from the output of one stage to the input of next stage, which means to isolate the DC conditions.

Types of Coupling

Joining one amplifier stage with the other in cascade, using coupling devices form a Multi-stage amplifier circuit. There are four basic methods of coupling, using these coupling devices such as resistors, capacitors, transformers etc. Let us have an idea about them.

Resistance-Capacitance Coupling

This is the mostly used method of coupling, formed using simple resistor-capacitor combination. The capacitor which allows AC and blocks DC is the main coupling element used here.

The coupling capacitor passes the AC from the output of one stage to the input of its next stage. While blocking the DC components from DC bias voltages to effect the next stage. Let us get into the details of this method of coupling in the coming chapters.

Impedance Coupling

The coupling network that uses inductance and capacitance as coupling elements can be called as Impedance coupling network.

In this impedance coupling method, the impedance of coupling coil depends on its inductance and signal frequency which is jwL. This method is not so popular and is seldom employed.

Transformer Coupling

The coupling method that uses a transformer as the coupling device can be called as Transformer coupling. There is no capacitor used in this method of coupling because the transformer itself conveys the AC component directly to the base of second stage.

The secondary winding of the transformer provides a base return path and hence there is no need of base resistance. This coupling is popular for its efficiency and its impedance matching and hence it is mostly used.

Direct Coupling

If the previous amplifier stage is connected to the next amplifier stage directly, it is called as direct coupling. The individual amplifier stage bias conditions are so designed that the stages can be directly connected without DC isolation.

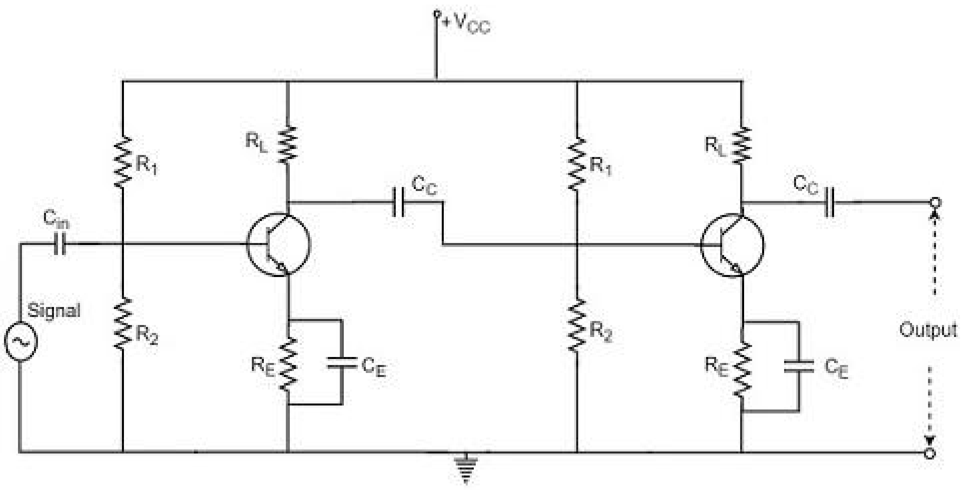
The direct coupling method is mostly used when the load is connected in series, with the output terminal of the active circuit element. For example, head-phones, loud speakers etc.

RC Coupling Amplifier

Construction of a Two-stage RC Coupled Amplifier

The constructional details of a two-stage RC coupled transistor amplifier circuit are as follows. The two stage amplifier circuit has two transistors, connected in CE configuration and a common power supply VCC 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. The figure below shows the circuit diagram of RC coupled amplifier.



Operation of RC Coupled Amplifier

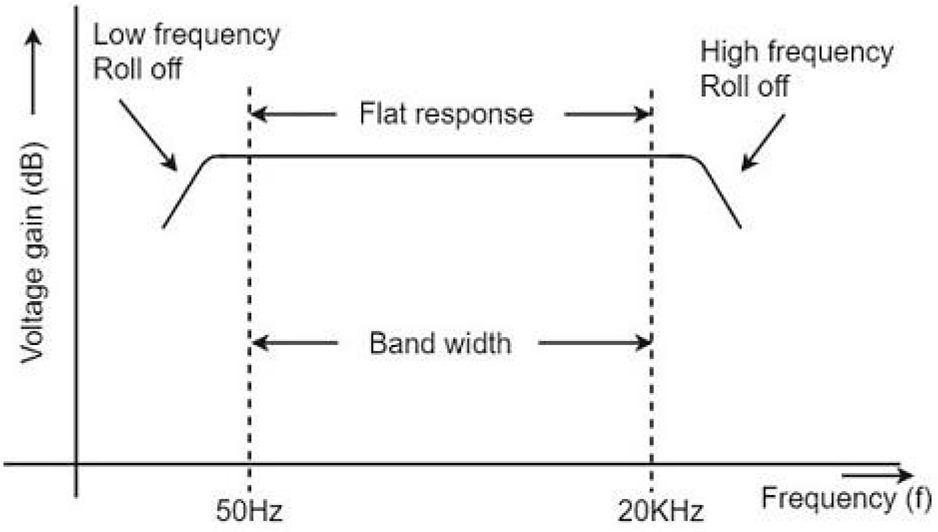
When an AC input signal is applied to the base of first transistor, it gets amplified and appears at the collector load RL which is then passed through the coupling capacitor CC to the next stage. This becomes the input of the next stage, whose amplified output again appears across its collector load.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

As we consider a two stage amplifier here, the output phase is same as input. Because the phase reversal is done two times by the two stage CE configured amplifier circuit.

Frequency Response of RC Coupled Amplifier

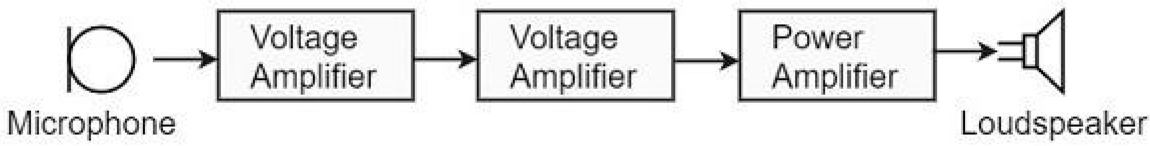
Frequency response curve is a graph that indicates the relationship between voltage gain and function of frequency. The frequency response of a RC coupled amplifier is as shown in the following graph.



From the above graph, it is understood that the frequency rolls off or decreases for the frequencies below 50Hz and for the frequencies above 20 KHz. whereas the voltage gain for the range of frequencies between 50Hz and 20 KHz is constant.

Unit-2: AUDIO POWER AMPLIFIERS

After the audio signal is converted into electrical signal, it has several voltage amplifications done, after which the power amplification of the amplified signal is done just before the loud speaker stage. This is clearly shown in the below figure.



While the voltage amplifier raises the voltage level of the signal, the power amplifier raises the power level of the signal. Besides raising the power level, it can also be said that a power amplifier is a device which converts DC power to AC power and whose action is controlled by the input signal.

The DC power is distributed according to the relation,

DC power input = AC power output + losses

The comparison between voltage and power amplifiers is given below in a tabular form.

S.No	Particular	Voltage Amplifier	Power Amplifier
1	β	High (>100)	Low (5 to 20)
2	RC	High (4-10 KΩ)	Low (5 to 20 Ω)
3	Coupling	Usually R-C coupling	Invariably transformer coupling
4	Input voltage	Low (a few m V)	High (2-4 V)
5	Collector current	Low (≈ 1 mA)	High (> 100 mA)
6	Power output	Low	High

7	Output impendence	High (≈ 12 K Ω)	Low (200 Ω)

<u>Classification Based on Frequencies</u>

Power amplifiers are divided into two categories, based on the frequencies they handle. They are as follows.

Audio Power Amplifiers − The audio power amplifiers raise the power level of signals that have audio frequency range (20 Hz to 20 KHz). They are also known as Small signal power amplifiers.

Radio Power Amplifiers − Radio Power Amplifiers or tuned power amplifiers raise the power level of signals that have radio frequency range (3 KHz to 300 GHz). They are also known as large signal power amplifiers.

Classification Based on Mode of Operation

On the basis of the mode of operation, i.e., the portion of the input cycle during which collector current flows, the power amplifiers may be classified as follows.

Class A Power amplifier − When the collector current flows at all times during the full cycle of signal, the power amplifier is known as class A power amplifier.

Class B Power amplifier − When the collector current flows only during the positive half cycle of the input signal, the power amplifier is known as class B power amplifier.

Class C Power amplifier − When the collector current flows for less than half cycle of the input signal, the power amplifier is known as class C power amplifier.

There forms another amplifier called Class AB amplifier, if we combine the class A and class B amplifiers so as to utilize the advantages of both.

The primary objective of a power amplifier is to obtain maximum output power. In order to achieve this, the important factors to be considered are **collector efficiency**, **power dissipation capability** and **distortion.**

Collector efficiency

The collector efficiency is defined as

η=averagea.c power output/average d.c power input to transistor

For example, if the battery supplies 15W and AC output power is 3W. Then the transistor efficiency will be 20%.

Power dissipation capability

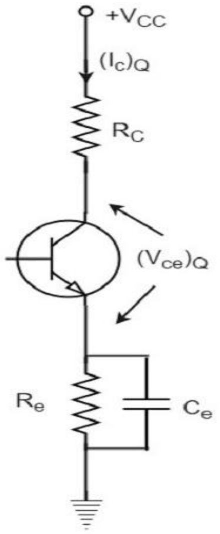
Power dissipation capability can be defined as the ability of a power transistor to dissipate the heat developed in it. Metal cases called heat sinks are used in order to dissipate the heat produced in power transistors.

Distortion

Distortion is defined as the change of output wave shape from the input wave shape of the amplifier. An amplifier that has lesser distortion, produces a better output and hence considered efficient.

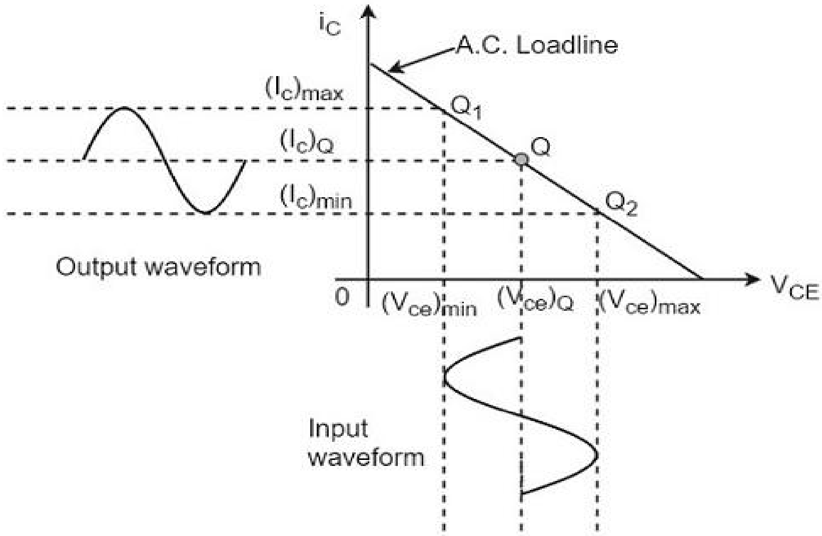
Class A power amplifier

A Class A power amplifier is one in which the output current flows for the entire cycle of the AC input supply. Hence the complete signal present at the input is amplified at the output. The following figure shows the circuit diagram for Class A Power amplifier.



From the above figure, it can be observed that the transformer is present at the collector as a load. The use of transformer permits the impedance matching, resulting in the transference of maximum power to the load e.g. loud speaker.

The operating point of this amplifier is present in the linear region. It is so selected that the current flows for the entire ac input cycle. The below figure explains the selection of operating point.



The output characteristics with operating point Q is shown in the figure above. Here (Ic)Q and (Vce)Q represent no signal collector current and voltage between collector and emitter respectively. When signal is applied, the Q-point shifts to Q1 and Q2. The output current increases to (Ic)max and decreases to (Ic)min. Similarly, the collector-emitter voltage increases to (Vce)max and decreases to (Vce)min.

The overall efficiency of the amplifier circuit is given by

$$=(P_O)ac/(Pin)dc$$

Collector Efficiency

The collector efficiency of the transistor is defined as

$$=(P_O)ac/(Ptr)dc$$

Expression for overall efficiency

$$(P_O)ac=Vrms\times Irms$$

$$\mathbf{(\eta)_{overall}=[(Vce)max-(Vce)min]\times[(I_C)max-(I_C)min]/8\times V_{CC}(I_C)_Q}$$

Advantages of Class A Amplifiers

The advantages of Class A power amplifier are as follows −

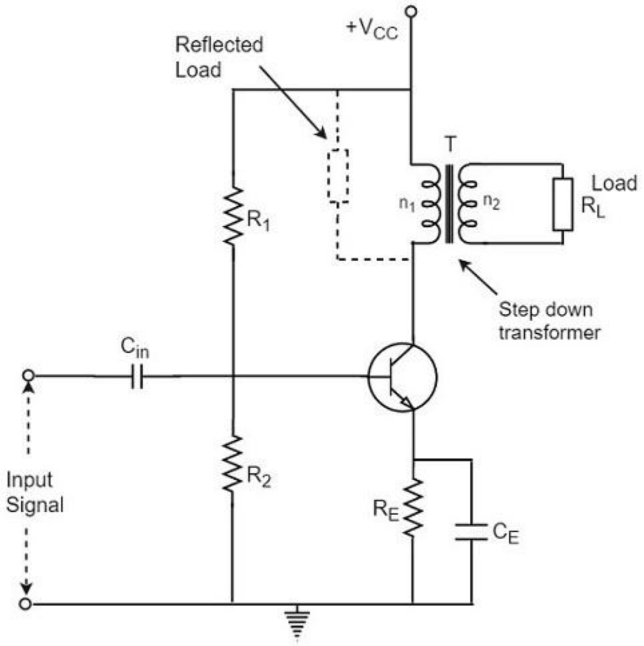
- The current flows for complete input cycle
- It can amplify small signals
- The output is same as input
- No distortion is present

Disadvantages of Class A Amplifiers

The disadvantages of Class A power amplifier are as follows −

- Low power output
- Low collector efficiency

The construction of class A power amplifier can be understood with the help of below figure. This is similar to the normal amplifier circuit but connected with a transformer in the collector load.



Here R1 and R2 provide potential divider arrangement. The resistor Re provides stabilization, Ce is the bypass capacitor and Re to prevent a.c. voltage. The transformer used here is a step-down transformer.

The high impedance primary of the transformer is connected to the high impedance collector circuit. The low impedance secondary is connected to the load (generally loud speaker).

Circuit Operation

If the peak value of the collector current due to signal is equal to zero signal collector current, then the maximum a.c. power output is obtained. So, in order to achieve complete amplification, the operating point should lie at the center of the load line.

The operating point obviously varies when the signal is applied. The collector voltage varies in opposite phase to the collector current. The variation of collector voltage appears across the primary of the transformer.

Circuit Analysis

The power loss in the primary is assumed to be negligible, as its resistance is very small.

The input power under dc condition will be

$(Pin)dc=(Ptr)dc=V_{CC}\times(I_C)_Q$

Under maximum capacity of class A amplifier, voltage swings from (Vce)max to zero and current from (Ic)max to zero.

Hence

$Vrms=1/\sqrt{2}[(Vce)max-(Vce)min/2]=1/\sqrt{2}[(Vce)max/2]=2V_{CC}/2\sqrt{2}$

$=V_{CC}/\sqrt{2}$

$Irms=1/\sqrt{2}[(I_C)max-(I_C)min/2]=1/\sqrt{2}[(I_C)max/2]=2(I_C)_Q/2\sqrt{2}$

$=(I_C)_Q/\sqrt{2}$

Therefore,

$(P_O)ac=Vrms\times Irms=V_{CC}/\sqrt{2}\times(I_C)_Q/\sqrt{2}=V_{CC}\times(I_C)_Q/2$

Therefore,

Collector Efficiency = $(P_O)ac(Ptr)dc$

$(\eta)collector=V_{CC}\times(I_C)_Q/2\times V_{CC}\times(I_C)_Q=1/2\times100=50\%$

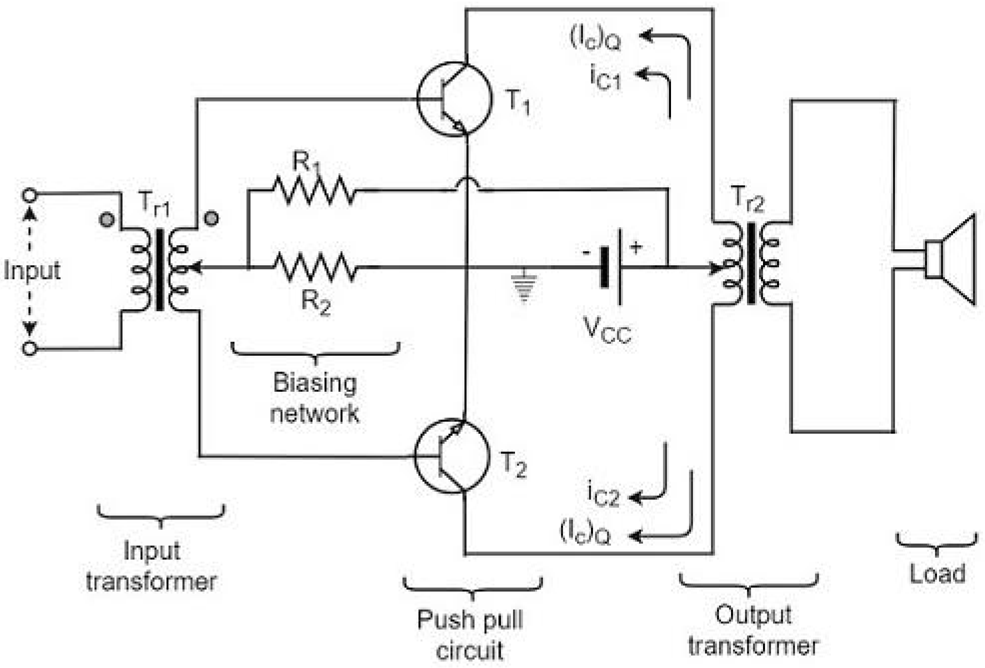
The efficiency of a class A power amplifier is nearly 30% whereas it has got improved to 50% by using the transformer coupled class A power amplifier.

It is possible to obtain greater power output and efficiency than that of the Class A amplifier by using a combinational transistor pair called as **Push-Pull configuration**.

In this circuit, we use two complementary transistors in the output stage with one transistor being an NPN or N-channel type while the other transistor is a PNP or P-channel (the complement) type connected in order to operate them like **PUSH a transistor to ON** and **PULL another transistor to OFF** at the same time. This push-pull configuration can be made in class A, class B, class C or class AB amplifiers.

Construction of Push-Pull Class A Power Amplifier

The construction of the class A power amplifier circuit in push-pull configuration is shown as in the figure below. This arrangement mainly reduces the harmonic distortion introduced by the non-linearity of the transfer characteristics of a single transistor amplifier.



In Push-pull arrangement, the two identical transistors T1 and T2 have their emitter terminals shorted. The input signal is applied to the transistors through the transformer Tr1 which provides opposite polarity signals to both the transistor bases. The collectors of both the transistors are connected to the primary of output transformer Tr2. Both the transformers are center tapped. The VCC supply is provided to the collectors of both the transistors through the primary of the output transformer.

The resistors R1 and R2 provide the biasing arrangement. The load is generally a loudspeaker which is connected across the secondary of the output transformer. The turns ratio of the output transformer is chosen in such a way that the load is well matched with the output impedance of the transistor. So maximum power is delivered to the load by the amplifier.

Circuit Operation

The output is collected from the output transformer Tr2. The primary of this transformer Tr2 has practically no dc component through it. The transistors T1 and T2 have their collectors connected to the primary of transformer Tr2 so that their currents are equal in magnitude and flow in opposite directions through the primary of transformer Tr2.

When the a.c. input signal is applied, the base of transistor T1 is more positive while the base of transistor T2 is less positive. Hence the collector current ic1 of transistor T1 increases while the collector current ic2 of transistor T2 decreases. These currents flow in opposite directions in two halves of the primary of output transformer. Moreover, the flux produced by these currents will also be in opposite directions.

Hence, the voltage across the load will be induced voltage whose magnitude will be proportional to the difference of collector currents.

$(ic1-ic2)$

Similarly, for the negative input signal, the collector current ic2 will be more than ic1. In this case, the voltage developed across the load will again be due to the difference

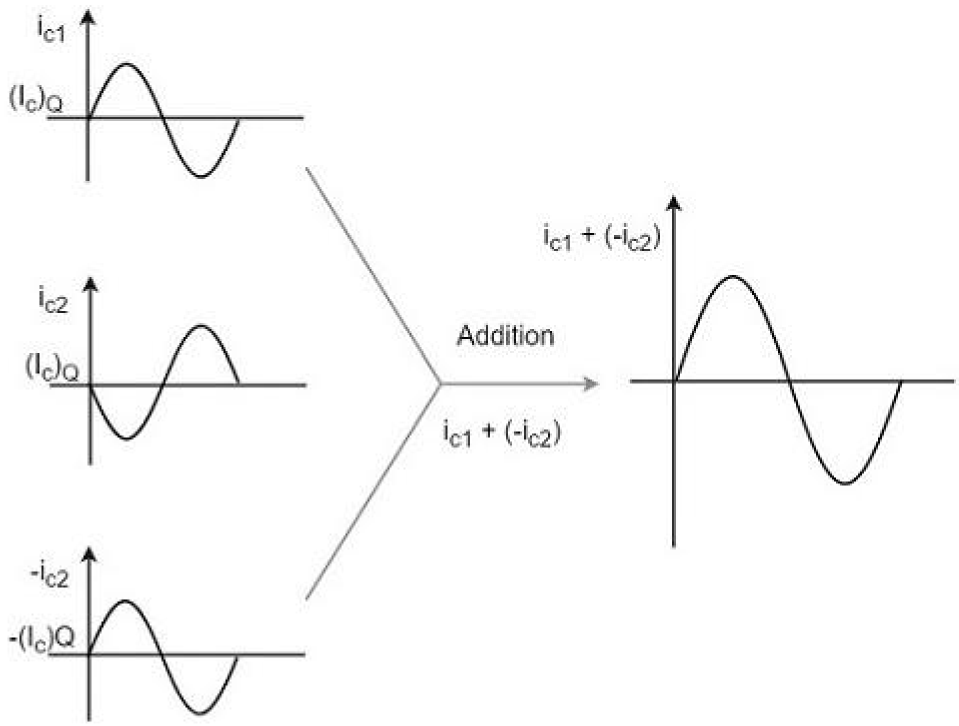
$(ic1-ic2$

As $ic2>ic1$

The polarity of voltage induced across load will be reversed.

$ic1-ic2=ic1+(-ic2)$

To have a better understanding, let us consider the below figure.



The overall operation results in an a.c. voltage induced in the secondary of output transformer and hence a.c. power is delivered to that load.

It is understood that, during any given half cycle of input signal, one transistor is being driven (or pushed) deep into conduction while the other being non-conducting (pulled out). Hence the name Push-pull amplifier. The harmonic distortion in Push-pull amplifier is minimized such that all the even harmonics are eliminated.

Advantages

- The advantages of class A Push-pull amplifier are as follows
- High a.c. output is obtained.
- The output is free from even harmonics.
- The effect of ripple voltages are balanced out. These are present in the power supply due to inadequate filtering.

Disadvantages

- The disadvantages of class A Push-pull amplifier are as follows
- The transistors are to be identical, to produce equal amplification.

Center-tapping is required for the transformers.

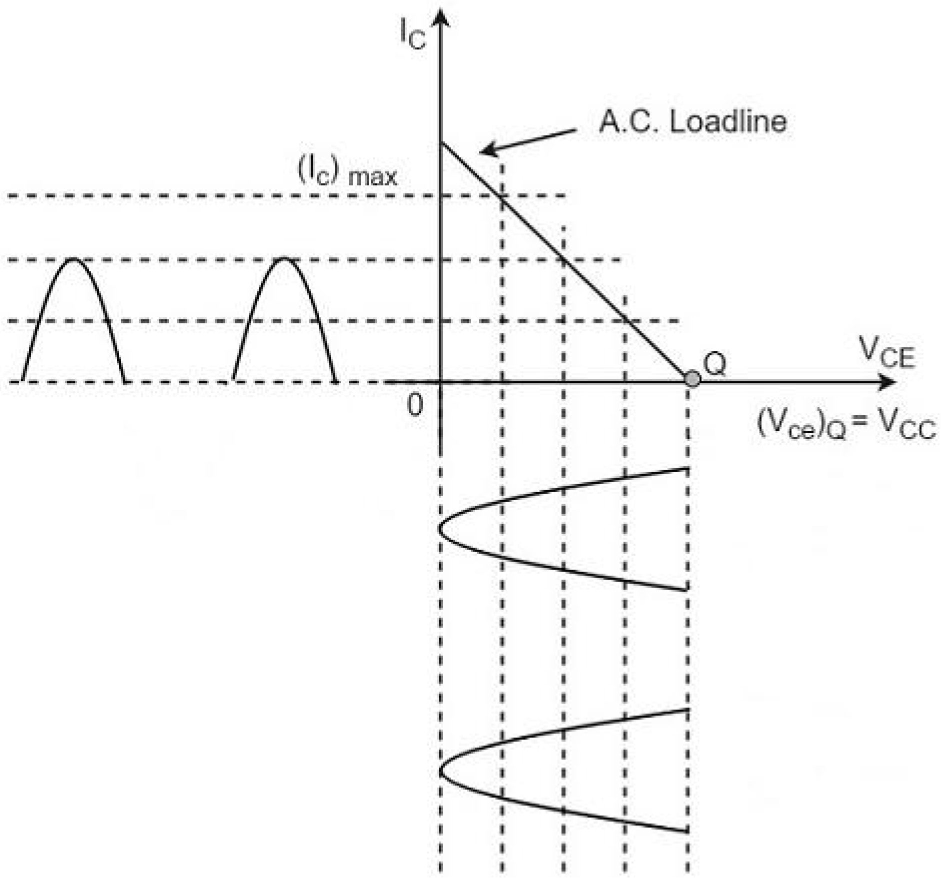
- The transformers are bulky and costly.

When the collector current flows only during the positive half cycle of the input signal, the power amplifier is known as class B power amplifier.

Class B Operation

The biasing of the transistor in class B operation is in such a way that at zero signal condition, there will be no collector current. The operating point is selected to be at collector cut off voltage. So, when the signal is applied, only the positive half cycle is amplified at the output.

The figure below shows the input and output waveforms during class B operation.



When the signal is applied, the circuit is forward biased for the positive half cycle of the input and hence the collector current flows. But during the negative half cycle of the input, the circuit is reverse biased and the collector current will be absent. Hence only the positive half cycle is amplified at the output.

As the negative half cycle is completely absent, the signal distortion will be high. Also, when the applied signal increases, the power dissipation will be more. But when compared to class A power amplifier, the output efficiency is increased.

Well, in order to minimize the disadvantages and achieve low distortion, high efficiency and high output power, the push-pull configuration is used in this class B amplifier.

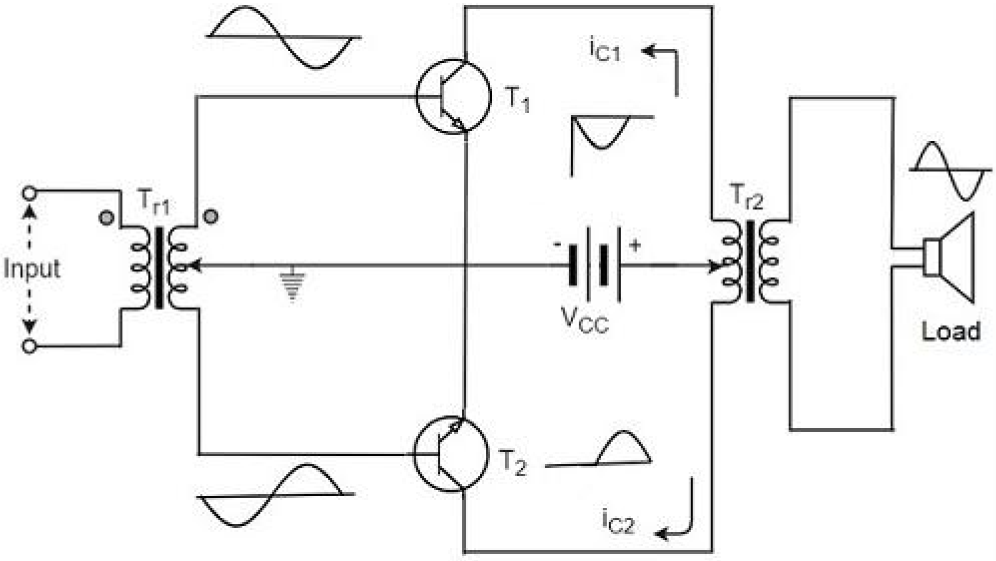
Class B Push-Pull Amplifier

Though the efficiency of class B power amplifier is higher than class A, as only one half cycle of the input is used, the distortion is high. Also, the input power is not completely utilized. In order to compensate these problems, the push-pull configuration is introduced in class B amplifier.

Construction

The circuit of a push-pull class B power amplifier consists of two identical transistors T1 and T2 whose bases are connected to the secondary of the center-tapped input transformer Tr1. The emitters are shorted and the collectors are given the VCC supply through the primary of the output transformer Tr2.

