MAR GREGORIOS COLLEGE OF ARTS & SCIENCE

Block No.8, College Road, Mogappair West, Chennai – 37

Affiliated to the University of Madras
Approved by the Government of Tamil Nadu
An ISO 9001:2015 Certified Institution

DEPARTMENT OF ELECTRONICS & COMMUNICATION SCIENCE

SUBJECT NAME: AMPLIFIERS & OSCILLATORS

SUBJET CODE: TAG3C

SEMESTER: III

PREPARED BY: PROF.SAVITHRI / PROF.S.SHANTHA

AMPLIFIERS AND OSCILLATORS (SYLLABUS)

UNIT I

AMPLIFIERS – General principles of small signal amplifiers – Load line analysis AC & DC – classifications of amplifiers – RC coupled amplifiers – Working - Frequency response (Qualitative Analysis) – Concept of Multistage amplifiers – Transformer coupled amplifiers – Working - Frequency response (Qualitative Analysis) - Direct Coupled amplifier – working - emitter follower.

POWER AMPLIFIERS - Class A – Single ended amplifier - Class B Transformer coupled pushpull amplifier – Crossover distortion – Complementary Symmetry Class-B Push-Pull Amplifier - power dissipation and output power calculations.

UNIT II

FEEDBACK AMPLIFIERS – Principles of feedback amplifiers – Transfer gain with feedback – General characteristics of negative feedback amplifier – effect of negative feedback on gain – gain stability – distortion and Bandwidth – Types of feedback.

UNIT III

OSCILLATORS – Feedback requirements of oscillators – Barkhausen Criterion for oscillation – Hartley, Colpitts, Phase shift and Wien bridge oscillators - Working – Frequency of oscillations – Crystal oscillator – UJT Relaxation oscillator.

UNIT IV

OPERATIONAL AMPLIFIERS – Op-Amp supply voltages – IC identification - Op-Amp parameters - Op-Amp as a voltage amplifier – Inverting amplifier – non-inverting amplifier – Voltage follower.

IC 555 timer - pin functions - internal architecture

UNIT- I

BJT Amplifier

Based on number of stages:

Depending upon the number of stages of Amplification, there are

Single-stage amplifiers and Multi-stage amplifiers

Single-stage Amplifiers: This has only one transistor circuit, which is a single stage amplification.

Multi-stage Amplifiers: This has multiple transistor circuit, which provides multistage amplification.

Based on its output :

Depending upon the parameter that is amplified at the output,

there are voltage and power amplifiers.

Voltage Amplifiers: The amplifier circuit that increases the voltage level of the input signal, is called as Voltage amplifier.

Power Amplifiers: The amplifier circuit that increases the power level of the input signal, is called as Power amplifier.

Based on the input signals :

Depending upon the magnitude of the input signal applied, they canbe categorized as Small signal and large signal amplifiers.

Small signal Amplifiers :When the input signal is so weak so as to produce small fluctuations in the collector current compared to its quiescent value, the amplifier is known as Small signal amplifier

Large signal amplifiers :When the fluctuations in collector current are large i.e. beyond the linear portion of the characteristics, the amplifier is known as large signal amplifier.

Based on the frequency range :

Depending upon the frequency range of the signals being used,there are audio and radio amplifier.

Audio Amplifiers: The amplifier circuit that amplifies the signals that lie in the audio frequency range i.e. from 20Hz to 20 KHz frequency range, is called as audio amplifier

Power Amplifiers: The amplifier circuit that amplifies the signals that lie in a very high frequency range, is called as Power amplifier.

Based on Biasing Conditions :

Depending upon their mode of operation, there are class A ,class B and class C amplifiers.

Class A amplifier: The biasing conditions in class A power amplifier are such that the collector current flows for the entire AC signal applied.

Class B amplifier: The biasing conditions in class B power amplifier are such that the collector current flows for half-cycle of input AC signal applied.

Class C amplifier: The biasing conditions in class C power amplifier are such that the collector current flows for less than half cycle of input AC signal applied.

Class AB amplifier: The class AB power amplifier is one which is created by combining both class A and class B in order to have all the advantages of both the classes and to minimize the problems they have.

Based on the Coupling method:

Depending upon the method of coupling one stage to the other, there arethree types of Amplifiers.

RC Coupled amplifier − A Multi-stage amplifier circuit that is coupled to the next stage using resistor and capacitor (RC) combination can be called as a RC coupled amplifier.

Transformer Coupled amplifier − A Multi-stage amplifier circuit that is coupled to the next stage, with the help of a transformer, can be called as a Transformer coupled amplifier.

Direct Coupled amplifier − A Multi-stage amplifier circuit that is coupled to the next stage directly, can be called as a direct coupled amplifier.

Based on the Transistor Configuration:

Depending upon the type of transistor configuration, there are CE, CB and CC amplifiers.

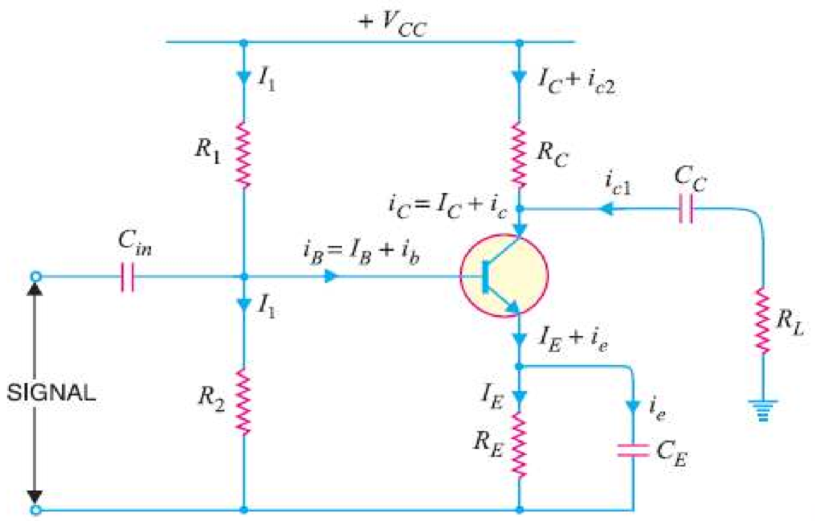
CE amplifier − The amplifier circuit that is formed using a CE configured transistor combination is called as CE amplifier.

CB amplifier − The amplifier circuit that is formed using a CB configured transistor combination is called as CB amplifier.

CC amplifier − The amplifier circuit that is formed using a CC configured transistor combination is called as CC amplifier

Single stage Transistor Amplifier

When an amplifier circuit has only one transistor for amplifying a weak signal, the circuit is known as single stage amplifier.



When a weak a.c. signal is applied to the base of the transistor, a small base current starts flowing in the input circuit.

Due to transistor action, a much larger (β times the base current) a.c. current flows through the the load Rc in the output circuit.

Since the value of load resistance Rc is very high, a large voltage will drop across it.

Thus, a weak signal applied in the base circuit appears in amplified form in the collector circuit. In this way the transistor acts as an amplifier.

Biasing Circuit:

The resistances R_1 , R_2 and R_E provide **biasing** and stabilization.

The biasing circuit must establish a proper operating point otherwise a part of the negative half cycle of the signal may be cut off in the output and you will get faithful amplification.

(ii) Input Capacitor (C_{in})

An electrolytic capacitor of value 10 μF is used to couple the signal to the base of the transistor.

Otherwise, the signal source resistance will come across R_2 and thus can change the bias.

This capacitor allows only the a.c. signal to flow but isolates the signal source from R_2.

Emitter Bypass Capacitor (C_E)

An emitter bypass capacitor of value 100 μF is used in parallel with R_E to provide a low reactance path to the amplified a.c. signal.

If this capacitor is not connected in the output circuit then the amplified a.c. signal will flow through R_E and cause a voltage drop across it, thereby reducing the output voltage.

(iv) Coupling Capacitor (C_C)

The coupling capacitor of value 10 μF is used to couple one stage of amplification to the next stage.

If it is not used, the bias condition of the next stage will be drastically changed due to the shunting effect of R_C. This is because R_C will come in parallel with the resistance R_1 of the biasing circuit of the next stage amplifier circuit and hence, alter the biasing condition of the next stage.

Therefore, the coupling capacitor is used to isolates the d.c. of one stage from the next stage and allows the a.c. signal only.

Various Circuit Currents:

(i) Base Current

When no signal is applied in the base circuit, d.c. base current I_B, also known as zero signal base current flows due to the biasing circuit.

When a.c. signal is applied, a.c. base current i_b flows in the base circuit.

Hence, the total base current i_B is given by :

$$i_B = I_B + i_b$$

(ii) Collector Current

When no signal is applied, a d.c. collector current I_C, also known as zero signal collector current flows due to the biasing circuit.

When a.c. signal is applied, a.c. collector current i_c also flows in the collector circuit.

Hence, the total collector current i_C is given by :

$$i_C = I_C + i_c$$

where

$$I_C = \beta I_B = \text{zero signal collector current}$$

$$i_c = \beta i_b = \text{collector current due to signal.}$$

(iii) Emitter Current

When no signal is applied, a d.c. emitter current I_E, flows due to the biasing circuit. When a.c. signal is applied, a.c. emitter current i_e also flows. Hence, the total emitter current i_E is given by :

$$i_E = I_E + i_e$$

$$I_E = I_B + I_C$$

$$i_e = i_b + i_c$$

Base current is usually very small, therefore, we can take the approximation :

$$I_E \simeq I_C \quad \text{and} \quad i_e \simeq i_c$$

D.C. and A.C. Equivalent Circuits

The analysis is divided into two parts such as; d.c. analysis and a.c. analysis.

In d.c. analysis, we will consider all the d.c. sources at the same time and work out the d.c. currents voltages in the circuit.

Similarly, in a.c. analysis, we will consider all the a.c. sources at the same time and work out the a.c. currents and voltages.

(i) D.C. Equivalent Circuit

In the d.c. equivalent circuit of a transistor amplifier, only d.c. conditions must be considered.

So let us assume there is no signal applied to the circuit.

Since, d.c. currents cannot pass through the capacitors, hence, all the capacitors look like open circuits in the d.c. equivalent circuit.

Therefore, to draw the d.c. equivalent circuit, the following two steps are applied to the transistor amplifier circuit :

1. Make all the a.c. sources zero/Remove all the a.c sources
2. Open all the capacitors

A.C. Equivalent Circuit:

In AC equivalent circuit of a transistor amplifier, only a.c. conditions must be considered.

DC voltage to be considered as zero.

The capacitors are used in the circuit to couple or bypass the a.c. signal.

To draw the a.c. equivalent circuit, the following two steps are applied to the transistor amplifier circuit :

Make all the d.c.sources zero/Remove all the d.c. sources

Short all the capacitors.

Now we can easily calculate the a.c. currents and voltages from this circuit.

Voltage Gain of Single stage Transistor Amplifier:

The voltage gain of a single stage transistor amplifier is the ratio of a.c. output voltage to a.c. input signal voltage.

In order to determine the voltage gain we must consider only the ac currents and voltages in the circuit.

As far as a.c. signal is concerned, load R_C appears in parallel with R_L.

Therefore,

$$\text{a.c. load, } R_{AC} = R_C \| R_L = \frac{R_C \times R_L}{R_C + R_L}$$

$$\text{Output voltage, } V_{out} = i_c R_{AC}$$

$$\text{Input voltage, } V_{in} = i_b R_{in}$$

$$\therefore \quad \text{Voltage gain, } A_v = V_{out}/V_{in} = \frac{i_c R_{AC}}{i_b R_{in}} = \beta \times \frac{R_{AC}}{R_{in}}$$

power gain is given by :

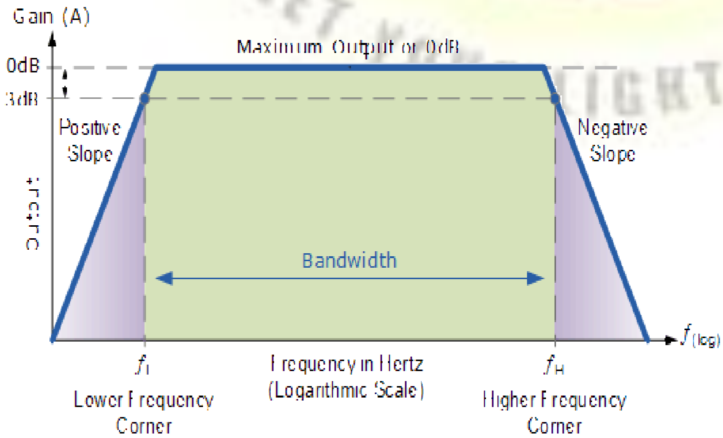
$$A_p = \frac{i_c^2 R_{AC}}{i_b^2 R_{in}} = \beta^2 \times \frac{R_{AC}}{R_{in}}$$

Frequency Response

Frequency Response of an amplifier or filter shows how the gain of the output responds to input signals at different frequencies

The frequency response of a given frequency dependent circuit can be displayed as a graphical sketch of magnitude (gain) against frequency (f). The horizontal frequency axis is usually plotted on a logarithmic scale while the vertical axis representing the voltage output or gain, is usually drawn as a linear scale in decimal divisions.

The frequency response is the variation in its performance with change in its frequency.



The range of frequencies either big or small between f_L and f_H is called the circuits bandwidth. Frequency points f_L and f_H lower cut-off frequency and the upper cut-off frequency points respectively were the circuits gain falls off at high and low frequencies. The bandwidth is given by:

$$\text{Bandwidth, (BW)} = f_H - f_L$$

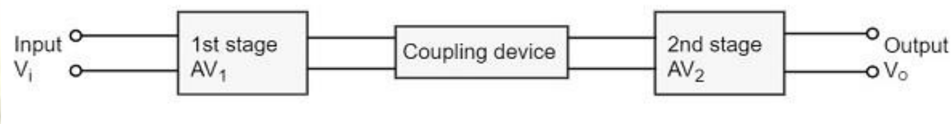
Multistage Amplifiers

If the voltage or power gain obtained from the single stage small amplifier is not sufficient for practical application, then we have to use more than one stage of amplification to achieve necessary power or voltage gain. Such an amplifier is known as multistage amplifier.

In multistage amplifier the output of first stage fed as input to next stage.

A multistage amplifier using two or more single stage common emitter amplifier is called as cascaded amplifier.

When the amplifiers are cascaded, it is necessary to use a coupling network between output of one amplifier and input of following amplifier. This type of coupling is called as inter stage couplin



g.

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

$$A_V = A_{V1} \times A_{V2} = V_2/V_1 \times V_0/V_2 = V_0/V_1$$

Where A_V = Overall gain,

A_{V1} = Voltage gain of 1st stage, and A_{V2} = Voltage gain of 2nd stage.

If there are n numbers of stages, the product of voltage gains of those n stages will be the overall gain of that multistage amplifier circuit.

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, this means to isolate the DC condition.

RC Coupled Amplifier

A Resistance Capacitance (RC) Coupled Amplifier is basically a multi-stage amplifier circuit extensively used in electronic circuits.

The individual stages of the amplifier are connected together using a **resistor–capacitor** combination.

The design of individual stages of the RC coupled amplifiers is similar to that in the case of common emitter amplifiers in which the resistors R_1 and R_2 form the biasing network while the emitter resistor RE form the stabilization network.

C_E is also called bypass capacitor which passes only AC while restricting DC, which causes only DC voltage to drop across R_E while the entire AC voltage will be coupled to the next stage.

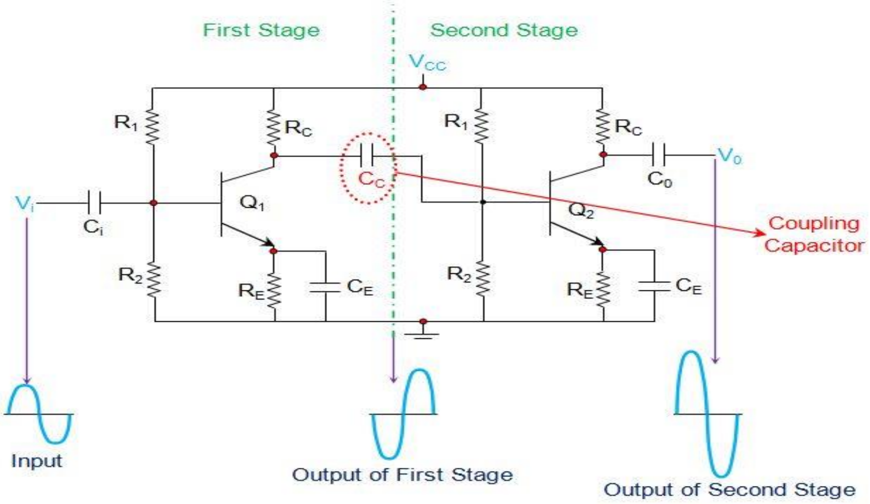


Figure 1 Two-Stage RC Coupled Amplifier

The coupling capacitor C_C also increases the stability of the network as it blocks the DC.

The input signal applied at the base of the **transistor** in stage 1 (Q_1) is amplified and appears at its collector terminal with a phase-shift of 180^o.

The AC component of this signal is coupled to the second stage of the RC coupled amplifier through the coupling capacitor C_C and thus appears as an input at the base of the second transistor Q_2.

This is further amplified and is passed-on as an output of the second stage and is available at the collector terminal of Q_2 after being shift by 180^o in its phase.

Thus the output of the second stage will be 360^o out-of-phase with respect to the input, which indicates that the phase of the input signal and the phase of the output signal obtained at stage II will be same.

The cascading of individual amplifier stages increases the gain of the overall circuit as the net gain will be the product of the gain offered by the individual stages.

When the number of stages are even, the output will be in-phase with the input while if the number of stages are odd, then the output and the input will be out-of-phase.

The frequency response of a RC coupled amplifier indicates that the gain of the amplifier is constant over a wide range of mid-frequencies while it decreases considerably both at low and high frequencies.

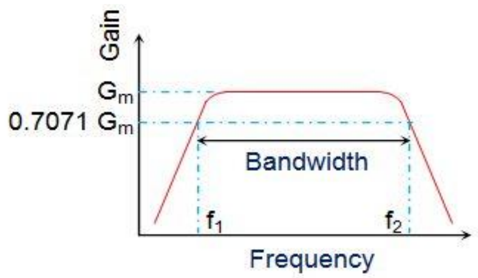


Figure 2 Frequency Response Curve of a RC Coupled Amplifier

At low frequencies, the reactance of coupling capacitor C_C is high which causes a small part of the signal to couple from one stage to the other.

At high frequencies, the reactance of C_C will be low which causes it to behave like a short circuit and thus reduces the voltage gain.

In mid-frequency range, as the frequency increases, the reactance of C_C goes on decreasing which would lead to the increase in gain.

Due to this reason, the gain of the amplifier remains uniform/constant throughout the mid-frequency band.

Advantages of RC Coupled Amplifier

Cheap, economical and compact as it uses only resistors and capacitors.

Offers a constant gain over a wide frequency band.

Disadvantages of RC Coupled Amplifier

Unsuitable for low-frequency amplification.

Low voltage and power gain as the effective load resistance (and hence the gain) is reduced due to the fact that the input of each stage presents a low resistance to its next stage.

Moisture-sensitive, making them noisy as time elapses.

Poor impedance matching as it has the output impedance several times larger than the device at its end-terminal (for example, a speaker in the case of a public address system).

Narrow bandwidth when compared to JFET amplifier.

Applications of RC Coupled Amplifier

RF Communications.

Optical Fiber Communications.

Public address systems as pre-amplifiers.

Controllers.

Radio or TV Receivers as small signal amplifiers.

Transformer Coupled Amplifier

To overcome the drawbacks of RC coupled amplifiers like high impedance at the output, the low impedance at the input, reduced resistance at the load, and reduced voltage gain, a transformer-coupled amplifier is introduced.

In multi-stage **amplifier** circuits, when the output of the first stage is connected to the input of the second stage in cascade through the transformer, then it is called as Transformer Coupled Amplifier.

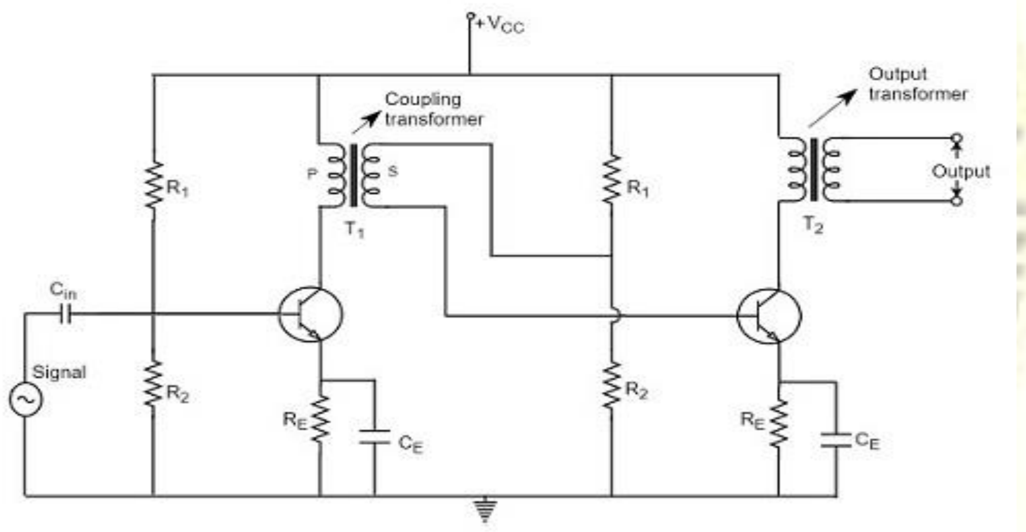
Transformer acts as a coupled device.

It has a high impedance at the input, the low impedance at the output, higher voltage gain, higher efficiency, and increased effective load resistance.

When the load is low and mainly used for power amplification.

The turns ratio of a transformer can be utilized for matching of impedance with the load.

This type of coupling can be used in high-frequency amplifier circuits in the range of Radio Frequency and Intermediate frequency.



Construction:

The coupling transformer T1 is connected between the output of the first stage and input of the second stage.

The collector of the transistor in the first stage is connected to the primary winding of T1 while the secondary winding of T1 is between potential divider R1 and the transistor base in the second stage.

Instead of **resistor** and capacitor, a transformer is used in a transformer-coupled amplifier circuit to achieve high voltage gain and efficiency.

Similarly, multiple stages can be formed by the coupling of the transformer.

R1, R2, and the resistor Re are the potential dividers used for biasing and stabilization purposes.

The coupling capacitor Cc is used to connect these two stages by isolating DC interference from one stage to another stage.

The input capacitor Cin in the first stage allows the AC signal to the base of the transistor.

Working

The AC input signal applied in the first stage flows through the input capacitor to the base of the transistor by eliminating DC components in the signal.