The circuit arrangement of class B push-pull amplifier, is same as that of class A push-pull amplifier except that the transistors are biased at cut off, instead of using the biasing resistors. The figure below gives the detailing of the construction of a push-pull class B power amplifier.

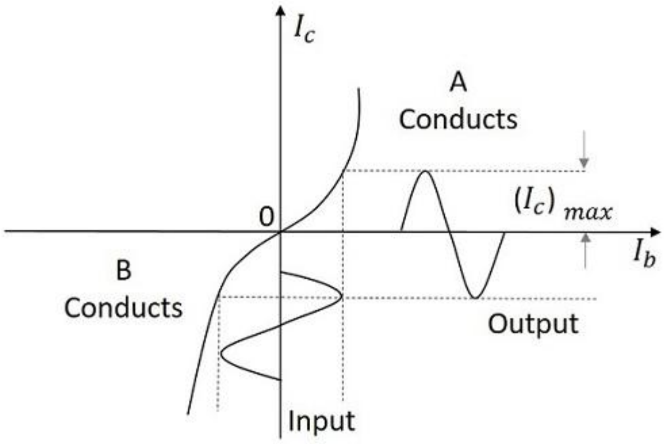


The circuit operation of class B push pull amplifier is detailed below.

Operation

The circuit of class B push-pull amplifier shown in the above figure clears that both the transformers are center-tapped. When no signal is applied at the input, the transistors T1 and T2 are in cut off condition and hence no collector currents flow. As no current is drawn from VCC, no power is wasted.

When input signal is given, it is applied to the input transformer Tr1 which splits the signal into two signals that are 180o out of phase with each other. These two signals are given to the two identical transistors T1 and T2. For the positive half cycle, the base of the transistor T1 becomes positive and collector current flows. At the same time, the transistor T2 has negative half cycle, which throws the transistor T2 into cutoff condition and hence no collector current flows. The waveform is produced as shown in the following figure.



For the next half cycle, the transistor T1 gets into cut off condition and the transistor T2 gets into conduction, to contribute the output. Hence for both the cycles, each transistor conducts alternately. The output transformer Tr3 serves to join the two currents producing an almost undistorted output waveform.

Power Efficiency of Class B Push-Pull Amplifier

The current in each transistor is the average value of half sine loop.

For half sine loop, Idc is given by

$Idc=(I_C)max/\pi$

Therefore,

$(pin)dc=2\times[(I_C)max/\pi\times V_{CC}]$

Here factor 2 is introduced as there are two transistors in push-pull amplifier.

R.M.S. value of collector current = $(I_C)max/2\sqrt{2}$

R.M.S. value of output voltage = $VCC/2\sqrt{2}$

Under ideal conditions of maximum power

Therefore,

$(P_O)ac=(I_C)max/\sqrt{2}\times V_{CC}/\sqrt{2}$

$=(I_C)max\times V_{CC}/2$

Now overall maximum efficiency

$\eta_{overall}=(P_O)ac/(Pin)dc$

$=(I_C)max\times V_{CC}/2/2\times[(I_C)max/\pi\times V_{CC}]$

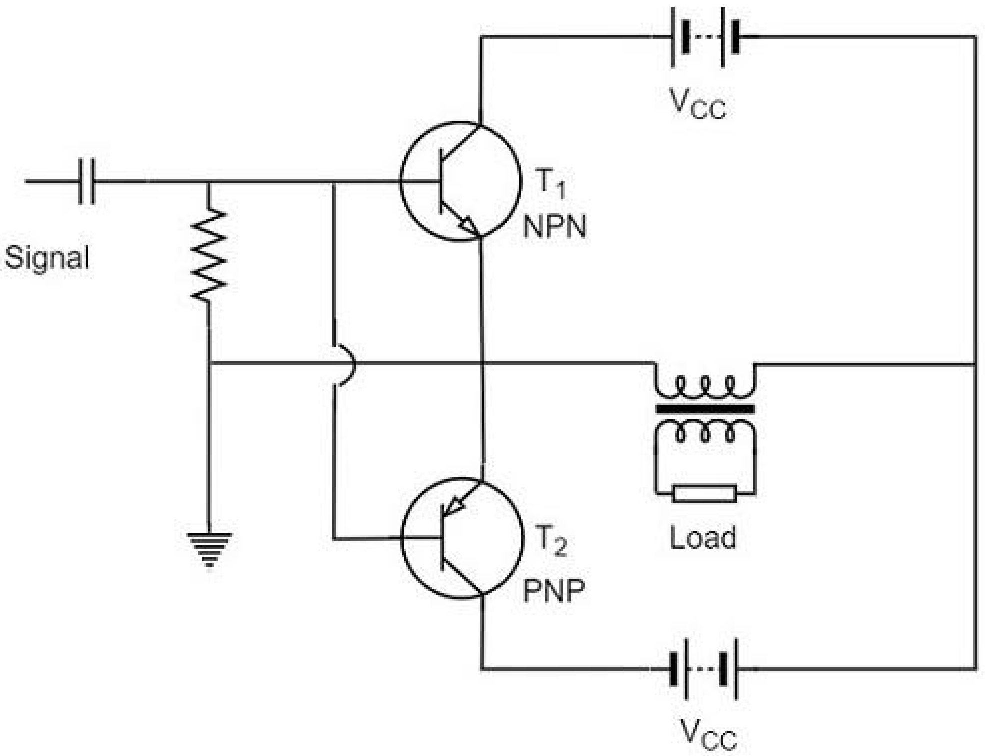
$=\pi/4=0.785=78.5\%$

The collector efficiency would be the same.

Hence the class B push-pull amplifier improves the efficiency than the class A push-pull amplifier.

Complementary Symmetry Push-Pull Class B Amplifier

The push pull amplifier which was just discussed improves efficiency but the usage of center-tapped transformers makes the circuit bulky, heavy and costly. To make the circuit simple and to improve the efficiency, the transistors used can be complemented, as shown in the following circuit diagram.



The above circuit employs a NPN transistor and a PNP transistor connected in push pull configuration. When the input signal is applied, during the positive half cycle of the input signal, the NPN transistor conducts and the PNP transistor cuts off. During the negative half cycle, the NPN transistor cuts off and the PNP transistor conducts.

In this way, the NPN transistor amplifies during positive half cycle of the input, while PNP transistor amplifies during negative half cycle of the input. As the transistors are both complement to each other, yet act symmetrically while being connected in push pull configuration of class B, this circuit is termed as Complementary symmetry push pull class B amplifier.

Advantages

The advantages of Complementary symmetry push pull class B amplifier are as follows.

As there is no need of center tapped transformers, the weight and cost are reduced.

Equal and opposite input signal voltages are not required.

Disadvantages

The disadvantages of Complementary symmetry push pull class B amplifier are as follows.

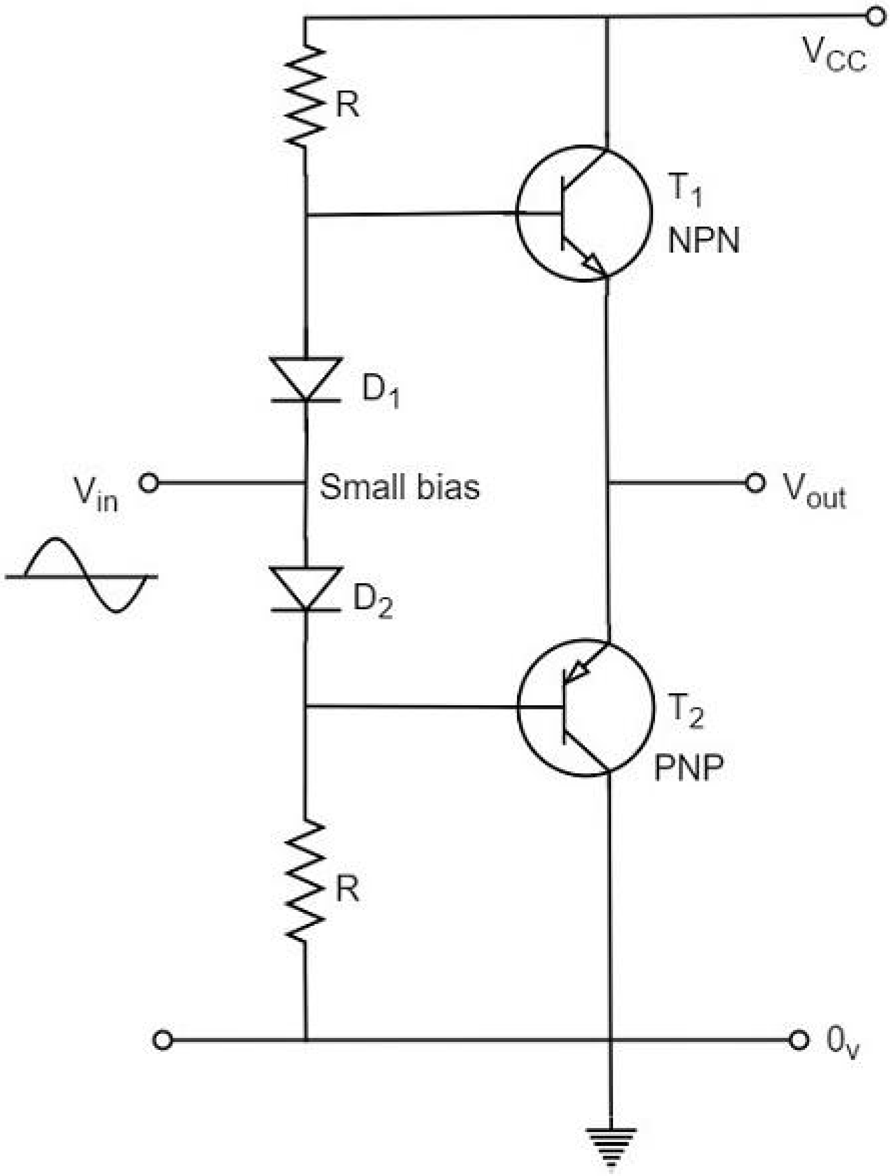
It is difficult to get a pair of transistors (NPN and PNP) that have similar characteristics.

We require both positive and negative supply voltages.

Class AB Power Amplifier

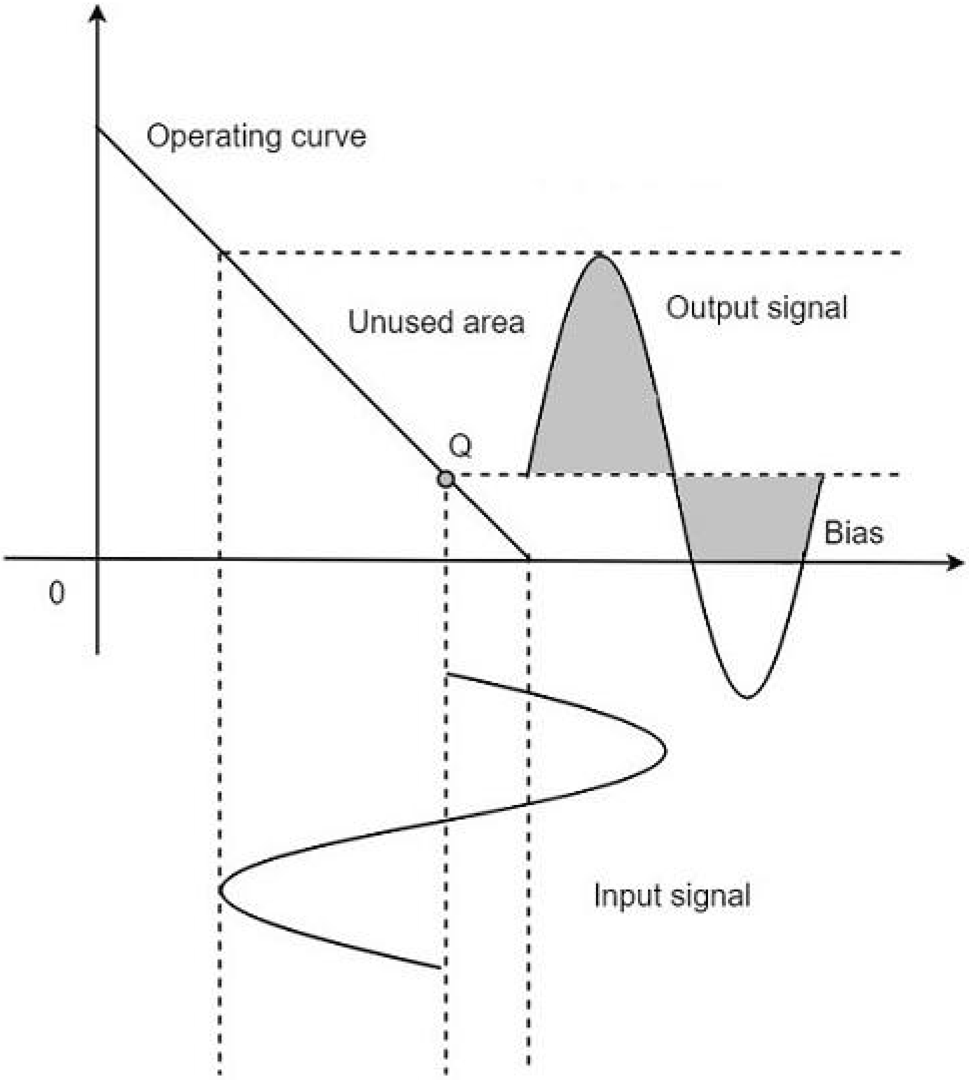
As the name implies, class AB is a combination of class A and class B type of amplifiers. As class A has the problem of low efficiency and class B has distortion problem, this class AB is emerged to eliminate these two problems, by utilizing the advantages of both the classes.

The cross over distortion is the problem that occurs when both the transistors are OFF at the same instant, during the transition period. In order to eliminate this, the condition has to be chosen for more than one half cycle. Hence, the other transistor gets into conduction, before the operating transistor switches to cut off state. This is achieved only by using class AB configuration, as shown in the following circuit diagram.



Therefore, in class AB amplifier design, each of the push-pull transistors is conducting for slightly more than the half cycle of conduction in class B, but much less than the full cycle of conduction of class A.

The conduction angle of class AB amplifier is somewhere between 180o to 360o depending upon the operating point selected. This is understood with the help of below figure.



The small bias voltage given using diodes D1 and D2, as shown in the above figure, helps the operating point to be above the cutoff point. Hence the output waveform of class AB results as seen in the above figure. The crossover distortion created by class B is overcome by this class AB, as well the inefficiencies of class A and B don't affect the circuit.

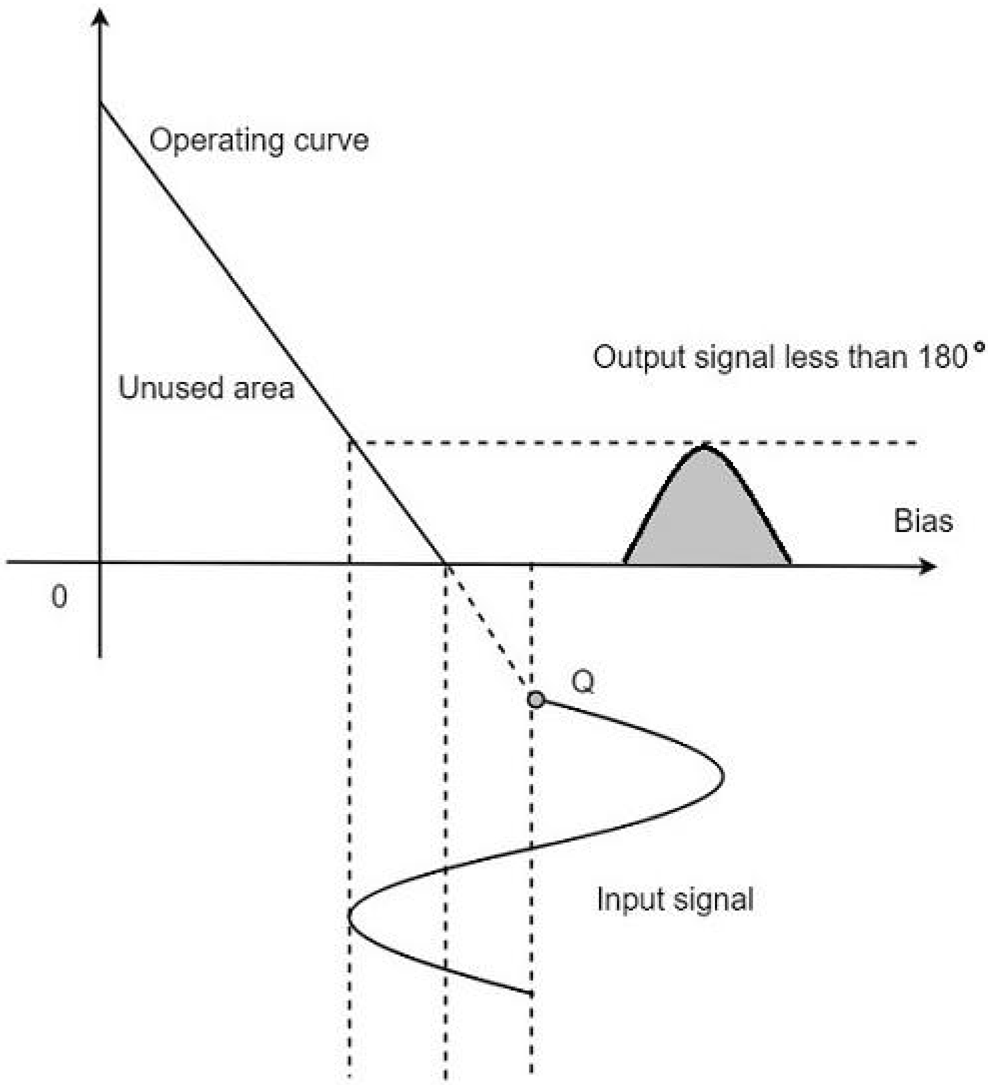
So, the class AB is a good compromise between class A and class B in terms of efficiency and linearity having the efficiency reaching about 50% to 60%. The class A, B and AB amplifiers are called as linear amplifiers because the output signal amplitude and phase are linearly related to the input signal amplitude and phase.

Class C Power Amplifier

When the collector current flows for less than half cycle of the input signal, the power amplifier is known as class C power amplifier.

The efficiency of class C amplifier is high while linearity is poor. The conduction angle for class C is less than 180o. It is generally around 90o, which means the transistor remains idle for more than half of the input signal. So, the output current will be delivered for less time compared to the application of input signal.

The following figure shows the operating point and output of a class C amplifier.



This kind of biasing gives a much improved efficiency of around 80% to the amplifier, but introduces heavy distortion in the output signal. Using the class C amplifier, the pulses produced at its output can be converted to complete sine wave of a particular frequency by using LC circuits in its collector circuit.

Unit-3: FIELD EFFECT TRANSISTOR (FET)

FET stands for Field Effect Transistor. The flow of current in the field effect transistors is due to only one type of charge carrier (either hole or electron). For this reason, they are known as unipolar transistors. The FET is also a three terminal device, where the names of the terminals are Source, Drain and . The Gate circuit symbol of FET is shown in Figure-2.

Figure 2 - Field Effect Transistor (FET)

Based on the channel between the source and drain provided for current to flow, the FET is of two types viz. N-Channel FET and P-Channel FET. In case of FET, there is no PN junction between the source and drain. The gate region of FET is made from alternate semiconductor material as compared to the channel between the source and drain terminals.

Difference between BJT and FET

The following table shows the major differences between bipolar junction transistor and field effect transistor.

Parameter	BJT	FET
Full form	BJT stands for Bipolar Junction Transistor.	FET stands for Field Effect Transistor.
Definition	A type of transistor which uses two types of charge carries viz. electrons and holes for conduction is known as bipolar junction transistor (BJT).	A type of transistor in which electric field is used to control the flow of current in a semiconductor is known as field effect transistor (FET).
Drive type	In BJT, the current flow is due to both majority and minority charge carriers. Thus, it is a bipolar device.	In FET, the electric current flows only due to majority charge carriers. Thus, it is a unipolar device.
Terminals	BJT has three terminals viz. Emitter, Base and Collector.	FET also has three terminals viz. Source,

Parameter	BJT	FET
		Drain and Gate.
PN junction	BJT consists of two PN junctions viz. emitter-base junction and collector-base junction.	FET does not have PN junctions.
Control element	BJT is a current-controlled device.	FET is a voltage controlled device.
Types	BJT are of two types: NPN transistor and PNP transistor.	FET are also of two types: N-channel FET and P-channel FET.
Configuration	BJT has three configurations: common emitter (CE), common base (CB) and common collector (CC).	FET also has three configurations: common source (CS), common gate (CG) and common drain (CD).
Size	BJT is large in size and hence requires more space. Therefore, it is more complicated to fabricate as an IC	FET is comparatively smaller in size. Hence, it is easier to fabricate as an IC.
Sensitivity	BJT is more sensitive to the changes in the applied voltage.	FET is less sensitive to the variations in the applied voltage.
Relationship between input and output	BJT has linear relationship between input and output.	FET has non-linear relationship between input and output.

Parameter	BJT	FET
Thermal noise	BJT has more thermal noise.	The thermal noise in case of FET is much lower.
Thermal runway	Thermal runway exits in BJT.	Thermal runway does not exists in FET.
Thermal stability	BJT has less thermal stability.	FET has good thermal stability due to absence of minority charge carriers.
Input impedance	In case of BJT, the input circuit is forward biased. Thus, the BJT has low input impedance.	FET has high input impedance due to reverse bias of input circuit.
Temperature coefficient at high current levels	BJT has positive temperature coefficient.	FET has negative temperature coefficient.
Suitability	BJT is suitable for low current applications.	FET is suitable for high current applications.
Switching speed	The switching speed of BJT is low.	FET has higher switching speed.
Effect of radiation	BJT is susceptible to radiation.	FET is relatively immune to radiation.
Gain bandwidth product	BJT has higher gain bandwidth product.	FET has lower gain bandwidth product.
Minority carrier storage effect	BJT suffers from minority carrier storage effect.	FET does not suffer from minority carrier storage

Parameter	BJT	FET
		effect.
Cost	BJT is cheaper to manufacture.	FET is relatively expensive to manufacture.
Installation	BJT does not require special handling during installation.	FET demands special handling during installation.
Applications	BJT is used as switch (in saturation and cut-off region) and amplifier (in active region).	FET is used as switch (in Ohmic and cut-off region) and as amplifier (in saturation region).

A Field Effect Transistor (FET)s operation is based on a controlled input voltage. By appearance JFET and bipolar transistors are very similar. However, BJT is a current controlled device and JFET is controlled by input voltage. Most commonly two types of FETs are available.

- **Junction Field Effect Transistor (JFET)**
- **Metal Oxide Semiconductor FET (IGFET)**

Junction Field Effect Transistor

The functioning of Junction Field Effect Transistor depends upon the flow of majority carriers (electrons or holes) only. Basically, JFETs consist of an N type or P type silicon bar containing PN junctions at the sides. Following are some important points to remember about FET −

Gate − By using diffusion or alloying technique, both sides of N type bar are heavily doped to create PN junction. These doped regions are called gate (G).

Source − It is the entry point for majority carriers through which they enter into the semiconductor bar.

Drain − It is the exit point for majority carriers through which they leave the semiconductor bar.

Channel − It is the area of N type material through which majority carriers pass from the source to drain.

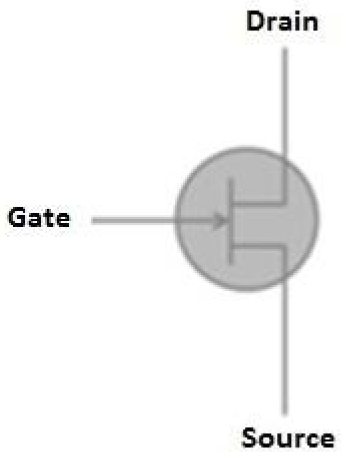
There are two types of JFETs commonly used in the field semiconductor devices: **N-Channel JFET** and **P-Channel JFET**.

N-Channel JFET

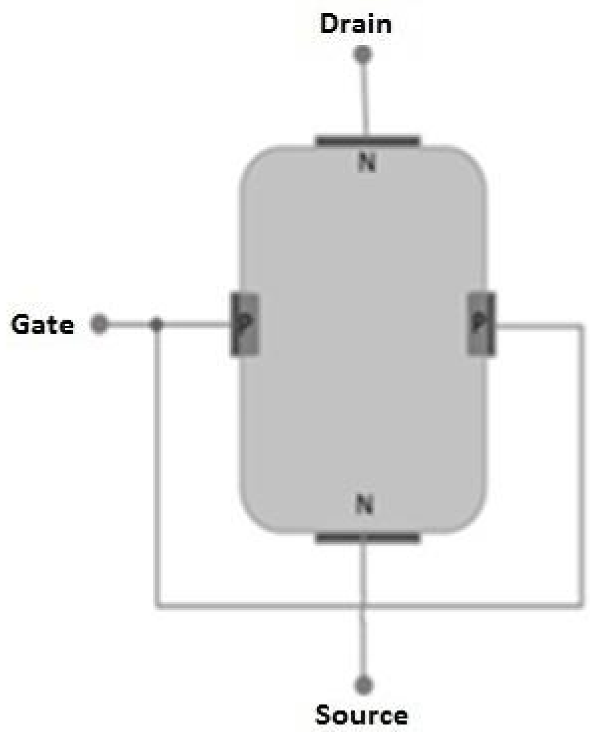
It has a thin layer of N type material formed on P type substrate. Following figure shows the crystal structure and schematic symbol of an N-channel JFET. Then the gate is formed on top of the N channel with P type material. At the end of the channel and the gate, lead wires are attached and the substrate has no connection.

When a DC voltage source is connected to the source and the drain leads of a JFET, maximum current will flow through the channel. The same amount of current will flow from the source and the drain terminals. The amount of channel current flow will be determined by the value of VDD and the internal resistance of the channel.

A typical value of source-drain resistance of a JFET is quite a few hundred ohms. It is clear that even when the gate is open full current conduction will take place in the channel. Essentially, the amount of bias voltage applied at ID, controls the flow of current carriers passing through the channel of a JFET. With a small change in gate voltage, JFET can be controlled anywhere between full conduction and cutoff state.



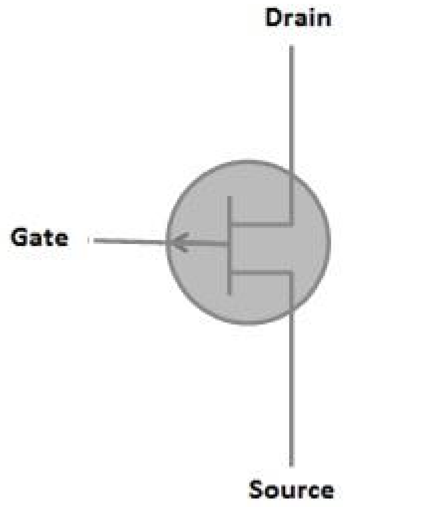
Symbol of N-Channel JFET



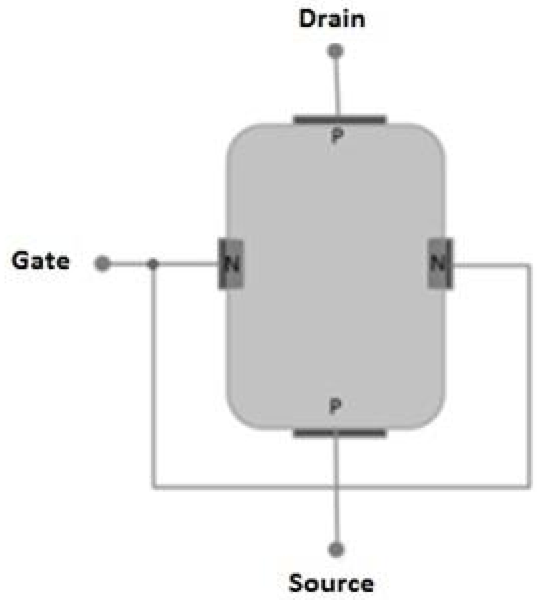
N-Channel JFET

P-Channel JFETs

It has a thin layer of P type material formed on N type substrate. The following figure shows the crystal structure and schematic symbol of an N-channel JFET. The gate is formed on top of the P channel with N type material. At the end of the channel and the gate, lead wires are attached. Rest of the construction details are similar to that of N-channel JFET.



Symbol of P-Channel JFET

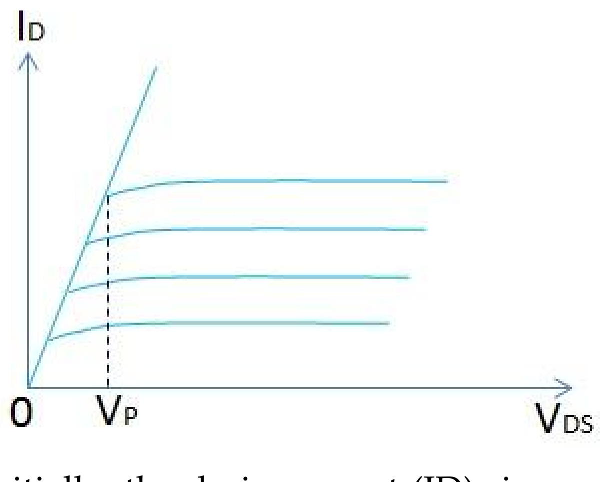


P-Channel JFET

Normally for general operation, the gate terminal is made positive with respect to the source terminal. The size of the P-N junction depletion layer depends upon fluctuations in the values of reverse biased gate voltage. With a small change in gate voltage, JFET can be controlled anywhere between full conduction and cutoff state.

Output Characteristics of JFET

The output characteristics of JFET are drawn between drain current (ID) and drain source voltage (VDS) at constant gate source voltage (VGS) as shown in the following figure.



Initially, the drain current (ID) rises rapidly with drain source voltage (VDS) however suddenly becomes constant at a voltage known as pinch-off voltage (VP). Above pinch-off voltage, the channel width becomes so narrow that it allows very small drain current to pass through it. Therefore, drain current (ID) remains constant above pinch-off voltage.

Parameters of JFET

The main parameters of JFET are −

- **AC drain resistance (Rd)**
- **Transconductance**
- **Amplification factor**

AC drain resistance (Rd) − It is the ratio of change in the drain source voltage (ΔV_{DS}) to the change in drain current (Δ_{ID}) at constant gate-source voltage. It can be expressed as,

$$Rd = (\Delta V_{DS})/(\Delta I_D) \text{ at Constant } V_{GS}$$

Transconductance (g_{fs}) − It is the ratio of change in drain current (ΔI_D) to the change in gate source voltage (ΔV_{GS}) at constant drain-source voltage. It can be expressed as,

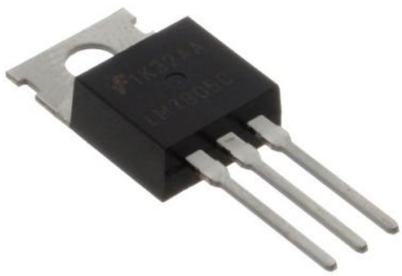
$$g_{fs} = (\Delta I_D)/(\Delta V_{GS}) \text{ at constant } V_{DS}$$

Amplification Factor (u) − It is the ratio of change in drain-source voltage (ΔV_{DS}) to the change in gate source voltage (ΔV_{GS}) constant drain current (ΔI_D). It can be expressed as,

$$u = (\Delta V_{DS})/(\Delta V_{GS}) \text{ at constant } I_D$$

MOSFET

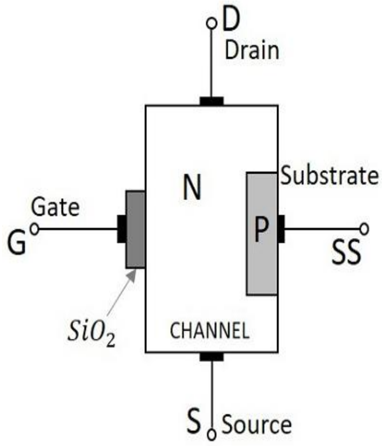
MOSFET stands for Metal Oxide Silicon Field Effect Transistor or Metal Oxide Semiconductor Field Effect Transistor. This is also called as IGFET meaning Insulated Gate Field Effect Transistor. The FET is operated in both depletion and enhancement modes of operation. The following figure shows how a practical MOSFET looks like.



Construction of a MOSFET

The construction of a MOSFET is a bit similar to the FET. An oxide layer is deposited on the substrate to which the gate terminal is connected. This oxide layer acts as an insulator (sio2 insulates from the substrate), and hence the MOSFET has another name as IGFET. In the construction of MOSFET, a lightly doped substrate, is diffused with a heavily doped region. Depending upon the substrate used, they are called as P-type and N-type MOSFETs.

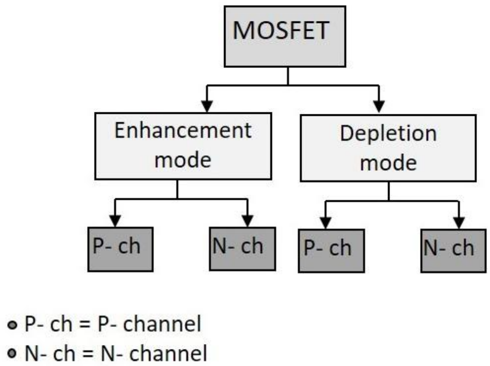
The following figure shows the construction of a MOSFET.



The voltage at gate controls the operation of the MOSFET. In this case, both positive and negative voltages can be applied on the gate as it is insulated from the channel. With negative gate bias voltage, it acts as depletion MOSFET while with positive gate bias voltage it acts as an Enhancement MOSFET.

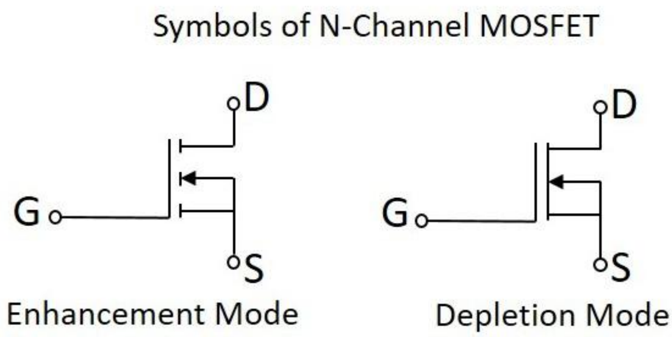
Classification of MOSFETs

Depending upon the type of materials used in the construction, and the type of operation, the MOSFETs are classified as in the following figure.



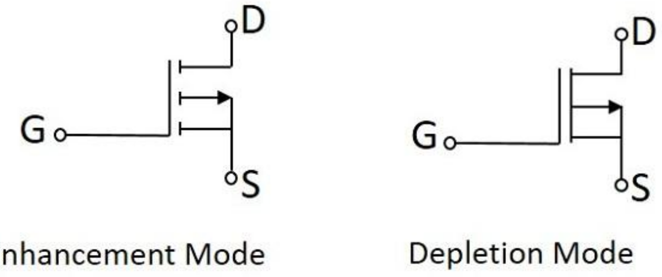
After the classification, let us go through the symbols of MOSFET.

The N-channel MOSFETs are simply called as NMOS. The symbols for N-channel MOSFET are as given below.



The P-channel MOSFETs are simply called as PMOS. The symbols for P-channel MOSFET are as given below.

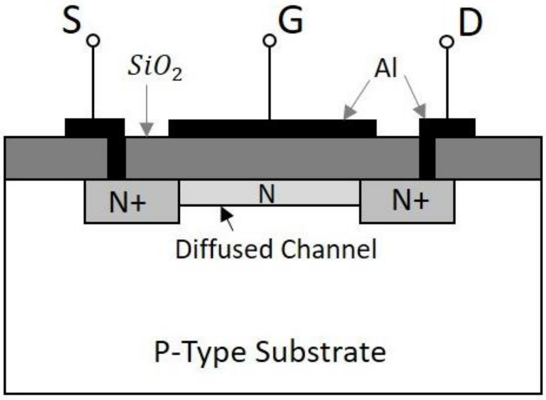
Symbols of P-Channel MOSFET



Now, let us go through the constructional details of an N-channel MOSFET. Usually an NChannel MOSFET is considered for explanation as this one is mostly used. Also, there is no need to mention that the study of one type explains the other too.

Construction of N- Channel MOSFET

Let us consider an N-channel MOSFET to understand its working. A lightly doped P-type substrate is taken into which two heavily doped N-type regions are diffused, which act as source and drain. Between these two N+ regions, there occurs diffusion to form an Nchannel, connecting drain and source.



Structure of N-channel MOSFET

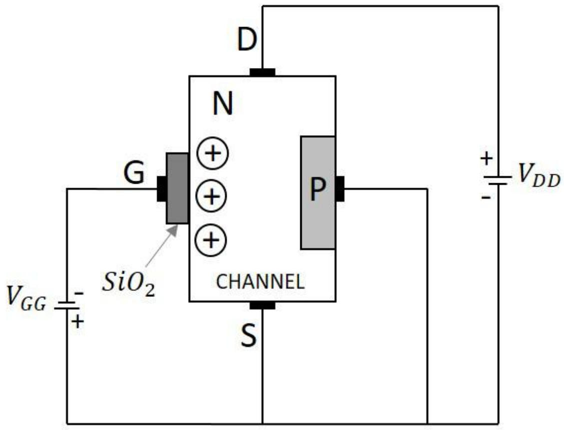
A thin layer of Silicon dioxide (SiO2) is grown over the entire surface and holes are made to draw ohmic contacts for drain and source terminals. A conducting layer of aluminum is laid over the entire channel, upon this SiO2 layer from source to drain which constitutes the gate. The SiO2 substrate is connected to the common or ground terminals.

Because of its construction, the MOSFET has a very less chip area than BJT, which is 5% of the occupancy when compared to bipolar junction transistor. This device can be operated in modes. They are depletion and enhancement modes. Let us try to get into the details.

Working of N - Channel depletion mode MOSFET

For now, we have an idea that there is no PN junction present between gate and channel in this, unlike a FET. We can also observe that, the diffused channel N between two N+ regions, the insulating dielectric SiO2 and the aluminum metal layer of the gate together form a parallel plate capacitor.

If the NMOS has to be worked in depletion mode, the gate terminal should be at negative potential while drain is at positive potential, as shown in the following figure.



Working of MOSFET in depletion mode

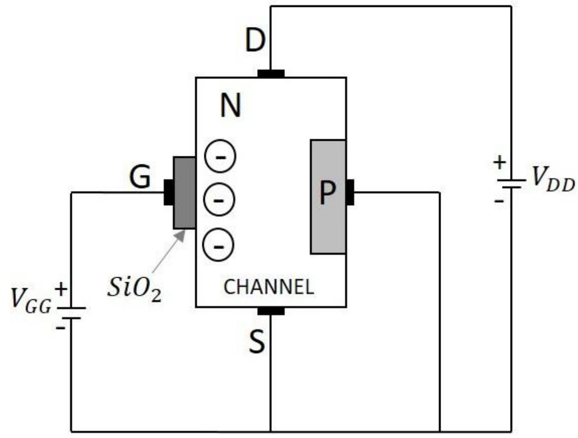
When no voltage is applied between gate and source, some current flows due to the voltage between drain and source. Let some negative voltage is applied at V_{GG}. Then the minority carriers i.e. holes, get attracted and settle near SiO2 layer. But the majority carriers, i.e., electrons get repelled.

With some amount of negative potential at V_{GG} a certain amount of drain current I_D flows through source to drain. When this negative potential is further increased, the electrons get depleted and the current I_D decreases. Hence the more negative the applied V_{GG}, the lesser the value of drain current I_D will be.

The channel nearer to drain gets more depleted than at source like in FET and the current flow decreases due to this effect. Hence it is called as depletion mode MOSFET.

Working of N-Channel MOSFET Enhancement Mode

The same MOSFET can be worked in enhancement mode, if we can change the polarities of the voltage V_{GG}. So, let us consider the MOSFET with gate source voltage V_{GG} being positive as shown in the following figure.



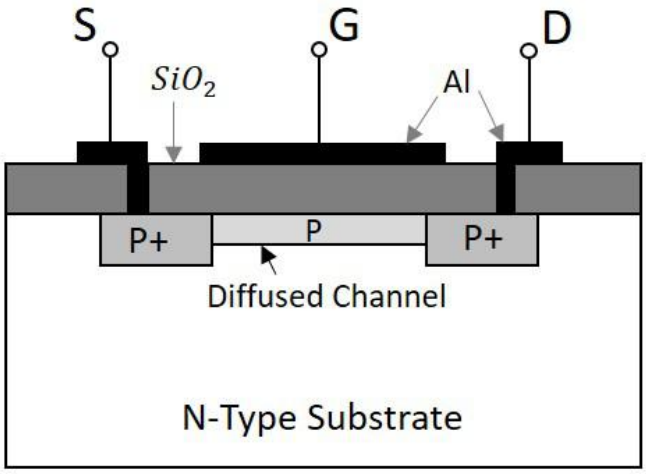
Working of MOSFET in Enhancement mode

When no voltage is applied between gate and source, some current flows due to the voltage between drain and source. Let some positive voltage is applied at V_{GG}. Then the minority carriers i.e. holes, get repelled and the majority carriers i.e. electrons gets attracted towards the SiO2 layer.

With some amount of positive potential at V_{GG} a certain amount of drain current I_D flows through source to drain. When this positive potential is further increased, the current I_D increases due to the flow of electrons from source and these are pushed further due to the voltage applied at V_{GG}. Hence the more positive the applied V_{GG}, the more the value of drain current I_D will be. The current flow gets enhanced due to the increase in electron flow better than in depletion mode. Hence this mode is termed as Enhanced Mode MOSFET.

P - Channel MOSFET

The construction and working of a PMOS is same as NMOS. A lightly doped n-substrate is taken into which two heavily doped P+ regions are diffused. These two P+ regions act as source and drain. A thin layer of SiO2 is grown over the surface. Holes are cut through this layer to make contacts with P+ regions, as shown in the following figure.



Structure of P-channel MOSFET

Working of PMOS

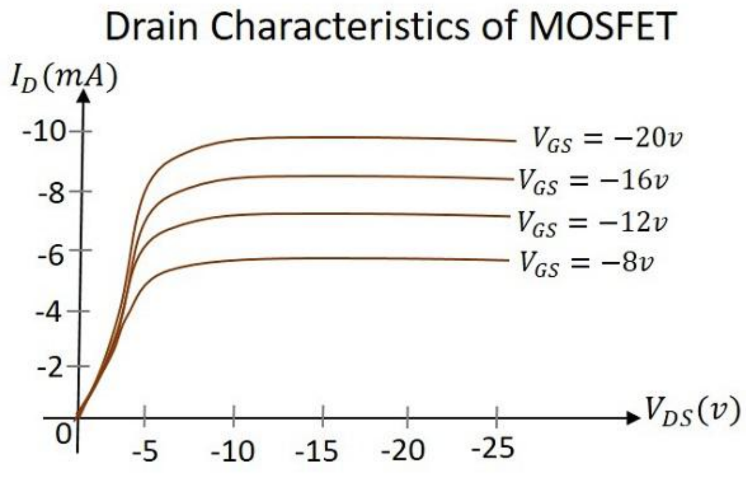
When the gate terminal is given a negative potential at V_{GG} than the drain source voltage V_{DD}, then due to the P+ regions present, the hole current is increased through the diffused P channel and the PMOS works in Enhancement Mode.

When the gate terminal is given a positive potential at V_{GG} than the drain source voltage V_{DD}, then due to the repulsion, the depletion occurs due to which the flow of current reduces. Thus PMOS works in Depletion Mode. Though the construction differs, the working is similar in both the type of MOSFETs. Hence with the change in voltage polarity both of the types can be used in both the modes.

This can be better understood by having an idea on the drain characteristics curve.

Drain Characteristics

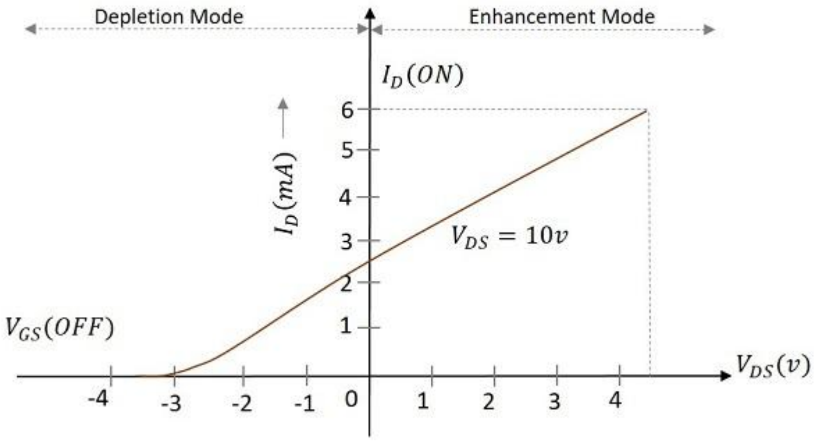
The drain characteristics of a MOSFET are drawn between the drain current I_D and the drain source voltage V_{DS}. The characteristic curve is as shown below for different values of inputs.



Actually when V_{DS} is increased, the drain current I_D should increase, but due to the applied V_{GS}, the drain current is controlled at certain level. Hence the gate current controls the output drain current.

Transfer Characteristics

Transfer characteristics define the change in the value of V_{DS} with the change in I_D and V_{GS} in both depletion and enhancement modes. The below transfer characteristic curve is drawn for drain current versus gate to source voltage.



Transfer Characteristics of a MOSFET

Comparison between BJT, FET and MOSFET

Now that we have discussed all the above three, let us try to compare some of their properties.

TERMS	BJT	FET	MOSFET
Device type	Current controlled	Voltage controlled	Voltage Controlled
Current flow	Bipolar	Unipolar	Unipolar
Terminals	Not interchangeable	Interchangeable	Interchangeable
Operational modes	No modes	Depletion mode only	Both Enhancement and Depletion modes
Input impedance	Low	High	Very high
Output	Moderate	Moderate	Low

resistance			
Operational speed	Low	Moderate	High
Noise	High	Low	Low
Thermal stability	Low	Better	High

Unit-4: FEED BACK AMPLIFIER & OSCILLATOR

Negative feedback in an amplifier is the method of feeding a portion of the amplified output to the input but in opposite phase. The phase opposition occurs as the amplifier provides 180o phase shift whereas the feedback network doesn't.

While the output energy is being applied to the input, for the voltage energy to be taken as feedback, the output is taken in shunt connection and for the current energy to be taken as feedback, the output is taken in series connection.

There are two main types of negative feedback circuits. They are −

- Negative Voltage Feedback
- Negative Current Feedback

Negative Voltage Feedback

In this method, the voltage feedback to the input of amplifier is proportional to the output voltage. This is further classified into two types −

Voltage-series feedback

Voltage-shunt feedback

Negative Current Feedback

In this method, the voltage feedback to the input of amplifier is proportional to the output current. This is further classified into two types.

Current-series feedback

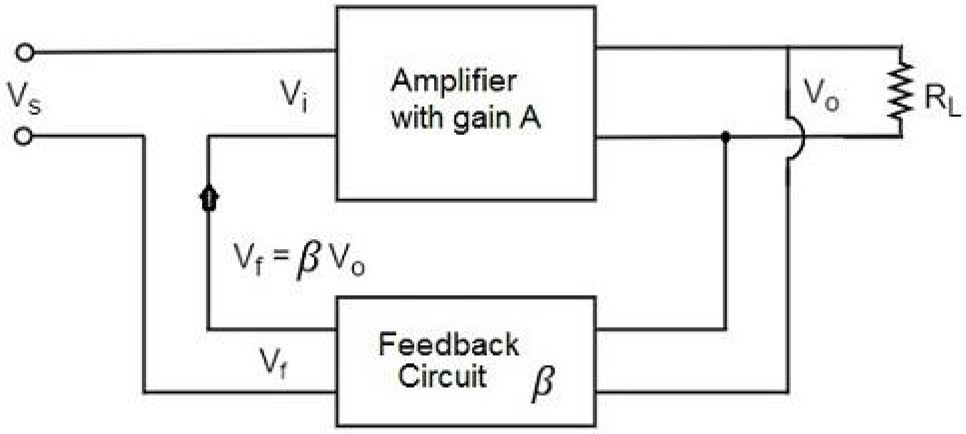
Current-shunt feedback

Let us have a brief idea on all of them.

Voltage-Series Feedback

In the voltage series feedback circuit, a fraction of the output voltage is applied in series with the input voltage through the feedback circuit. This is also known as shunt-driven series-fed feedback, i.e., a parallel-series circuit.

The following figure shows the block diagram of voltage series feedback, by which it is evident that the feedback circuit is placed in shunt with the output but in series with the input.

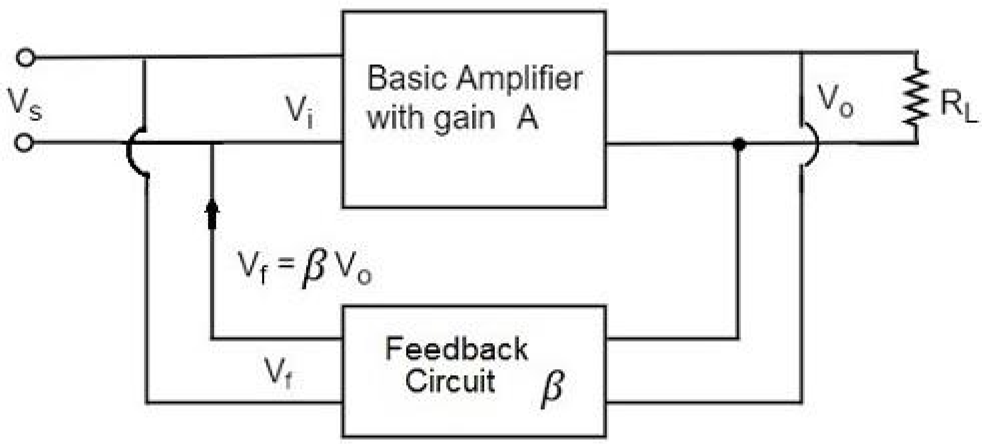


As the feedback circuit is connected in shunt with the output, the output impedance is decreased and due to the series connection with the input, the input impedance is increased.

Voltage-Shunt Feedback

In the voltage shunt feedback circuit, a fraction of the output voltage is applied in parallel with the input voltage through the feedback network. This is also known as shunt-driven shunt-fed feedback i.e., a parallel-parallel proto type.

The below figure shows the block diagram of voltage shunt feedback, by which it is evident that the feedback circuit is placed in shunt with the output and also with the input.

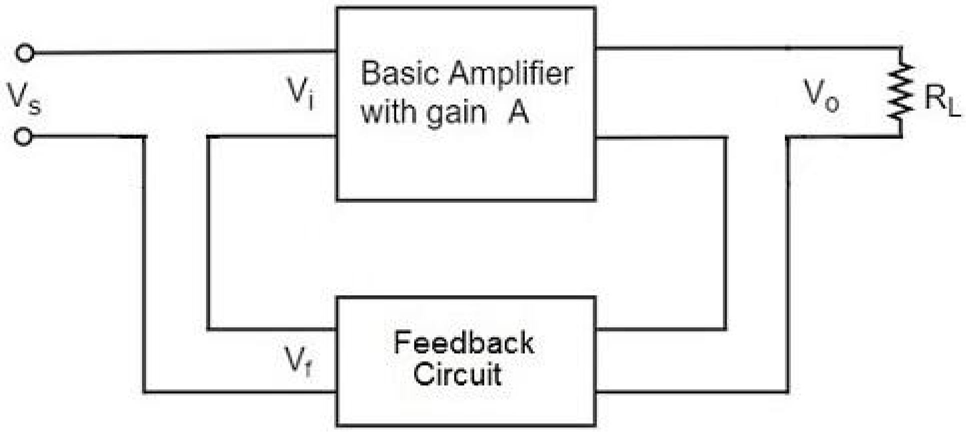


As the feedback circuit is connected in shunt with the output and the input as well, both the output impedance and the input impedance are decreased.

Current-Series Feedback

In the current series feedback circuit, a fraction of the output voltage is applied in series with the input voltage through the feedback circuit. This is also known as series-driven series-fed feedback i.e., a series-series circuit.

The following figure shows the block diagram of current series feedback, by which it is evident that the feedback circuit is placed in series with the output and also with the input.

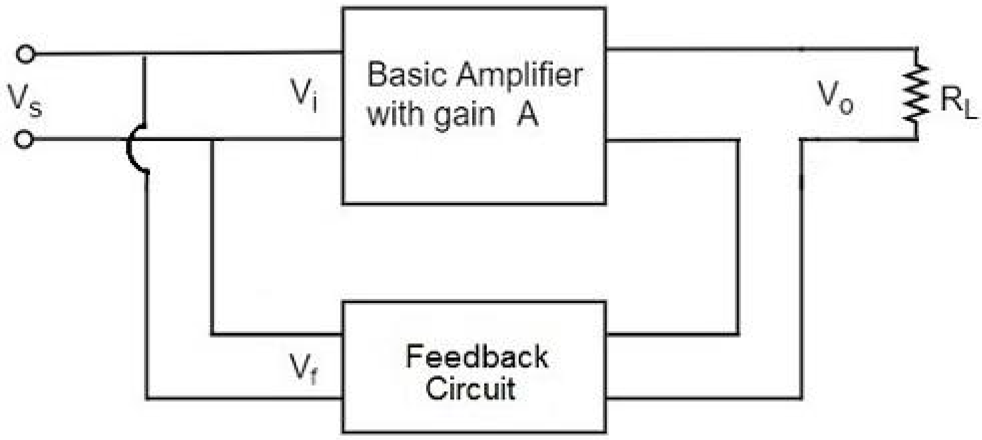


As the feedback circuit is connected in series with the output and the input as well, both the output impedance and the input impedance are increased.

Current-Shunt Feedback

In the current shunt feedback circuit, a fraction of the output voltage is applied in series with the input voltage through the feedback circuit. This is also known as series-driven shunt-fed feedback i.e., a series-parallel circuit.

The below figure shows the block diagram of current shunt feedback, by which it is evident that the feedback circuit is placed in series with the output but in parallel with the input.



As the feedback circuit is connected in series with the output, the output impedance is increased and due to the parallel connection with the input, the input impedance is decreased.

Let us now tabulate the amplifier characteristics that get affected by different types of negative feedbacks.

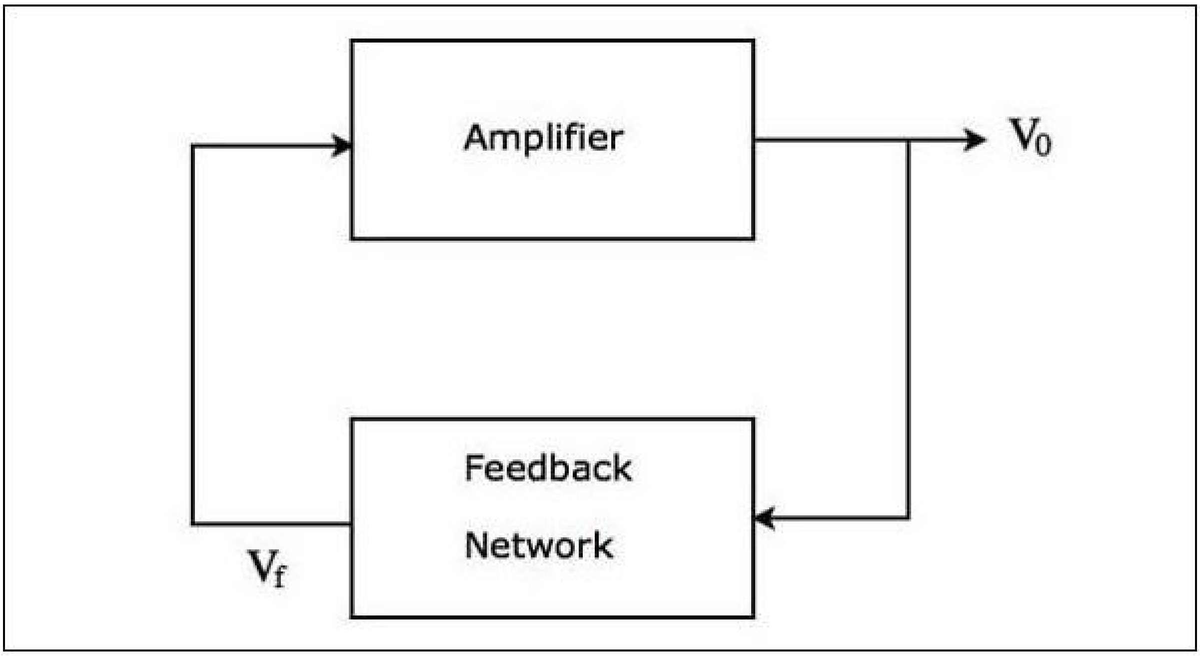
Characteristics	**Types of Feedback**			
	Voltage-Series	**Voltage-Shunt**	**Current-Series**	**Current-Shunt**

Voltage Gain	Decreases	Decreases	Decreases	Decreases
Bandwidth	Increases	Increases	Increases	Increases
Input resistance	Increases	Decreases	Increases	Decreases
Output resistance	Decreases	Decreases	Increases	Increases
Harmonic distortion	Decreases	Decreases	Decreases	Decreases
Noise	Decreases	Decreases	Decreases	Decreases

Oscillator

An oscillator is an electronic circuit that produces a periodic signal. If the oscillator produces sinusoidal oscillations, it is called as a sinusoidal oscillator. It converts the input energy from a DC source into an AC output energy of a periodic signal. This periodic signal will be having a specific frequency and amplitude.

The block diagram of a sinusoidal oscillator is shown in the following figure −



The above figure mainly consists of two blocks: an amplifier and a feedback network.The feedback network takes a part of the output of amplifier as an input to it and produces a voltage signal. This voltage signal is applied as an input to the amplifier.

The block diagram of a sinusoidal oscillator shown above produces sinusoidal oscillations, when the following two conditions are satisfied −

The loop gain $A_v\beta$ of the above block diagram of sinusoidal oscillator must be greater than or equal to unity. Here, A_v and β are the gain of amplifier and gain of the feedback network, respectively.

The total phase shift around the loop of the above block diagram of a sinusoidal oscillator must be either 0^0 or 360^0.

The above two conditions together are called as **Barkhausen criteria**.