This input signal gets amplified by the transistor in the first stage and it is collected by the primary winding of the transformer T1.

Here T1 transformer is used for coupling two stages. The signal, which is amplified in the first stage is fed to the input of the second stage through the secondary winding of T1.

The voltage of the amplified signal at the primary winding can be converted in accordance with the turns ratio of the secondary winding of T1.

The output of the circuit has low DC resistance, higher efficiency, higher voltage gain, high input impedance, and low output impedance.

The primary winding of T1 acts as a load on the first stage of the transistor while the secondary winding of T1 is for impedance matching for the next stage transistor.

Consider N1 and N2 are the turns ratio of the transformer, Z1 and Z2 are the impedances of primary and secondary windings

Then the relation between primary winding impedance and secondary winding impedance is given by, $Z_2 = Z_1 * (N_2/N_1)^2$

Frequency Response of Transformer Coupled Amplifier

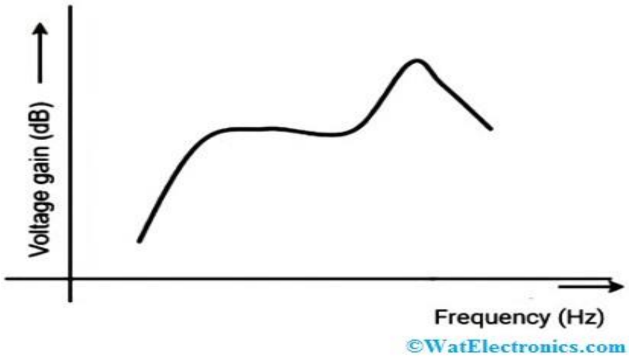
The frequency response is drawn between voltage gain in dB and frequency in Hz.

Transformer-coupled amplifier offers constant gain over a small range of frequencies.

In the lower-frequency range, the reactance of primary winding starts reducing, this results in a decreased gain.

At higher frequencies, the bypass condenser (capacitance between the turns of two windings) reduces the voltage at the output and gain.

So, the signal cannot be amplified and some distortion is occurred, called frequency distortion.



Advantages:

No loss in signal power at base and collector resistors.

Excellent impedance matching properties of the coupling transformer.

High voltage gain due to impedance matching.

The voltage gain of a perfectly designed single-stage transformer-coupled amplifier is equal to the gain of two stages of the RC coupled amplifier.

Operating efficiency is excellent.

Disadvantages:

It offers poor frequency responses than the RC coupled amplifier, so gain varies according to the frequencies.

The coupling can be done by using transformers. So looks bulky and expensive for audio frequencies.

There will be high frequency distortions in the speech signal, audio signal, music, etc.

The coupling transformer makes some noise (hum) in the output.

Applications:

Used in impedance matching applications

Used in the amplification of power.

Used in devices where maximum power could be transferred to the output. Ex: loudspeaker

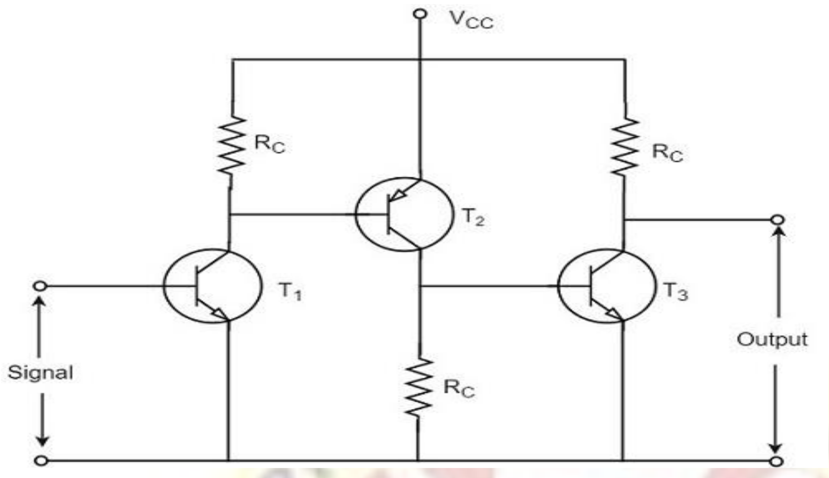
Used in high-frequency amplifiers (RF and IF range) such as radio receivers and TV receivers.

Direct Coupled Amplifier

As no coupling devices are used, the coupling of the amplifier stages is done directly and hence called as Direct coupled amplifier.

Construction

The figure below indicates the three stage direct coupled transistor amplifier. The output of first stage transistor T1 is connected to the input of second stage transistor T2.



The transistor in the first stage will be an NPN transistor, while the transistor in the next stage will be a PNP transistor and so on. This is because; the variations in one transistor tend to cancel the variations in the other. The rise in the collector current and the variation in β of one transistor gets cancelled by the decrease in the other.

Operation

The input signal when applied at the base of transistor T1, it gets amplified due to the transistor action and the amplified output appears at the collector resistor Rc of transistor T1. This output is applied to the base of transistor T2 which further amplifies the signal. In this way, a signal is amplified in a direct coupled amplifier circuit.

Advantages

The circuit arrangement is simple because of minimum use of resistors.

The circuit is of low cost because of the absence of expensive coupling devices.

Disadvantages

It cannot be used for amplifying high frequencies.

The operating point is shifted due to temperature variations.

Applications

Low frequency amplifications.

Low current amplifications.

Emitter Follower

Emitter follower circuit has a prominent place in feedback amplifiers. Emitter follower is a case of negative current feedback circuit. This is mostly used as a last stage amplifier in signal generator circuits.

The important features of Emitter Follower are

It has high input impedance

It has low output impedance

It is ideal circuit for impedance matching

Construction

The constructional details of an emitter follower circuit are nearly similar to a normal amplifier. The main difference is that the load R_L is absent at the collector terminal, but present at the emitter terminal of the circuit. Thus the output is taken from the emitter terminal instead of collector terminal.

Operation

The input signal voltage applied between base and emitter, develops an output voltage V_o across R_E, which is in the emitter section. Therefore,

$V_o = I_E R_E$

The whole of this output current is applied to the input through feedback. Hence,

$V_f = V_o$

As the output voltage developed across R_L is proportional to the emitter current, this emitter follower circuit is a current feedback circuit.

$\beta = Vf/Vo = 1$

The input signal voltage to the transistor (= V_i) is equal to the difference of V_s and V_o i.e.,

$Vi = Vs - Vo$

Hence the feedback is negative.

Characteristics

No voltage gain. In fact, the voltage gain is nearly 1.

Relatively high current gain and power gain.

High input impedance and low output impedance.

Input and output ac voltages are in phase.

Voltage Gain

The AC resistance r_E of the emitter circuit is given by

$r_E = r'_E + R_E$

The input voltage is applied across the ac resistance of the emitter circuit i.e.,

($r'_E + R_E$). Assuming the emitter diode to be ideal, the output voltage V_{out} will be

Vout=ieRE

Input voltage V_{in} will be

Vin=ie(r′e+RE)

Therefore, the Voltage Gain of emitter follower is

AV = Vout/Vin = ieRE/ie(r′e+RE) = RE(r′e+RE)

In practical applications,

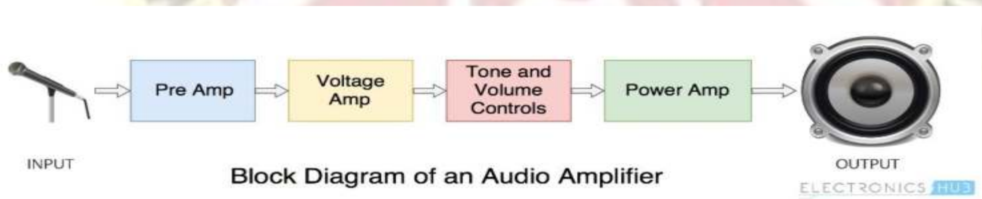
RE≫r′e

So, $A_V \approx 1$. In practice, the voltage gain of an emitter follower is between 0.8 and 0.999.

Power Amplifiers

A power amplifier is an electronic amplifier designed to increase the magnitude of power of a given input signal. The power of the input signal is increased to a level high enough to drive loads of output devices like speakers, headphones, RF transmitters etc.

The input signal to a power amplifier needs to be above a certain threshold. So instead of directly passing the raw audio/RF signal to the power amplifier, it is first pre-amplified using current/voltage amplifiers and is sent as input to the power amp after making necessary modifications.



In this case a microphone is used as an input source. The magnitude of signal from the microphone is not enough for the power amplifier.

So first it is pre-amplified where its voltage and current are increased slightly. Then the signal is passed through tone and volume controls circuit which makes aesthetic adjustments to the audio waveform.

Finally the signal is passed through a power amplifier and the output from power amp is fed to a speaker.

Power Transistor

For Power amplification, a normal transistor cannot be used. A transistor that is manufactured to suit the purpose of power amplification is called as a Power transistor.

A Power transistor differs from the other transistors, in the following factors:

It is larger in size, in order to handle large powers.

The collector region of the transistor is made large and a heat sink is placed at the collector-base junction in order to minimize heat generated.

The emitter and base regions of a power transistor are heavily doped.

Due to the low input resistance, it requires low input power.

Hence there is a lot of difference in voltage amplification and power amplification. So, let us now try to get into the details to understand the differences between a voltage amplifier and a power amplifier.

Classification of Power Amplifiers

Classification Based on Frequencies

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.

Performance Characteristics

Collector Efficiency

This explains how well an amplifier converts DC power to AC power. When the DC supply is given by the battery but no AC signal input is given, the collector output at such a condition is observed as collector efficiency.

The collector efficiency is defined as

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

Power Dissipation Capacity

Every transistor gets heated up during its operation. As a power transistor handles large currents, it gets more heated up. This heat increases the temperature of the transistor, which alters the operating point of the transistor. So, in order to maintain the operating point stability, the temperature of the transistor has to be kept in permissible limits. For this, the heat produced has to be dissipated. Such a capacity is called as Power dissipation capability.

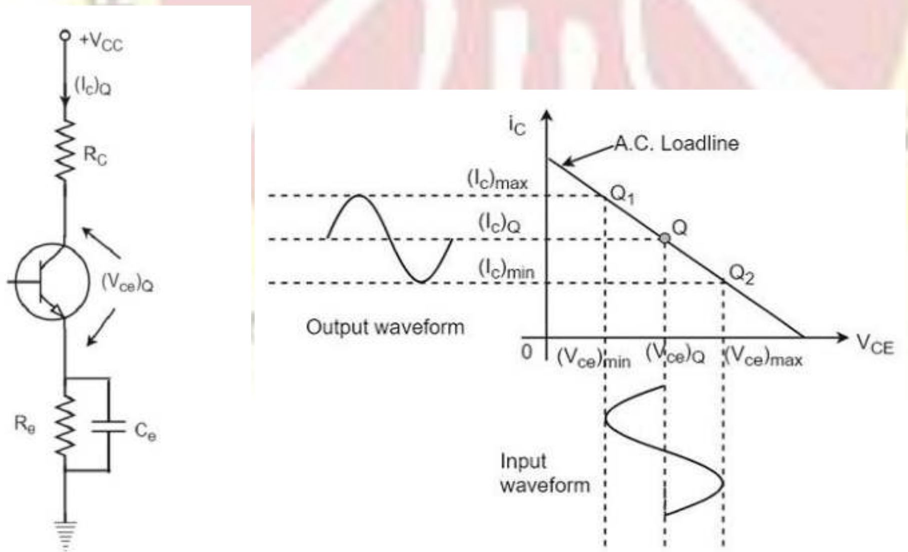
Power dissipation capability can be defined as the ability of a power transistor to dissipate the heat developed in it. Metal cases called heat sinks are used in order to dissipate the heat produced in power transistors.

Distortion

It 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.

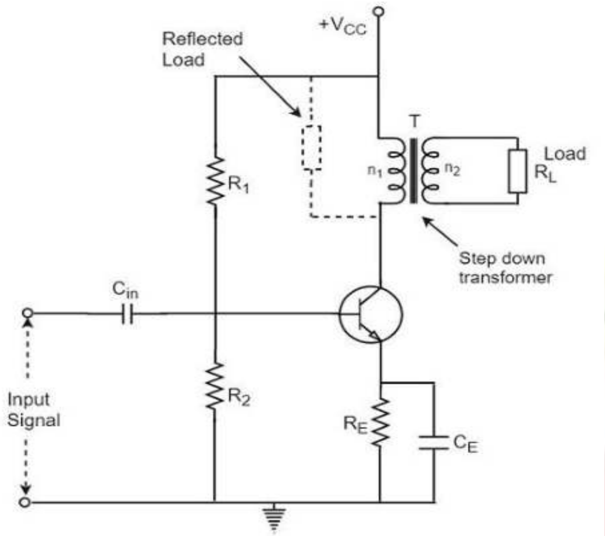


The transformer is present at the collector acts as a load. The use of transformer permits the impedance matching, resulting in the transference of maximum power to the load e.g. loud speaker.

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 amplifier is connected with a transformer in the collector load. Here R_1 and R_2 provide potential divider arrangement. The resistor Re provides stabilization, C_e is the bypass capacitor and R_e 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 Analysis

We know that

$$\frac{V_1}{V_2} = \frac{n_1}{n_2}\ and\ \frac{I_1}{I_2} = \frac{n_1}{n_2}$$

Or

$$V_1 = \frac{n_1}{n_2} V_2\ and\ I_1 = \frac{n_1}{n_2} I_2$$

Hence

$$\frac{V_1}{I_1} = \left(\frac{n_1}{n_2}\right)^2 \frac{V_2}{I_2}$$

But $V_1/I_1 = R_L{}'$ = effective input resistance

And $V_2/I_2 = R_L$ = effective output resistance

Therefore,

$$R'_L = \left(\frac{n_1}{n_2}\right)^2 R_L = n^2 R_L$$

Where

$$n = \frac{number\ of\ turns\ in\ primary}{number\ of\ turns\ in\ secondary} = \frac{n_1}{n_2}$$

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

$$(P_{in})_{dc} = (P_{tr})_{dc} = V_{CC} \times (I_C)_Q$$

Under maximum capacity of class A amplifier, voltage swings from $(V_{ce})_{max}$ to zero and current from $(I_C)_{max}$ to zero.

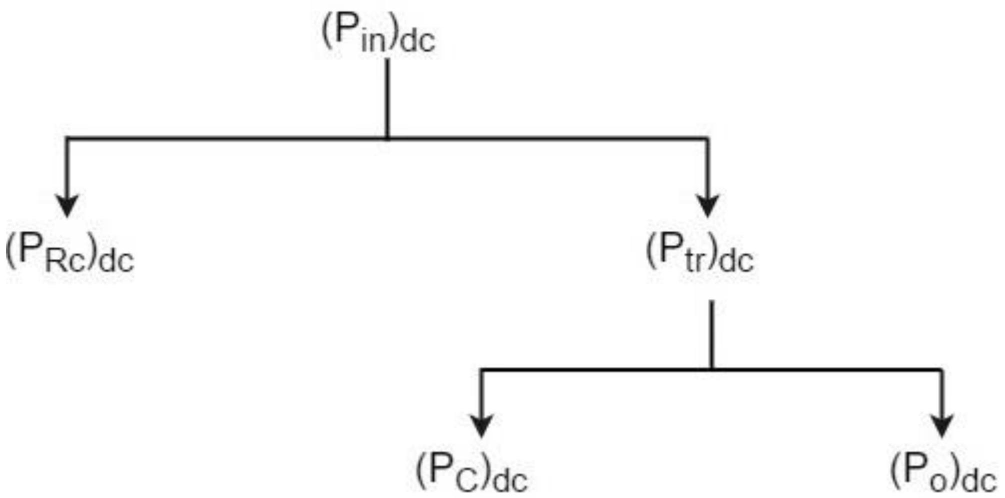
Hence

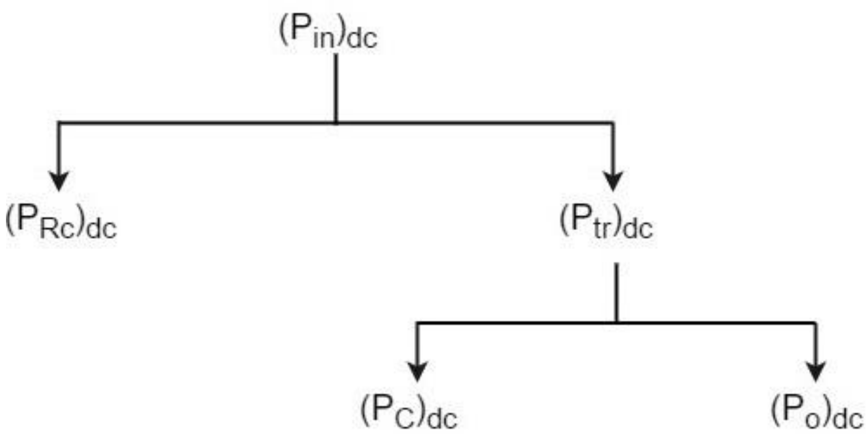
$$V_{rms} = \frac{1}{\sqrt{2}}\left[\frac{(V_{ce})_{max} - (V_{ce})_{min}}{2}\right] = \frac{1}{\sqrt{2}}\left[\frac{(V_{ce})_{max}}{2}\right] = \frac{2V_{CC}}{2\sqrt{2}} = \frac{V_{CC}}{\sqrt{2}}$$

$$I_{rms} = \frac{1}{\sqrt{2}}\left[\frac{(I_C)_{max} - (I_C)_{min}}{2}\right] = \frac{1}{\sqrt{2}}\left[\frac{(I_C)_{max}}{2}\right] = \frac{2(I_C)_Q}{2\sqrt{2}} = \frac{(I_C)_Q}{\sqrt{2}}$$

Therefore,

$$(P_O)_{ac} = V_{rms} \times I_{rms} = \frac{V_{CC}}{\sqrt{2}} \times \frac{(I_C)_Q}{\sqrt{2}} = \frac{V_{CC} \times (I_C)_Q}{2}$$





Therefore,

$$\text{Collector Efficiency} = \frac{(P_O)_{ac}}{(P_{tr})_{dc}}$$

Or,

$$(\eta)_{collector} = \frac{V_{CC} \times (I_C)_Q}{2 \times V_{CC} \times (I_C)_Q} = \frac{1}{2}$$

$$= \frac{1}{2} \times 100 = 50\%$$

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

Advantages

The advantages of transformer coupled class A power amplifier are as follows.

No loss of signal power in the base or collector resistors.

Excellent impedance matching is achieved.

Gain is high.

DC isolation is provided.

Disadvantages

Low frequency signals are less amplified comparatively.

Hum noise is introduced by transformers.

Transformers are bulky and costly.

Poor frequency response.

Applications

This circuit is where impedance matching is the main criterion.

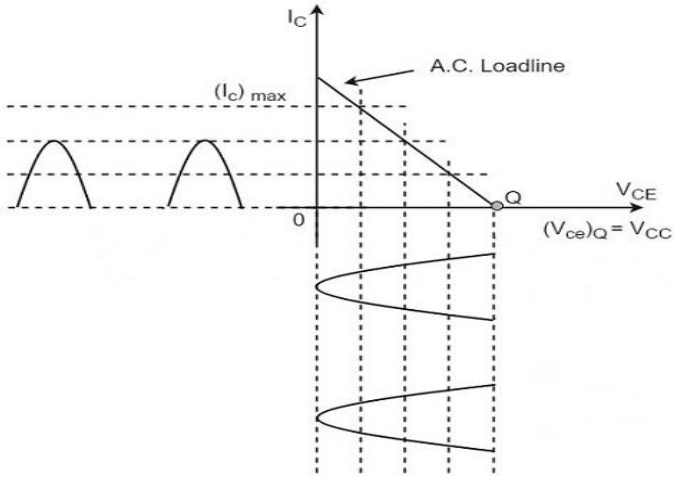
These are used as driver amplifiers and sometimes as output amplifiers.

CLASS B 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 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.



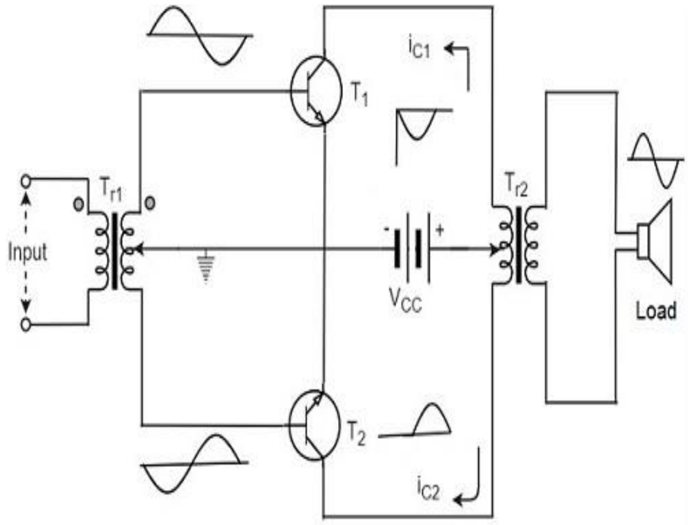
When the signal is applied, the circuit is forward biased for the positive half cycle of the input and hence the collector current flows. But during the negative half cycle of the input, the circuit is reverse biased and the collector current will be absent. Hence only the positive half cycle is amplified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

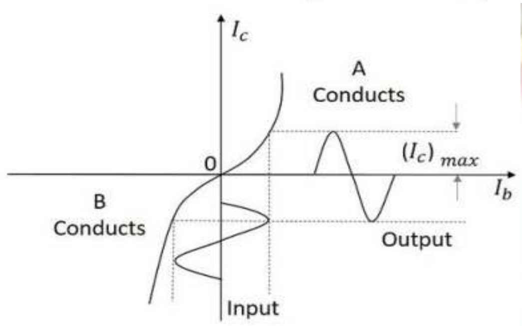
As the negative half cycle is completely absent, the signal distortion will be high. Also, when the applied signal increases, the power dissipation will be more. But when compared to class A power amplifier, the output efficiency is increased.

Class B Push-Pull Amplifier

The circuit of a push-pull class B power amplifier consists of two identical transistors T_1 and T_2 whose bases are connected to the secondary of the center-tapped input transformer T_{r1}. The emitters are shorted and the collectors are given the V_{CC} supply through the primary of the output transformer T_{r2}.

The transistors are biased at cut off, instead of using the biasing resistors.





When no signal is applied at the input, the transistors T_1 and T_2 are in cut off condition and hence no collector currents flow. As no current is drawn from V_{CC}, no power is wasted.

When input signal is given, it is applied to the input transformer T_{r1} which splits the signal into two signals that are 180^o out of phase with each other. These two signals are given to the two identical transistors T_1 and T_2. For the positive half cycle, the base of the transistor T_1 becomes positive and collector current flows. At the same time, the transistor T_2 has negative half cycle, which throws the transistor T_2 into cutoff condition and hence no collector current flows.

For the next half cycle, the transistor T_1 gets into cut off condition and the transistor T_2 gets into conduction, to contribute the output. Hence for both the cycles, each transistor conducts alternately. The output transformer T_{r3} 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, I_{dc} is given by

$$I_{dc} = \frac{(I_C)_{max}}{\pi}$$

Therefore,

$$(p_{in})_{dc} = 2 \times \left[\frac{(I_C)_{max}}{\pi} \times V_{CC}\right]$$

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

R.M.S. value of collector current = (IC)max/2–√(IC)max/2

R.M.S. value of output voltage = VCC/2–√VCC/2

Under ideal conditions of maximum power

Therefore,

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

Now overall maximum efficiency

$$\eta_{overall} = \frac{(P_O)_{ac}}{(P_{in})_{dc}}$$

$$= \frac{(I_C)_{max} \times V_{CC}}{2} \times \frac{\pi}{2(I_C)_{max} \times V_{CC}}$$

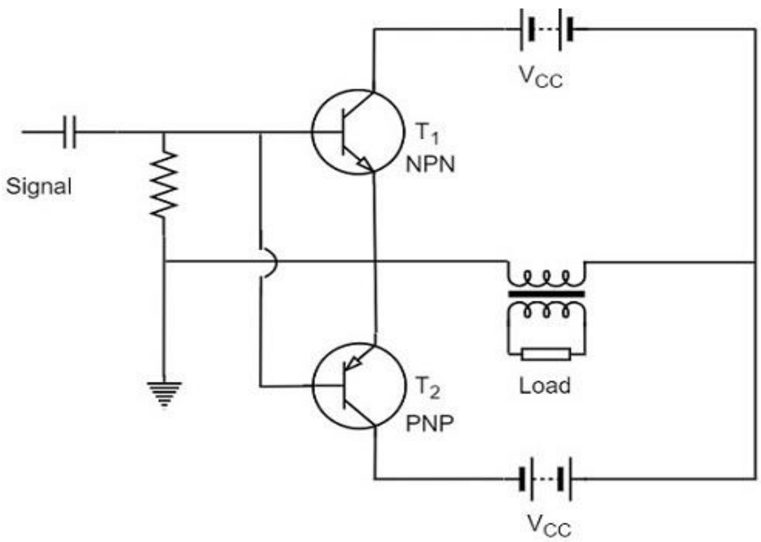
$$= \frac{\pi}{4} = 0.785 = 78.5\%$$

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 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,

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.



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

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

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

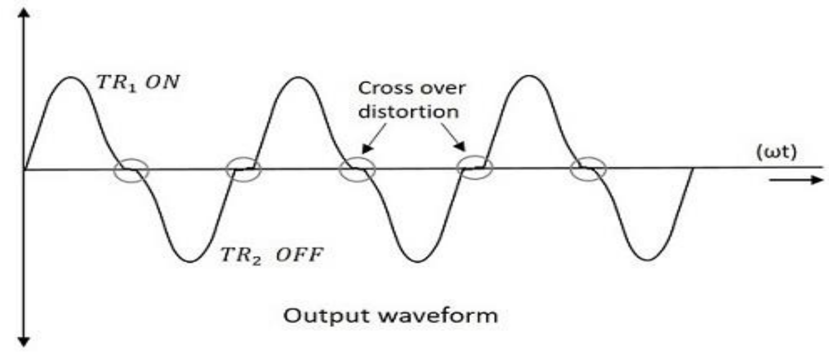
We require both positive and negative supply voltages.

Cross-over Distortion

In the push-pull configuration, the two identical transistors get into conduction, one after the other and the output produced will be the combination of both.

When the signal changes or crosses over from one transistor to the other at the zero voltage point, it produces an amount of distortion to the output wave shape. For a transistor in order to conduct, the base emitter junction should cross 0.7v, the cut off voltage. The time taken for a transistor to get ON from OFF or to get OFF from ON state is called the transition period.

At the zero voltage point, the transition period of switching over the transistors from one to the other, has its effect which leads to the instances where both the transistors are OFF at a time. Such instances can be called as Flat spot or Dead band on the output wave shape.



The above figure clearly shows the cross over distortion which is prominent in the output waveform. This is the main disadvantage. This cross over distortion effect also reduces the overall peak to peak value of the output waveform which in turn reduces the maximum power output.

This cross over distortion can be eliminated if the conduction of the amplifier is more than one half cycle, so that both the transistors won't be OFF at the same time.

UNIT II

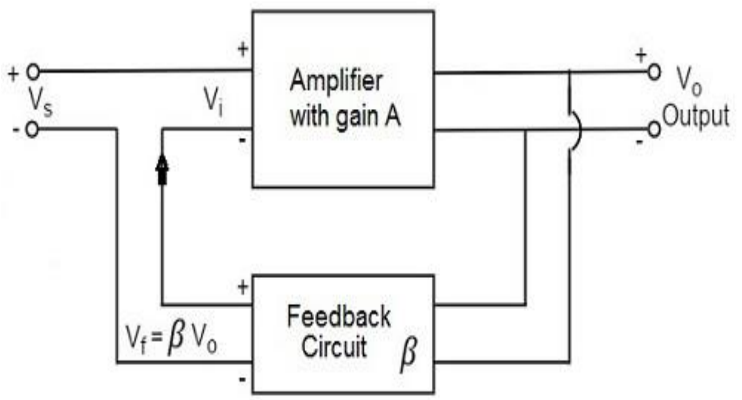
FEEDBACK AMPLIFIERS

Definition

The process by which some part or fraction of output is combined with the input is known as feedback. There exists a feedback factor between the generated feedback signal and the applied input signal.

Principle of Feedback Amplifier

A feedback amplifier generally consists of two parts. They are the amplifier and the feedback circuit. The feedback circuit usually consists of resistors.



The gain of the amplifier is represented as A. the gain of the amplifier is the ratio of output voltage V_o to the input voltage V_i. the feedback network extracts a voltage

$V_f = \beta V_o$ from the output V_o of the amplifier.

This voltage is added for positive feedback and subtracted for negative feedback, from the signal voltage V_s.

$$V_i = V_s + V_f = V_s + \beta V_o$$

$$V_i = V_s - V_f = V_s - \beta V_o$$

The quantity $\beta = V_f/V_o$ is called as feedback ratio or feedback fraction.

Types of Feedback

The process of injecting a fraction of output energy of some device back to the input is known as Feedback. It has been found that feedback is very useful in reducing noise and making the amplifier operation stable.

Depending upon whether the feedback signal aids or opposes the input signal, there are two types of feedbacks used.

Positive Feedback

Negative Feedback

Positive Feedback

The feedback in which the feedback energy i.e., either voltage or current is in phase with the input signal and thus aids it is called as Positive feedback.Both the input signal and feedback signal introduces a phase shift of 180° thus making a 360° resultant phase shift around the loop, to be finally in phase with the input signal.

It has the disadvantages such as

Increasing distortion

Instability

Negative Feedback

The feedback in which the feedback energy i.e., either voltage or current is out of phase with the input and thus opposes it, is called as negative feedback.

In negative feedback, the amplifier introduces a phase shift of 180° into the circuit while the feedback network is so designed that it produces no phase shift or zero phase shift. Thus the resultant feedback voltage V_f is 180° out of phase with the input signal V_{in}.

Let us consider the case of negative feedback. The output V_o must be equal to the input voltage (V_s - βV_o) multiplied by the gain A of the amplifier.

Hence $(V_s - \beta V_o)A = V_o$

$$AV_s - A\beta V_o = V_o$$

$$AV_s = V_o(1 + A\beta)$$

Therefore,

$$\frac{V_o}{V_s} = \frac{A}{1 + A\beta}$$

Let A_f be the overall gain (gain with the feedback) of the amplifier. This is defined as the ratio of output voltage V_o to the applied signal voltage V_s, i.e.,

$$A_f = \frac{Output\ voltage}{Input\ signal\ voltage} = \frac{V_o}{V_s}$$

So, from the above two equations, we can understand that,

The equation of gain of the feedback amplifier, with negative feedback is given by

$$A_f = \frac{A}{1 + A\beta}$$

The equation of gain of the feedback amplifier, with positive feedback is given by

$$A_f = \frac{A}{1 - A\beta}$$

The voltage gain of the positive feedback must be

$(1- \beta Av) <1$

Then $A'v> Av$

When the product βAv becomes unity then $(1- \beta Av) = 0$ which makes $Av = \infty$.

Thus positive feedback produces infinite gain.

This feedback is called as regenerative feedback.

In a negative feedback $(1 + \beta Av) >1$ which makes the gain smaller. Thus negative feedback reduces the gain of the amplifier. This is called as degenerative feedback.

Types of feedback amplifiers:

Voltage Series Feedback Amplifier

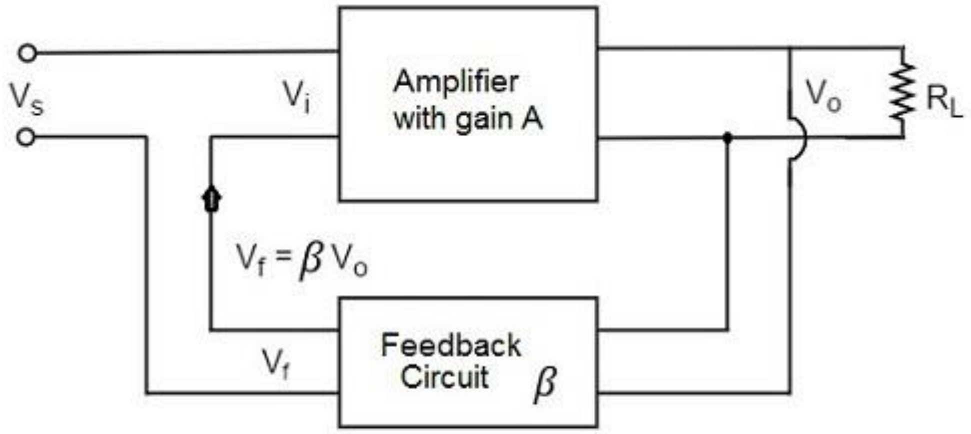
Voltage Shunt Feedback Amplifier

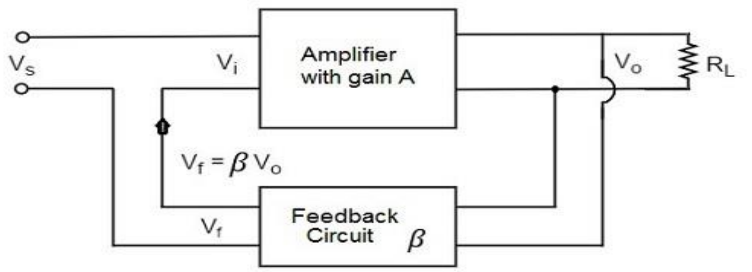
Current Series Feedback Amplifier

Current Shunt Feedback Amplifier

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.





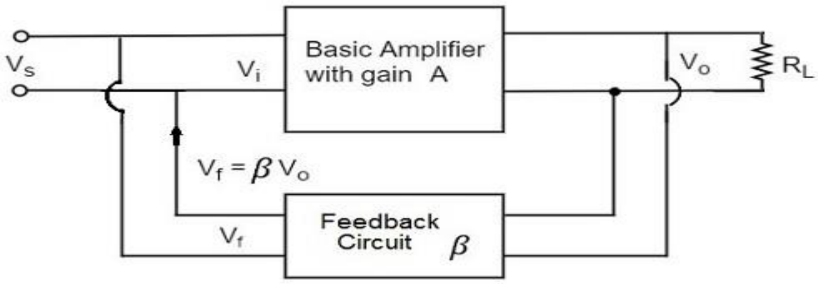
In this circuit the feedback circuit is placed in shunt with the output but in series with

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.

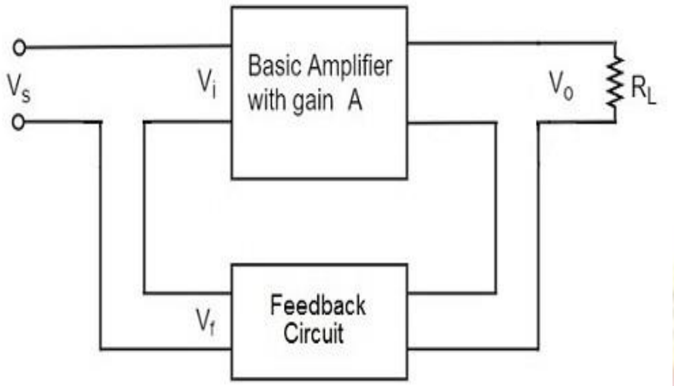
In voltage shunt feedback, the feedback circuit is placed in shunt with the output and also with the input.



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

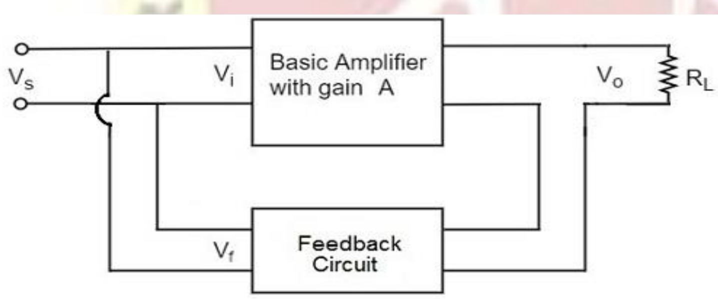
In the current series feedback circuit, a fraction of the output voltage is applied in series with the input voltage through the feedback circuit. This is also known as series-driven series-fed feedback i.e., a series-series circuit.



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.

Feedback Topology	Input Resistance	Output Resistance
Voltage Series	Increases Rif = Ri*(1+A*β)	Decreases Rof=Ro/(1+A*β)
Current Series	Increases Rif = Ri*(1+A*β)	Increases Rof = Ro*(1+A*β)
Current Shunt	Decreases Rif = Ri/(1+A*β)	Increases Rof = Ro*(1+A*β)
Voltage Shunt	Decreases Rif = Ri*(1+A*β)	Decreases Rof=Ro/(1+A*β)

UNIT III

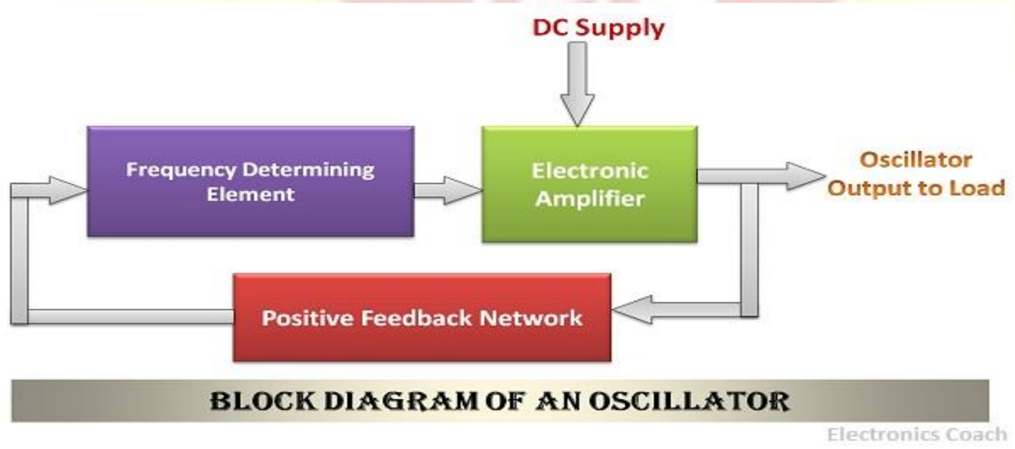
OSCILLATORS

Definition

Oscillator is a circuit which utilizes positive feedback amplifier to generate sinusoidal waveforms of fixed amplitude and frequency.

It is the major source of power in electrical and electronic instruments.

The amplifier provided with the positive feedback can generate the sinusoidal signal even in the absence of any input. These signals are termed as oscillations, and hence the device is known as an oscillator.



TYPE OF OSCILLATORS	APPROXIMATE RANGE
Audio-frequency Oscillators	20 Hz - 20 kHz
Radio-Frequency Oscillators	20 kHz - 30 MHz
Very low-frequency Oscillators	15 - 100 kHz
Low frequency Oscillators	100 - 500 kHz
Broadcast Oscillators	500 kHz - 1.5 MHz
Video - Frequency Oscillators	0 - 5 MHz
High - Frequency Oscillators	1.5 - 30 MHz
Very high - frequency Oscillators	30 - 300 MHz
Ultra - high frequency Oscillators	300 - 3000 MHz
Microwave Oscillators	Beyond 3 GHz (3000 MHz)

The positive feedback here implies the addition of some part of the signal from the output with the input signal voltage. This is allowed to pass through the amplifier circuit. The amplifier passes it with the input signal coming from the source. The amplifier does nothing other than adding the signal coming from the feedback path and the input signal.

Thus, continuous oscillations are generated, and at a time a stage is reached when without any input signal the oscillator circuit generates waveforms.

The oscillator does not generate the energy of its own, to generate oscillation but uses DC source to convert the DC power into AC. Therefore it is also termed as an inverter which is the opposite of rectifier.

The oscillators are available in wide range of frequencies. According to the frequency of the oscillation, the oscillators are named accordingly.

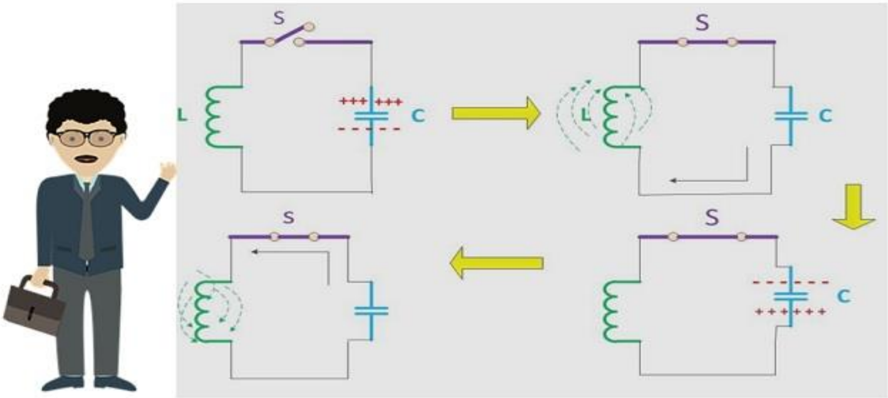
Oscillatory Circuit

The circuit of the oscillator is termed as the tank circuit. It consists of the capacitor and an inductor.

The capacitor is already charged and connected to an inductor with the switch between them. When the switch is open, nothing will happen.

When the switch is closed the charge stored by the capacitor will start discharging, and the electrons will start flowing in the circuit.

The direction of flow of current in the circuit will be opposite to the direction of flow of electrons.



When the current flows in the circuit, the current will pass from inductor due to which magnetic field is generated.

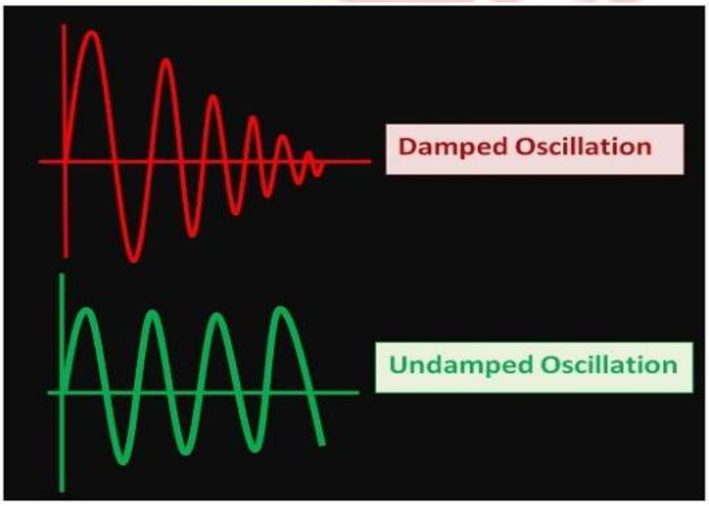
The current flowing in the circuit will create the flux which in result creates the magnetic field.Due to the magnetic fields around inductor, the energy is stored in the inductor in the form of magnetic field.

When the capacitor is fully discharged the current flowing in the circuit will get stopped and the magnetic field generated around the inductor will generate emf.

Lenz law which states the magnetic field created will oppose the cause which has created it. The emf thus induced will cause current to flow again in the LC circuit.

As a result of induced emf the charge will flow and it will be stored in the capacitor. The capacitor will store the energy in the form of the electrostatic field.

This process of charging and discharging of the capacitor and inductor will continue until the process is terminated externally.



In this way, the capacitor will charge at a time, and then inductor will charge at another point of time. Therefore, oscillations will generate continuously.

The oscillations generated by an oscillator are damped oscillation. Ideally, it is considered that the oscillation generated by the oscillator is undamped and continuous sinusoidal but practically, this is not possible.

The resistor suffers from dielectric loss while inductor suffers from radiation and resistive loss.

Frequency of the Oscillator Circuit

The frequency of the Oscillator circuit is called as the resonant frequency. The resonant frequency is expressed in terms of inductance and capacitance. The resonant frequency is inversely proportional to capacitance as well as inductance.

Frequency of Oscillation (resonant frequency) f_r

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

Capacitance of Oscillator (C)

$$f_r \propto \frac{1}{\sqrt{C}}$$

Inductance of Coil (L)

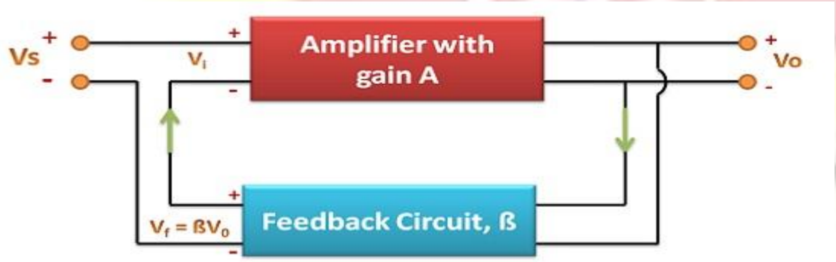
$$f_r \propto \frac{1}{\sqrt{L}}$$

Combining above two equation

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

Principle of Oscillator and Barkhausen Criterion

The principle of the oscillator is that when the feedback factor or the loop gain is one, then the overall gain of the oscillator circuit will be infinite.



Thus even when there is no input the oscillator will continue to generate the output.