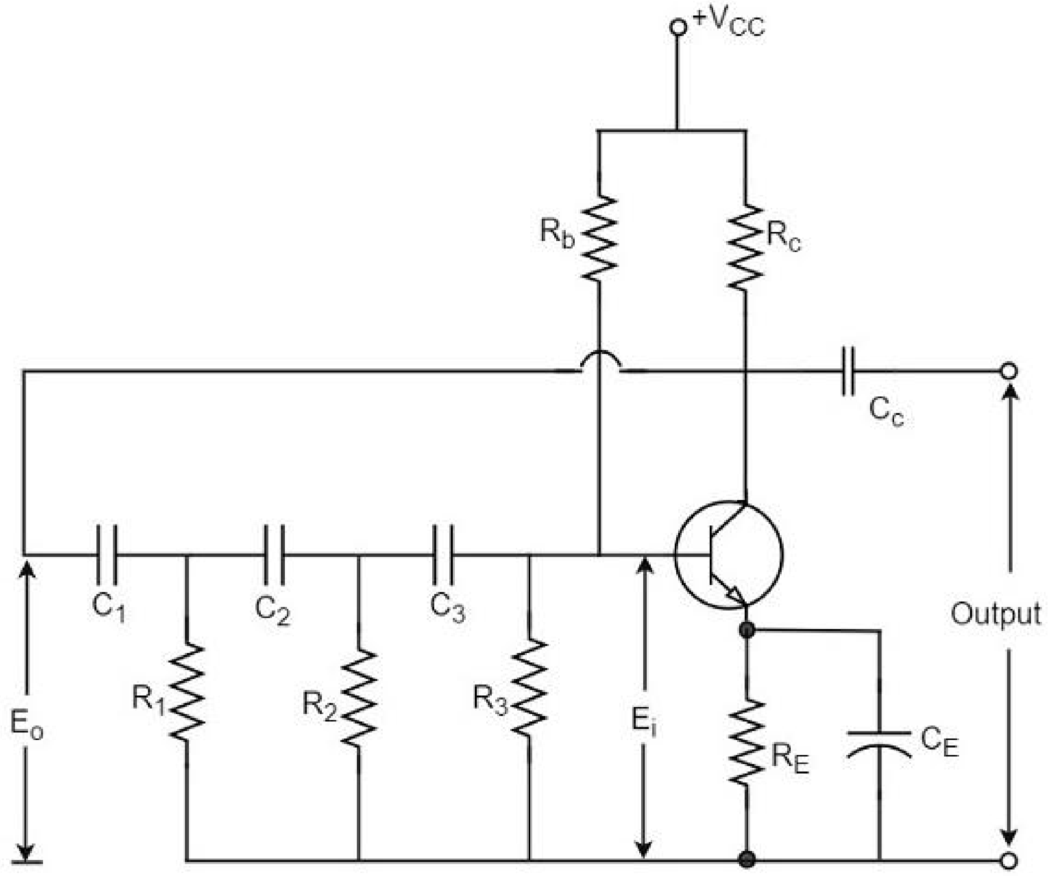
RC Phase-shift Oscillator Circuit

The oscillator circuit that produces a sine wave using a phase-shift network is called as a Phase-shift oscillator circuit. The constructional details and operation of a phase-shift oscillator circuit are as given below.

Construction

The phase-shift oscillator circuit consists of a single transistor amplifier section and a RC phase-shift network. The phase shift network in this circuit, consists of three RC sections. At the resonant frequency fo, the phase shift in each RC section is 60o so that the total phase shift produced by RC network is 180o.

The following circuit diagram shows the arrangement of an RC phase-shift oscillator.



The frequency of oscillations is given by

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

Where

R1=R2=R3=R

C1=C2=C3=C

Operation

The circuit when switched ON oscillates at the resonant frequency fo. The output Eo of the amplifier is fed back to RC feedback network. This network produces a phase shift of 180^0 and a voltage Ei appears at its output. This voltage is applied to the transistor amplifier.

The feedback applied will be

$$m = Ei/Eo$$

The feedback is in correct phase, whereas the transistor amplifier, which is in CE configuration, produces a 180o phase shift. The phase shift produced by network and the transistor add to form a phase shift around the entire loop which is 360^0.

Advantages

The advantages of RC phase shift oscillator are as follows −

- It does not require transformers or inductors.
- It can be used to produce very low frequencies.
- The circuit provides good frequency stability.

Disadvantages

The disadvantages of RC phase shift oscillator are as follows −

- Starting the oscillations is difficult as the feedback is small.
- The output produced is small.

Wien Bridge Oscillator

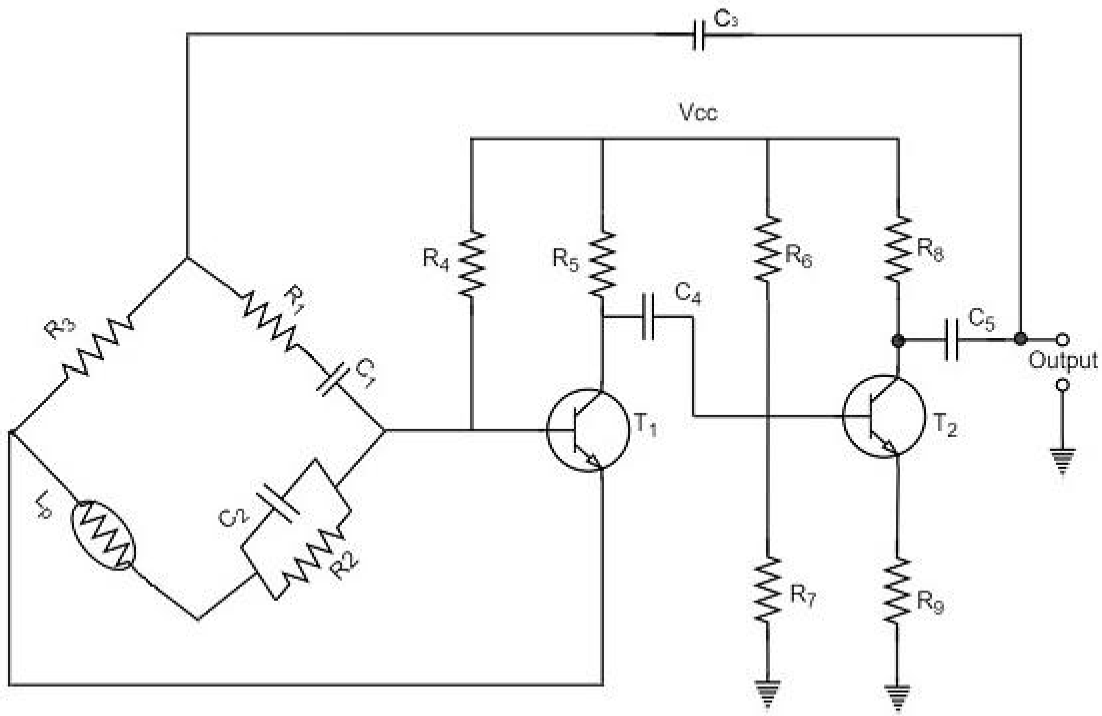
Another type of popular audio frequency oscillator is the Wien bridge oscillator circuit. This is mostly used because of its important features. This circuit is free from the circuit fluctuations and the ambient temperature.

The main advantage of this oscillator is that the frequency can be varied in the range of 10Hz to about 1MHz whereas in RC oscillators, the frequency is not varied.

Construction

The circuit construction of Wien bridge oscillator can be explained as below. It is a two-stage amplifier with RC bridge circuit. The bridge circuit has the arms R1C1, R3, R2C2 and the tungsten lamp Lp. Resistance R3 and the lamp Lp are used to stabilize the amplitude of the output.

The following circuit diagram shows the arrangement of a Wien bridge oscillator.



The transistor T1 serves as an oscillator and an amplifier while the other transistor T2 serves as an inverter. The inverter operation provides a phase shift of 180o. This circuit provides positive feedback through R1C1, C2R2 to the transistor T1 and negative feedback through the voltage divider to the input of transistor T2.

The frequency of oscillations is determined by the series element R1C1 and parallel element R2C2 of the bridge.

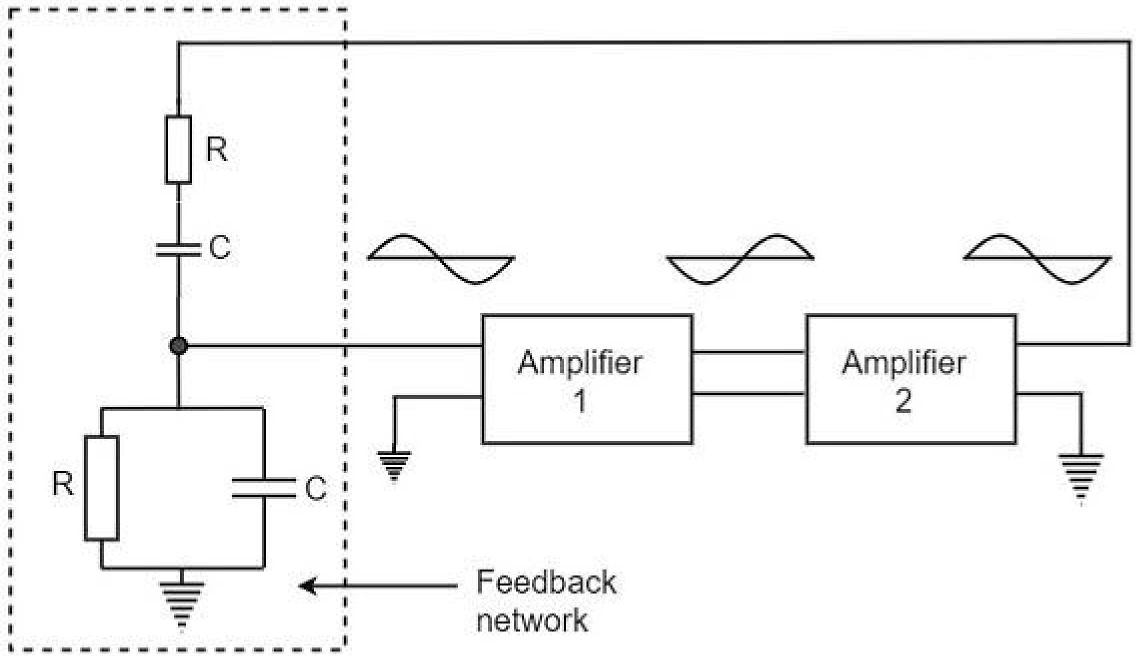
$$f=1/2\pi\sqrt{R_1C_1R_2C_2}$$

If $R_1 = R_2 = R$ and $C_1 = C_2 = C$

Then,

$$f=1/2\pi RC$$

Now, we can simplify the above circuit as follows −



The oscillator consists of two stages of RC coupled amplifier and a feedback network. The voltage across the parallel combination of R and C is fed to the input of amplifier 1. The net phase shift through the two amplifiers is zero.

The usual idea of connecting the output of amplifier 2 to amplifier 1 to provide signal regeneration for oscillator is not applicable here as the amplifier 1 will amplify signals over a wide range of frequencies and hence direct coupling would result in poor frequency stability. By adding Wien bridge feedback network, the oscillator becomes sensitive to a particular frequency and hence frequency stability is achieved.

Operation

When the circuit is switched ON, the bridge circuit produces oscillations of the frequency stated above. The two transistors produce a total phase shift of 360^0 so that proper positive feedback is ensured. The negative feedback in the circuit ensures constant output. This is achieved by temperature sensitive tungsten lamp Lp. Its resistance increases with current.

If the amplitude of the output increases, more current is produced and more negative feedback is achieved. Due to this, the output would return to the original value. Whereas, if the output tends to decrease, reverse action would take place.

Advantages

The advantages of Wien bridge oscillator are as follows −

- The circuit provides good frequency stability.
- It provides constant output.
- The operation of circuit is quite easy.
- The overall gain is high because of two transistors.
- The frequency of oscillations can be changed easily.
- The amplitude stability of the output voltage can be maintained more accurately, by replacing R2 with a thermistor.

Disadvantages

The disadvantages of Wien bridge oscillator are as follows −

- The circuit cannot generate very high frequencies.
- Two transistors and number of components are required for the circuit construction

Hartley Oscillator

A very popular local oscillator circuit that is mostly used in radio receivers is the Hartley Oscillator circuit. The constructional details and operation of a Hartley oscillator are as discussed below.

Construction

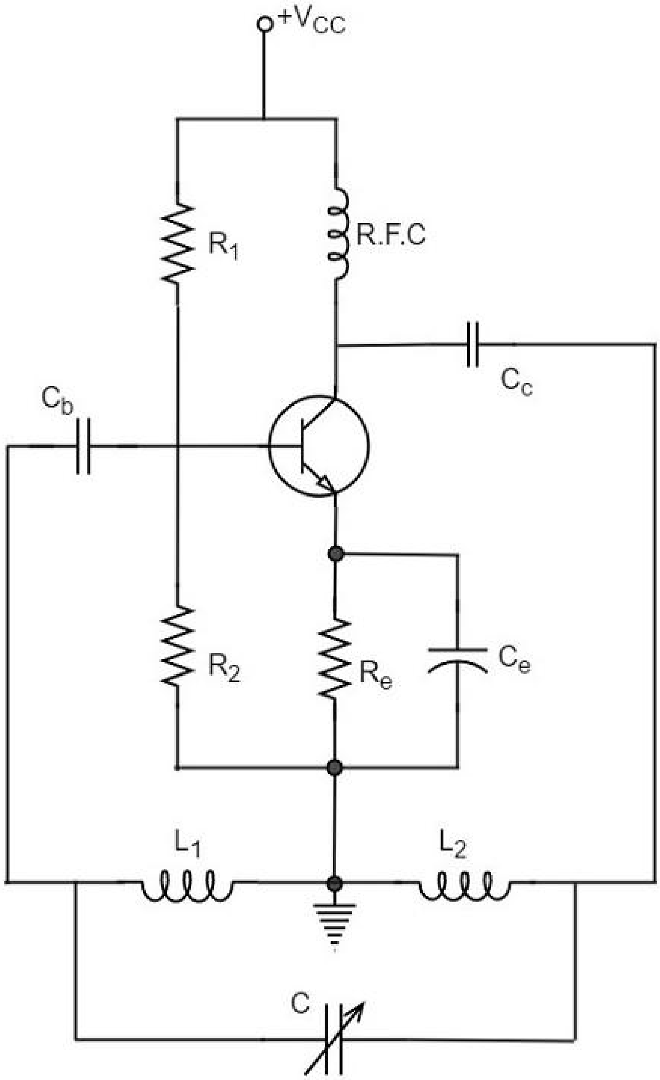
In the circuit diagram of a Hartley oscillator shown below, the resistors R1, R2 and Re provide necessary bias condition for the circuit. The capacitor Ce provides a.c. ground thereby providing any signal degeneration. This also provides temperature stabilization.

The capacitors Cc and Cb are employed to block d.c. and to provide an a.c. path. The radio frequency choke (R.F.C) offers very high impedance to high frequency currents which means it shorts for d.c. and opens for a.c. Hence it provides d.c. load for collector and keeps a.c. currents out of d.c. supply source

Tank Circuit

The frequency determining network is a parallel resonant circuit which consists of the inductors L1 and L2 along with a variable capacitor C. The junction of L1 and L2 are earthed. The coil L1 has its one end connected to base via Cc and the other to emitter via Ce. So, L2 is in the output circuit. Both the coils L1 and L2 are inductively coupled and together form an Auto-transformer.

The following circuit diagram shows the arrangement of a Hartley oscillator. The tank circuit is shunt fed in this circuit. It can also be a series-fed.



Operation

When the collector supply is given, a transient current is produced in the oscillatory or tank circuit. The oscillatory current in the tank circuit produces a.c. voltage across L1.

The auto-transformer made by the inductive coupling of L1 and L2 helps in determining the frequency and establishes the feedback. As the CE configured transistor provides 180o phase shift, another 180o phase shift is provided by the transformer, which makes 360o phase shift between the input and output voltages.

This makes the feedback positive which is essential for the condition of oscillations. When the loop gain |βA| of the amplifier is greater than one, oscillations are sustained in the circuit.

Frequency

The equation for frequency of Hartley oscillator is given as

$f=1/2\pi\sqrt{L_T C}$

Where $L_T=L1+L2+2M$

Here,

L_T is the total cumulatively coupled inductance;

L1 and L2 represent inductances of 1st and 2nd coils; and M represents mutual inductance.

Mutual inductance is calculated when two windings are considered.

Advantages

The advantages of Hartley oscillator are

- Instead of using a large transformer, a single coil can be used as an auto-transformer.
- Frequency can be varied by employing either a variable capacitor or a variable inductor.
- Less number of components are sufficient.
- The amplitude of the output remains constant over a fixed frequency range.

Disadvantages

- The disadvantages of Hartley oscillator are
- It cannot be a low frequency oscillator.
- Harmonic distortions are present.

Applications

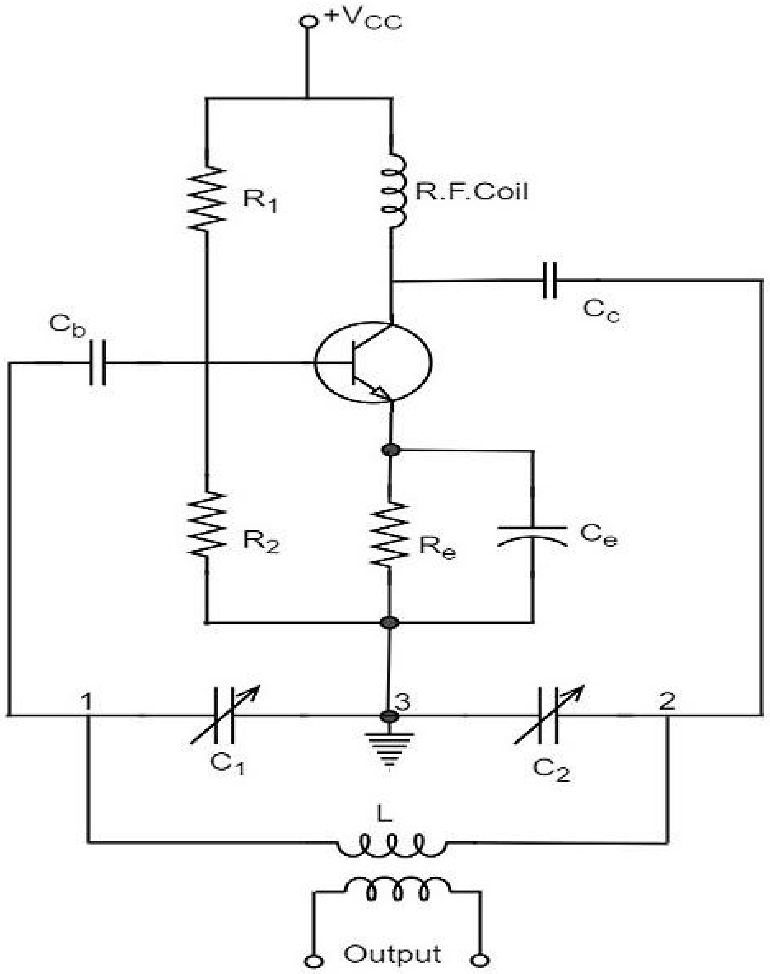
- The applications of Hartley oscillator are
- It is used to produce a sinewave of desired frequency.
- Mostly used as a local oscillator in radio receivers.
- It is also used as R.F. Oscillator.

Colpitts Oscillator

A Colpitts oscillator looks just like the Hartley oscillator but the inductors and capacitors are replaced with each other in the tank circuit. The constructional details and operation of a colpitts oscillator are as discussed below.

Construction

Let us first take a look at the circuit diagram of a Colpitts oscillator.



The resistors R1, R2 and Re provide necessary bias condition for the circuit. The capacitor Ce provides a.c. ground thereby providing any signal degeneration. This also provides temperature stabilization.
The capacitors Cc and Cb are employed to block d.c. and to provide an a.c. path. The radio frequency choke (R.F.C) offers very high impedance to high frequency currents which means it shorts for d.c. and opens for a.c. Hence it provides d.c. load for collector and keeps a.c. currents out of d.c. supply source.

Tank Circuit

The frequency determining network is a parallel resonant circuit which consists of variable capacitors C1 and C2 along with an inductor L. The junction of C1 and C2 are earthed. The capacitor C1 has its one end connected to base via Cc and the other to emitter via Ce. the voltage developed across C1 provides the regenerative feedback required for the sustained oscillations.

Operation

When the collector supply is given, a transient current is produced in the oscillatory or tank circuit. The oscillatory current in the tank circuit produces a.c. voltage across C1 which are applied to the base emitter junction and appear in the amplified form in the collector circuit and supply losses to the tank circuit.

If terminal 1 is at positive potential with respect to terminal 3 at any instant, then terminal 2 will be at negative potential with respect to 3 at that instant because terminal 3 is grounded. Therefore, points 1 and 2 are out of phase by 180o.
As the CE configured transistor provides 180o phase shift, it makes 360o phase shift between the input and output voltages. Hence, feedback is properly phased to produce continuous Undamped oscillations. When the loop gain $|\beta A|$ of the amplifier is greater than one, oscillations are sustained in the circuit.

Frequency

The equation for frequency of Colpitts oscillator is given as

$f=1/2\pi\sqrt{LC_T}$

C_T is the total capacitance of C1 and C2 connected in series.

$1/C_T=1/C_1+1/C_2$

$C_T=C_1\times C_2/(C1+C2)$

Advantages

The advantages of Colpitts oscillator are as follows −

- Colpitts oscillator can generate sinusoidal signals of very high frequencies.
- It can withstand high and low temperatures.
- The frequency stability is high.
- Frequency can be varied by using both the variable capacitors.
- Less number of components are sufficient.
- The amplitude of the output remains constant over a fixed frequency range.

The Colpitts oscillator is designed to eliminate the disadvantages of Hartley oscillator and is known to have no specific disadvantages. Hence there are many applications of a colpitts oscillator.

Applications

The applications of Colpitts oscillator are as follows −

- Colpitts oscillator can be used as High frequency sinewave generator.
- This can be used as a temperature sensor with some associated circuitry.
- Mostly used as a local oscillator in radio receivers.
- It is also used as R.F. Oscillator.
- It is also used in Mobile applications.
- It has got many other commercial applications.

Whenever an oscillator is under continuous operation, its frequency stability gets affected. There occur changes in its frequency. The main factors that affect the frequency of an oscillator are

- Power supply variations
- Changes in temperature
- Changes in load or output resistance

In RC and LC oscillators the values of resistance, capacitance and inductance vary with temperature and hence the frequency gets affected. In order to avoid this problem, the piezo electric crystals are being used in oscillators.

The use of piezo electric crystals in parallel resonant circuits provide high frequency stability in oscillators. Such oscillators are called as Crystal Oscillators.

Crystal Oscillators

The principle of crystal oscillators depends upon the Piezo electric effect. The natural shape of a crystal is hexagonal. When a crystal wafer is cur perpendicular to X-axis, it is called as X-cut and when it is cut along Y-axis, it is called as Y-cut.

The crystal used in crystal oscillator exhibits a property called as Piezo electric property. So, let us have an idea on piezo electric effect.

Piezo Electric Effect

The crystal exhibits the property that when a mechanical stress is applied across one of the faces of the crystal, a potential difference is developed across the opposite faces of the crystal. Conversely, when a potential difference is applied across one of the faces, a mechanical stress is produced along the other faces. This is known as Piezo electric effect.

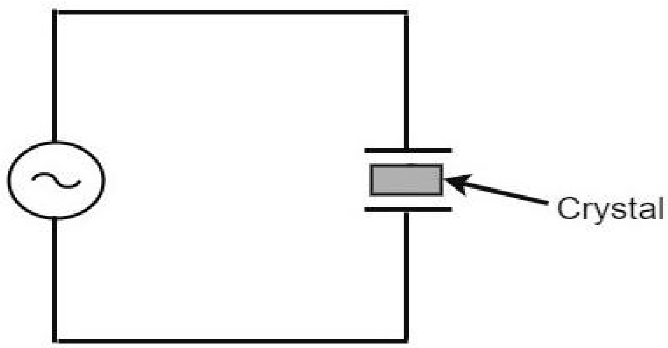
Certain crystalline materials like Rochelle salt, quartz and tourmaline exhibit piezo electric effect and such materials are called as Piezo electric crystals. Quartz is the most commonly used piezo electric crystal because it is inexpensive and readily available in nature.

When a piezo electric crystal is subjected to a proper alternating potential, it vibrates mechanically. The amplitude of mechanical vibrations becomes maximum when the frequency of alternating voltage is equal to the natural frequency of the crystal.

Working of a Quartz Crystal

In order to make a crystal work in an electronic circuit, the crystal is placed between two metal plates in the form of a capacitor. Quartz is the mostly used type of crystal because of its availability and strong nature while being inexpensive. The ac voltage is applied in parallel to the crystal.

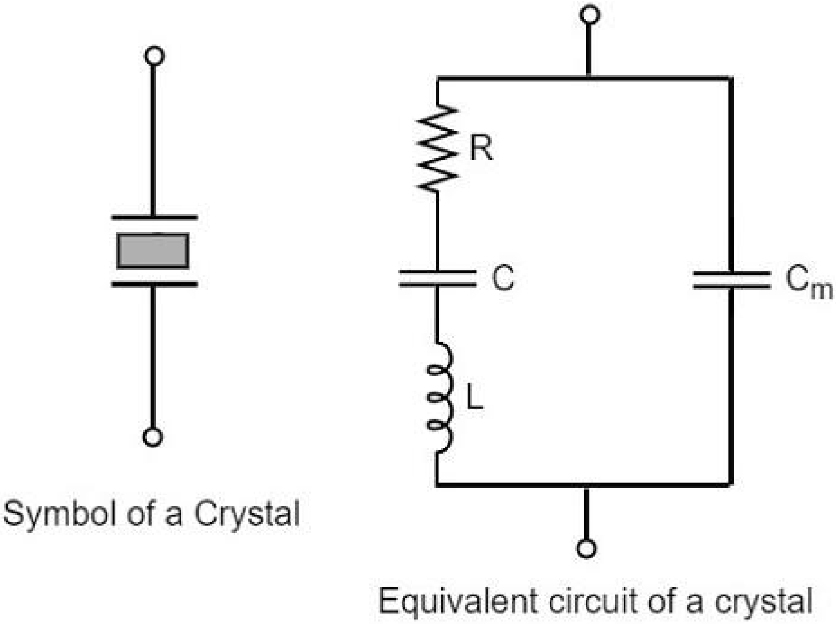
The circuit arrangement of a Quartz Crystal will be as shown below −



If an AC voltage is applied, the crystal starts vibrating at the frequency of the applied voltage. However, if the frequency of the applied voltage is made equal to the natural frequency of the crystal, resonance takes place and crystal vibrations reach a maximum value. This natural frequency is almost constant.

Equivalent circuit of a Crystal

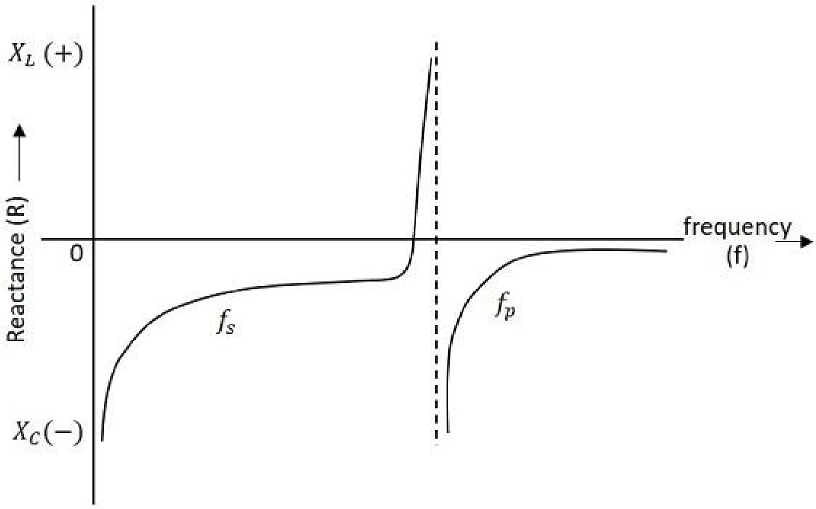
If we try to represent the crystal with an equivalent electric circuit, we have to consider two cases, i.e., when it vibrates and when it doesn't. The figures below represent the symbol and electrical equivalent circuit of a crystal respectively.



The above equivalent circuit consists of a series R-L-C circuit in parallel with a capacitance Cm. When the crystal mounted across the AC source is not vibrating, it is equivalent to the capacitance Cm. When the crystal vibrates, it acts like a tuned R-L-C circuit.

Frequency response

The frequency response of a crystal is as shown below. The graph shows the reactance (X_L or X_C) versus frequency (f). It is evident that the crystal has two closely spaced resonant frequencies.