BARKHAUSEN CRITERION

$$V_i = V_s + V_f = V_s + ß\, V_0, \text{ here } ß = \frac{V_f}{V_0}$$

$$(V_s + ßV_0)A = V\, ß\, V_0$$

$$AV_s + A\, ß\, V_0 = V_0$$

$$AV_s = V_0\,(1\text{-}Aß)$$

$$\text{Therefore, } \frac{V_0}{V_s} = \frac{A}{1-Aß}$$

$$A_f = \frac{Output\ Voltage}{Input\ Voltage} = \frac{V_0}{V_s}$$

$$A_f = \frac{A}{1-Aß}$$

Where Aß is the feedback factor or loop gain, if Aß is 1 then $A_f = \infty$, thus the gain becomes infinity, that is there if output without any input. Therefore, amplifier in this condition works as an oscillator. This condition of Aß = 1, is called Barkhausen Criterion.

This is the essential condition for the amplifier to acts as the feedback.

Thus an amplifier which uses positive feedback and which possess infinite overall gain is termed as the oscillator circuit.

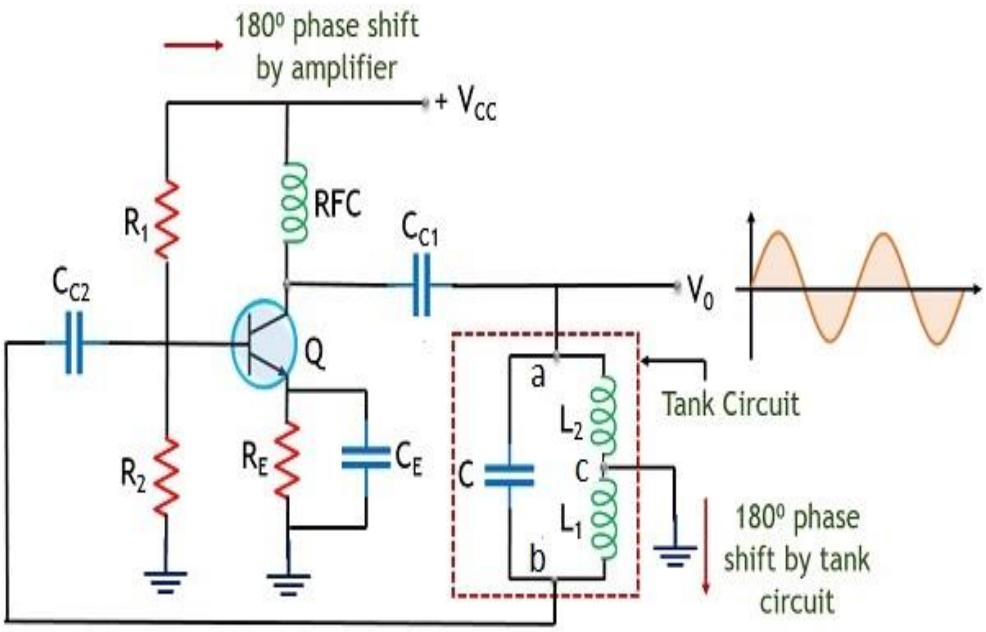
Hartley Oscillator (LC Oscillator)

Hartley oscillator is a type of LC oscillator that generates undamped sinusoidal oscillations whose tank circuit consists of 2 inductors and a capacitor. In the tank circuit, the two inductive coils are serially connected together forming a parallel combination with the capacitor.

This oscillator is a type of harmonic oscillator. The Hartley Oscillator produces the waves of radio frequencies therefore it is also referred to as radio frequency oscillators. In normal LC oscillators, the circuit generates an uncontrollable amplitude of oscillations. But Hartley oscillator uses an LC parallel feedback configuration that has a self-tuning base oscillator circuit.

Construction

The circuit consists of NPN transistor connected in common emitter configuration which acts as an amplifier.R1 and R2 are biasing resistors and RFC is the radio frequency choke which provides the isolation between AC and DC operation. R1 AND R2 resistors form voltage divider bias network for the transistor in common- emitter configuration.



Circuit Diagram of Hartley Oscillator

L1, L2 and C form the tank circuit. C_{C1} and C_{C2} are the coupling capacitors.A phase shift of 180⁰ is provided by the transistor amplifier present in the circuit.

In high-frequency applications, the reactance of RFC becomes very large, thus can be considered as open-circuited.

Working

When dc supply voltage V_{cc} is provided to the circuit, then with the increase in the collector current of the transistor, the capacitor in the tank circuit starts charging. Once the capacitor gets fully charged then the capacitor begins to discharge through inductor L_1 and L_2.This results in charging of the inductor. The inductor stores the charge in the form of the magnetic field. This continuous charging and discharging of capacitor and inductor will provide sinusoidal oscillations at the output.

These oscillations are damped oscillations, as amplitude is decreasing continuously. Thus it is required to amplify the oscillations to make it a damped one.

To overcome this problem, the output of the tank circuit is provided as input to the common emitter configuration transistor. The sinusoidal signal when provided to the transistor gets amplified.

Thus the oscillating frequency is given as

$$f = \frac{1}{2\pi\sqrt{L_{eq}C}}$$

Advantages of Hartley Oscillator

It provides sinusoidal oscillations of constant amplitude.

The oscillating frequency can be changed by the use of a variable capacitor.

The circuit is not complex.

The amplitude of the output remains constant over the working frequency range.

Practically a single tapped coil can also be used in place of 2 inductors in the circuit.

Disadvantages of Hartley Oscillator

Due to the presence of harmonics, sometimes distorted sinusoidal oscillations are generated.

It cannot be used in low-frequency applications.

Applications of Hartley Oscillator

Hartley oscillators widely used in the generation of sinusoidal waveforms of a certain frequency. Thus these are suitable for radio-frequency applications, thus find their use in radio receivers.

The Hartley oscillator is used as a local oscillator in radio receivers. This oscillator is suitable for oscillations in Radio Frequency (RF) range up to 30MHz.

Colpitts Oscillator (LC Oscillator)

Colpitts oscillator is a type of LC oscillator that generates undamped sinusoidal oscillations whose tank circuit consists of 2 capacitors and an inductor.In the tank circuit, the two capacitors are serially connected together forming a parallel combination with the Inductor.It is another type of sinusoidal LC oscillator and is basically a harmonic oscillator, which has a lot of applications.

The frequency of oscillations is determined by the value of the capacitors and inductor in the tank circuit.Colpitts oscillator is generally used in RF applications and the typical operating range is 20KHz to 300MHz.In Colpitts oscillator, the capacitive voltage divider setup in the tank circuit works as the feed back source and this arrangement gives better frequency stability when compared to the Hartley oscillator.

Construction

The circuit consists of NPN transistor connected in common emitter configuration which acts as an amplifier.R1 and R2 are biasing resistors and RFC is the radio frequency choke which provides the isolation between AC and DC operation. C1, C2 and L form the tank circuit. C_{C1} and C_{C2} are the coupling capacitors. A phase shift of 180^o is provided by the transistor amplifier present in the circuit.

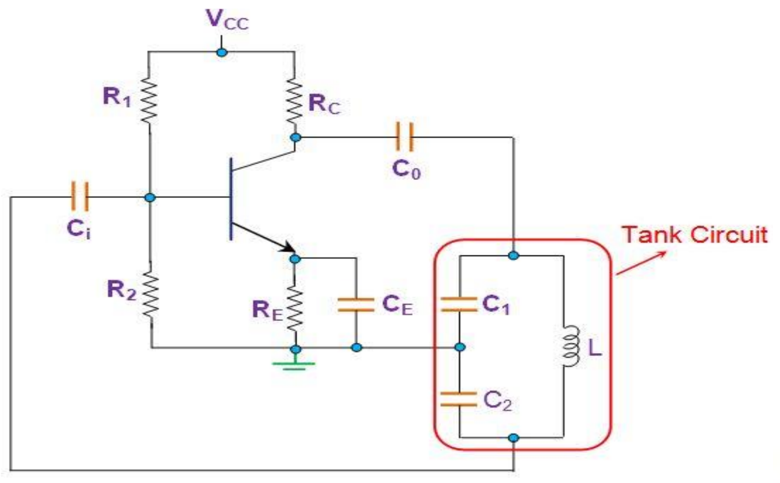


Figure 1 Colpitts Oscillator

When the power supply is switched on, the capacitors C1 and C2 start charging and after the capacitors get fully charged, the capacitors starts discharging through the inductor L1 in the circuit causing damped harmonic oscillations in the tank circuit.During this process, the electrostatic energy stored in the capacitor gets converted into **magnetic flux** which in turn is stored within the inductor in the form of electromagnetic energy.

The inductor starts to discharge which **charges the capacitors** once again and the cycle continues which gives rise to the oscillations in the tank circuit.

These oscillations are damped oscillations, as amplitude is decreasing continuously. Thus it is required to amplify the oscillations to make it a undamped one.To overcome this problem, the output of the tank circuit is provided as input to the common emitter configuration transistor. The sinusoidal signal when provided to the transistor, gets amplified. This amplified output compensates for the losses generated by the tank circuit. Thus the tank circuit provides continuous sinusoidal oscillations of constant amplitude at the output.

The tank circuit provides a phase shift of $180^o$and also, a phase shift of 180^o is provided by the transistor amplifier present in the circuit.

Frequency of Oscillations in Colpitts Oscillator

The frequency of oscillations of the sinusoidal signal generated by the tank circuit is given as

$$f = \frac{1}{2\pi\sqrt{LC}}$$

This circuit has 2 capacitors in the tank circuit thus equivalent capacitance will be given as

$$\frac{1}{C_T} = \frac{1}{C_1} + \frac{1}{C_2}$$

$$C_T = \frac{C_1 \times C_2}{C_1 + C_2}$$

The equation for frequency of Colpitts oscillator is given by

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

Advantages:

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.

Applications

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.

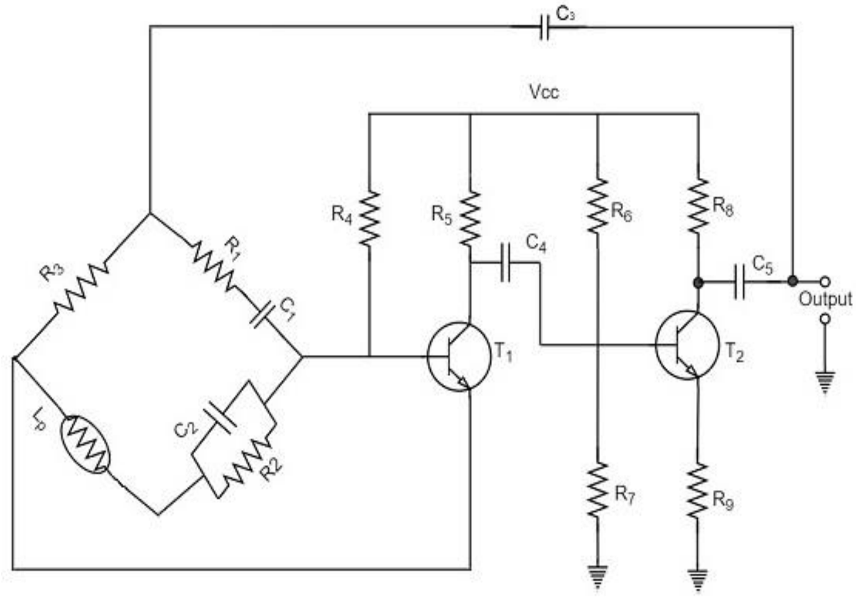
Wien Bridge Oscillator (RC Oscillator)

Wien Bridge Oscillator is an oscillator which uses RC network so as to produce a sine wave at the output. These are basically the low-frequency oscillator that generates audio and sub audio frequency that ranges between 20 Hz to 20 KHz.This **oscillator** circuit uses the Wien Bridge to provide feedback with the desired phase shift. It gives highly stable oscillation frequency and does not vary much with supply or temperature variation.

The Wien bridge feedback network is used so as to make the oscillator sensitive to signal of only a particular frequency.At this particular frequency, the Wien bridge gets balanced and provides a phase shift of 0^o.The Wien bridge circuit is a lead-lag network as with the rise in frequency phase shift lags and with the reduction in frequency, it leads.

Construction

The circuit mainly comprised of two transistors Q_1 and Q_2 and Wien bridge circuit. A series of RC circuit R_1C_1 is connected with a parallel RC circuit consisting of R_2C_2.At low-frequency range, the reactance of serially connected capacitor C_1 is very high due to which it acts as an open circuit that results in blocking of an input signal which resultantly gives no signal at the output.



Similarly, at a higher frequency, the reactance of parallel capacitor C_2 becomes very low thus behaving like a short circuit across the output, which again results in no signal at the output.So, there is a need to choose a frequency point in between the above two conditions.

The frequency at which the oscillator provides maximum output is known as resonant frequency.

Let us derive an expression for resonant frequency using the bridge.

$$R_3\left[\frac{R_2}{1+j\omega C_2R_2}\right] = R_4\left[R_1 - \frac{j}{\omega C_1}\right]$$

$$R_2R_3 = R_4(1 + j\omega C_2R_2)(R_1 - j/\omega C_1)$$

$$R_2R_3 - R_4R_1 - R_2R_4 + \frac{jR_4}{\omega C_1} - j\omega C_2R_2R_1R_4 = 0$$

By separating real and imaginary terms we can get,

$$R_2R_3 - R_4R_1 - \frac{C_2}{C_1} R_2R_4 = 0$$

$$or\ \frac{C_2}{C_1} = \frac{R_3}{R_4} - \frac{R_1}{R_2}$$

$$\frac{R_4}{\omega C_1} - \omega C_2R_2R_1R_4 = 0$$

$$\omega^2 = \frac{1}{C_1C_2R_1R_2}$$

$$\omega = \frac{1}{\sqrt{C_1C_2R_1R_2}}$$

$$f = \frac{1}{2\pi\sqrt{R_1R_2C_1C_2}}$$

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

$$f = \frac{1}{2\pi CR}$$

$$and\ R_3 = 2R_4$$

This frequency is known as the resonant frequency of the oscillator.

At this frequency the reactance of the circuit becomes equal to its resistance thus the resulting phase difference between input and output will be 0^o and in this condition magnitude of output becomes maximum. Here the transistor Q_1 behaves as an oscillator and amplifier whereas; Q_2 behaves as an inverter that causes a phase shift of 180^o.

The circuit uses both positive feedback and negative feedback. The positive feedback is given through R_1, C_1, R_2, C_2 to transistor Q_1 and the negative feedback is given through voltage divider R_3-R_4 to emitter section of transistor Q_1.The amplitude of the output is stabilized by resistor R_3 and R_4.

Suppose we have resistor R = 20KΩ and capacitor C = 1000pF.So the frequency of oscillations is given by:

$$f = \frac{1}{2\pi RC} = \frac{1}{2\pi X 20,00 X 1 X 10^{-9}} = 7.9\ KHz$$

Advantages

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 circuit cannot generate very high frequencies.

Two transistors and number of components are required for the circuit construction.

Phase-shift Oscillator (RC Oscillator)

The oscillator circuit that produces a sine wave using a phase-shift network is called as a Phase-shift oscillator circuit.

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 60^o so that the total phase shift produced by RC network is 180^o.

Operation

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

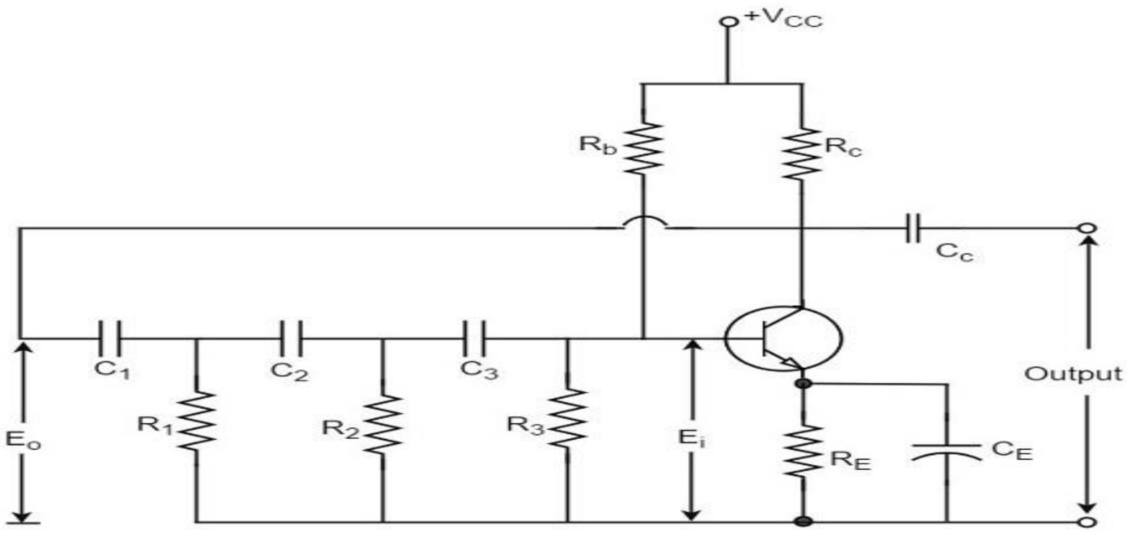
The frequency of oscillations is given by

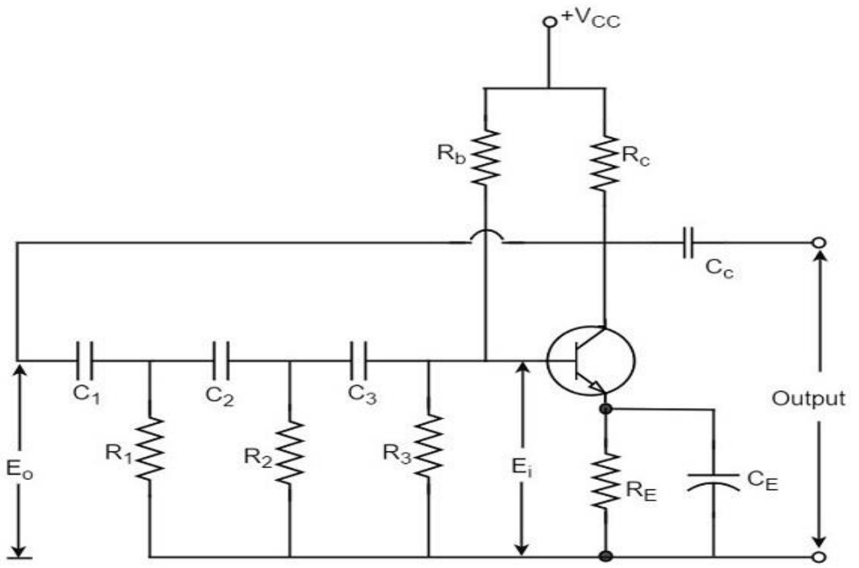
$$f_o = \frac{1}{2\pi RC\sqrt{6}}$$

Where

$$R_1 = R_2 = R_3 = R$$

$$C_1 = C_2 = C_3 = C$$





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

Advantages

It does not require transformers or inductors.

It can be used to produce very low frequencies.

The circuit provides good frequency stability.

Disadvantages

Starting the oscillations is difficult as the feedback is small.

The output produced is small.

UNIT IV

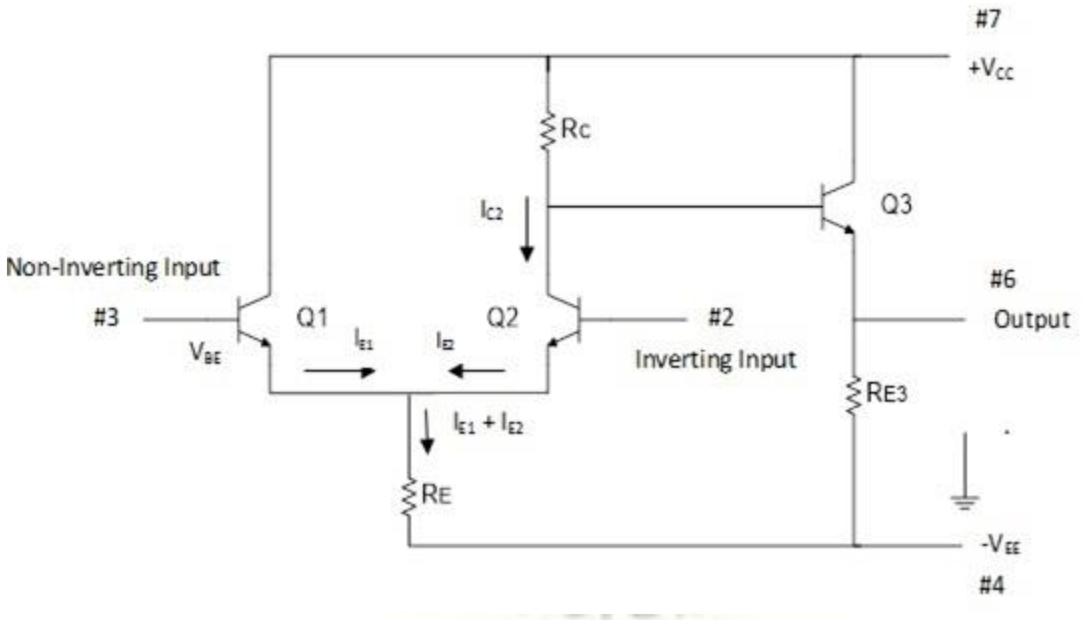
Operational Amplifier

Introduction

An operational amplifier commonly known as op-amp is a two-input single-output differential voltage amplifier which is characterized by high gain, high input impedance and low output impedance.

The operational amplifier is called so because it has its origins in analog computers, and was mainly used to perform mathematical operations. Depending on its feedback circuit and biasing, an op-amp can be made to add, subtract, multiply, divide, negate, and interestingly even perform calculus operations like differentiation and integration.

Today, op-amps are very popular building blocks in electronic circuits. Op-amps are used for a variety of applications such as AC and DC signal amplification, filters, oscillators, voltage regulators, comparators and in most of the consumer and industrial devices. Op-amps exhibit little dependence on temperature-changes or manufacturing variations, which makes them ideal building blocks in electronic circuits.



The basic circuit of an operational amplifier is as shown in the figure above. An op-amp has a differential amplifier input stage and emitter follower output stage. Practical op-amp circuits are much more complicated than the above shown basic op-amp circuit.

Transistors Q1 and Q2 forms a differential amplifier, where the difference input voltage is applied to the base terminals of Q1and Q2. Transistor Q3 operates as an emitter follower and provides low output impedance.

The output of the basic op-amp circuit V_{OUT} is given as,

$V_{OUT} = V_{CC} - V_{RC} - V_{BE3}$

$V_{OUT} = V_{CC} - I_{C2}R_C - V_{BE}$

Where, V_{RC} is the voltage across the resistor R_C and V_{BE3} is the base-emitter voltage of transistor Q3.

Assume that the transistors Q1 and Q2 are matched transistors i.e., they have equal V_{BE} levels and equal current gains. If both transistors base terminals are connected to ground, the emitter currents I_{E1} and I_{E2} are equal and both I_{E1} and I_{E2} flow through the common resistor R_E. The emitter current is given by the relation,

$I_{E1} + I_{E2} = V_{RE} / R_E$

If both Q1 and Q2 bases connected to ground,

$0 – V_{BE} – V_{RE} + V_{EE} = 0$

i.e. $V_{RE} = V_{EE} – V_{BE}$

Therefore, $I_{E1} + I_{E2} = (V_{EE} – V_{BE}) / R_E$

When a positive voltage is applied to the non-inverting input terminal, the base of Q1 is pulled up by the input voltage and its emitter terminal follows the input signal. Since Q1 and Q2 emitters are connected together, the emitter of Q2 also gets pulled up by the positive input at the non-inverting terminal. The base of Q2 is grounded, so the positive voltage at its emitter causes a reduction in its base-emitter voltage V_{BE2}. The reduction in V_{BE2} causes the emitter current I_{E2} to decrease and consequently I_{C2} also reduced.

It can be noted that a positive input at pin #3 gives a positive output, hence the name non-inverting input terminal.

741 Op Amp (Operational Amplifier)

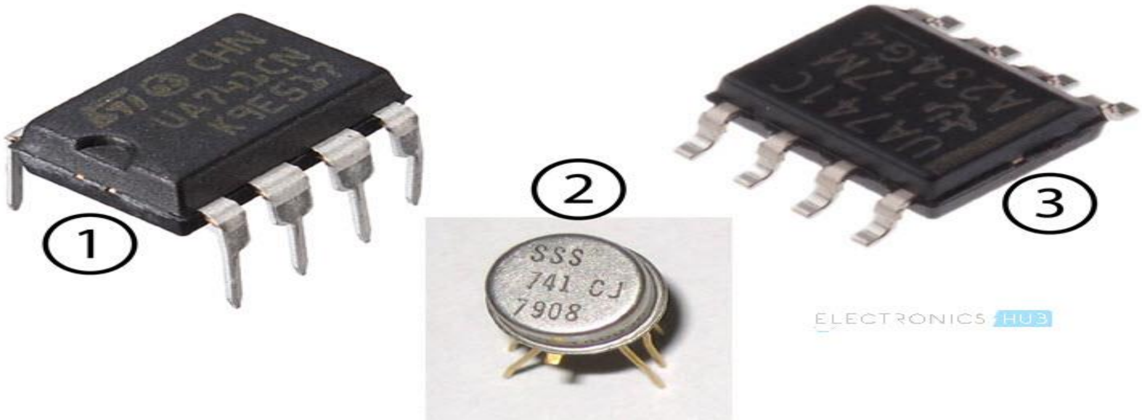
The 741 Op Amp IC is a monolithic integrated circuit, comprising of a general-purpose Operational Amplifier. It was first manufactured by Fairchild semiconductors in the year 1963. The number 741 indicates that this operational amplifier IC has 7 functional pins, 4 pins capable of taking input and 1 output pin.

IC 741 Op Amp can provide high voltage gain and can be operated over a wide range of voltages, which makes it the best choice for use in integrators, summing amplifiers and general feedback applications. It also features short circuit protection and internal frequency compensation circuits built in it. This Op-amp IC comes in the following form factors:

8 Pin DIP Package

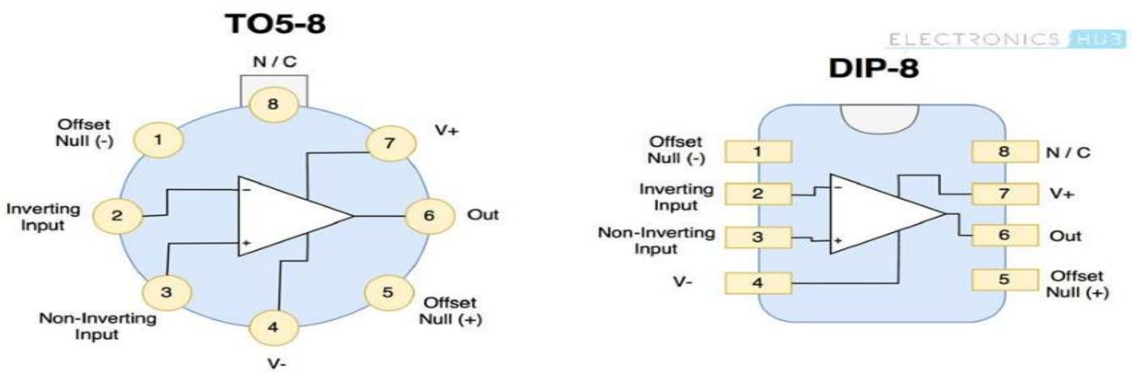
TO5-8 Metal can package

8 Pin SOIC



Pinout of IC 741 Op Amp and their Functions

The below figure illustrates the pin configurations and internal block diagram of IC 741 in 8 pin DIP and TO5-8 metal can package.



Pin-Outs of 741 IC

Functions of Different Pins Of 741 IC:

Pin4 & Pin7 (Power Supply): Pin7 is the positive voltage supply terminal and Pin4 is the negative voltage supply terminal. The 741 IC draws in power for its operation from these pins. The voltage between these two pins can be anywhere between 5V and 18V.

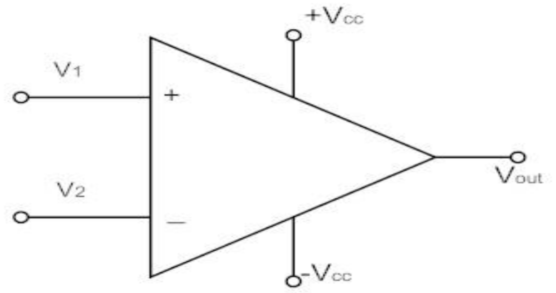
Pin6 (Output): This is the output pin of IC 741. The voltage at this pin depends on the signals at the input pins and the feedback mechanism used. If the output is said to be high, it means that voltage at the output is equal to positive supply voltage. Similarly, if the output is said to be low, it means that voltage at the output is equal to negative supply voltage.

Pin2 & Pin3 (Input): These are input pins for the IC. Pin2 is the inverting input and Pin3 is the non-inverting input. If the voltage at Pin2 is greater than the voltage at Pin3, i.e., the voltage at inverting input is higher, the output signal stays low. Similarly, if the voltage at Pin3 is greater than the voltage at Pin2, i.e., the voltage at non-inverting input is high, the output goes high.

Pin1 & Pin5 (Offset Null): Because of high gain provided by 741 Op-Amp, even slight differences in voltages at the inverting and non-inverting inputs, caused due to irregularities in manufacturing process or external disturbances, can influence the output. To nullify this effect, an offset voltage can be applied at pin1 and pin5, and is usually done using a potentiometer.

Pin8 (N/C): This pin is not connected to any circuit inside 741 IC. It's just a dummy lead used to fill the void space in standard 8 pin packages.

Op-Amp Symbol



Op-Amp Symbol

Note:

1. If an input signal is applied to either of the input terminals to the other input terminal connected to ground, the operation is called "single-ended".

In single-ended operation a single input applied, drives both the transistors due to the common-emitter connection. The output obtained is thus driven by both the collectors.

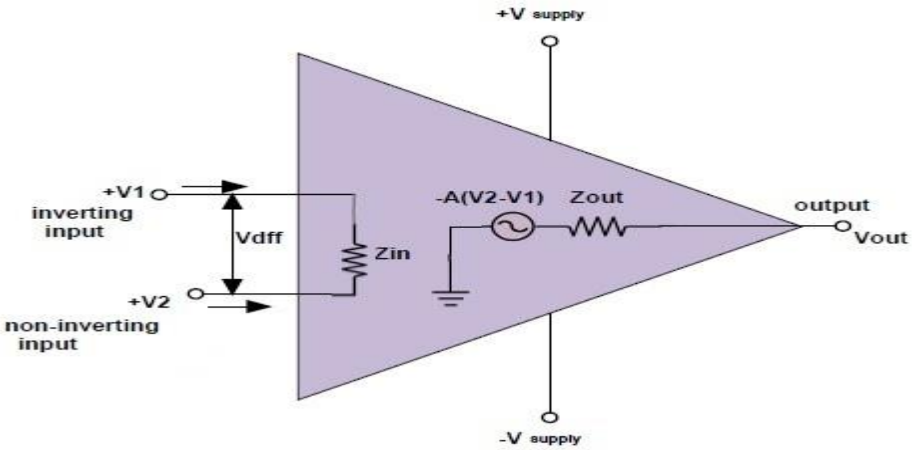
2. If two input signals are applied to the two input terminals the operation is referred to as "double-ended".

If the same input is applied to both inputs, the operation is called "common mode". In common-mode operation the common input signal at both the input terminals results in opposite signals at each collector. These signals get cancelled, resulting in an output signal zero. Practically, the opposite signals do not completely cancel each other and a small signal is resulted inthe output.

Equivalent Circuit of an Ideal Op-Amp

The equivalent circuit of an ideal op-amp is shown above. The input voltage V_{DIFF} is the difference voltage (V_1-V_2). Z_{in} is the input impedance and Z_{out} is the output impedance. The gain parameter A is called the open loop gain. If an op-amp does not have any feedback from the output to either of the inputs,it is said to be operating in open-loop configuration.

An ideal op-amp exhibits infinite open loop gain, infinite input impedance, zero output impedance, infinite voltage swing, infinite bandwidth, infinite slew rate and zero input offset voltage.



Operational Amplifier Characteristics

Input Impedance (Z_{in})

An ideal op-amp has infinite input impedance to prevent any flow of current from the supply into the op-amp circuit. But when the op-amp is used in linear applications, some form of negative feedback is provided externally. Due to this negative feedback, the input impedance becomes

$Z_{in} = (1 + A_{OL} \beta) Z_i$

Where, Z_{in} is the input impedance without feedback

A_{OL} is the open-loop gain

β is the feedback factor (1 for voltage follower)

Output Impedance (Z_{out})

An ideal op-amp has zero output impedance. This means that the output voltage is independent of output current. Thus an ideal op-amp can act as a perfect internal voltage source with zero internal resistance, so that maximum current can be driven to the load.

Practically, the output impedance of the op-amp is affected by the negative feedback and is given by,

$Z_{out} = Z_o / (1 + A_{OL} \beta)$

Where, Z_o is the output impedance of op-amp without feedback

A_{OL} is the open-loop gain

β is the feedback factor

Load impedances connected at the output of the op-amp must be much larger than the circuit output impedance, to avoid any significant loss of output as a voltage drop across Z_{out}.

Open-Loop Gain (A_{VO})

Open-loop gain of an op-amp is defined as the gain of the op-amp when there is no feedback from the output to either of its inputs. For an ideal op-amp, the gain will be infinite theoretically, but practical value range from 20,000 to 200,000.

Bandwidth (BW)

An ideal op-amp can amplify any frequency signal from DC to highest AC frequencies, thus it has an infinite frequency response. Therefore, the bandwidth of an ideal op-amp should be infinite. In practical circuits, the bandwidth of the op-amp is limited by the gain-bandwidth product (GB).

CMRR (Common Mode Rejection Ratio)

CMRR is defined as the ability of an op-amp to reject the common mode input signal. CMRR is an important measure of an op-amp. An ideal op-amp will have infinite CMRR. In practical circuits, CMRR is given by

$$CMRR = 20 \log_{10} (A_D/|A_C|) \text{ dB}$$

Where, A_D is the differential gain and A_C is the common mode gain of the op-amp.

Offset Voltage (V_{IO})

The input offset voltage defines the differential DC voltage required between the input terminals to make the output zero volts with respect to ground. An Ideal op-amp will have zero offset voltage, whereas practical op-amps show some small offset.

Slew Rate

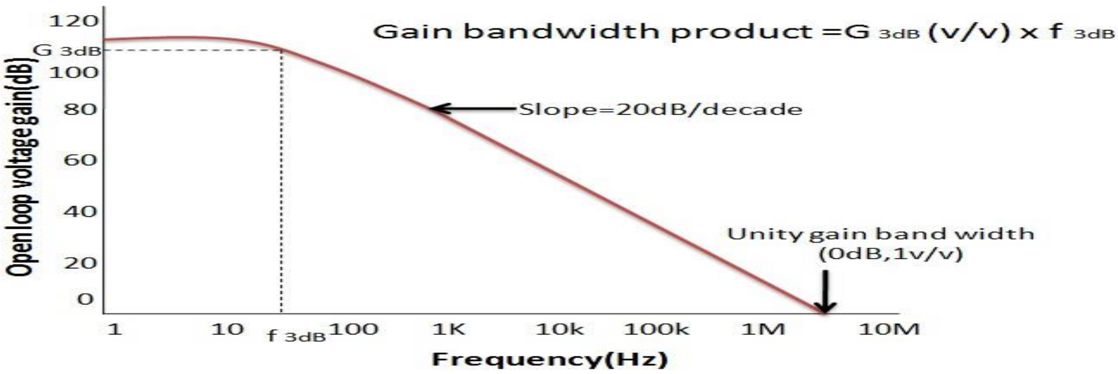
Slew rate is defined as the maximum change of output voltage per unit time and is expressed as volts per second. An ideal op-amp will have an infinite slew rate. In practical op-amps, the slew rate is inherently limited by the small internal drive currents of the op-amp and also by the internal capacitances designed to compensate for high frequency oscillations.

Op-Amp Characteristic Table

Parameter	Symbol	Ideal Op-Amp	Practical Op-Amp
DC Open loop gain	A_{OL}	∞	100 dB
Input Impedance	Z_{IN}	∞	2MΩ
Output Impedance	Z_{out}	0	75Ω
Input Offset Voltage	V_{IO}	0	1mV
Slew rate	SR	∞	Depends on input signal frequency
Bandwidth	BW	∞	Depends on input signal frequency
CMRR	ρ	∞	90 dB

Op-amp Frequency Response

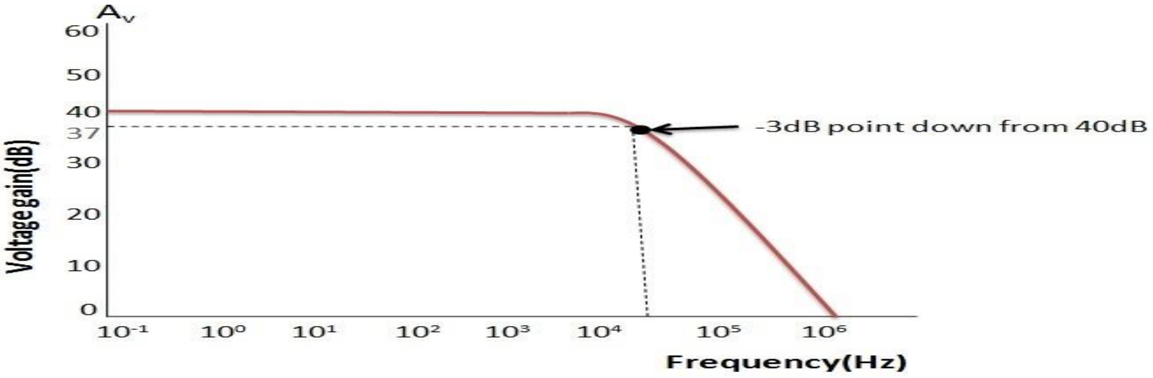
The open loop gain A_{OL} is not constant for all frequencies. Real op-amps have a frequency-dependant open-loop gain. The frequency response curve of a practical op-amp is as shown below.



From the above curve, we can note that the product of gain and frequency is constant at any point along the curve. This constant is known as the Gain-Bandwidth product (GB). Also, the gain of the amplifier at any point along the curve is determined by unity-gain (0 dB) frequency.

Operational Amplifier Bandwidth

The bandwidth of the operational amplifier is defined as the frequency range over which the voltage gain of the amplifier is above -3dB (maximum is 0dB) of its maximum output value.



In the above figure, the -3dB of the $A_{V(max)}$ is shown as 37dB. The 37dB line intersects with the curve at just over 10 kHz frequencies. This frequency can be more accurately calculated if the GB product of the amplifier is known. It can be noted that the open-loop gain decreases as the frequency of the input signal increases. Frequency is plotted in logarithmic scale and the gain decreases linearly as frequency increases logarithmically. The rate of fall of the gain in op amp is known to be 20 dB per decade.

Op-Amp Applications

Operational amplifiers are popular building blocks in electronic circuits and they find applications in most of the consumer and industrial electronic systems. Op-amps can be configured to work as different types of signal amplifiers like inverting, non-inverting, differential, summing, etc. as well as it is used to perform mathematical operations like addition, subtraction, multiplication, division and also differentiation and integration.

Operational amplifiers can be used in construction of active filters, providing high-pass, low-pass, band-pass, band-reject and delay functions. The high input impedance and gain of an op-amp allow straightforward calculation of element values, allows accurate implementation of any desired filter topology with little concern for the loading effects of stages in the filter or of subsequent stages.

An operational amplifier can, if necessary, be forced to act as a comparator. The smallest difference between the input voltages will be amplified considerably.

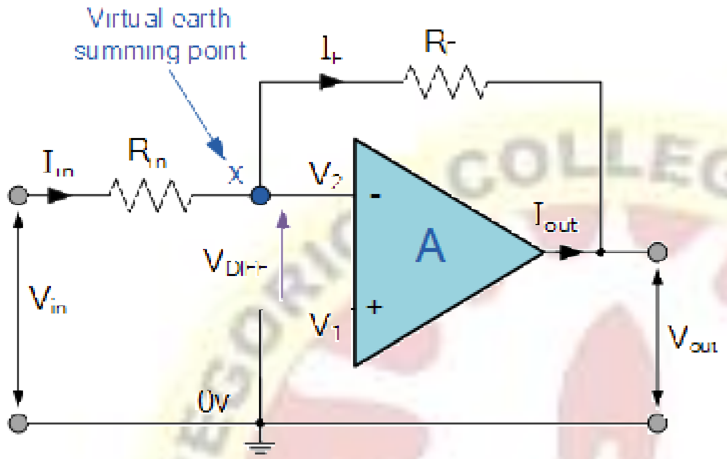
Op-amps are used in the construction of oscillators, like a Wein bridge oscillator. Op-amps are also used in non-linear circuits such as logarithmic and anti-logarithmic amplifiers.

Op-amps find applications as Voltage sources, Current sources, and Current sinks and also as DC & AC Voltmeters. Op-amps are also used in signal processing circuits such as Precision Rectifiers, Clamping circuits and Sample-and-Hold circuits.

Inverting Operational Amplifiers

The inverting amplifier is an important circuit configuration using op-amps and it uses a negative feedback connection. An inverting amplifier, like the name suggests, inverts the input signal as wells as amplifies it. A positive-going signal at the input of an inverting amplifier would result in a negative-going signal at the output and vice versa. An AC sinusoidal signal at the input would produce 180° out of phase sinusoidal signal at the output.

Inverting Operational Amplifier Circuit



In this Inverting Amplifier circuit the operational amplifier is connected with feedback to produce a closed loop operation. When dealing with operational amplifiers there are two very important rules to remember about inverting amplifiers, these are: “No current flows into the input terminal” and that “V1 always equals V2”. However, in real world op-amp circuits both of these rules are slightly broken.

This is because the junction of the input and feedback signal (X) is at the same potential as the positive (+) input which is at zero volts or ground then, the junction is a “Virtual Earth”. Because of this virtual earth node, the input resistance of the amplifier is equal to the value of the input resistor, Rin and the closed loop gain of the inverting amplifier can be set by the ratio of the two external resistors.

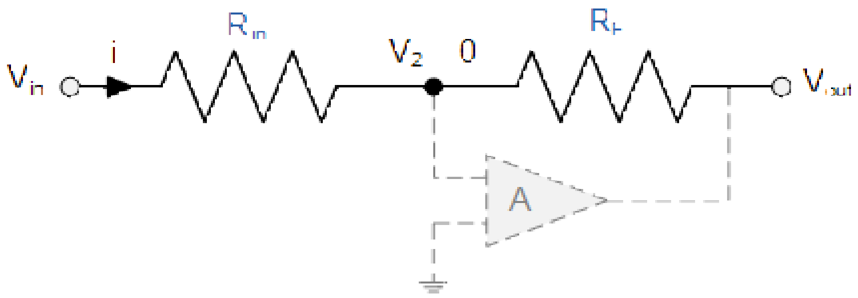
We said above that there are two very important rules to remember about Inverting Amplifiers or any operational amplifier for that matter and these are.

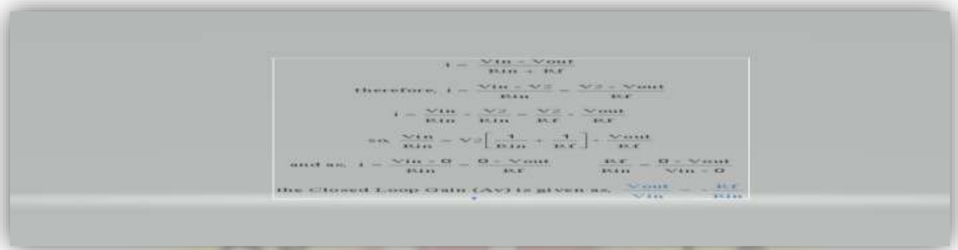
No Current Flows into the Input Terminals

The Differential Input Voltage is Zero as V1 = V2 = 0 (Virtual Earth)

Then by using these two rules we can derive the equation for calculating the closed-loop gain of an inverting amplifier, using first principles.

Current (i) flows through the resistor network as shown.





$$i = \frac{Vin - Vout}{Rin + Rf}$$

$$\text{therefore, } i = \frac{Vin - V2}{Rin} = \frac{V2 - Vout}{Rf}$$

$$i = \frac{Vin}{Rin} - \frac{V2}{Rin} = \frac{V2}{Rf} - \frac{Vout}{Rf}$$

$$\text{so, } \frac{Vin}{Rin} = V2\left[\frac{1}{Rin} + \frac{1}{Rf}\right] - \frac{Vout}{Rf}$$

$$\text{and as, } i = \frac{Vin - 0}{Rin} = \frac{0 - Vout}{Rf} \qquad \frac{Rf}{Rin} = \frac{0 - Vout}{Vin - 0}$$

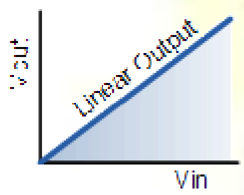
the Closed Loop Gain (Av) is given as, $\frac{Vout}{Vin} = -\frac{Rf}{Rin}$

Then, the Closed-Loop Voltage Gain of an Inverting Amplifier is given as.

$$\text{Gain (Av)} \quad \frac{V_{out}}{V_{in}} \quad = \; -\frac{R_f}{R_{in}}$$

and this can be transposed to give Vout as:

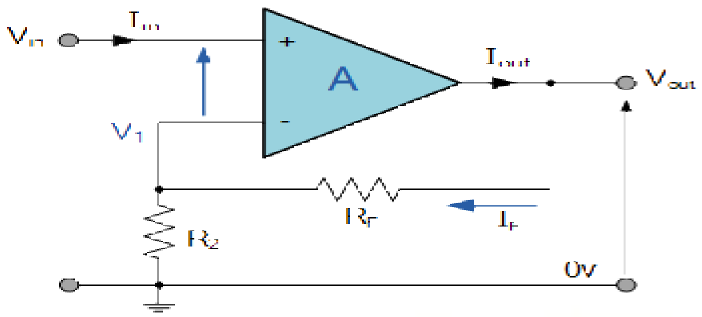
$$Vout = -\frac{Rf}{Rin} \times Vin$$



Linear Output

The negative sign in the equation indicates an inversion of the output signal with respect to the input as it is 180° out of phase. This is due to the feedback being negative in value.