The first one is the series resonant frequency (fs), which occurs when reactance of the inductance (L) is equal to the reactance of the capacitance C. In that case, the impedance of the equivalent circuit is equal to the resistance R and the frequency of oscillation is given by the relation,

$$f=1/2\pi\sqrt{L.C}$$

The second one is the parallel resonant frequency (fp), which occurs when the reactance of R-L-C branch is equal to the reactance of capacitor Cm. At this frequency, the crystal offers a very high impedance to the external circuit and the frequency of oscillation is given by the relation.

$$fp=1/2\pi\sqrt{L.C_T}$$

Where

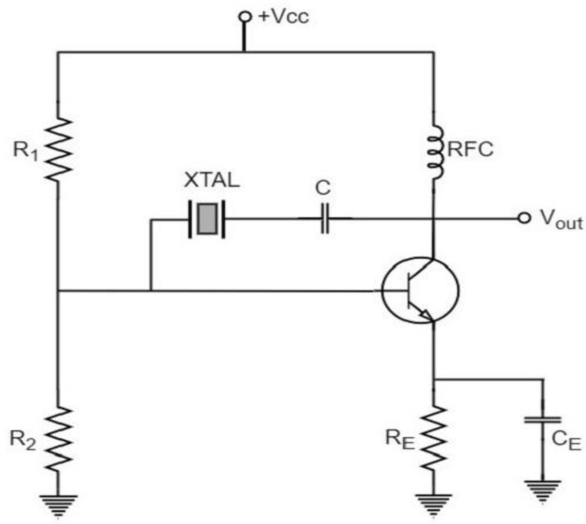
$$C_T=CCm/(C+Cm)$$

The value of Cm is usually very large as compared to C. Therefore, the value of CT is approximately equal to C and hence the series resonant frequency is approximately equal to the parallel resonant frequency (i.e., fs = fp).

Crystal Oscillator Circuit

A crystal oscillator circuit can be constructed in a number of ways like a Crystal controlled tuned collector oscillator, a Colpitts crystal oscillator, a Clap crystal oscillator etc. But the transistor pierce crystal oscillator is the most commonly used one. This is the circuit which is normally referred as a crystal oscillator circuit.

The following circuit diagram shows the arrangement of a transistor pierce crystal oscillator.



In this circuit, the crystal is connected as a series element in the feedback path from collector to the base. The resistors R1, R2 and R_E provide a voltage-divider stabilized d.c. bias circuit. The capacitor C_E provides a.c. bypass of the emitter resistor and RFC (radio frequency choke) coil provides for d.c. bias while decoupling any a.c. signal on the power lines from affecting the output signal. The coupling capacitor C has negligible impedance at the circuit operating frequency. But it blocks any d.c. between collector and base.

The circuit frequency of oscillation is set by the series resonant frequency of the crystal and its value is given by the relation,

$$fo=1/2\pi\sqrt{L.C}$$

It may be noted that the changes in supply voltage, transistor device parameters etc. have no effect on the circuit operating frequency, which is held stabilized by the crystal.

Advantages

- The advantages of crystal oscillator are as follows −
- They have a high order of frequency stability.
- The quality factor (Q) of the crystal is very high.

Disadvantages

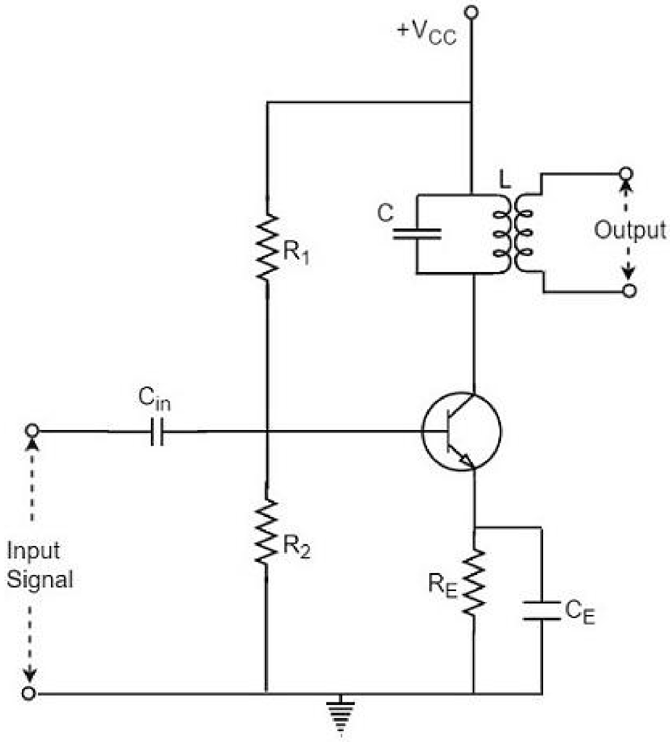
- The disadvantages of crystal oscillator are as follows −
- They are fragile and can be used in low power circuits.
- The frequency of oscillations cannot be changed appreciably.

Unit-5: TUNED AMPLIFIER & WAVE SHAPING CIRCUIT

What is a Tuned Amplifier?

Tuned amplifiers are the amplifiers that are employed for the purpose of tuning. Tuning means selecting. Among a set of frequencies available, if there occurs a need to select a particular frequency, while rejecting all other frequencies, such a process is called Selection. This selection is done by using a circuit called as **Tuned circuit**.

When an amplifier circuit has its load replaced by a tuned circuit, such an amplifier can be called as a Tuned amplifier circuit. The basic tuned amplifier circuit looks as shown below.



The tuner circuit is nothing but a LC circuit which is also called as resonant or tank circuit. It selects the frequency. A tuned circuit is capable of amplifying a signal over a narrow band of frequencies that are centered at resonant frequency.

When the reactance of the inductor balances the reactance of the capacitor, in the tuned circuit at some frequency, such a frequency can be called as resonant frequency. It is denoted by fr.

The formula for resonance is

$$2\pi fL=1/2\pi fc$$

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

Types of Tuned Circuits

A tuned circuit can be Series tuned circuit (Series resonant circuit) or Parallel tuned circuit (parallel resonant circuit) according to the type of its connection to the main circuit.

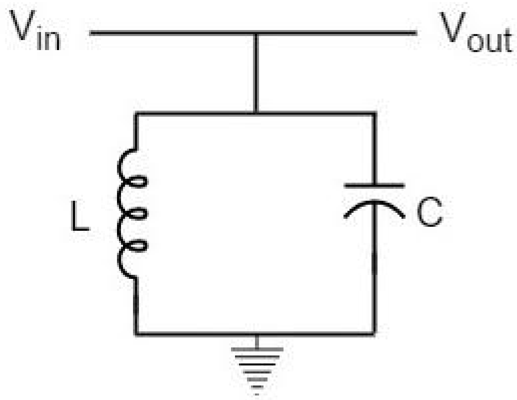
Series Tuned Circuit

The inductor and capacitor connected in series make a series tuned circuit, as shown in the following circuit diagram.

At resonant frequency, a series resonant circuit offers low impedance which allows high current through it. A series resonant circuit offers increasingly high impedance to the frequencies far from the resonant frequency.

Parallel Tuned Circuit

The inductor and capacitor connected in parallel make a parallel tuned circuit, as shown in the below figure.



At resonant frequency, a parallel resonant circuit offers high impedance which does not allow high current through it. A parallel resonant circuit offers increasingly low impedance to the frequencies far from the resonant frequency.

Characteristics of a Parallel Tuned Circuit

The frequency at which parallel resonance occurs (i.e. reactive component of circuit current becomes zero) is called the resonant frequency fr. The main characteristics of a tuned circuit are as follows.

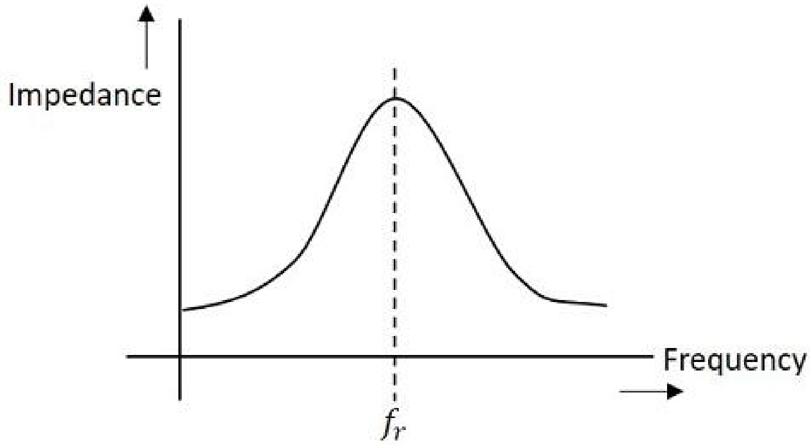
Impedance

The ratio of supply voltage to the line current is the impedance of the tuned circuit. Impedance offered by LC circuit is given by

Supply voltage/Line equation=V/I

At resonance, the line current increases while the impedance decreases.

The below figure represents the impedance curve of a parallel resonance circuit.



Impedance of the circuit decreases for the values above and below the resonant frequency fr. Hence the selection of a particular frequency and rejection of other frequencies is possible.

To obtain an equation for the circuit impedance, let us consider

$$\text{Line Current } I = I_L \cos \phi$$

$$V/Zr = V/Z_L \times R/Z_L$$

$$1/Zr = R/Z_L^2$$

$$1/Zr = R/(L/C) = CR/L$$

$$\text{Since, } Z_L^2 = L/C$$

Therefore, circuit impedance Zr is obtained as

$$Z_R = L/CR$$

Thus at parallel resonance, the circuit impedance is equal to L/CR.

Circuit Current

At parallel resonance, the circuit or line current I is given by the applied voltage divided by the circuit impedance Zr i.e.,

$$\text{Line Current } I = V/Zr$$

$$\text{Where } Zr = L/CR$$

Because Zr is very high, the line current I will be very small.

Quality Factor

For a parallel resonance circuit, the sharpness of the resonance curve determines the selectivity. The smaller the resistance of the coil, the sharper the resonant curve will be. Hence the inductive reactance and resistance of the coil determine the quality of the tuned circuit.

The ratio of inductive reactance of the coil at resonance to its resistance is known as Quality factor. It is denoted by Q.

$$Q = X_L/R = 2\pi f r L/R$$

The higher the value of Q, the sharper the resonance curve and the better the selectivity will be.

Advantages of Tuned Amplifiers

The following are the advantages of tuned amplifiers.

- The usage of reactive components like L and C, minimizes the power loss, which makes the tuned amplifiers efficient.
- The selectivity and amplification of desired frequency is high, by providing higher impedance at resonant frequency.
- A smaller collector supply VCC would do, because of its little resistance in parallel tuned circuit.

It is important to remember that these advantages are not applicable when there is a high resistive collector load.

Frequency Response of Tuned Amplifier

For an amplifier to be efficient, its gain should be high. This voltage gain depends upon β, input impedance and collector load. The collector load in a tuned amplifier is a tuned circuit.

The voltage gain of such an amplifier is given by

$$\textbf{Voltage gain} = \beta Z_C/Zin$$

Where Z_C = effective collector load and Zin = input impedance of the amplifier.

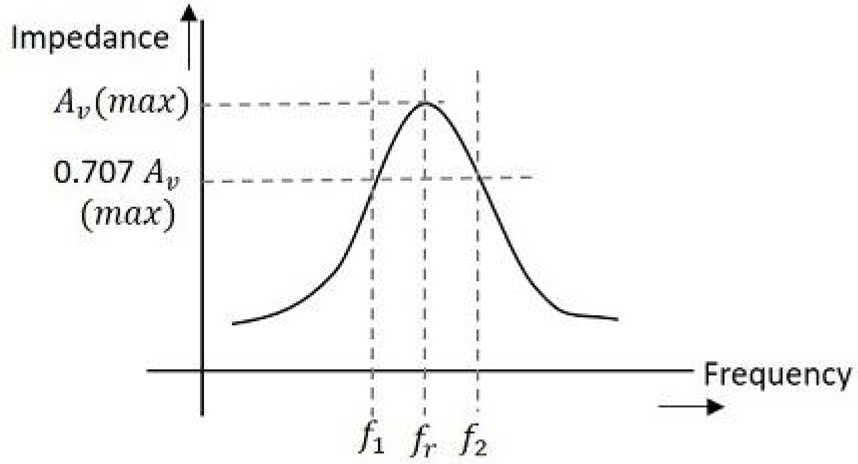
The value of Z_C depends upon the frequency of the tuned amplifier. As Z_C is maximum at resonant frequency, the gain of the amplifier is maximum at this resonant frequency.

Bandwidth

The range of frequencies at which the voltage gain of the tuned amplifier falls to 70.7% of the maximum gain is called its Bandwidth.

The range of frequencies between f1 and f2 is called as bandwidth of the tuned amplifier. The bandwidth of a tuned amplifier depends upon the Q of the LC circuit i.e., upon the sharpness of the frequency response. The value of Q and the bandwidth are inversely proportional.

The figure below details the bandwidth and frequency response of the tuned amplifier.



Relation between Q and Bandwidth

The quality factor Q of the bandwidth is defined as the ratio of resonant frequency to bandwidth, i.e.,

$$Q=fr/BW$$

In general, a practical circuit has its Q value greater than 10.

Under this condition, the resonant frequency at parallel resonance is given by

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

There are two main types of tuned amplifiers. They are −

- **Single tuned amplifier**
- **Double tuned amplifier**

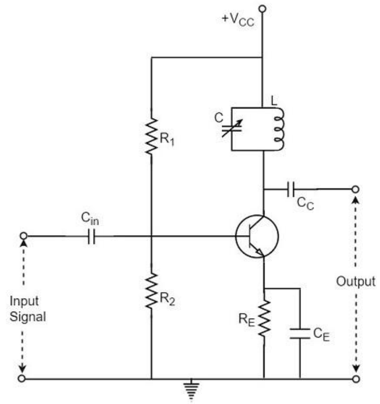
Single Tuned Amplifier

An amplifier circuit with a single tuner section being at the collector of the amplifier circuit is called as Single tuner amplifier circuit.

Construction

A simple transistor amplifier circuit consisting of a parallel tuned circuit in its collector load, makes a single tuned amplifier circuit. The values of capacitance and inductance of the tuned circuit are selected such that its resonant frequency is equal to the frequency to be amplified.

The following circuit diagram shows a single tuned amplifier circuit.



The output can be obtained from the coupling capacitor CC as shown above or from a secondary winding placed at L.

Operation

The high frequency signal that has to be amplified is applied at the input of the amplifier. The resonant frequency of the parallel tuned circuit is made equal to the frequency of the signal applied by altering the capacitance value of the capacitor C, in the tuned circuit.

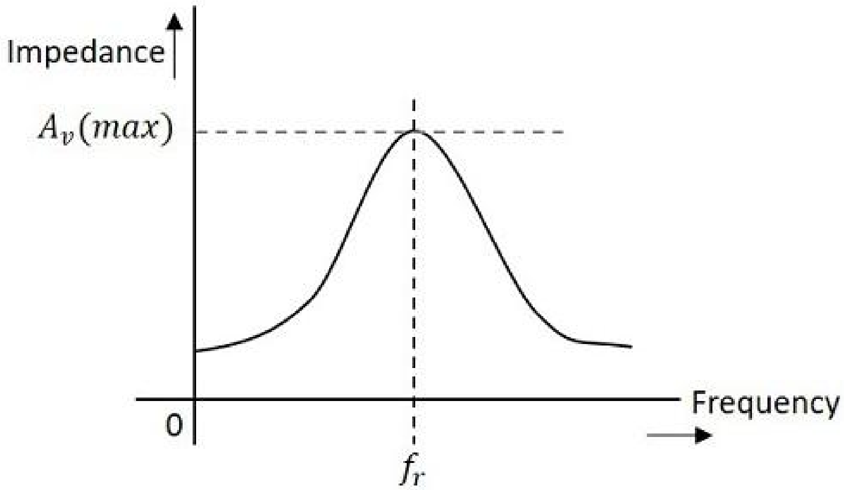
At this stage, the tuned circuit offers high impedance to the signal frequency, which helps to offer high output across the tuned circuit. As high impedance is offered only for the tuned frequency, all the other frequencies which get lower impedance are rejected by the tuned circuit. Hence the tuned amplifier selects and amplifies the desired frequency signal.

Frequency Response

The parallel resonance occurs at resonant frequency fr when the circuit has a high Q. the resonant frequency fr is given by

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

The following graph shows the frequency response of a single tuned amplifier circuit.



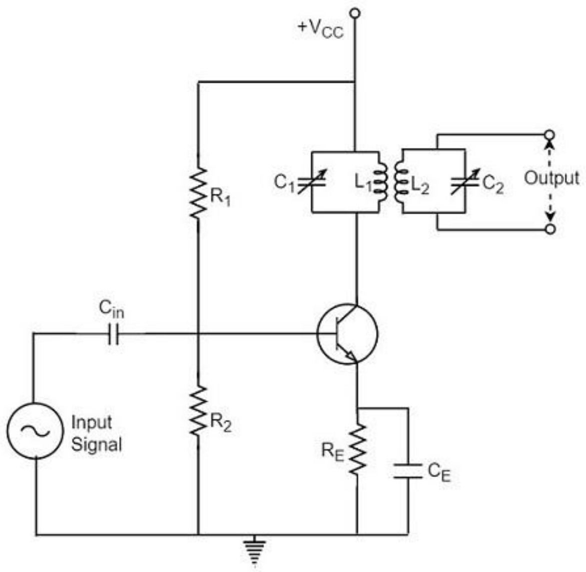
At resonant frequency fr the impedance of parallel tuned circuit is very high and is purely resistive. The voltage across RL is therefore maximum, when the circuit is tuned to resonant frequency. Hence the voltage gain is maximum at resonant frequency and drops off above and below it. The higher the Q, the narrower will the curve be.

Double Tuned Amplifier

An amplifier circuit with a double tuner section being at the collector of the amplifier circuit is called as Double tuner amplifier circuit.

Construction

The construction of double tuned amplifier is understood by having a look at the following figure. This circuit consists of two tuned circuits L1C1 and L2C2 in the collector section of the amplifier. The signal at the output of the tuned circuit L1C1 is coupled to the other tuned circuit L2C2 through mutual coupling method. The remaining circuit details are same as in the single tuned amplifier circuit, as shown in the following circuit diagram.



Operation

The high frequency signal which has to be amplified is given to the input of the amplifier. The tuning circuit L1C1 is tuned to the input signal frequency. At this condition, the tuned circuit offers high reactance to the signal frequency. Consequently, large output appears at the output of the tuned circuit L1C1 which is then coupled to the other tuned circuit L2C2 through mutual induction. These double tuned circuits are extensively used for coupling various circuits of radio and television receivers.

Frequency Response of Double Tuned Amplifier

The double tuned amplifier has the special feature of coupling which is important in determining the frequency response of the amplifier. The amount of mutual inductance between the two tuned circuits states the degree of coupling, which determines the frequency response of the circuit.

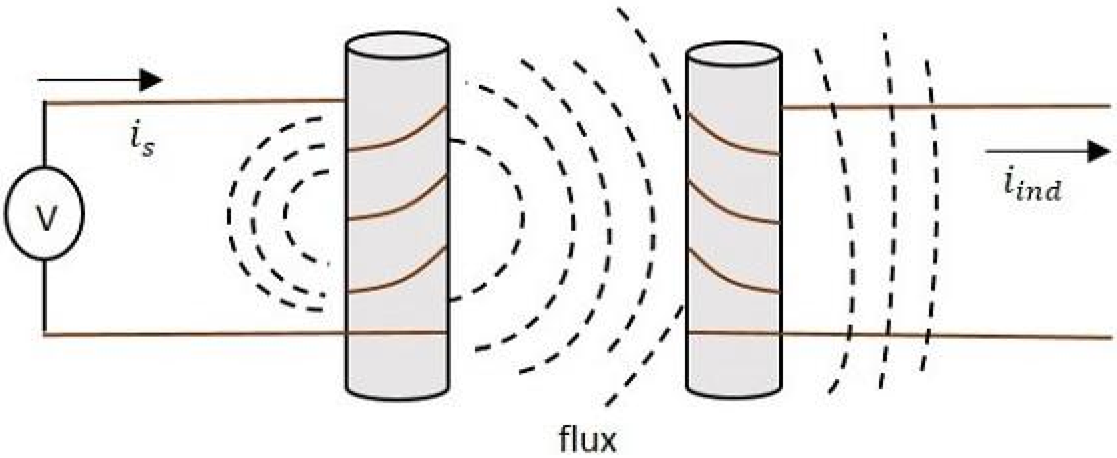
In order to have an idea on the mutual inductance property, let us go through the basic principle.

Mutual Inductance

As the current carrying coil produces some magnetic field around it, if another coil is brought near this coil, such that it is in the magnetic flux region of the primary, then the varying magnetic flux induces an EMF in the second coil. If this first coil is called as Primary coil, the second one can be called as a Secondary coil.

When the EMF is induced in the secondary coil due to the varying magnetic field of the primary coil, then such phenomenon is called as the Mutual Inductance.

The figure below gives an idea about this.

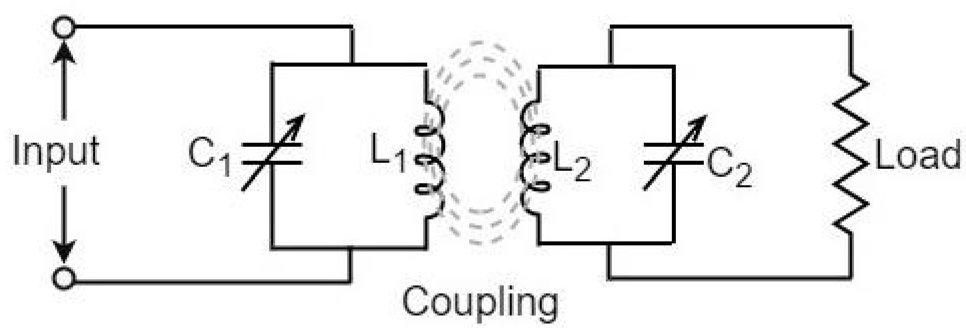


The current is in the figure indicate the source current while iind indicates the induced current. The flux represents the magnetic flux created around the coil. This spreads to the secondary coil also.

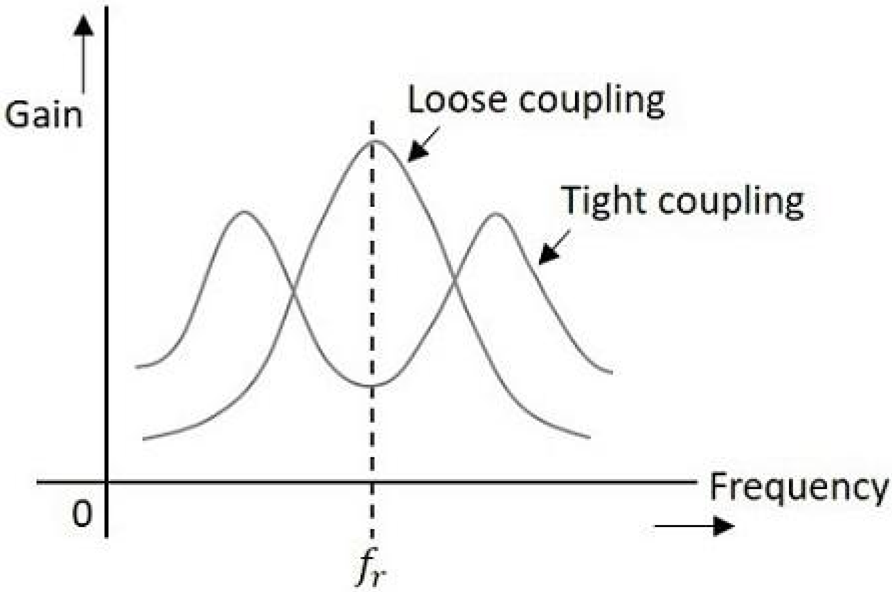
With the application of voltage, the current is flows and flux gets created. When the current is varies the flux gets varied, producing iind in the secondary coil, due to the Mutual inductance property.

Coupling

Under the concept of mutual inductance coupling will be as shown in the figure below.



When the coils are spaced apart, the flux linkages of primary coil L1 will not link the secondary coil L2. At this condition, the coils are said to have Loose coupling. The resistance reflected from the secondary coil at this condition is small and the resonance curve will be sharp and the circuit Q is high as shown in the figure below.



On the contrary, when the primary and secondary coils are brought close together, they have Tight coupling. Under such conditions, the reflected resistance will be large and the circuit Q is lower. Two positions of gain maxima, one above and the other below the resonant frequency are obtained.

Bandwidth of Double Tuned Circuit

The above figure clearly states that the bandwidth increases with the degree of coupling. The determining factor in a double tuned circuit is not Q but the coupling.

We understood that, for a given frequency, the tighter the coupling the greater the bandwidth will be.

The equation for bandwidth is given as

$$\mathbf{BW_{dt}=k\ fr}$$

Where BW_{dt} = bandwidth for double tuned circuit, K = coefficient of coupling, and fr = resonant frequency.

Wave shaping circuits are the electronic circuits, which produce the desired shape at the output from the applied input wave form. These circuits perform two functions −

Attenuate the applied wave

Alter the dc level of the applied wave.

There are two types of wave shaping circuits: **Clippers** and **Clampers**.

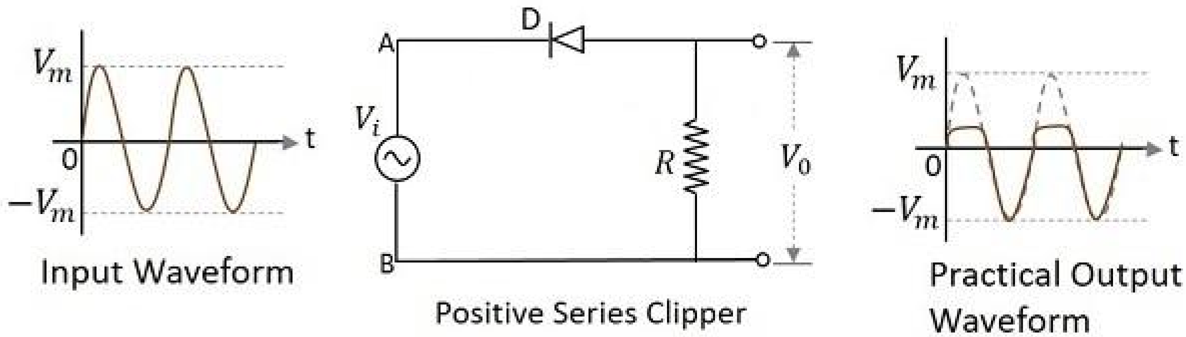
The Clipper circuit that is intended to attenuate positive portions of the input signal can be termed as a Positive Clipper. Among the positive diode clipper circuits, we have the following types −

- Positive Series Clipper
- Positive Series Clipper with positive Vr reference voltage
- Positive Series Clipper with negative Vr
- Positive Shunt Clipper
- Positive Shunt Clipper with positive Vr
- Positive Shunt Clipper with negative Vr

Let us discuss each of these types in detail.

Positive Series Clipper

A Clipper circuit in which the diode is connected in series to the input signal and that attenuates the positive portions of the waveform, is termed as Positive Series Clipper. The following figure represents the circuit diagram for positive series clipper.

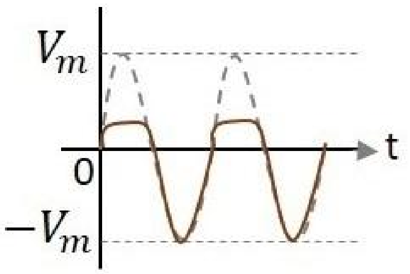


Positive Cycle of the Input − When the input voltage is applied, the positive cycle of the input makes the point A in the circuit positive with respect to the point B. This makes the diode reverse biased and hence it behaves like an open switch. Thus the voltage across the load resistor becomes zero as no current flows through it and hence V_0 will be zero.

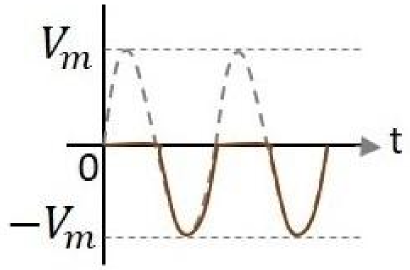
Negative Cycle of the Input − The negative cycle of the input makes the point A in the circuit negative with respect to the point B. This makes the diode forward biased and hence it conducts like a closed switch. Thus the voltage across the load resistor will be equal to the applied input voltage as it completely appears at the output V_0.

Waveforms

In the above figures, if the waveforms are observed, we can understand that only a portion of the positive peak was clipped. This is because of the voltage across V_0. But the ideal output was not meant to be so. Let us have a look at the following figures.



Practical Output Waveform

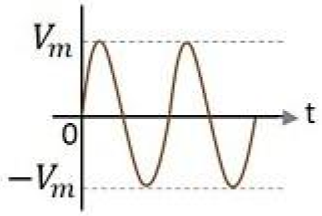


Ideal Output Waveform

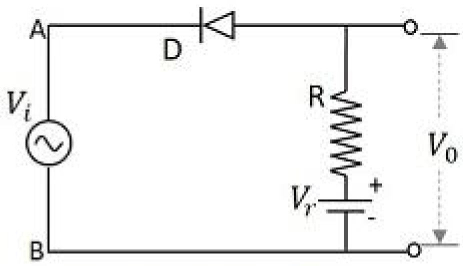
Unlike the ideal output, a bit portion of the positive cycle is present in the practical output due to the diode conduction voltage which is 0.7v. Hence there will be a difference in the practical and ideal output waveforms.

Positive Series Clipper with positive Vr

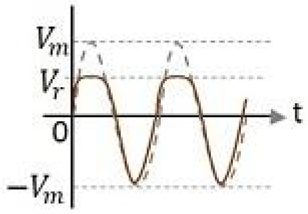
A Clipper circuit in which the diode is connected in series to the input signal and biased with positive reference voltage Vr and that attenuates the positive portions of the waveform, is termed as Positive Series Clipper with positive Vr. The following figure represents the circuit diagram for positive series clipper when the reference voltage applied is positive.