Non-inverting Operational Amplifier

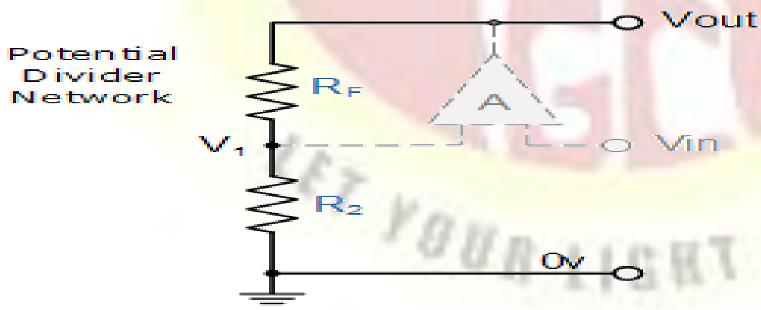


In this configuration, the input voltage signal, (VIN) is applied directly to the non-inverting (+) input terminal which means that the output gain of the amplifier becomes "Positive" in value in contrast to the "Inverting Amplifier" circuit whose output gain is negative in value. The result of this is that the output signal is "in-phase" with the input signal.

for an ideal op-amp "No current flows into the input terminal" of the amplifier and that "V1 always equals V2". This was because the junction of the input and feedback signal (V1) are at the same potential.

In other words, the junction is a "virtual earth" summing point. Because of this virtual earth node, the resistors, Rƒ and R2 form a simple potential divider network across the non-inverting amplifier with the voltage gain of the circuit being determined by the ratios of R2 and Rƒ as shown below.

Equivalent Potential Divider Network



Then using the formula to calculate the output voltage of a potential divider network, we can calculate the closed-loop voltage gain (AV) of the **Non-inverting Amplifier** as follows:

$$V_1 \quad \frac{R_2}{R_2 + R_F} \quad V_{OUT}$$

$$\text{Ideal Summing Point: } V_1 \quad V_{IN}$$

$$\text{Voltage Gain, } A_{(V)} \text{ is equal to: } \frac{V_{OUT}}{V_{IN}}$$

$$\text{Then, } A_{(V)} \quad \frac{V_{OUT}}{V_{IN}} \quad \frac{R_2 \quad R_F}{R_2}$$

$$\text{Transpose to give: } A_{(V)} = \frac{V_{OUT}}{V_{IN}} = 1 - \frac{R_F}{R_2}$$

Then the closed loop voltage gain of a **Non-inverting Operational Amplifier** will be given as:

$$A_{(V)} - 1 \quad \frac{R_F}{R_2}$$

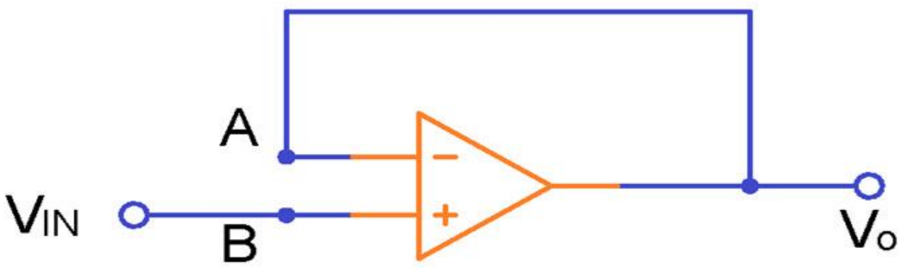
We can see from the equation above, that the overall closed-loop gain of a non-inverting amplifier will always be greater but never less than one (unity), it is positive in nature and is determined by the ratio of the values of $Rƒ$ and R2.

If the value of the feedback resistor $Rƒ$ is zero, the gain of the amplifier will be exactly equal to one (unity). If resistor R2 is zero the gain will approach infinity, but in practice it will be limited to the operational amplifiers open-loop differential gain, (A_O).

Voltage Follower Circuit (UNITY GAIN)

Voltage follower is one of the simplest uses of an operational amplifier, where the output voltage is exactly same as the input voltage applied to the circuit. In other words, the gain of a voltage follower circuit is unity.

The output of the op-amp is directly connected to the inverting input terminal, and the input voltage is applied to the non-inverting input terminal. The voltage follower, like a non-inverting amplifier, has very high input impedance and very low output impedance. The circuit diagram of a voltage follower is shown in the figure below.



It can be seen that the above configuration is the same as the non-inverting amplifier circuit, with the exception that there are no resistors used. The gain of a non-inverting amplifier is given as,

$A_{CL} = 1 + (R_1 / R_2)$

In the voltage follower, the resistor R1 is equal to zero and R2 is infinite. So the gain of the voltage follower will be equal to 1. A Voltage follower is also commonly known as a Unity Gain Buffer. The voltage follower or unity gain buffer circuit is commonly used to isolate different circuits, i.e. to separate one stage of the circuit from another and also used in impedance matching applications.

In practice, the output voltage of a voltage follower will not be exactly equal to the input voltage applied and there will be a slight difference. This difference is due to the high internal voltage gain of the op-amp.

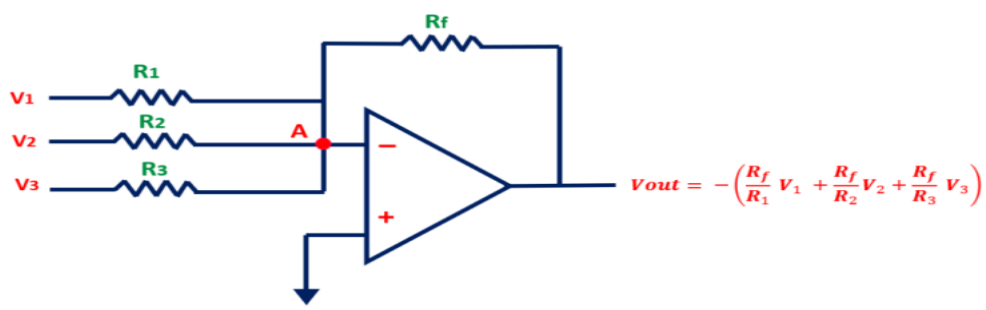
Summing Amplifier

One of the important applications of an Operational Amplifier is the Summing Amplifier or otherwise known as Adder. As the name suggests, a Summing Amplifier is an Op-amp based circuit where multiple input signals of different voltages are added.

Many applications in electronic circuits require two or more analog signals to be added or combined into a single output. The summing amplifier does the exact same thing.For this reason, summing amplifier is also called as Voltage adder since its output is the addition of voltages present at its input terminal.

INVERTING SUMMING AMPLIFIER

The summing amplifier uses an inverting amplifier configuration, i.e. the input is applied to the inverting input terminal of the op-amp, while the non-inverting input terminal is connected to ground. Due to this configuration, the output of voltage adder is out of phase with respect to the input by



As shown in Fig. the op-amp is used as a summing amplifier in the **inverting configuration**. The inputs to the op-amp (V1, V2, and V3) are applied using the resistors R1, R2, and R3 respectively. The output of the op-amp can be given as

$$Vout = -\left(\frac{R_f}{R_1} V_1 + \frac{R_f}{R_2} V_2 + \frac{R_f}{R_3} V_3\right)$$

Proof:

Here, assume the op-amp is an ideal op-amp. As shown in Fig.2, let's say, the current flowing through resistor R1, R2 and R3 are I1, I2, and I3. And current flowing through resistor R_f is I_F.

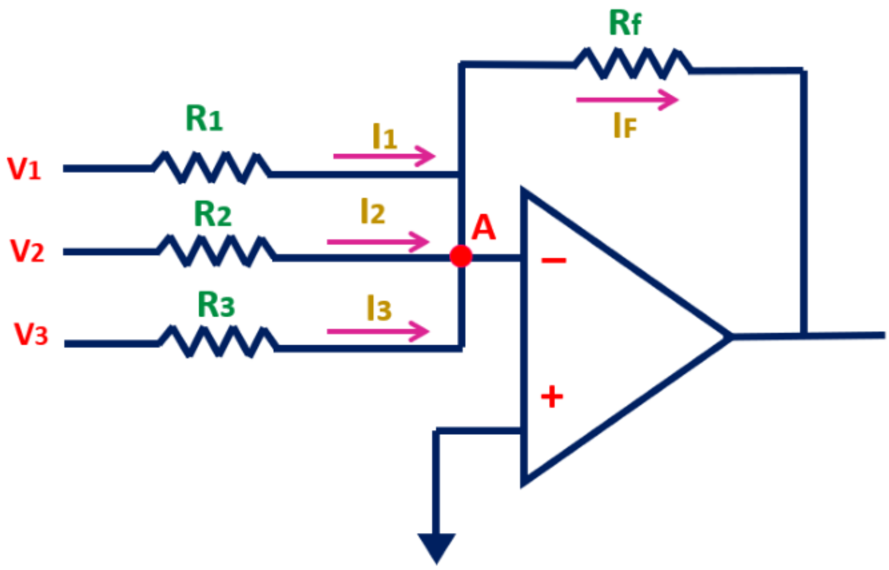


Fig.2 Currents in the Inverting Summing Amplifier

For an ideal op-amp, no current is flowing into the op-amp terminal. So, applying KCL at node A, it can be written as

$$I_1 + I_2 + I_3 = I_f$$

$$\frac{V_1 - 0}{R_1} + \frac{V_2 - 0}{R_2} + \frac{V_3 - 0}{R_3} = \frac{0 - V_{out}}{R_f} \qquad \frac{V_1}{R_1} + \frac{V_2}{R_2} + \frac{V_3}{R_3} = \frac{-V_{out}}{R_f}$$

$$Vout = -\left(\frac{R_f}{R_1} V_1 + \frac{R_f}{R_2} V_2 + \frac{R_f}{R_3} V_3\right)$$

If R1 = R2 = R3 = R then

$$Vout = -\frac{R_f}{R}(V_1 + V_2 + V_3)$$

When $R_f = R$, then Vout = – ($V_1 + V_2 + V_3$)

And when **R1 = R2 = R3 = R** and the **ratio of Rf and R** is selected such that,

$$\frac{R_f}{R} = \frac{1}{n}$$

where n=number of inputs being applied to the inverting terminal.

Then it can be used as a averaging circuit. For example, in the above circuit when $R_1 = R_2 = R_3 = R$ and $R = 3 R_f$ then $Vo = -(V_1 + V_2 + V_3)/3$

That means output is the average of the three input signals.

Applications of Summing Amplifier

Summing, Averaging, and Scaling

For providing DC offset

Digital to Analog Converters

Audio Mixer

Non-Inverting Summing Amplifier

A non-inverting summing amplifier can also be constructed, using the non-inverting amplifier configuration. That is, the input voltages are applied to the non-inverting input terminal and a part of the output is fed back to the inverting input terminal, through voltage-divider-bias feedback.

A non-inverting summing circuit is shown in the figure below.

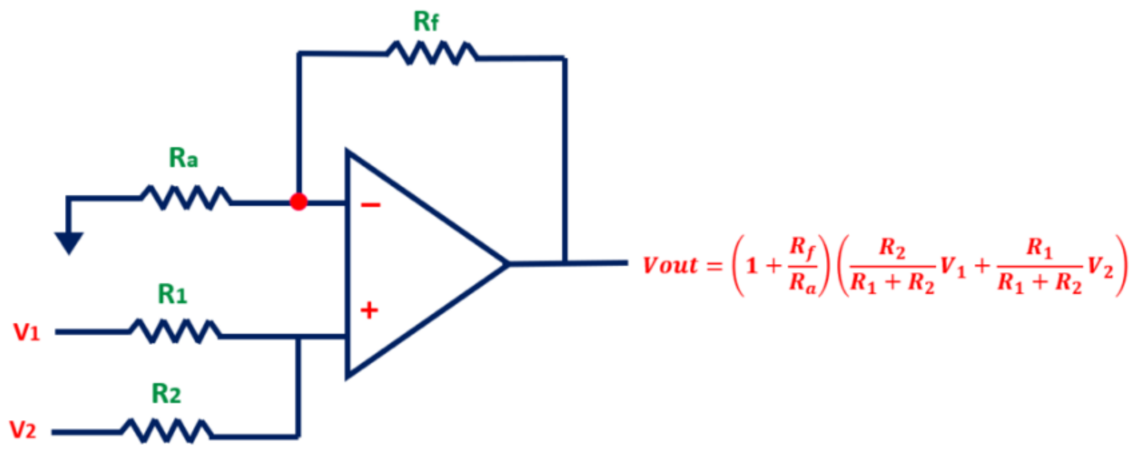


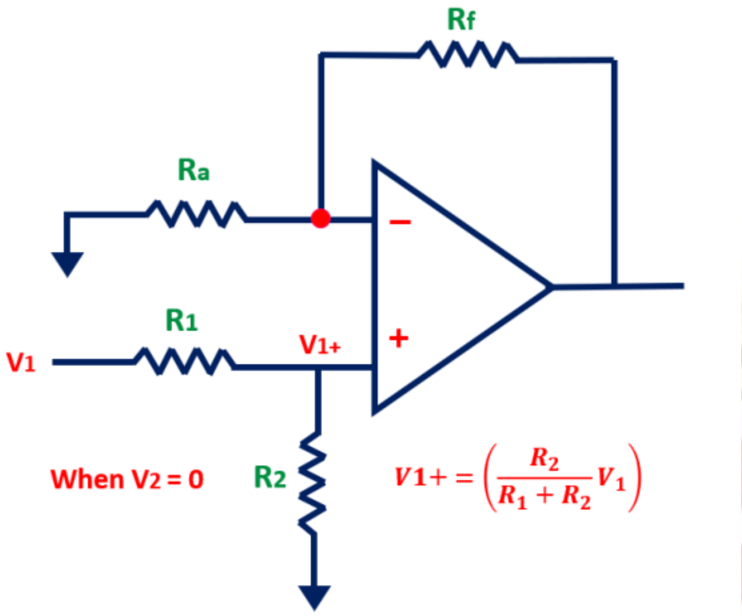
Fig Shows the **non-inverting** summing amplifier. Where the two inputs V1 and V2 are applied to the non-inverting op-amp terminal through resistor R_1 and R_2.

The output of the op-amp can be given as

$$Vout = \left(1 + \frac{R_f}{R_a}\right)\left(\frac{R_2}{R_1 + R_2}V_1 + \frac{R_1}{R_1 + R_2}V_2\right)$$

Proof:

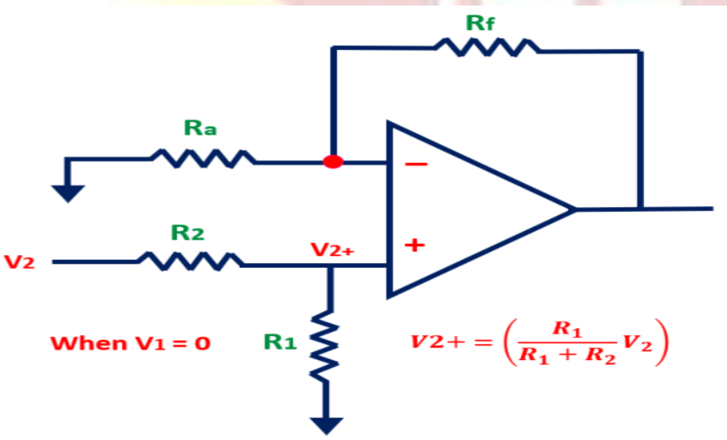
The output can be found by applying the superposition theorem. To find the output, let's consider only one input at a time. If V_1 is acting alone and $V_2 = 0$ (as shown in fig.4), then voltage V_1+ at the non-inverting input is



(When $V_2 = 0$)

$$V_1^+ = \frac{R_2}{R_1 + R_2} V_1$$

Similarly, as shown in Fig.5, when voltage V1 = 0 and V2 is acting alone then voltage V_2+ can be given as



(When $V_1 = 0$)

$$V_2^+ = \frac{R_1}{R_1 + R_2} V_2$$

Considering both inputs simultaneously, the total voltage at the non-inverting input terminal

$$V^{+} = V_1^{+} + V_2^{+}$$

$$V^{+} = \frac{R_2}{R_1 + R_2} V_1 + \frac{R_1}{R_1 + R_2} V_2$$

That means overall output voltage of the op-amp is

$$Vout = \left(1 + \frac{R_f}{R_a}\right) V^{+}$$

$$Vout = \left(1 + \frac{R_f}{R_a}\right)\left(\frac{R_2}{R_1 + R_2} V_1 + \frac{R_1}{R_1 + R_2} V_2\right)$$

When $R_1 = R_2 = R$ then output of the op-amp

$$Vout = \left(1 + \frac{R_f}{R_a}\right)\left(\frac{V_1 + V_2}{2}\right)$$

And when $R_f = R_a$ then output

$$Vout = V_1 + V_2$$

The Differential Amplifier

A difference amplifier or differential amplifier amplifies the difference between the two input signals. An operational amplifier is a difference amplifier; it has an inverting input and a non-inverting input. But the open loop voltage gain of an operational amplifier is too high (ideally infinite), to be used without a feedback connection.So, a practical difference amplifier uses a negative feedback connection to c

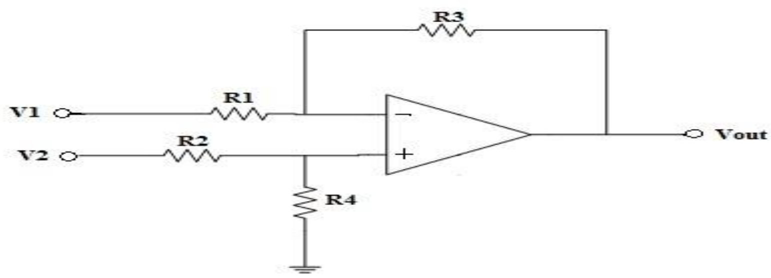


Fig.- Differential Amplifier Circuit

The difference amplifier shown in the above circuit is a combination of both inverting and non-inverting amplifiers. If the non-inverting terminal is connected to ground, the circuit operates as an inverting amplifier and the input signal V_1 is amplified by – (R_3 / R_1).

Similarly, if the inverting input terminal is connected to ground, the circuit behaves as a non-inverting amplifier. With the inverting input terminal grounded, R_3 and R_1 function as the feedback components of a non-inverting amplifier.

Input V_2 is potentially divided across resistors R_2 and R_4 to give V_{R4}, and then V_{R4} is amplified by $(R_3 + R_1) / R_1$.

With $V_2 = 0$,

$V_{O1} = -(R_3 / R_1) * V_1$

With $V_1 = 0$,

$V_{R4} = \{R_4/(R_2+R_4)\}*V_2$

and

$V_{O2} = \{(R_1+R_3)/R_1\}*V_{R4}$

Therefore,

$V_{O2} = \{(R_1 + R_3) / R_1\} * \{R_4 / (R_2 + R_4) \} * V_2$

If the input resistances are chosen such that, $R_2 = R_1$ and $R_4 = R_3$, then

$V_{O2} = \{R3 / R_1\} * V_2$

Now, according to superposition principle if both the input signals V_1 and V_2 are present, then the output voltage is

$V_O = V_{O1} + V_{O2}$

$= \{-(R_3 / R_1) * V_1\} + \{R3 / R_1\} * V_2$

Which results in,

$V_O = (R_3 / R_1) * \{V_2 – V_1\}$

When the resistors R_3 and R_1 are of the same value, the output is the direct difference of the input voltages applied. By selecting R_3 greater than R_1, the output can be made an amplified version of the difference of the input voltages.

The output voltage of a difference amplifier is given as,

$V_O = A_D (V_1 – V_2)$

where, $A_D = – (R_3 / R_1)$ is the differential gain of the amplifier.

Characteristics of a Differential Amplifier

High Differential Voltage Gain

Low Common Mode Gain

High Input Impedance

Low Output Impedance

High CMRR

Large Bandwidth

Low offset voltages and currents

What is a Comparator?

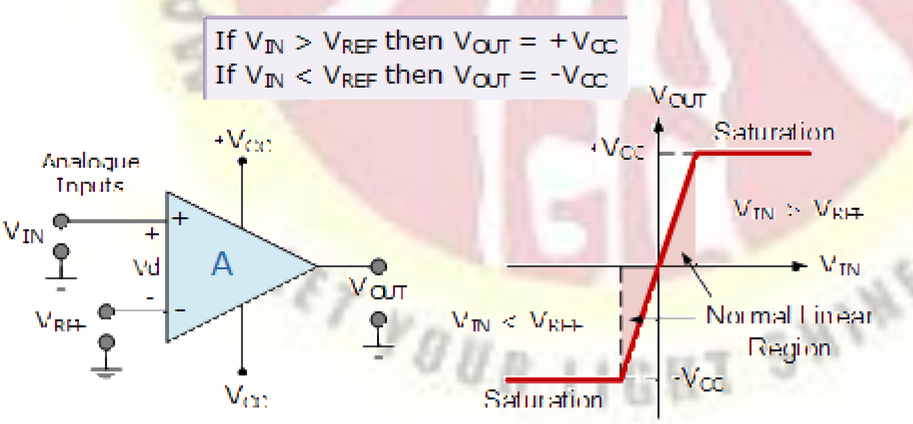
In general, a comparator can be defined as a device that consists of two input terminals, one terminal is fed with the reference input signal and the other input is fed with the actual value of the signal. Then, based on the difference between these two inputs, it generates an output signal as either 0 (low) or 1 (high).

The two analog input terminals are V+ & V- and digital output is V0. This output signal can be defined as

V0 = 1, if V+ > V- and
V0 = 0, if V+ < V-

Comparators are frequently used in the devices that are used to measure analog signals, relaxation oscillators, and analog to digital converters (ADCs). These comparators comprise of high-gain differential amplifier and the comparators can be built using operational amplifiers.