Input Waveform



Positive Series Clipper with positive V_r

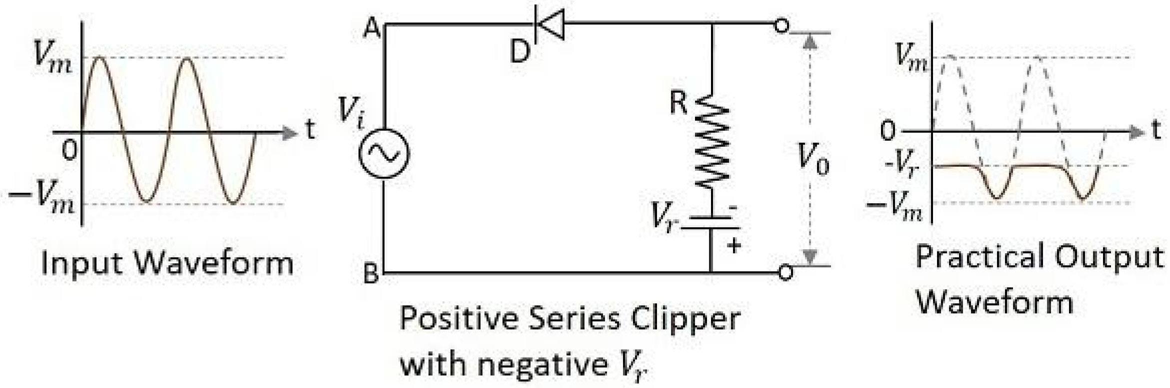


Practical Output Waveform

During the positive cycle of the input the diode gets reverse biased and the reference voltage appears at the output. During its negative cycle, the diode gets forward biased and conducts like a closed switch. Hence the output waveform appears as shown in the above figure.

Positive Series Clipper with negative Vr

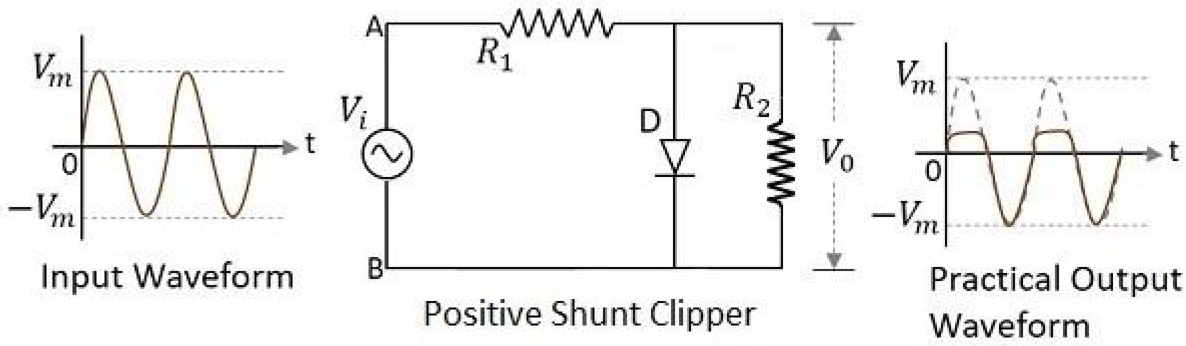
A Clipper circuit in which the diode is connected in series to the input signal and biased with negative reference voltage Vr and that attenuates the positive portions of the waveform, is termed as Positive Series Clipper with negative Vr. The following figure represents the circuit diagram for positive series clipper, when the reference voltage applied is negative.



During the positive cycle of the input the diode gets reverse biased and the reference voltage appears at the output. As the reference voltage is negative, the same voltage with constant amplitude is shown. During its negative cycle, the diode gets forward biased and conducts like a closed switch. Hence the input signal that is greater than the reference voltage, appears at the output.

Positive Shunt Clipper

A Clipper circuit in which the diode is connected in shunt to the input signal and that attenuates the positive portions of the waveform, is termed as Positive Shunt Clipper. The following figure represents the circuit diagram for positive shunt clipper.

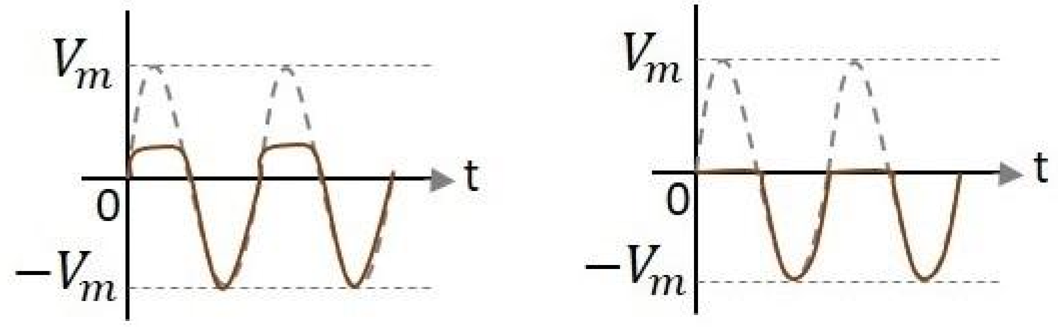


Positive Cycle of the Input − When the input voltage is applied, the positive cycle of the input makes the point A in the circuit positive with respect to the point B. This makes the diode forward biased and hence it conducts like a closed switch. Thus the voltage across the load resistor becomes zero as no current flows through it and hence V_0 will be zero.

Negative Cycle of the Input − The negative cycle of the input makes the point A in the circuit negative with respect to the point B. This makes the diode reverse biased and hence it behaves like an open switch. Thus the voltage across the load resistor will be equal to the applied input voltage as it completely appears at the output V_0.

Waveforms

In the above figures, if the waveforms are observed, we can understand that only a portion of the positive peak was clipped. This is because of the voltage across V_0. But the ideal output was not meant to be so. Let us have a look at the following figures.

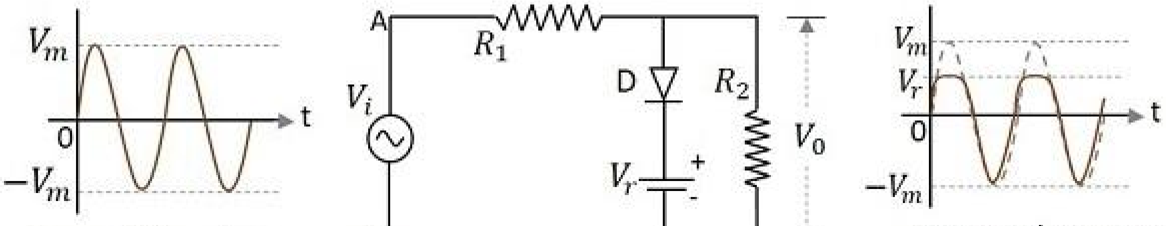


Practical Output Waveform   Ideal Output Waveform

Unlike the ideal output, a bit portion of the positive cycle is present in the practical output due to the diode conduction voltage which is 0.7v. Hence there will be a difference in the practical and ideal output waveforms.

Positive Shunt Clipper with positive Vr

A Clipper circuit in which the diode is connected in shunt to the input signal and biased with positive reference voltage Vr and that attenuates the positive portions of the waveform, is termed as Positive Shunt Clipper with positive Vr. The following figure represents the circuit diagram for positive shunt clipper when the reference voltage applied is positive.



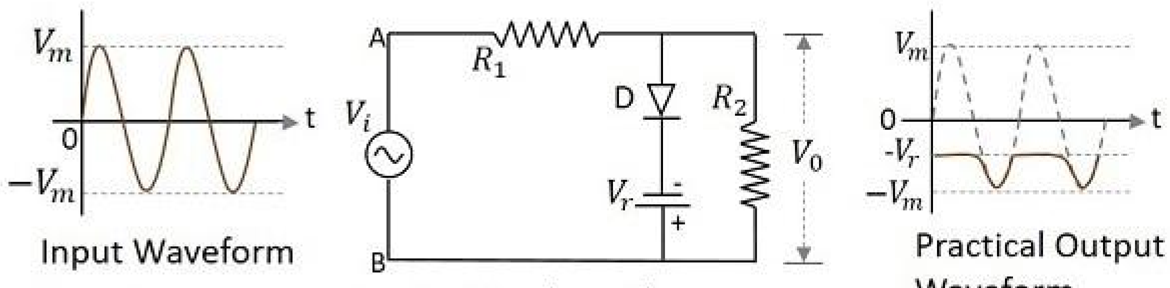
Input Waveform   Positive Shunt Clipper with positive V_r Practical Output Waveform

During the positive cycle of the input the diode gets forward biased and nothing but the reference voltage appears at the output. During its negative cycle, the diode gets reverse biased and behaves as an open switch. The whole of the input appears at the output. Hence the output waveform appears as shown in the above figure.

Positive Shunt Clipper with negative Vr

A Clipper circuit in which the diode is connected in shunt to the input signal and biased with negative reference voltage Vr and that attenuates the positive portions of the waveform, is termed as Positive Shunt Clipper with negative Vr.

The following figure represents the circuit diagram for positive shunt clipper, when the reference voltage applied is negative.



Positive Shunt Clipper with negative V_r

During the positive cycle of the input, the diode gets forward biased and the reference voltage appears at the output. As the reference voltage is negative, the same voltage with constant amplitude is shown. During its negative cycle, the diode gets reverse biased and behaves as an open switch. Hence the input signal that is greater than the reference voltage, appears at the output.

The Clipper circuit that is intended to attenuate negative portions of the input signal can be termed as a Negative Clipper. Among the negative diode clipper circuits, we have the following types.

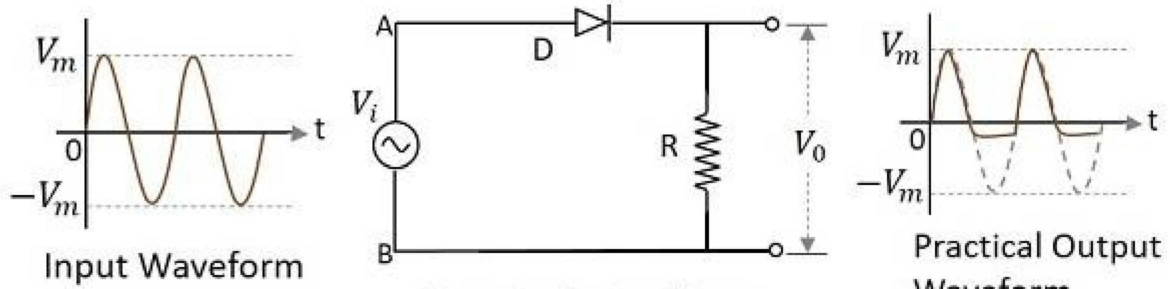
Negative Series Clipper

- Negative Series Clipper with positive Vr reference voltage
- Negative Series Clipper with negative Vr
- Negative Shunt Clipper
- Negative Shunt Clipper with positive Vr
- Negative Shunt Clipper with negative Vr

Let us discuss each of these types in detail.

Negative Series Clipper

A Clipper circuit in which the diode is connected in series to the input signal and that attenuates the negative portions of the waveform, is termed as Negative Series Clipper. The following figure represents the circuit diagram for negative series clipper.



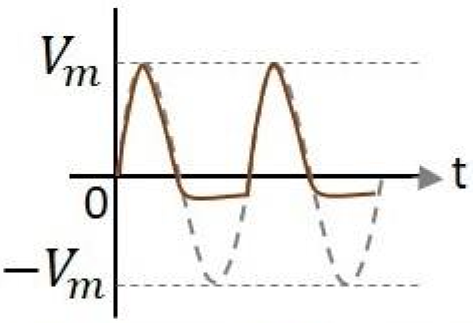
Negative Series Clipper

Positive Cycle of the Input − When the input voltage is applied, the positive cycle of the input makes the point A in the circuit positive with respect to the point B. This makes the diode forward biased and hence it acts like a closed switch. Thus the input voltage completely appears across the load resistor to produce the output V_0.

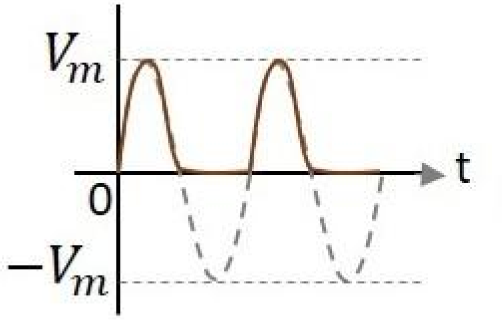
Negative Cycle of the Input − The negative cycle of the input makes the point A in the circuit negative with respect to the point B. This makes the diode reverse biased and hence it acts like an open switch. Thus the voltage across the load resistor will be zero making V_0 zero.

Waveforms

In the above figures, if the waveforms are observed, we can understand that only a portion of the negative peak was clipped. This is because of the voltage across V0�0. But the ideal output was not meant to be so. Let us have a look at the following figures.



Practical Output waveform

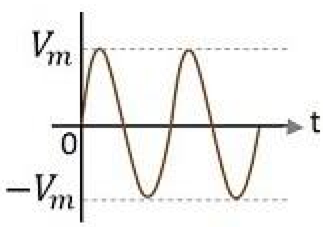


Ideal Output Waveform

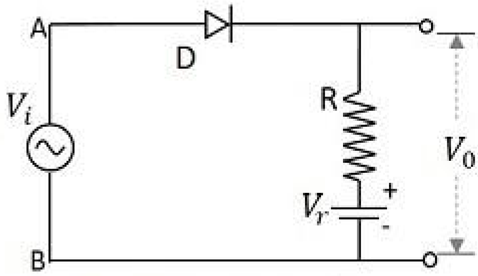
Unlike the ideal output, a bit portion of the negative cycle is present in the practical output due to the diode conduction voltage which is 0.7v. Hence there will be a difference in the practical and ideal output waveforms.

Negative Series Clipper with positive Vr

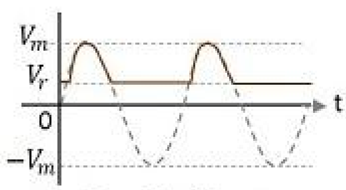
A Clipper circuit in which the diode is connected in series to the input signal and biased with positive reference voltage Vr and that attenuates the negative portions of the waveform, is termed as Negative Series Clipper with positive Vr. The following figure represents the circuit diagram for negative series clipper when the reference voltage applied is positive.



Input Waveform



Negative Series Clipper with positive V_r

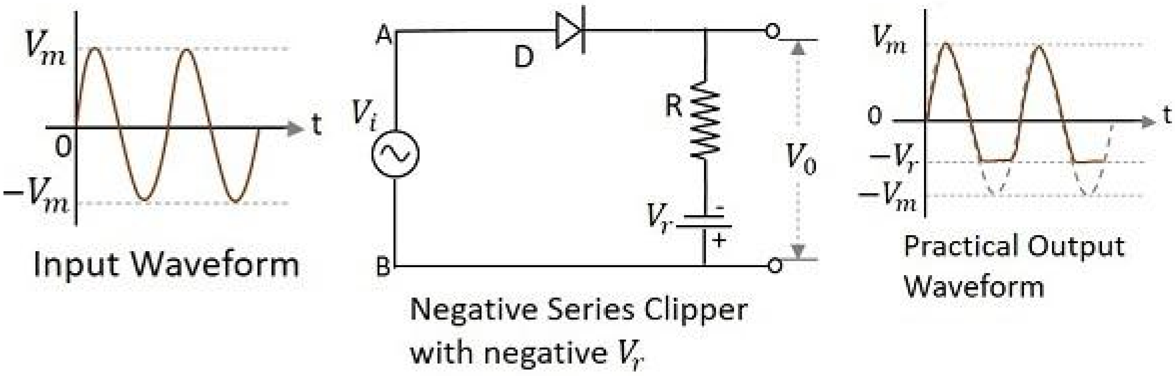


Practical Output Waveform

During the positive cycle of the input, the diode starts conducting only when the anode voltage value exceeds the cathode voltage value of the diode. As the cathode voltage equals the reference voltage applied, the output will be as shown.

Negative Series Clipper with negative Vr

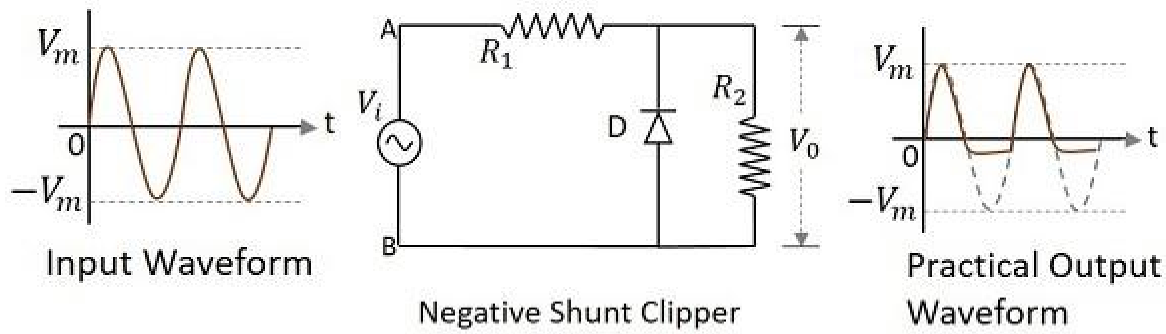
A Clipper circuit in which the diode is connected in series to the input signal and biased with negative reference voltage Vr and that attenuates the negative portions of the waveform, is termed as Negative Series Clipper with negative Vr. The following figure represents the circuit diagram for negative series clipper, when the reference voltage applied is negative.



During the positive cycle of the input the diode gets forward biased and the input signal appears at the output. During its negative cycle, the diode gets reverse biased and hence will not conduct. But the negative reference voltage being applied, appears at the output. Hence the negative cycle of the output waveform gets clipped after this reference level.

Negative Shunt Clipper

A Clipper circuit in which the diode is connected in shunt to the input signal and that attenuates the negative portions of the waveform, is termed as Negative Shunt Clipper. The following figure represents the circuit diagram for negative shunt clipper.

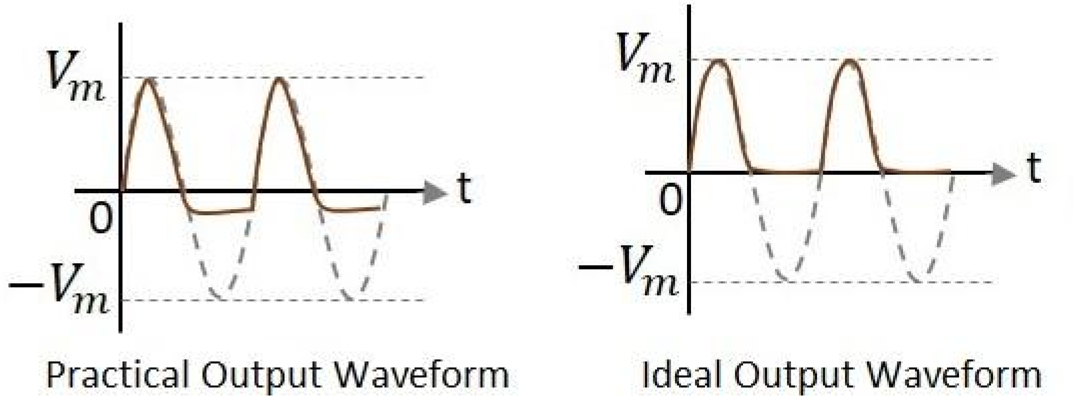


Positive Cycle of the Input − When the input voltage is applied, the positive cycle of the input makes the point A in the circuit positive with respect to the point B. This makes the diode reverse biased and hence it behaves like an open switch. Thus the voltage across the load resistor equals the applied input voltage as it completely appears at the output V_0

Negative Cycle of the Input − The negative cycle of the input makes the point A in the circuit negative with respect to the point B. This makes the diode forward biased and hence it conducts like a closed switch. Thus the voltage across the load resistor becomes zero as no current flows through it.

Waveforms

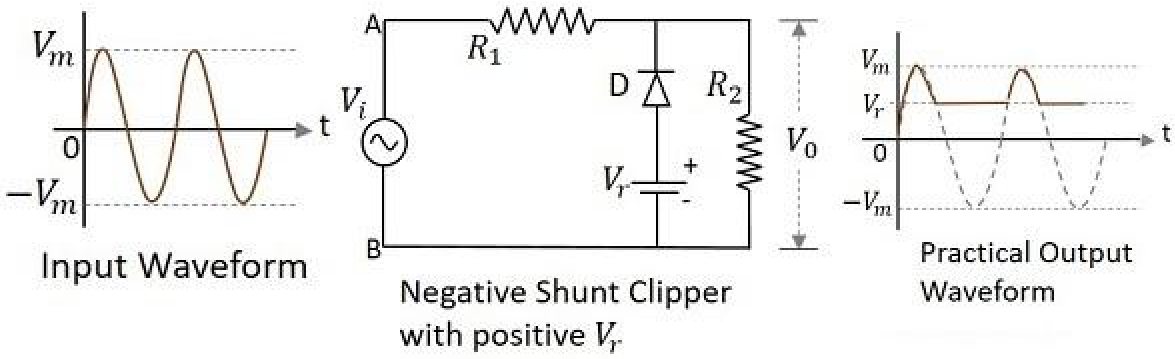
In the above figures, if the waveforms are observed, we can understand that just a portion of the negative peak was clipped. This is because of the voltage across V_0. But the ideal output was not meant to be so. Let us have a look at the following figures.



Unlike the ideal output, a bit portion of the negative cycle is present in the practical output due to the diode conduction voltage which is 0.7v. Hence there will be a difference in the practical and ideal output waveforms.

Negative Shunt Clipper with positive Vr

A Clipper circuit in which the diode is connected in shunt to the input signal and biased with positive reference voltage Vr and that attenuates the negative portions of the waveform, is termed as Negative Shunt Clipper with positive Vr. The following figure represents the circuit diagram for negative shunt clipper when the reference voltage applied is positive.

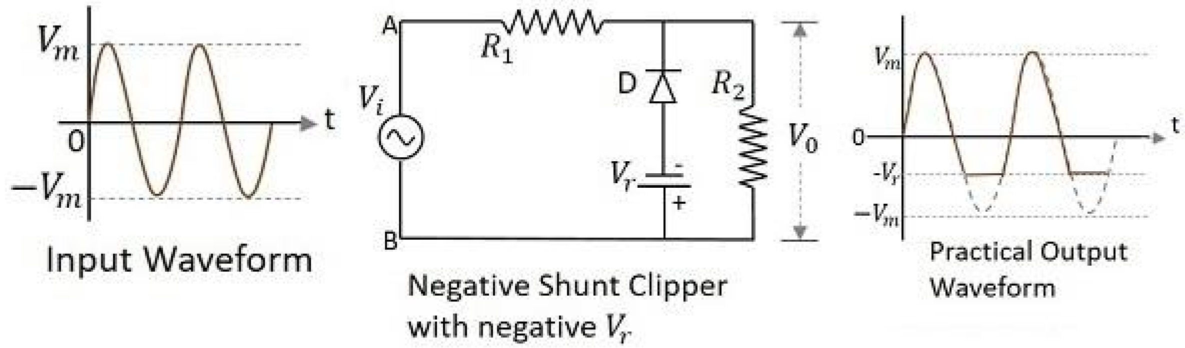


During the positive cycle of the input the diode gets reverse biased and behaves as an open switch. So whole of the input voltage, which is greater than the reference voltage applied, appears at the output. The signal below reference voltage level gets clipped off.

During the negative half cycle, as the diode gets forward biased and the loop gets completed, no output is present.

Negative Shunt Clipper with negative Vr

A Clipper circuit in which the diode is connected in shunt to the input signal and biased with negative reference voltage Vr and that attenuates the negative portions of the waveform, is termed as Negative Shunt Clipper with negative Vr. The following figure represents the circuit diagram for negative shunt clipper, when the reference voltage applied is negative.

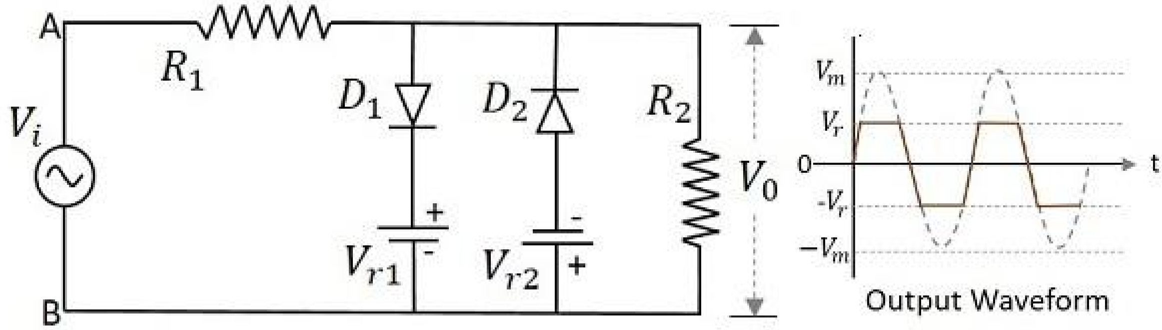


During the positive cycle of the input the diode gets reverse biased and behaves as an open switch. So whole of the input voltage, appears at the output Vo. During the negative half cycle, the diode gets forward biased. The negative voltage up to the reference voltage, gets at the output and the remaining signal gets clipped off.

Two-way Clipper

This is a positive and negative clipper with a reference voltage Vr. The input voltage is clipped two-way both positive and negative portions of the input waveform with two reference voltages. For this, two diodes D_1 and D_2 along with two reference voltages Vr1 and Vr2 are connected in the circuit.

This circuit is also called as a Combinational Clipper circuit. The figure below shows the circuit arrangement for a two-way or a combinational clipper circuit along with its output waveform.



During the positive half of the input signal, the diode D_1 conducts making the reference voltage Vr1 appear at the output. During the negative half of the input signal, the diode D_2 conducts making the reference voltage Vr1 appear at the output. Hence both the diodes conduct alternatively to clip the output during both the cycles. The output is taken across the load resistor.

Clamper Circuit

A Clamper Circuit is a circuit that adds a DC level to an AC signal. Actually, the positive and negative peaks of the signals can be placed at desired levels using the clamping circuits. As the DC level gets shifted, a clamper circuit is called as a Level Shifter.

Clamper circuits consist of energy storage elements like capacitors. A simple clamper circuit comprises of a capacitor, a diode, a resistor and a dc battery if required.

Clamper Circuit

A Clamper circuit can be defined as the circuit that consists of a diode, a resistor and a capacitor that shifts the waveform to a desired DC level without changing the actual appearance of the applied signal.

In order to maintain the time period of the wave form, the tau must be greater than, half the time period discharging time of the capacitor should be slow.

$$\tau = Rc$$

Where

R is the resistance of the resistor employed

C is the capacitance of the capacitor used

The time constant of charge and discharge of the capacitor determines the output of a clamper circuit.

In a clamper circuit, a vertical shift of upward or downward takes place in the output waveform with respect to the input signal.

The load resistor and the capacitor affect the waveform. So, the discharging time of the capacitor should be large enough.

The DC component present in the input is rejected when a capacitor coupled network is used as a capacitor blocks dc. Hence when dc needs to be restored, clamping circuit is used.

Types of Clampers

There are few types of clamper circuits, such as

Positive Clamper

- Positive clamper with positive Vr
- Positive clamper with negative Vr

Negative Clamper

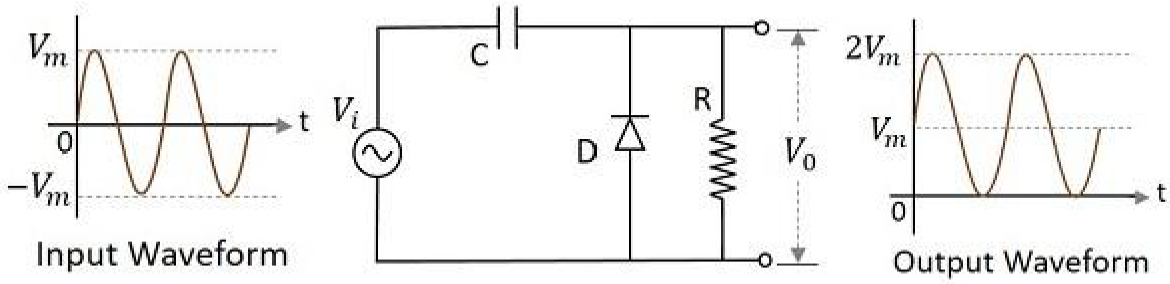
- Negative clamper with positive Vr
- Negative clamper with negative Vr

Let us go through them in detail.

Positive Clamper Circuit

A Clamping circuit restores the DC level. When a negative peak of the signal is raised above to the zero level, then the signal is said to be positively clamped.

A Positive Clamper circuit is one that consists of a diode, a resistor and a capacitor and that shifts the output signal to the positive portion of the input signal. The figure below explains the construction of a positive clamper circuit.



Positive Clamper circuit

Initially when the input is given, the capacitor is not yet charged and the diode is reverse biased. The output is not considered at this point of time. During the negative half cycle, at the peak value, the capacitor gets charged with negative on one plate and positive on the other. The capacitor is now charged to its peak value Vm. The diode is forward biased and conducts heavily.

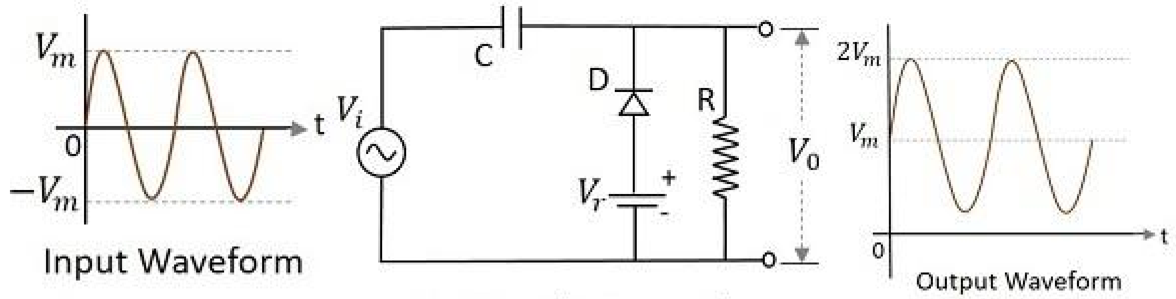
During the next positive half cycle, the capacitor is charged to positive Vm while the diode gets reverse biased and gets open circuited. The output of the circuit at this moment will be

$$V_0 = Vi + Vm$$

Hence the signal is positively clamped as shown in the above figure. The output signal changes according to the changes in the input, but shifts the level according to the charge on the capacitor, as it adds the input voltage.

Positive Clamper with Positive Vr

A Positive clamper circuit if biased with some positive reference voltage, that voltage will be added to the output to raise the clamped level. Using this, the circuit of the positive clamper with positive reference voltage is constructed as below.

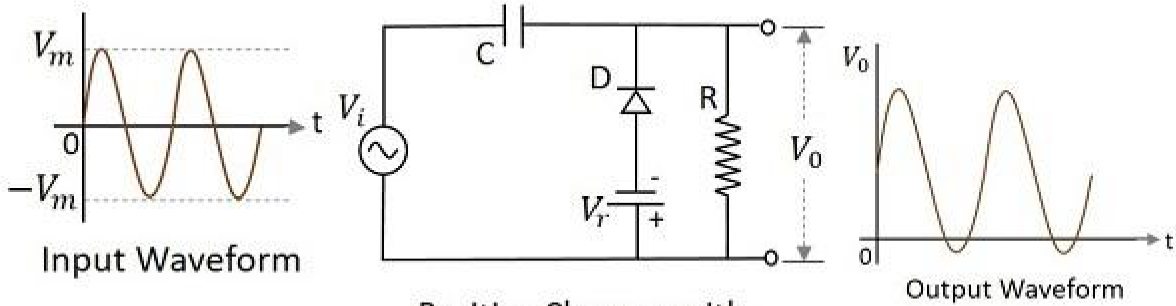


Positive Clamper with positive reference V_r

During the positive half cycle, the reference voltage is applied through the diode at the output and as the input voltage increases, the cathode voltage of the diode increase with respect to the anode voltage and hence it stops conducting. During the negative half cycle, the diode gets forward biased and starts conducting. The voltage across the capacitor and the reference voltage together maintain the output voltage level.

Positive Clamper with Negative Vr

A Positive clamper circuit if biased with some negative reference voltage, that voltage will be added to the output to raise the clamped level. Using this, the circuit of the positive clamper with positive reference voltage is constructed as below.

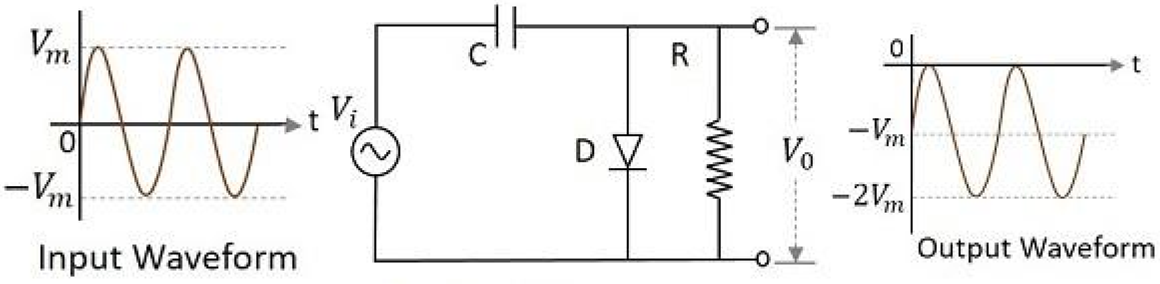


Positive Clamper with negative reference V_r

During the positive half cycle, the voltage across the capacitor and the reference voltage together maintain the output voltage level. During the negative half-cycle, the diode conducts when the cathode voltage gets less than the anode voltage. These changes make the output voltage as shown in the above figure.

Negative Clamper

A Negative Clamper circuit is one that consists of a diode, a resistor and a capacitor and that shifts the output signal to the negative portion of the input signal. The figure below explains the construction of a negative clamper circuit.



Negative Clamper circuit

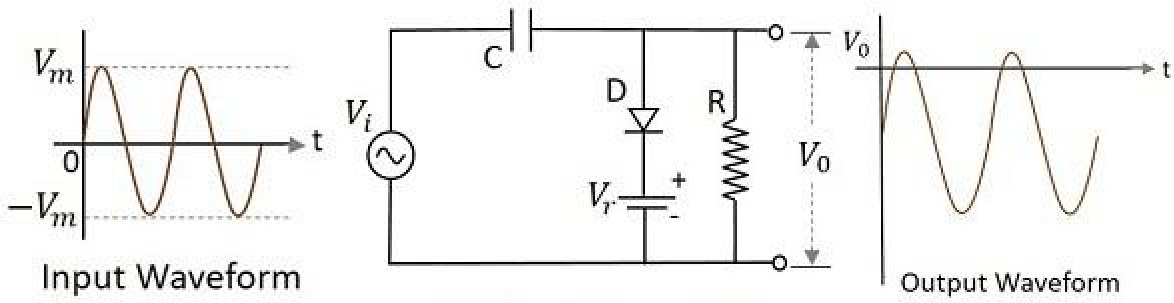
During the positive half cycle, the capacitor gets charged to its peak value Vm. The diode is forward biased and conducts. During the negative half cycle, the diode gets reverse biased and gets open circuited. The output of the circuit at this moment will be

$$V_0\text{=Vi+Vm}$$

Hence the signal is negatively clamped as shown in the above figure. The output signal changes according to the changes in the input, but shifts the level according to the charge on the capacitor, as it adds the input voltage.

Negative clamper with positive Vr

A Negative clamper circuit if biased with some positive reference voltage, that voltage will be added to the output to raise the clamped level. Using this, the circuit of the negative clamper with positive reference voltage is constructed as below.

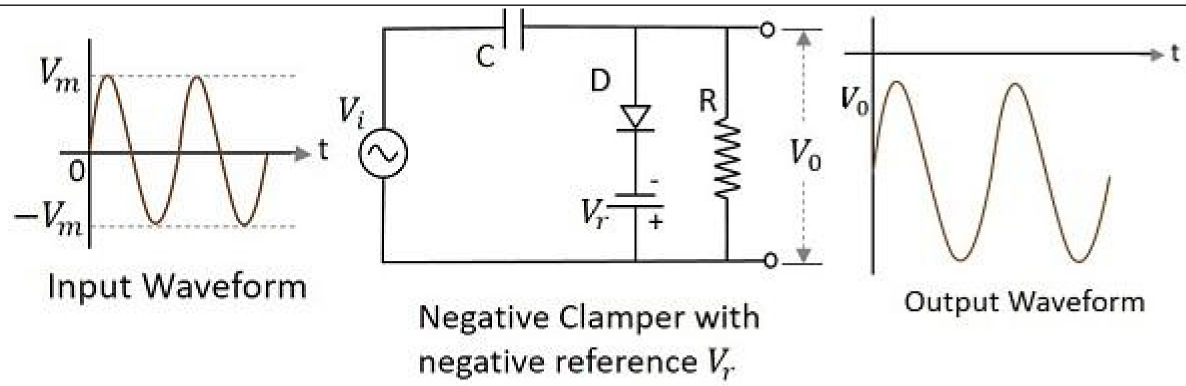


Negative Clamper with positive reference V_r

Though the output voltage is negatively clamped, a portion of the output waveform is raised to the positive level, as the applied reference voltage is positive. During the positive half-cycle, the diode conducts, but the output equals the positive reference voltage applied. During the negative half cycle, the diode acts as open circuited and the voltage across the capacitor forms the output.

Negative Clamper with Negative Vr

A Negative clamper circuit if biased with some negative reference voltage, that voltage will be added to the output to raise the clamped level. Using this, the circuit of the negative clamper with negative reference voltage is constructed as below.



The cathode of the diode is connected with a negative reference voltage, which is less than that of zero and the anode voltage. Hence the diode starts conducting during positive half cycle, before the zero voltage level. During the negative half cycle, the voltage across the capacitor appears at the output. Thus the waveform is clamped towards the negative portion.

Applications

There are many applications for both Clippers and Clampers such as

Clippers

- Used for the generation and shaping of waveforms
- Used for the protection of circuits from spikes
- Used for amplitude restorers
- Used as voltage limiters
- Used in television circuits
- Used in FM transmitters

Clampers

- Used as direct current restorers
- Used to remove distortions
- Used as voltage multipliers
- Used for the protection of amplifiers
- Used as test equipment
- Used as base-line stabilizer

MULTIVIBRATOR

A multivibrator circuit is nothing but a switching circuit. It generates non-sinusoidal waves such as Square waves, Rectangular waves and Saw tooth waves etc. Multivibrators are used as frequency generators, frequency dividers and generators of time delays and also as memory elements in computers etc.

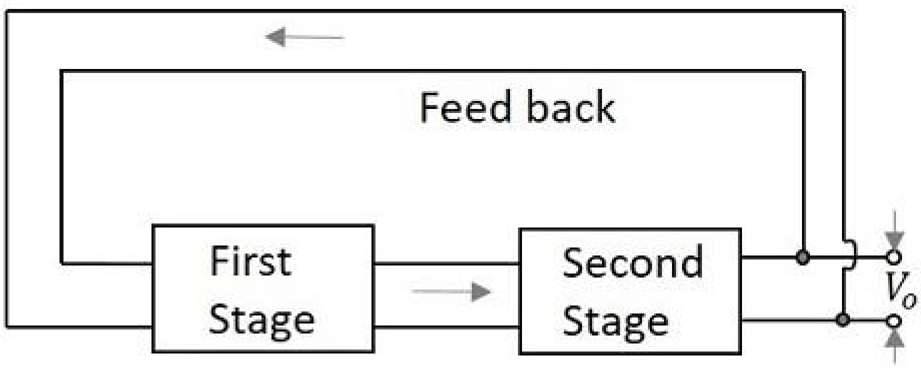
A Transistor basically functions as an amplifier in its linear region. If a transistor amplifier output stage is joined with the previous amplifier stage, such a connection is said to be coupled. If a resistor is used in coupling two stages of such an amplifier circuit, it is called as Resistance coupled amplifier. For more details, refer to the AMPLIFIERS tutorial.

What is a Multivibrator?

According to the definition, **A Multivibrator is a two-stage resistance coupled amplifier with positive feedback from the output 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.

The following figure shows the block diagram of a Multivibrator.



Types of Multivibrators

There are two possible states of a Multivibrator. In first stage, the transistor Q1 turns ON while the transistor Q2 turns OFF. In second stage, the transistor Q1 turns OFF while the transistor Q2 turns ON. These two states are interchanged for certain time periods depending upon the circuit conditions.

Depending upon the manner in which these two states are interchanged, the Multivibrators are classified into three types. They are

Astable Multivibrator

An Astable Multivibrator is such a circuit that it automatically switches between the two states continuously without the application of any external pulse for its operation. As this produces a continuous square wave output, it is called as a Free-running Multivibrator. The dc power source is a common requirement.

The time period of these states depends upon the time constants of the components used. As the Multivibrator keeps on switching, these states are known as quasi-stable or halfstable states. Hence there are two quasi-stable states for an Astable Multivibrator.

Monostable Multivibrator

A Monostable Multivibrator has a stable state and a quasi-stable state. This has a trigger input to one transistor. So, one transistor changes its state automatically, while the other one needs a trigger input to change its state.

As this Multivibrator produces a single output for each trigger pulse, this is known as One-shot Multivibrator. This Multivibrator cannot stay in quasi-stable state for a longer period while it stays in stable state until the trigger pulse is received.

Bistable Multivibrator

A Bistable Multivibrator has both the two states stable. It requires two trigger pulses to be applied to change the states. Until the trigger input is given, this Multivibrator cannot change its state. It’s also known as flip-flop multivibrator.

As the trigger pulse sets or resets the output, and as some data, i.e., either high or low is stored until it is disturbed, this Multivibrator can be called as a Flip-flop.

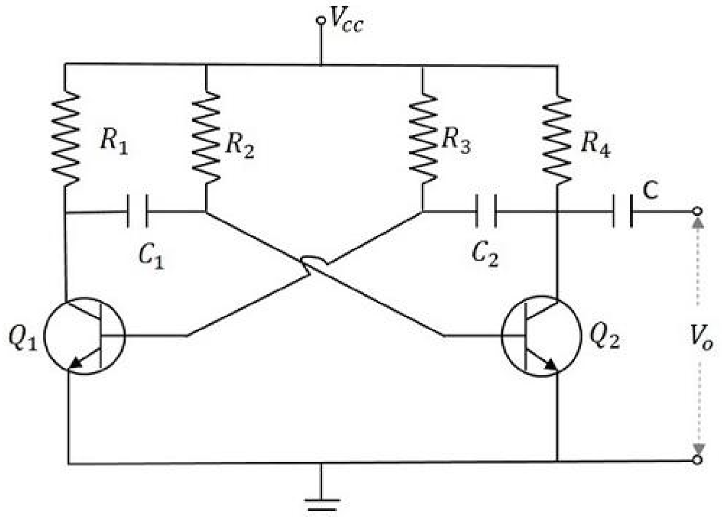
Astable Multivibrator

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 RC time constants. A dc power supply or Vcc is given to the circuit for its operation.

Construction of Astable Multivibrator

Two transistors named Q1 and Q2 are connected in feedback to one another. The collector of transistor Q1 is connected to the base of transistor Q2 through the capacitor C1 and vice versa. The emitters of both the transistors are connected to the ground. The collector load resistors R1 and R4 and the biasing resistors R2 and R3 are of equal values. The capacitors C1 and C2 are of equal values.

The following figure shows the circuit diagram for Astable Multivibrator.



Operation of Astable Multivibrator

When Vcc is applied, the collector current of the transistors increase. As the collector current depends upon the base current,

$I_C = \beta I_B$

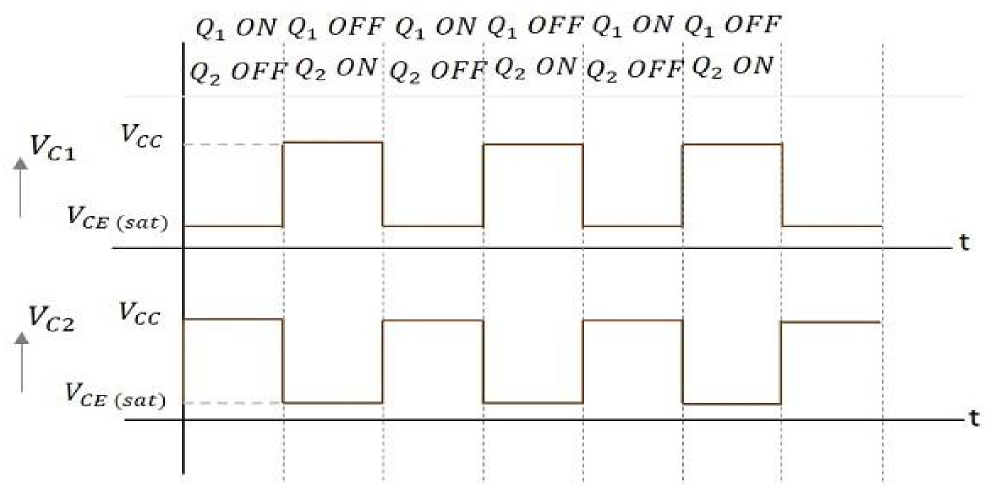
As no transistor characteristics are alike, one of the two transistors say Q1 has its collector current increase and thus conducts. The collector of Q1 is applied to the base of Q2 through C1. This connection lets the increased negative voltage at the collector of Q1 to get applied at the base of Q2 and its collector current decreases. This continuous action makes the collector current of Q2 to decrease further. This current when applied to the base of Q1 makes it more negative and with the cumulative actions Q1 gets into saturation and Q2 to cut off. Thus the output voltage of Q1 will be V_{CE} (sat) and Q2 will be equal to V_{CC}.

The capacitor C1 charges through R1 and when the voltage across C1 reaches 0.7v, this is enough to turn the transistor Q2 to saturation. As this voltage is applied to the base of Q2, it gets into saturation, decreasing its collector current. This reduction of voltage at point B is applied to the base of transistor Q1 through C2 which makes the Q1 reverse bias. A series of these actions turn the transistor Q1 to cut off and transistor Q2 to saturation. Now point A has the potential V_{CC}. The capacitor C2 charges through R2. The voltage across this capacitor C2 when gets to 0.7v, turns on the transistor Q1 to saturation.

Hence the output voltage and the output waveform are formed by the alternate switching of the transistors Q1 and Q2. The time period of these ON/OFF states depends upon the values of biasing resistors and capacitors used, i.e., on the RC values used. As both the transistors are operated alternately, the output is a square waveform, with the peak amplitude of V_{CC}.

Waveforms

The output waveforms at the collectors of Q1 and Q2 are shown in the following figures.



Frequency of Oscillations

The ON time of transistor Q1 or the OFF time of transistor Q2 is given by

$t_1 = 0.69R_1C_1$

Similarly, the OFF time of transistor Q1 or ON time of transistor Q2 is given by

$t_2 = 0.69R_2C_2$

Hence, total time period of square wave

$t = t_1 + t_2 = 0.69(R_1C_1 + R_2C_2)$

As $R_1 = R_2 = R$ and $C_1 = C_2 = C$, the frequency of square wave will be

$f=1/t=1/1.38RC=0.7/RC$

Advantages

The advantages of using an astable multivibrator are as follows −

- No external triggering required.
- Circuit design is simple
- Inexpensive
- Can function continuously

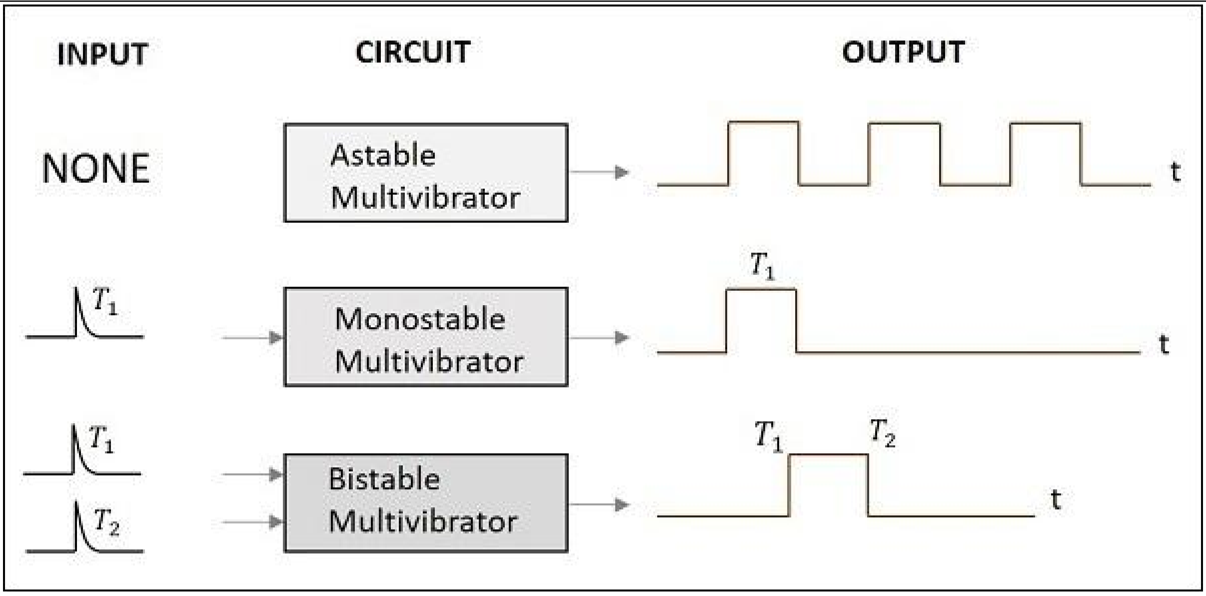
Disadvantages

The drawbacks of using an astable multivibrator are as follows −

- Energy absorption is more within the circuit.
- Output signal is of low energy.
- Duty cycle less than or equal to 50% can't be achieved.

Applications

Astable Multivibrators are used in many applications such as amateur radio equipment, Morse code generators, timer circuits, analog circuits, and TV systems.



Monostable Multivibrator

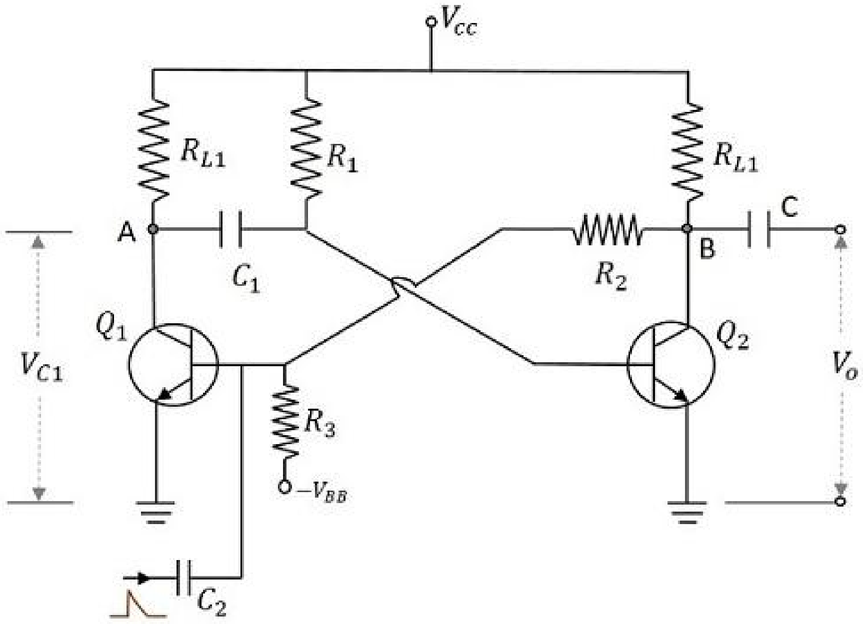
A monostable multivibrator, as the name implies, has only one stable state. When the transistor conducts, the other remains in non-conducting state. A stable state is such a state where the transistor remains without being altered, unless disturbed by some external trigger pulse. As Monostable works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

Construction of Monostable Multivibrator

Two transistors Q1 and Q2 are connected in feedback to one another. The collector of transistor Q1 is connected to the base of transistor Q2 through the capacitor C1. The base Q1 is connected to the collector of Q2 through the resistor R2 and capacitor C. Another dc supply voltage –VBB is given to the base of transistor Q1 through the resistor R3. The trigger pulse is given to the base of Q1 through the capacitor C2 to change its state. RL1 and RL2 are the load resistors of Q1 and Q2.

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.

The following figure shows the circuit diagram of a Monostable Multivibrator.



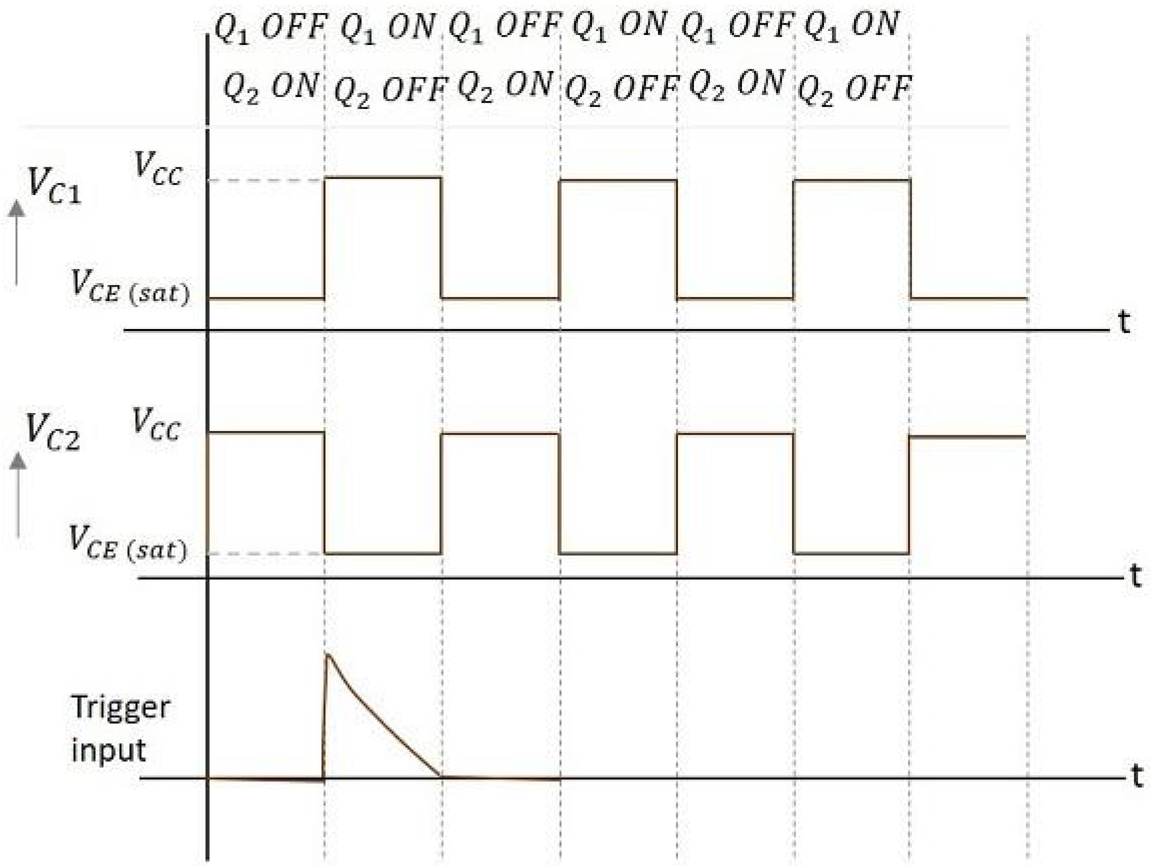
Operation of Monostable Multivibrator

Firstly, when the circuit is switched ON, transistor Q1 will be in OFF state and Q2 will be in ON state. This is the stable state. As Q1 is OFF, the collector voltage will be V_{CC} at point A and hence C1 gets charged. A positive trigger pulse applied at the base of the transistor Q1 turns the transistor ON. This decreases the collector voltage, which turns OFF the transistor Q2. The capacitor C1 starts discharging at this point of time. As the positive voltage from the collector of transistor Q2 gets applied to transistor Q1, it remains in ON state. This is the quasi-stable state or Meta-stable state.

The transistor Q2 remains in OFF state, until the capacitor C1 discharges completely. After this, the transistor Q2 turns ON with the voltage applied through the capacitor discharge. This turn ON the transistor Q1, which is the previous stable state.

Output Waveforms

The output waveforms at the collectors of Q1 and Q2 along with the trigger input given at the base of Q1 are shown in the following figures.



The width of this output pulse depends upon the RC time constant. Hence it depends on the values of R_1C_1. The duration of pulse is given by

$$T=0.69R_1C_1$$

The trigger input given will be of very short duration, just to initiate the action. This triggers the circuit to change its state from Stable state to Quasi-stable or Meta-stable or Semi-stable state, in which the circuit remains for a short duration. There will be one output pulse for one trigger pulse.

Advantages

The advantages of Monostable Multivibrator are as follows −

- One trigger pulse is enough.
- Circuit design is simple
- Inexpensive

Disadvantages

The major drawback of using a monostable multivibrator is that the time between the applications of trigger pulse T has to be greater than the RC time constant of the circuit.

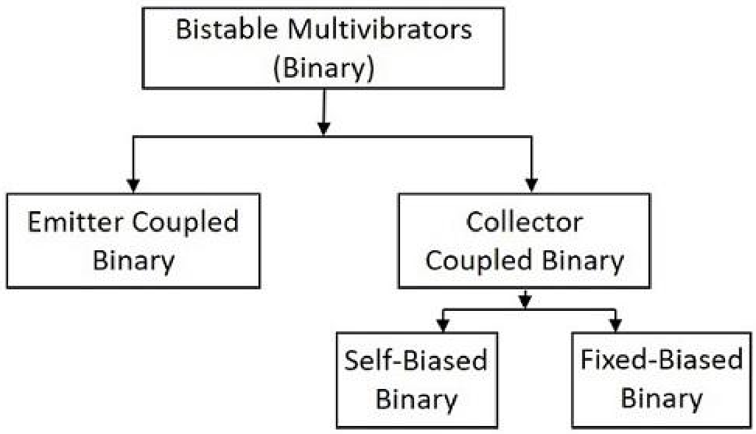
Applications

Monostable Multivibrators are used in applications such as television circuits and control system circuits.

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.

There are few types in Bistable Multivibrators. They are as shown in the following figure.