Op-amp Comparator Circuit



With reference to the op-amp comparator circuit above, lets first assume that V_{IN} is less than the DC voltage level at V_{REF}, ($V_{IN} < V_{REF}$). As the non-inverting (positive) input of the comparator is less than the inverting (negative) input, the output will be LOW and at the negative supply voltage, -Vcc resulting in a negative saturation of the output.

If we now increase the input voltage, V_{IN} so that its value is greater than the reference voltage V_{REF} on the inverting input, the output voltage rapidly switches HIGH towards the positive supply voltage, +Vcc resulting in a positive saturation of the output. If we reduce again the input voltage V_{IN}, so that it is slightly less than the reference voltage, the op-amp's output switches back to its negative saturation voltage acting as a threshold detector.

Then we can see that the op-amp voltage comparator is a device whose output is dependent on the value of the input voltage, V_{IN} with respect to some DC voltage level as the output is HIGH when the voltage on the non-inverting input is greater than the voltage on the inverting input, and LOW when the non-inverting input is less than the inverting input voltage. This condition is true regardless of whether the input signal is connected to the inverting or the non-inverting input of the comparator.

Operational Amplifier as Integrator

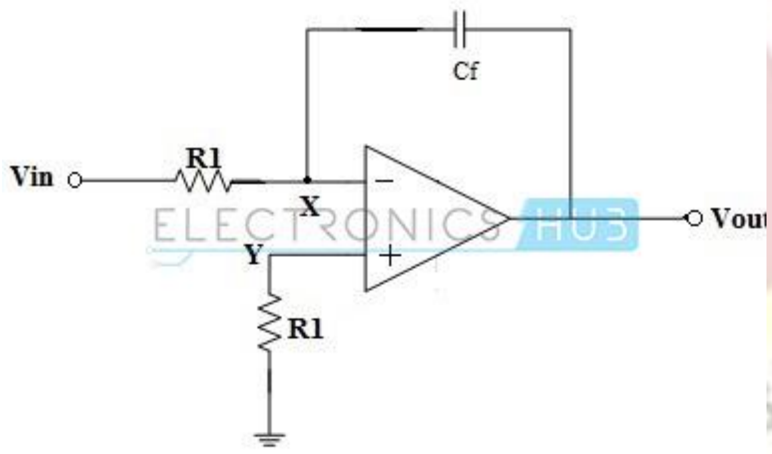
Another major application of Op-amp is its use in mathematical applications. An Operational Amplifier can be configured to perform the mathematical operations of Integration and Differentiation. In this tutorial, we will learn and analyze the working of an Operational Amplifier as Integrator

Operational amplifier can be configured to perform calculus operations such as differentiation and integration. In an integrating circuit, the output is the integration of the input voltage with respect to time..

Ideal Op-amp Integrator Circuit

An op-amp integrating circuit produces an output voltage which is proportional to the area (amplitude multiplied by time) contained under the waveform.

An ideal op-amp integrator uses a capacitor C1, connected between the output and the op-amp inverting input terminal, as shown in the figure below.



An Ideal Op-amp Integrator

The negative feedback to the inverting input terminal ensures that the node X is held at ground potential (virtual ground). If the input voltage is 0 V, there will be no current through the input resistor R1, and the capacitor is uncharged.

Hence, the output voltage is ideally zero.

If a constant positive voltage (DC) is applied to the input of the integrating amplifier, the output voltage will fall negative at a linear rate, in an attempt to keep the inverting input terminal at ground potential.

Conversely, a constant negative voltage at the input results in a linearly rising (positive) voltage at the output. The rate of change of the output voltage is proportional to the value of the applied input voltage.

Output Voltage Calculation

From the circuit, it is seen that node Y is grounded through a compensating resistor R1. Node X will also be at ground potential, due to the virtual ground.

$V_X = V_Y = 0$

Since the input current to an op-amp is ideally zero, the current flowing through the input resistor, due to Vin, also flows through the capacitor Cf.

From the input side, the current I is given as,

$I = (V_{in} – V_X) / R_1 = V_{in} / R_1$

From the output side, the current I is given as,

$I = C_f [d(V_X – V_{out})/dt] = -C_f [d(V_{out})/dt]$

Equating the above two equations of I, we get,

$[V_{in} / R_1] = – C_f [d(V_{out})/dt]$

Integrating both the sides of the above equation,

$$\int_0^t \frac{Vin}{R1} \, . dt = - C_f \int_0^t d \frac{Vout}{dt} \; dt$$

$$\int_0^t \frac{Vin}{R1} \, . dt = -C_f . V_{out}$$

$$\text{Therefore, } V_{out} = - \frac{1}{R1.Cf} \int_0^t Vin \, . dt$$

In the above equation, the output is $-\{1/(R_1.C_f)\}$ times the integral of the input voltage, where the term $(R_1.C_f)$ is known as the time constant of the integrator.

The negative sign indicates that there is a phase shift of 180^o between input and output, because the input is provided to the inverting input terminal of the op-amp.

The main advantage of an active integrator is the large time constant, which results in the accurate integration of the input signal.

.

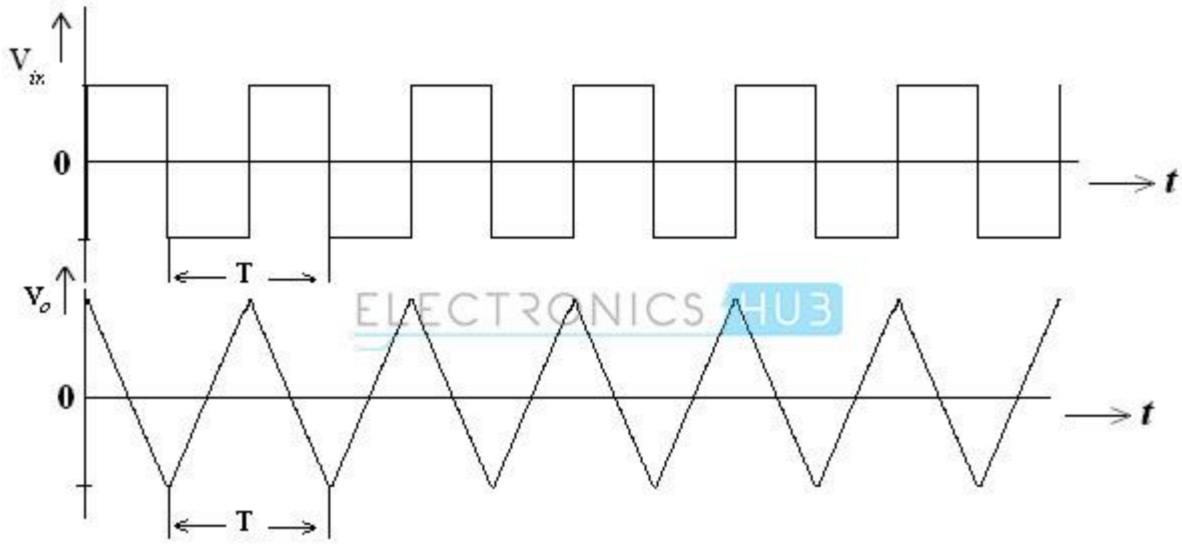


Fig: Input and Output Waveforms of an Integrator

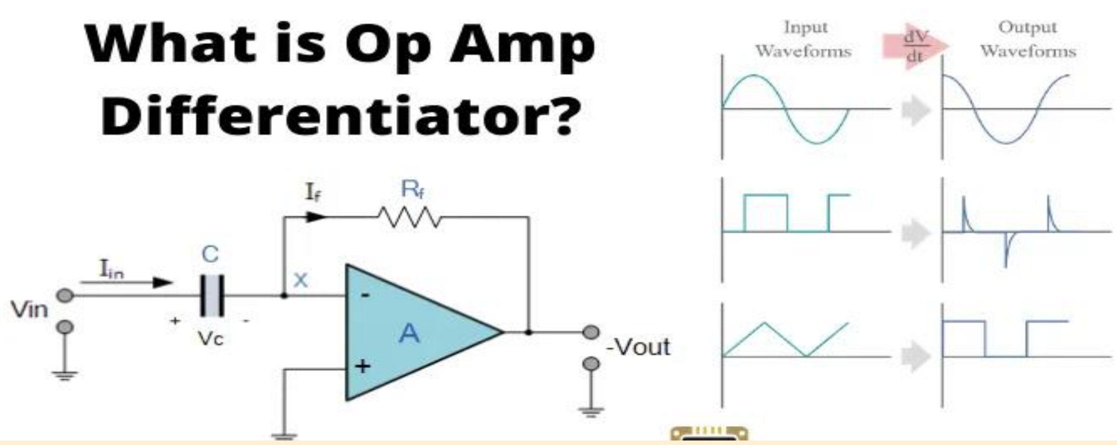
Op-amp Integrator Applications

Op-amp integrating amplifiers are used to perform calculus operations in analogue computers.

Integrating circuits are most commonly used in analogue-to-digital converters, ramp generators and also in wave shaping applications.

Another application would be to integrate a signal representing water flow, producing a signal representing the total quantity of water that has passed by the flow meter. This application of an integrator is sometimes called a totalizer in the industrial instrumentation trade.

Op Amp Differentiator



Differentiator is an op amp based circuit, whose output signal is proportional to differentiation of input signal.

we will first assume that the op amp used here is an ideal op amp. We know that the voltage at both inverting and non-inverting terminals of an ideal op amp is same. As the electric potential at non inverting terminal is zero since it is grounded. The electric potential of inverting terminal is also zero, as the opamp is ideal. It is also known to us that the current entering through inverting and non-inverting terminal of an ideal op amp is zero

n the above circuit, the op-amp node voltage at inverting terminal is zero then the flow of current through **the capacitor** can be written.

$\mathbf{I_{in} = I_f}$

Where $\mathbf{I_f = -V_{out}/R_f}$

The capacitor charge equals the voltage with capacitance times across the capacitor

$\mathbf{Q = C \times V_{in}}$

Therefore the charge rate change is

$\mathbf{dQ/dt = C\, dV_{in}/dt}$

But the dQ/dt is the current through the capacitor

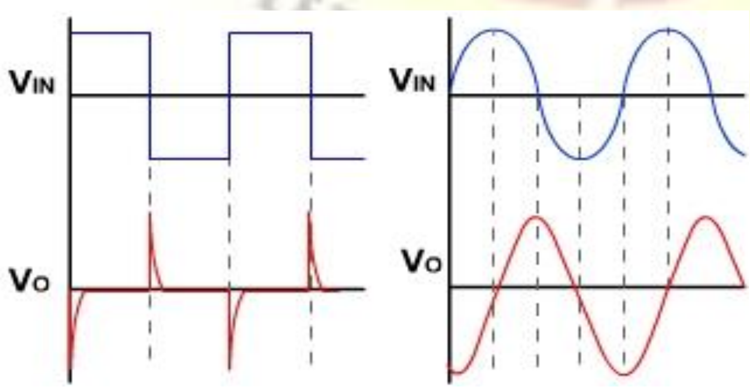
$\mathbf{I_{in} = C\, dV_{in}/dt = I_f}$

$\mathbf{-V_{out}/R_f = C\, dV_{in}/dt}$

An ideal output voltage (Vout) for the operational amplifier differentiator is written as

$\mathbf{Vout = -R_f C\, dV_{in}/dt}$

Thus, the output voltage is a constant input voltage derivative – $R_f C$ times of the input Vin voltage with respect to time. Here sign minus (–) specifies the phase shift (180°) as the input signal is given to the input inverting terminal of the op-amp.



ACTIVE FILTERS

- Filters are circuits that are capable of *passing signals within a band* of frequencies while *rejecting or blocking* signals of frequencies *outside this band*. This property of filters is also called "frequency selectivity".
- Filter can be passive or active filter.

Passive filters: The circuits built using RC, RL, or RLC circuits.

Active filters : The circuits that employ one or more op-amps in the design an addition to resistors and capacitors

TYPES OF FILTERS

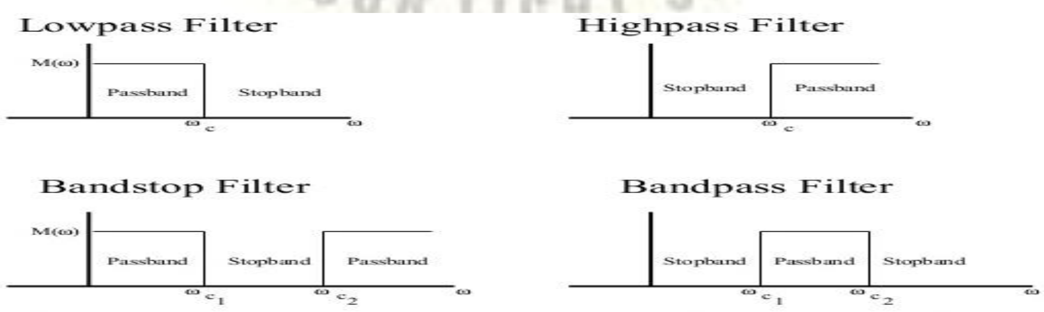
There are two broad categories of filters:

- An *analog filter* processes continuous-time signals
- A *digital filter* processes discrete-time signals.

The analog or digital filters can be subdivided into four categories:

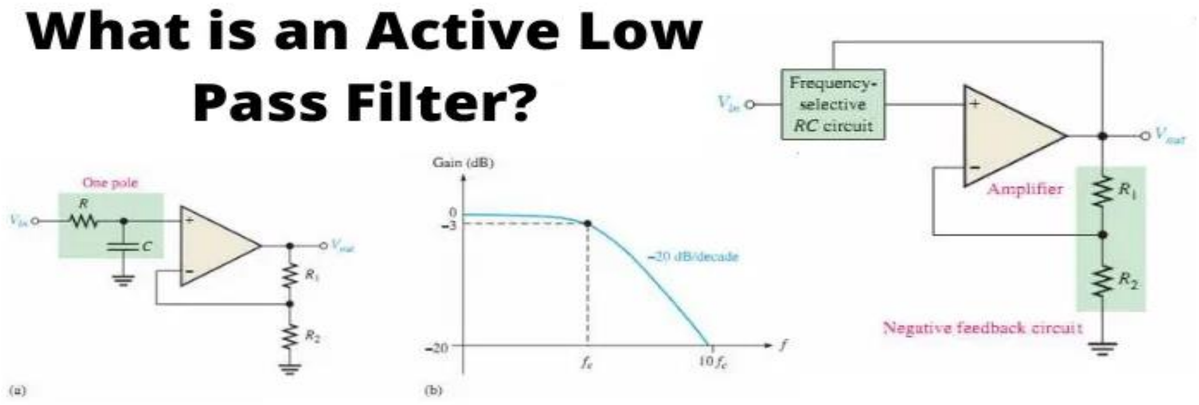
- Low pass Filters
- High pass Filters
- Band stop Filters
- Band pass Filters

IDEAL FILTER



APPLICATION

- Active filters are mainly used in communication and signal processing circuits.
- They are also employed in a wide range of applications such as entertainment, medical electronics, etc.



What is an Active Low Pass Filter?

If an active filter permits only low-frequency components and denies all other high-frequency components, then it is termed as an **Active Low Pass Filter**. Active low pass filters are made up of Op-Amp. The input to the Op-Amp is high impedance signals, which produces a low impedance signal as output.

The performance of the amplifier plays a very important factor when designing an active low pass filter. There are two primary kinds of active low-pass filters namely switched **capacitor** type and continuous capacitor type. The filters are available from first-order until the eighth order of design.

The passband begins from 0Hz or DC for a low pass filter and continues at -3dB to the designated cut-off point. Signals are attenuated beyond the cut off frequency. Active low pass filters are grouped according to the order of the filter..

The inverse of a low pass filter is a high pass filter, that permits signals with frequencies higher than the cut-off frequency and blocks all frequencies below this cut-off frequency. There are also bandpass filters, which combine the functionality of low pass filters and high pass filters to only allow frequencies within a specific frequency range.

Gain of a first-order low pass filter

$$\text{Voltage Gain, } (Av) = \frac{Vout}{Vin} = \frac{A_F}{\sqrt{1 + \left(\frac{f}{fc}\right)^2}}$$

Where:

AF = the pass band gain of the filter, (1 + R2/R1)

ƒ = the frequency of the input signal in Hertz, (Hz)

*ƒ*c = the cut-off frequency in Hertz, (Hz)

Thus, the operation of a low pass active filter can be verified from the frequency gain equation above as:

1. At very low frequencies, *ƒ* < *ƒ*c $\frac{Vout}{Vin} \cong A_F$

2. At the cut-off frequency, *ƒ* = *ƒ*c $\frac{Vout}{Vin} = \frac{A_F}{\sqrt{2}} = 0.707\,A_F$

3. At very high frequencies, *ƒ* > *ƒ*c $\frac{Vout}{Vin} < A_F$

Thus, the **Active Low Pass Filter** has a constant gain A_F from 0Hz to the high frequency cut-off point, f_C. At f_C the gain is $0.707A_F$, and after f_C it decreases at a constant rate as the frequency increases. That is, when the frequency is increased tenfold (one decade), the voltage gain is divided by 10.

In other words, the gain decreases 20dB (= 20*log(10)) each time the frequency is increased by 10. When dealing with filter circuits the magnitude of the pass band gain of the circuit is generally expressed in *decibels* or *dB* as a function of the voltage gain, and this is defined as:

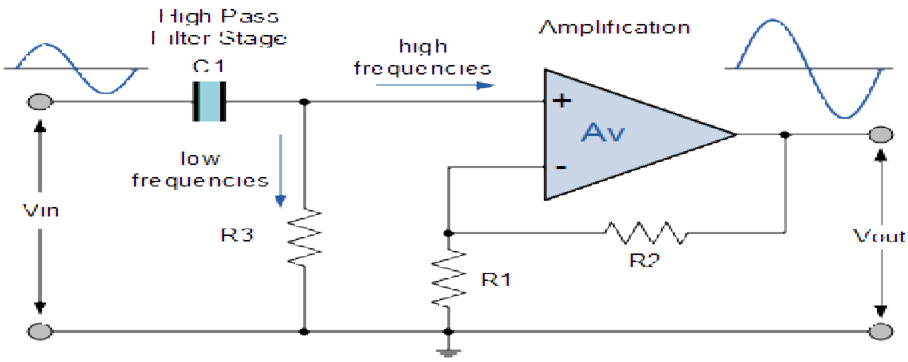
Magnitude of Voltage Gain in (dB)

$$Av(dB) = 20\log_{10}\left(\frac{Vout}{Vin}\right)$$

$$\therefore\ -3dB = 20\log_{10}\left(0.707\frac{Vout}{Vin}\right)$$

Active High Pass Filter

An Active High Pass Filter can be created by combining a passive RC filter network with an operational amplifier to produce a high pass filter with amplification



first-order (single-pole) **Active High Pass Filter** as its name implies, attenuates low frequencies and passes high frequency signals. It consists simply of a passive filter section followed by a non-inverting operational amplifier. The frequency response of the circuit is the same as that of the passive filter, except that the amplitude of the signal is increased by the gain of the amplifier and for a non-inverting amplifier the value of the pass band voltage gain is given as 1 + R2/R1, the same as for the low pass filter circuit.

For a non-inverting amplifier circuit, the magnitude of the voltage gain for the filter is given as a function of the feedback resistor (R2) divided by its corresponding input resistor (R1) value and is given as:

Gain for an Active High Pass Filter

$$\text{Voltage Gain, } (Av) = \frac{Vout}{Vin} = \frac{A_F\left(\frac{f}{fc}\right)}{\sqrt{1+\left(\frac{f}{fc}\right)^2}}$$

Where:

A_F = the Pass band Gain of the filter, (1 + R2/R1)

f = the Frequency of the Input Signal in Hertz, (Hz)

fc = the Cut-off Frequency in Hertz, (Hz)

Just like the low pass filter, the operation of a high pass active filter can be verified from the frequency gain equation above as:

1. At very low frequencies, $f < fc$ $\quad \frac{Vout}{Vin} < A_F$

2. At the cut-off frequency, $f = fc$ $\quad \frac{Vout}{Vin} = \frac{A_F}{\sqrt{2}} = 0.707\,A_F$

3. At very high frequencies, $f > fc$ $\quad \frac{Vout}{Vin} = A_F$

Then, the **Active High Pass Filter** has a gain A_F that increases from 0Hz to the low frequency cut-off point, fc at 20dB/decade as the frequency increases. At fc the gain is $0.707*A_F$, and after fc all frequencies are pass band frequencies so the filter has a constant gain A_F with the highest frequency being determined by the closed loop bandwidth of the op-amp.

When dealing with filter circuits the magnitude of the pass band gain of the circuit is generally expressed in *decibels* or *dB* as a function of the voltage gain, and this is defined as:

Magnitude of Voltage Gain in (dB)

$$Av(dB) = 20\log_{10}\left(\frac{Vout}{Vin}\right)$$

$$\therefore\ -3dB = 20\log_{10}\left[0.707\frac{Vout}{Vin}\right]$$

For a first-order filter the frequency response curve of the filter increases by 20dB/decade or 6dB/octave up to the determined cut-off frequency point which is always at -3dB below the maximum gain value. As with the previous filter circuits, the lower cut-off or corner frequency (fc) can be found by using the same formula:

$$fc = \frac{1}{2\pi RC}\ Hz$$

The corresponding phase angle or phase shift of the output signal is the same as that given for the passive RC filter and leads that of the input signal. It is equal to +45° at the cut-off frequency fc value and is given as:

$$\text{Phase Shift } \varphi = \tan^{-1}\left(\frac{1}{2\pi fRC}\right)$$

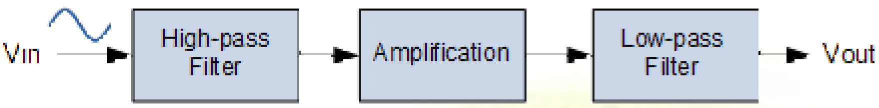
A simple first-order active high pass filter can also be made using an inverting operational amplifier configuration as well, and an example of this circuit design is given along with its corresponding frequency response curve. A gain of 40dB has been assumed for the circuit.