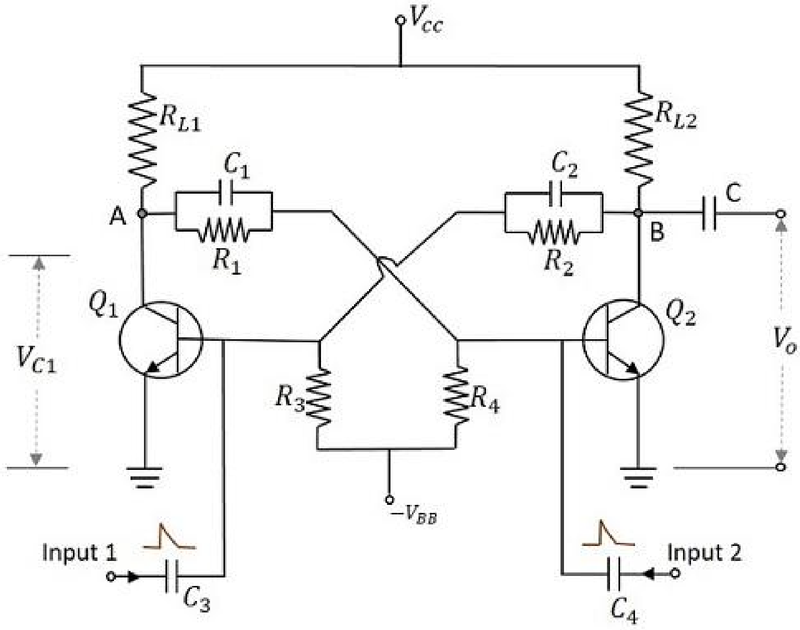


Construction of Bistable Multivibrator

Two similar transistors Q1 and Q2 with load resistors RL1 and RL2 are connected in feedback to one another. The base resistors R3 and R4 are joined to a common source –V_{BB}. The feedback resistors R1 and R2 are shunted by capacitors C1 and C2 known as Commutating Capacitors. The transistor Q1 is given a trigger input at the base through the capacitor C3 and the transistor Q2 is given a trigger input at its base through the capacitor C4.

The capacitors C1 and C2 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.

The following figure shows the circuit diagram of a self-biased Bistable Multivibrator.



Operation of Bistable Multivibrator

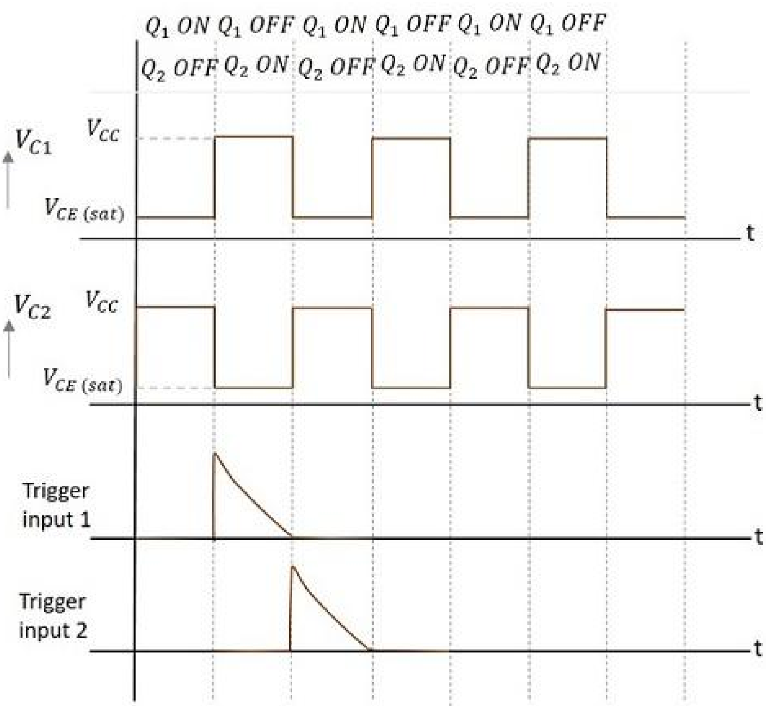
When the circuit is switched ON, due to some circuit imbalances as in Astable, one of the transistors, say Q1 gets switched ON, while the transistor Q2 gets switched OFF. This is a stable state of the Bistable Multivibrator.

By applying a negative trigger at the base of transistor Q1 or by applying a positive trigger pulse at the base of transistor Q2, this stable state is unaltered. So, let us understand this by considering a negative pulse at the base of transistor Q1. As a result, the collector voltage increases, which forward biases the transistor Q2. The collector current of Q2 as applied at the base of Q1, reverse biases Q1 and this cumulative action, makes the transistor Q1 OFF and transistor Q2 ON. This is another stable state of the Multivibrator.

Now, if this stable state has to be changed again, then either a negative trigger pulse at transistor Q2 or a positive trigger pulse at transistor Q1 is applied.

Output Waveforms

The output waveforms at the collectors of Q1 and Q2 along with the trigger inputs given at the bases of QW and Q2 are shown in the following figures.



Advantages

The advantages of using a Bistable Multivibrator are as follows −

- Stores the previous output unless disturbed.
- Circuit design is simple

Disadvantages

The drawbacks of a Bistable Multivibrator are as follows −

◆ Two kinds of trigger pulses are required.

◆ A bit costlier than other Multivibrators.

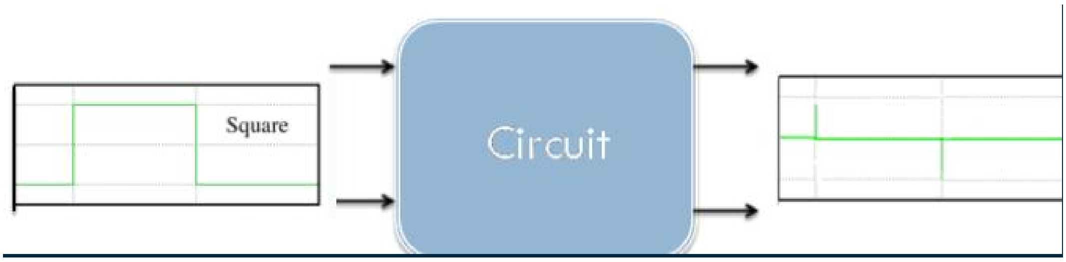
Applications

Bistable Multivibrators are used in applications such as pulse generation and digital operations like counting and storing of binary information.

What Is Integrator?

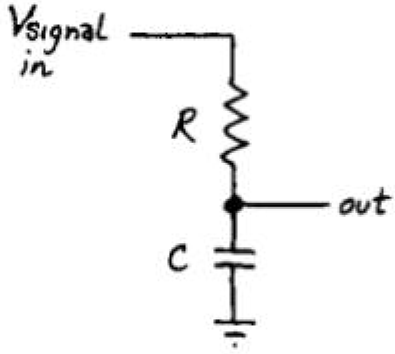
The Integrator is a circuit that converts or 'integrates' a square wave input signal into triangular waveform output.

What Is Differentiator?

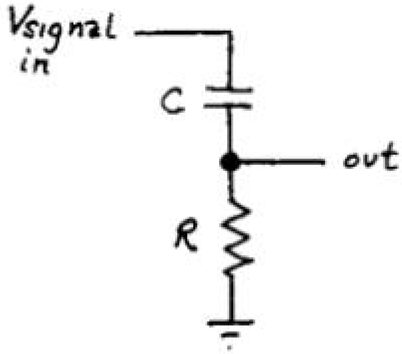


The Differentiator circuit converts or 'differentiates' a square wave input signal into high frequency spikes at its output.

RC As Integrators And Differentiator ?



integrator ?



differentiator?

Unit-6: OPERATIONAL AMPLIFIER CIRCUITS & FEEDBACK CONFIGURATIONS

Operational Amplifier, also called as an Op-Amp, is an integrated circuit, which can be used to perform various linear, non-linear, and mathematical operations. An op-amp is a direct coupled high gain amplifier. You can operate op-amp both with AC and DC signals. This chapter discusses the characteristics and types of op-amps.

Construction of Operational Amplifier

An op-amp consists of differential amplifier(s), a level translator and an output stage. A differential amplifier is present at the input stage of an op-amp and hence an op-amp consists of two input terminals. One of those terminals is called as the inverting terminal and the other one is called as the non-inverting terminal. The terminals are named based on the phase relationship between their respective inputs and outputs.

Characteristics of Operational Amplifier

The important characteristics or parameters of an operational amplifier are as follows −

- **Open loop voltage gain**
- **Output offset voltage**
- **Common Mode Rejection Ratio**
- **Slew Rate**

Open loop voltage gain

The open loop voltage gain of an op-amp is its differential gain without any feedback path.

Mathematically, the open loop voltage gain of an op-amp is represented as −

$$Av=v_0/(v_1-v_2)$$

Output offset voltage

The voltage present at the output of an op-amp when its differential input voltage is zero is called as output offset voltage.

Common Mode Rejection Ratio

Common Mode Rejection Ratio (CMRR) of an op-amp is defined as the ratio of the closed loop differential gain, Ad and the common mode gain, Ac.

Mathematically, CMRR can be represented as −

$$\mathbf{CMRR=Ad/Ac}$$

Note that the common mode gain, Ac of an op-amp is the ratio of the common mode output voltage and the common mode input voltage.

Slew Rate

Slew rate of an op-amp is defined as the maximum rate of change of the output voltage due to a step input voltage.

Mathematically, slew rate (SR) can be represented as −

$$SR = \text{Maximum of } dV_0/dt$$

Where, V_0 is the output voltage. In general, slew rate is measured in either V/μSec or V/mSec.

Types of Operational Amplifiers

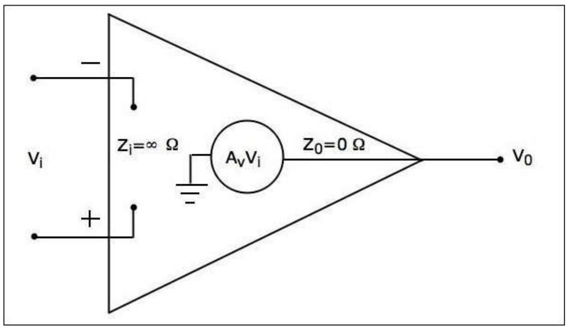
An op-amp is represented with a triangle symbol having two inputs and one output.

Op-amps are of two types: **Ideal Op-Amp** and **Practical Op-Amp**.

They are discussed in detail as given below −

Ideal Op-Amp

An ideal op-amp exists only in theory, and does not exist practically. The equivalent circuit of an ideal op-amp is shown in the figure given below −

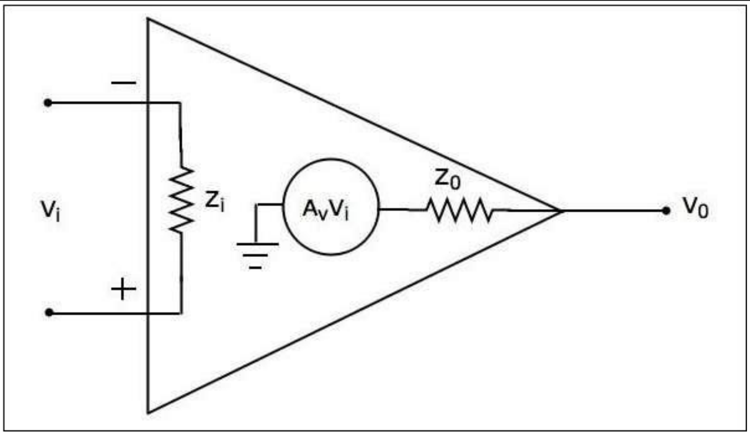


An ideal op-amp exhibits the following characteristics −

- Input impedance Zi=∞Ω
- Output impedance Z_0=0Ω
- Open loop voltage gaine Av=∞
- If (the differential) input voltage Vi=0V, then the output voltage will be V_0=0V
- Bandwidth is infinity. It means, an ideal op-amp will amplify the signals of any frequency without any attenuation.
- Common Mode Rejection Ratio (CMRR) is infinity.
- Slew Rate (SR) is infinity. It means, the ideal op-amp will produce a change in the output instantly in response to an input step voltage.

Practical Op-Amp

Practically, op-amps are not ideal and deviate from their ideal characteristics because of some imperfections during manufacturing. The equivalent circuit of a practical op-amp is shown in the following figure −



A practical op-amp exhibits the following characteristics −

Input impedance, Zi in the order of Mega ohms.
Output impedance, Z_0 in the order of few ohms..
Open loop voltage gain, Av will be high.

When we choose a practical op-amp, you should check whether it satisfies the following conditions −

- Input impedance, Zi should be as high as possible.
- Output impedance, Z_0 should be as low as possible.
- Open loop voltage gain, Av should be as high as possible.
- Output offset voltage should be as low as possible.
- The operating Bandwidth should be as high as possible.
- CMRR should be as high as possible.
- Slew rate should be as high as possible.

A circuit is said to be linear, if there exists a linear relationship between its input and the output. Similarly, a circuit is said to be non-linear, if there exists a non-linear relationship between its input and output.

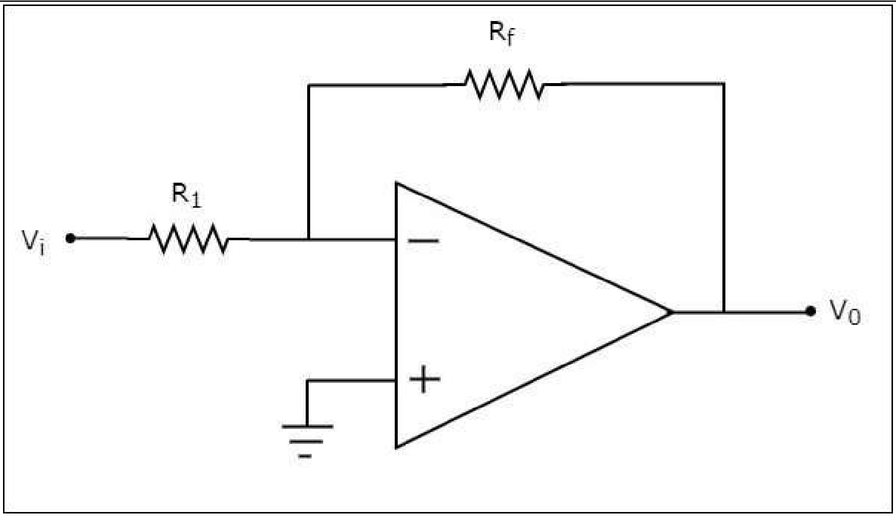
Op-amps can be used in both linear and non-linear applications. The following are the basic applications of op-amp −

- **Inverting Amplifier**
- **Non-inverting Amplifier**
- **Voltage follower**

Inverting Amplifier

An inverting amplifier takes the input through its inverting terminal through a resistor R_1, and produces its amplified version as the output. This amplifier not only amplifies the input but also inverts it (changes its sign).

The circuit diagram of an inverting amplifier is shown in the following figure −



Note that for an op-amp, the voltage at the inverting input terminal is equal to the voltage at its non-inverting input terminal. Physically, there is no short between those two terminals but virtually, they are in short with each other.
In the circuit shown above, the non-inverting input terminal is connected to ground. That means zero volts is applied at the non-inverting input terminal of the op-amp.
According to the virtual short concept, the voltage at the inverting input terminal of an op-amp will be zero volts.
The nodal equation at this terminal's node is as shown below −

$$(0-Vi)/R_1+(0-V_0)/R_f=0$$

$$=>-Vi/R_1=V_0R_f$$

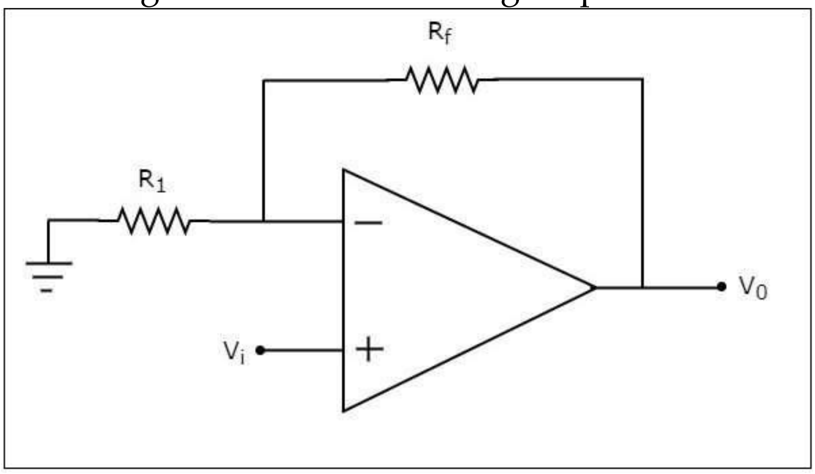
$$=>V_0=(-R_f/R_1)Vt$$

$$=>V_0/Vi=-R_f/R_1$$

The ratio of the output voltage V_0 and the input voltage Vi is the voltage-gain or gain of the amplifier. Therefore, the gain of inverting amplifier is equal to $-R_f/R_1$.
Note that the gain of the inverting amplifier is having a negative sign. It indicates that there exists a 180^0 phase difference between the input and the output.

Non-Inverting Amplifier

A non-inverting amplifier takes the input through its non-inverting terminal, and produces its amplified version as the output. As the name suggests, this amplifier just amplifies the input, without inverting or changing the sign of the output.
The circuit diagram of a non-inverting amplifier is shown in the following figure −



In the above circuit, the input voltage Vi is directly applied to the non-inverting input terminal of op-amp. So, the voltage at the non-inverting input terminal of the op-amp will be Vi.

By using voltage division principle, we can calculate the voltage at the inverting input terminal of the op-amp as shown below −

=>$V_1=V_0R1/(R1+Rf)$

According to the virtual short concept, the voltage at the inverting input terminal of an op-amp is same as that of the voltage at its non-inverting input terminal.

=>$V_1=Vi$

=>$V_0R1/(R1+Rf)=Vi$

=>$V_0/Vi=(R1+Rf)/R_1$

=>$V_0/Vi=1+R_f/R_1$

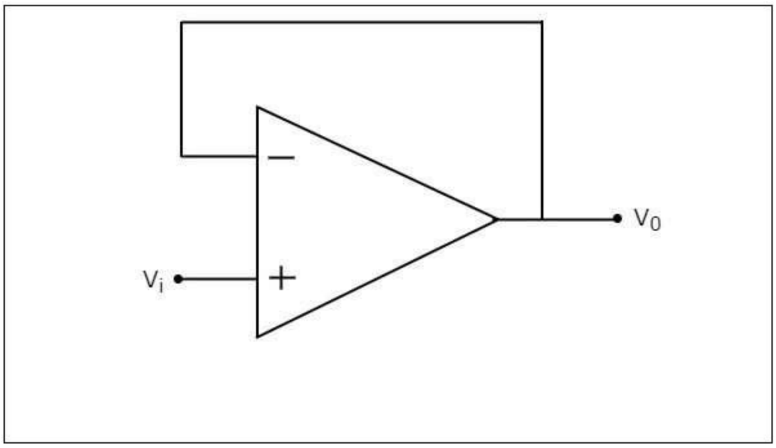
Now, the ratio of output voltage V_0 and input voltage Vi or the voltage-gain or gain of the non-inverting amplifier is equal to $1+R_f/R_1$.

Note that the gain of the non-inverting amplifier is having a positive sign. It indicates that there is no phase difference between the input and the output.

Voltage follower

A voltage follower is an electronic circuit, which produces an output that follows the input voltage. It is a special case of non-inverting amplifier.

If we consider the value of feedback resistor, Rf as zero ohms and (or) the value of resistor, 1 as infinity ohms, then a non-inverting amplifier becomes a voltage follower. The circuit diagram of a voltage follower is shown in the following figure −



In the above circuit, the input voltage Vi is directly applied to the non-inverting input terminal of the op-amp. So, the voltage at the non-inverting input terminal of op-amp is equal to Vi. Here, the output is directly connected to the inverting input terminal of opamp. Hence, the voltage at the inverting input terminal of op-amp is equal to V_0.

According to the virtual short concept, the voltage at the inverting input terminal of the op-amp is same as that of the voltage at its non-inverting input terminal.

=>$V_0=Vi$

So, the output voltage V_0 of a voltage follower is equal to its input voltage Vi.

Thus, the gain of a voltage follower is equal to one since, both output voltage V_0 and input voltage Vi of voltage follower are same.

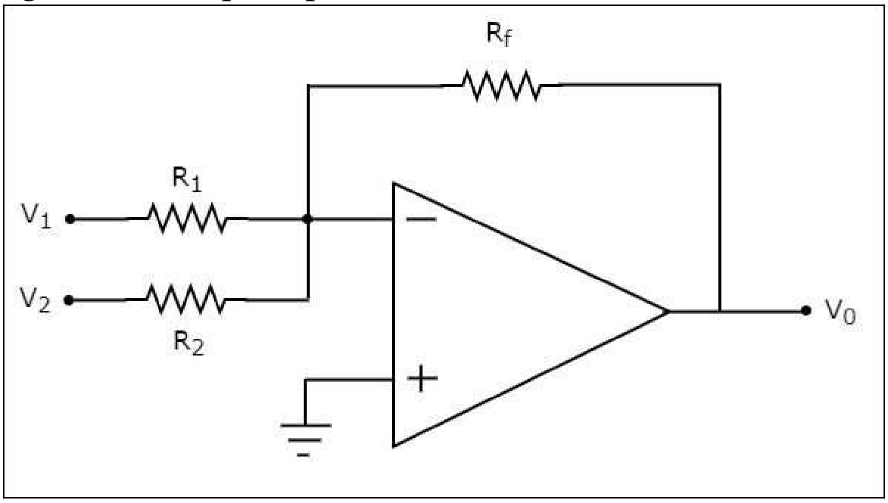
The electronic circuits, which perform arithmetic operations are called as arithmetic circuits. Using op-amps, you can build basic arithmetic circuits such as an adder and a subtractor. In this chapter, you will learn about each of them in detail.

Adder

An adder is an electronic circuit that produces an output, which is equal to the sum of the applied inputs. This section discusses about the op-amp based adder circuit.

An op-amp based adder produces an output equal to the sum of the input voltages applied at its inverting terminal. It is also called as a summing amplifier, since the output is an amplified one.

The circuit diagram of an op-amp based adder is shown in the following figure −



In the above circuit, the non-inverting input terminal of the op-amp is connected to ground. That means zero volts is applied at its non-inverting input terminal.

According to the virtual short concept, the voltage at the inverting input terminal of an op-amp is same as that of the voltage at its non-inverting input terminal. So, the voltage at the inverting input terminal of the op-amp will be zero volts.

The nodal equation at the inverting input terminal's node is

$$\mathbf{0-V_1/R_1+0-V_2/R_2+0+V_0/Rf=0}$$

$$\mathbf{=>V_1/R_1-V_2/R_2=V_0/Rf}$$

$$\mathbf{=>V_0=Rf(V_1/R_1+V_2/R_2)}$$

If $Rf=R_1=R_2=R$, then the output voltage V_0 will be −

$$\mathbf{V_0=-R(V_1/R+V_2/R)}$$

$$\mathbf{=>V_0=-(V_1+V_2)}$$

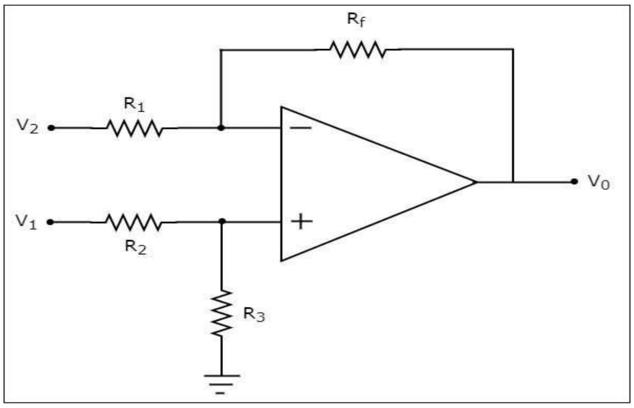
Therefore, the op-amp based adder circuit discussed above will produce the sum of the two input voltages v_1 and v_2, as the output, when all the resistors present in the circuit are of same value. Note that the output voltage V_0 of an adder circuit is having a negative sign, which indicates that there exists a 180^0 phase difference between the input and the output.

Subtractor

A subtractor is an electronic circuit that produces an output, which is equal to the difference of the applied inputs. This section discusses about the op-amp based subtractor circuit.

An op-amp based subtractor produces an output equal to the difference of the input voltages applied at its inverting and non-inverting terminals. It is also called as a difference amplifier, since the output is an amplified one.

The circuit diagram of an op-amp based subtractor is shown in the following figure −

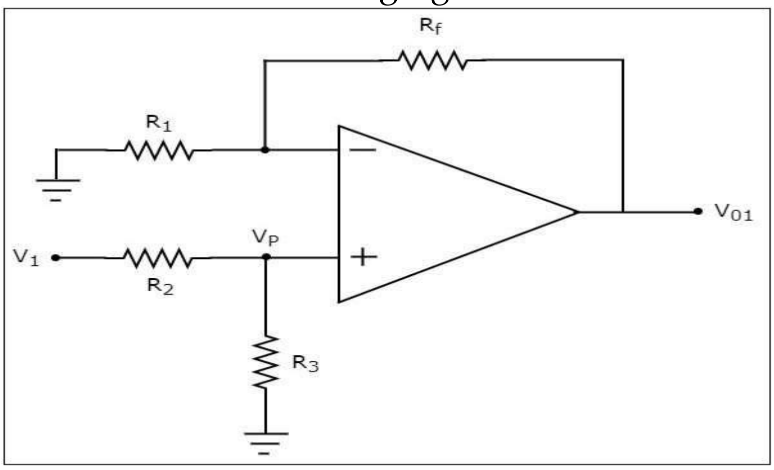


Now, let us find the expression for output voltage V_0 of the above circuit using superposition theorem using the following steps −

Step 1

Firstly, let us calculate the output voltage V_0 by considering only V_1.

For this, eliminate V_2 by making it short circuit. Then we obtain the modified circuit diagram as shown in the following figure −



Now, using the voltage division principle, calculate the voltage at the non-inverting input terminal of the op-amp.

$$\Rightarrow V_p = V_1 R_3 (R_2 + R_3)$$

Now, the above circuit looks like a non-inverting amplifier having input voltage Vp��. Therefore, the output voltage V_{01} of above circuit will be

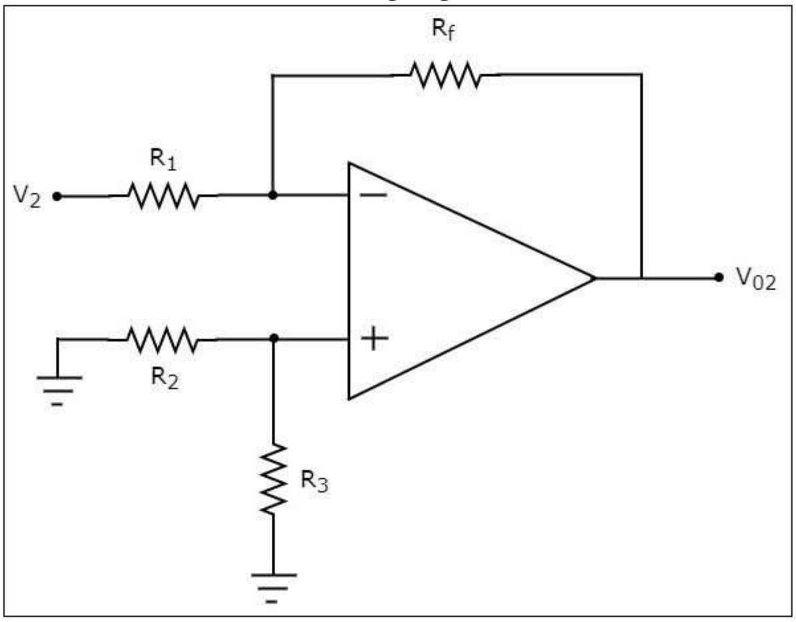
$$V_{01} = V_p (1 + R_f/R_1)$$

Substitute, the value of Vp in above equation, we obtain the output voltage V_{01} by considering only V_1, as −

$$V_{01}=V_1R3/(R2+R_3)(1+Rf/R_1)$$

Step 2

In this step, let us find the output voltage, V_{02} by considering only V_2. Similar to that in the above step, eliminate V_1 by making it short circuit. The modified circuit diagram is shown in the following figure.



You can observe that the voltage at the non-inverting input terminal of the op-amp will be zero volts. It means, the above circuit is simply an inverting op-amp. Therefore, the output voltage V_{02} of above circuit will be −

$$V_{02}=(-Rf/R_1)V_2$$

Step 3

In this step, we will obtain the output voltage V_0 of the subtractor circuit by adding the output voltages obtained in Step1 and Step2. Mathematically, it can be written as

$$V_0=V_{01}+V_{02}$$

Substituting the values of V_{01} and V_{02} in the above equation, we get −

$V_0=V_1R_3/(R_2+R_3)(1+Rf/R_1)+(-Rf/R_1)V_2$

$=>V_0=V_1R_3/(R_2+R_3)(1+Rf/R_1)-(Rf/R_1)V_2$

If $Rf=R_1=R_2=R_3=R$, then the output voltage V_0 will be

$V_0=V_1R/(R+R)(1+R/R)-(R/R)V_2$

$=>V_0=V_1(R/2R)(2)-(1)V_2$

$V_0=V_1-V_2$

Thus, the op-amp based subtractor circuit discussed above will produce an output, which is the difference of two input voltages V_1 and V_2, when all the resistors present in the circuit are of same value.