Inverting Operational Amplifier Circuit

Active Band Pass Filter

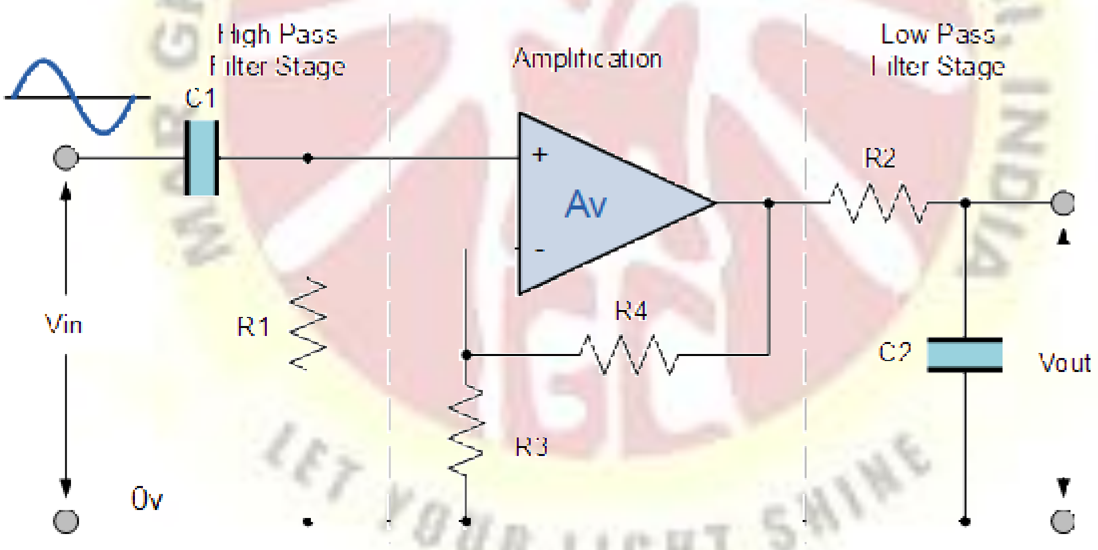
The principal characteristic of a **Band Pass Filter** or any filter for that matter, is its ability to pass frequencies relatively unattenuated over a specified band or spread of frequencies called the "Pass Band".

Simple Active Band Pass Filter can be easily made by cascading together a single Low Pass Filter with a single High Pass Filter as shown.

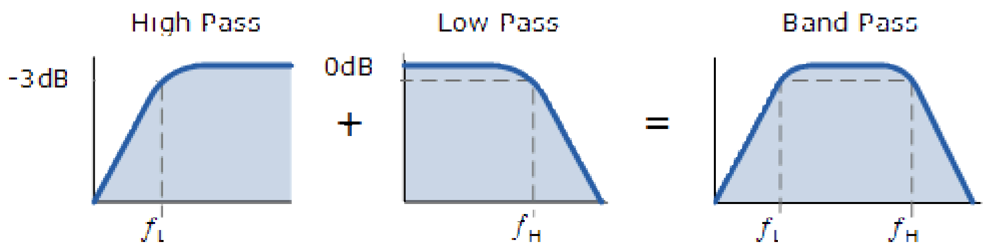


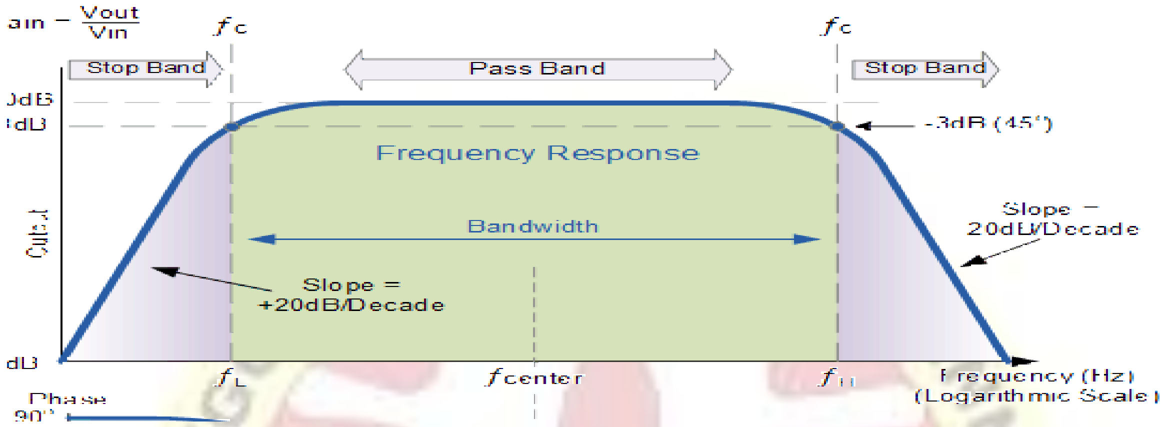
he cut-off or corner frequency of the low pass filter (LPF) is higher than the cut-off frequency of the high pass filter (HPF) and the difference between the frequencies at the -3dB point will determine the "bandwidth" of the band pass filter while attenuating any signals outside of these points. One way of making a very simple **Active Band Pass Filter** is to connect the basic passive high and low pass filters we look at previously to an amplifying op-amp circuit as shown.

Active Band Pass Filter Circuit



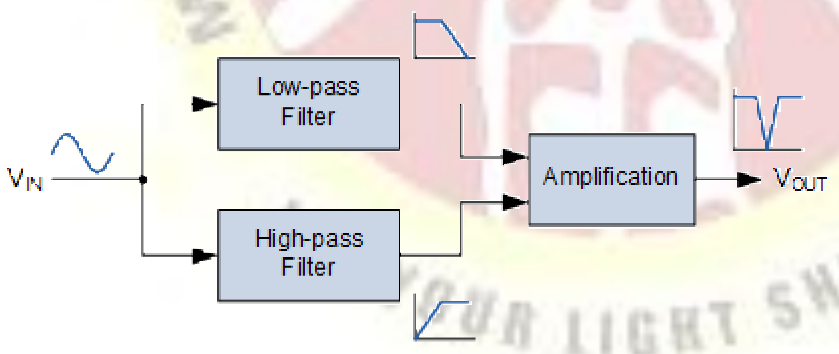
This cascading together of the individual low and high pass passive filters produces a low "Q-factor" type filter circuit which has a wide pass band. The first stage of the filter will be the high pass stage that uses the capacitor to block any DC biasing from the source. This design has the advantage of producing a relatively flat asymmetrical pass band frequency response with one half representing the low pass response and the other half representing high pass response as shown.

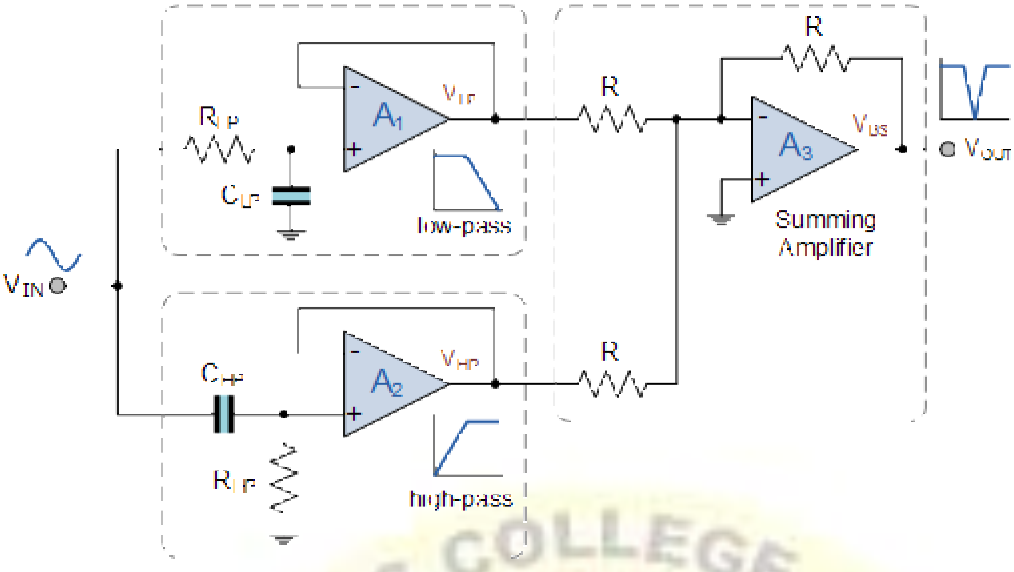




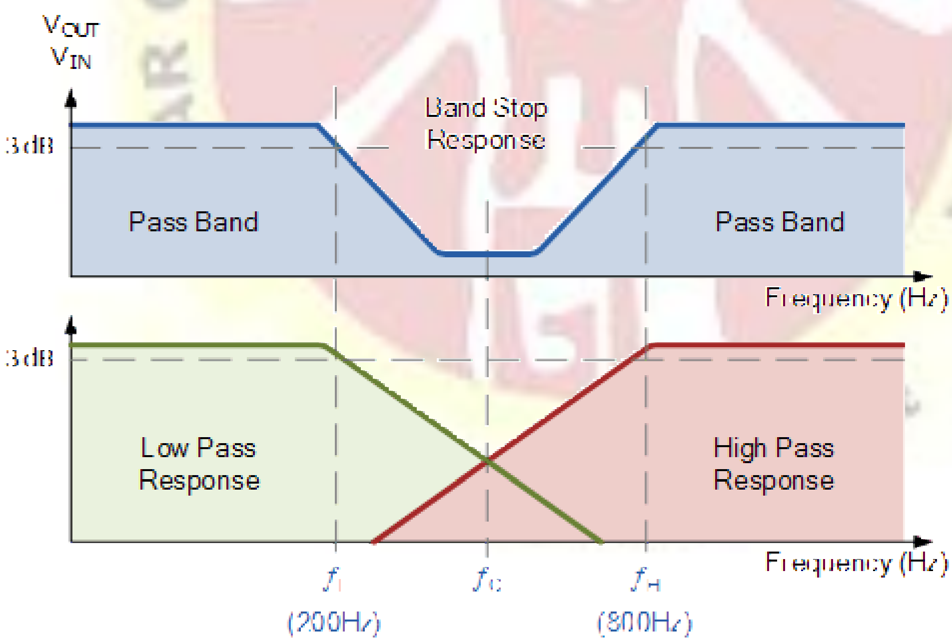
Generally band-pass filters are constructed by combining a low pass filter (LPF) in series with a high pass filter (HPF). Band stop filters are created by combining together the low pass and high pass filter sections in a “parallel” type configuration as shown.

Typical Band Stop Filter Configuration





Band Stop Filter Characteristics



.

INTRODUCTION 555 TIMER

The 555 timer IC was introduced in the year 1970 by Signetic Corporation and gave the name **SE/NE 555 timer**. It is basically a monolithic timing circuit that produces accurate and highly stable time delays or oscillation. When compared to the applications of an op-amp in the same areas, the 555IC is also equally reliable and is cheap in cost. Apart from its applications as a **monostable multivibrator** and **astable multivibrator**, a 555 timer can also be used in **dc-dc converters**, digital logic probes, **waveform generators**, analog frequency meters and tachometers, **temperature measurement** and control devices, **voltage regulators** etc. The timer IC is set up to work in either of the two modes – one-shot or monostable or as a free-running or astable multivibrator. The **SE 555** can be used for temperature ranges between – 55°C to 125 °. The **NE 555** can be used for a temperature range between 0° to 70°C.

The important features of the 555 timer are :

It operates from a wide range of **power supplies** ranging from + 5 Volts to + 18 Volts supply voltage.

Sinking or sourcing 200 mA of load current.

The external components should be selected properly so that the timing intervals can be made into several minutes along with the frequencies exceeding several hundred kilohertz.

The output of a 555 timer can drive a transistor-transistor logic (TTL) due to its high current output.

It has a temperature stability of 50 parts per million (ppm) per degree Celsius change in temperature, or equivalently 0.005 %/ °C.

The duty cycle of the timer is adjustable.

The maximum power dissipation per package is 600 mW and its trigger and reset inputs has logic compatibility. More features are listed in the datasheet.

2. IC Pin Configuration

PIN FUNCTION

Pin 1: **Grounded Terminal:** All the voltages are measured with respect to the Ground terminal.

Pin 2: Trigger Terminal: The trigger pin is used to feed the trigger input hen the 555 IC is set up as a monostable multivibrator. This pin is an inverting input of a comparator and is responsible for the transition of **flip-flop** from set to reset. The output of the timer depends on the amplitude of the external trigger pulse applied to this pin. A negative pulse with a dc level greater than Vcc/3 is applied to this terminal. In the negative edge, as the trigger passes through Vcc/3, the output of the lower comparator becomes high and the complimentary of Q becomes zero. Thus the 555 IC output gets a high voltage, and thus a quasi-stable state.

Pin 3: Output Terminal: Output of the timer is available at this pin. There are two ways in which a load can be connected to the output terminal. One way is to connect between output pin (pin 3) and ground pin (pin 1) or between pin 3 and supply pin (pin 8). The load connected between output and ground supply pin is called the ***normally on load*** and that connected between output and ground pin is called the ***normally off load***.

Pin 4: Reset Terminal: Whenever the timer IC is to be reset or disabled, a negative pulse is applied to pin 4, and thus is named as reset terminal. The output is reset irrespective of the input condition. When this pin is not to be used for reset purpose, it should be connected to + V_{CC} to avoid any possibility of false triggering.

Pin 5: Control Voltage Terminal: The threshold and trigger levels are controlled using this pin. The pulse width of the output waveform is determined by connecting a POT or bringing in an external voltage to this pin. The external voltage applied to this pin can also be used to modulate the output waveform. Thus, the amount of voltage applied in this terminal will decide when the comparator is to be switched, and thus changes the pulse width of the output. When this pin is not used, it should be bypassed to ground through a 0.01 micro-Farad to avoid any noise problem.

Pin 6: **Threshold Terminal:** This is the non-inverting input terminal of comparator 1, which compares the voltage applied to the terminal with a reference voltage of 2/3 V_{CC}. The amplitude of voltage applied to this terminal is responsible for the set state of flip-flop. When the voltage applied in this terminal is greater than 2/3Vcc, the upper comparator switches to +Vsat and the output gets reset.

Pin 7 : **Discharge Terminal:** This pin is connected internally to the collector of transistor and mostly a capacitor is connected between this terminal and ground. It is called discharge terminal because when transistor saturates, capacitor discharges through the transistor. When the transistor is cut-off, the capacitor charges at a rate determined by the external resistor and capacitor.

Pin 8: Supply Terminal: A supply voltage of + 5 V to + 18 V is applied to this terminal with respect to ground (pin 1).

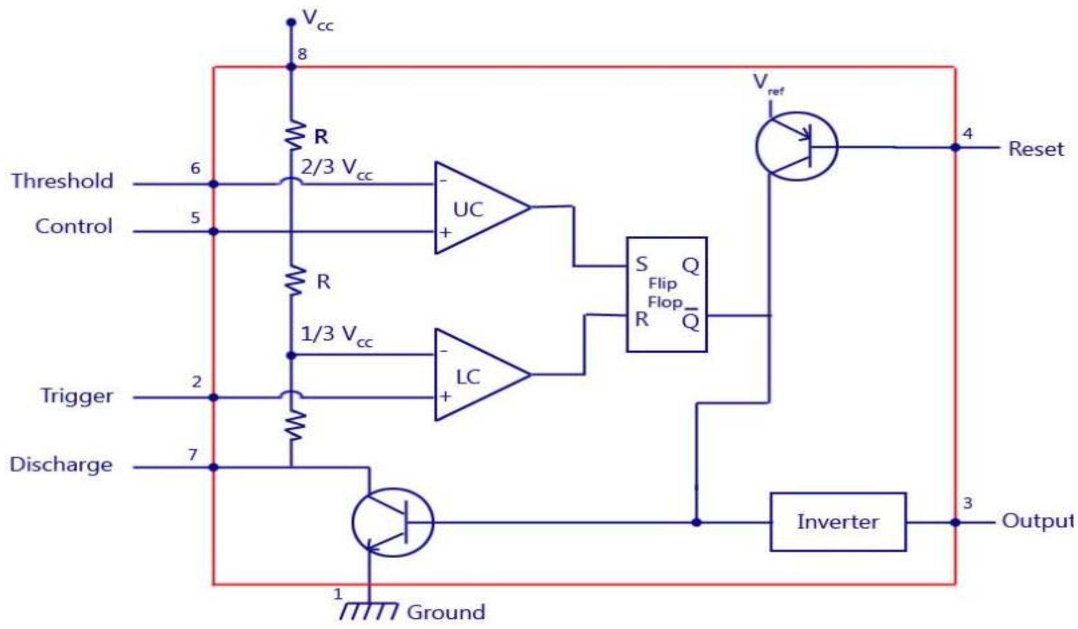
555 IC Timer Block Diagram

The block diagram of a 555 timer is shown in the above figure. A 555 timer has two comparators, which are basically 2 op-amps), an R-S flip-flop, two transistors and a resistive network.

Resistive network consists of three equal resistors and acts as a voltage divider.

Comparator 1 compares threshold voltage with a reference voltage + 2/3 V_{CC} volts.

Comparator 2 compares the trigger voltage with a reference voltage + 1/3 V_{CC} volts. Output of both the comparators is supplied to the flip-flop. Flip-flop assumes its state according to the output of the two comparators. One of the two transistors is a discharge transistor of which collector is connected to **pin** 7. This transistor saturates or cuts-off according to the output state of the flip-flop. The saturated transistor provides a discharge path to a capacitor connected externally. Base of another transistor is connected to a reset terminal. A pulse applied to this terminal resets the whole timer irrespective of any input.



Working Principle

Refer Block Diagram of 555 timer IC given above:

The internal resistors act as a voltage divider network, providing (2/3)Vcc at the non-inverting terminal of the upper comparator and (1/3)Vcc at the inverting terminal of the lower comparator. In most applications, the control input is not used, so that the control voltage equals +(2/3) V_{CC}. Upper comparator has a threshold input (pin 6) and a control input (pin 5). Output of the upper comparator is applied to set (S) input of the flip-flop. Whenever the threshold voltage exceeds the control voltage, the upper comparator will set the flip-flop and its output is high. *A* high output from the flip-flop when given to the base of the discharge transistor saturates it and thus discharges the transistor that is connected externally to the discharge pin 7. The complementary signal out of the flip-flop goes to pin 3, the output. The output available at pin 3 is low. These conditions will prevail until lower comparator triggers the flip-flop. Even if the voltage at the threshold input falls below (2/3) V_{CC}, that is upper comparator cannot cause the flip-flop to change again. It means that the upper comparator can only force the flip-flop's output high.

To change the output of flip-flop to low, the voltage at the trigger input must fall below +(1/3) Vcc. When this occurs, lower comparator triggers the flip-flop, forcing its output low. The low output from the flip-flop turns the discharge transistor off and forces the power amplifier to output a high. These conditions will continue independent of the voltage on the trigger input. Lower comparator can only cause the flip-flop to output low.

From the above discussion, it is concluded that for the having low output from the timer 555, the voltage on the threshold input must exceed the control voltage or + (2/3) V_{CC}. This also turns the discharge transistor on. To force the output from the timer high, the voltage on the trigger input must drop below +(1/3) V_{CC}. This turns the discharge transistor off.

A voltage may be applied to the control input to change the levels at which the switching occurs. When not in use, a 0.01 nano Farad capacitor should be connected between pin 5 and ground to prevent noise coupled onto this pin from causing false triggering.

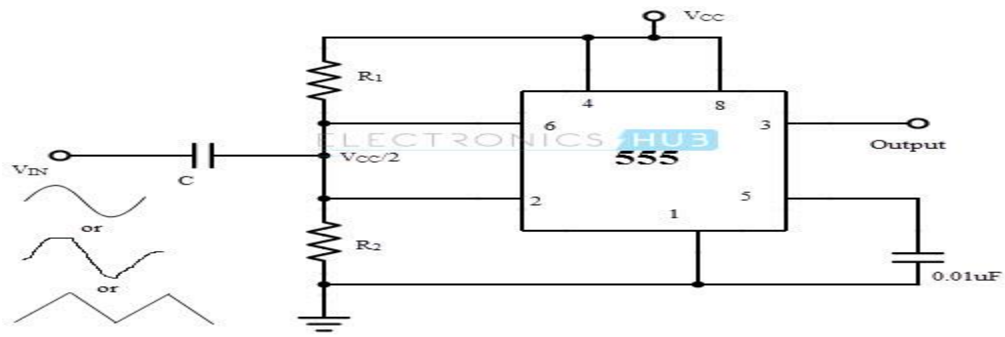
Connecting the reset (pin 4) to a logic low will place a high on the output of flip-flop. The discharge transistor will go on and the power amplifier will output a low. This condition will continue until reset is taken high. This allows the synchronization or resetting of the circuit's operation. When not in use, reset should be tied to +V_{CC}.

Schmitt trigger

It is an electronic circuit, which acts as a comparator ,that is used to detect a given reference level.It has two stable states logic 1,logic 0.Whenever the input is sine,triangle,or any periodic waveform the output of the schmitt trigger come out to be a rectangular or square wave form which has sharp edges.The advantage of the schmitt trigger is the desirable rise and fall times for all digital circuits

Circuit of 555 timer as Schmitt Trigger

The following circuit shows the structure of a 555 timer used as a Schmitt trigger.



Pins 4 and 8 are connected to the supply (VCC). The pins 2 and 6 are tied together and the input is given to this common point through a capacitor C. this common point is supplied with an

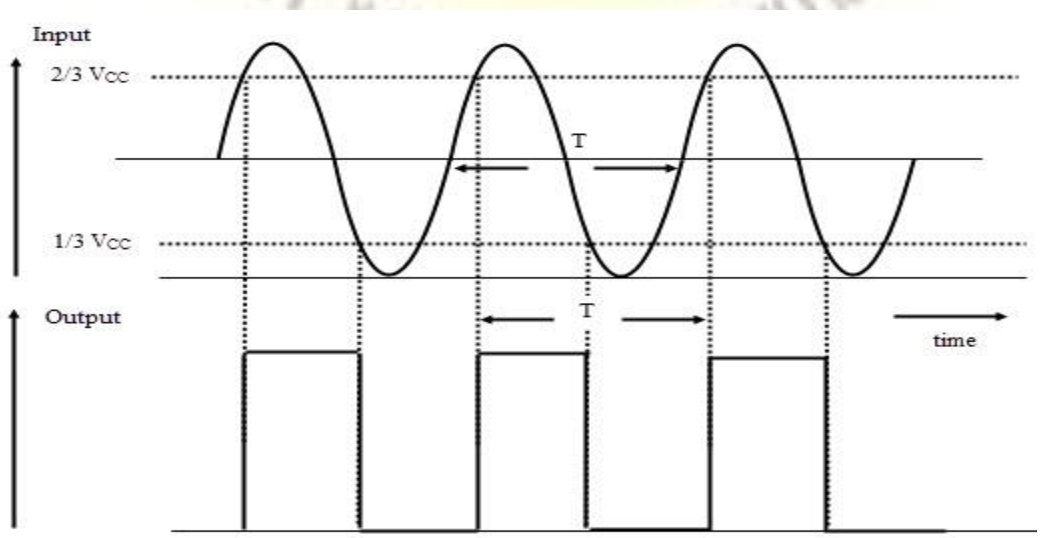
external bias voltage of VCC / 2 with the help of the voltage divider circuit formed by the resistors R1 and R2.

The important characteristic of the Schmitt trigger is Hysteresis. The output of the Schmitt trigger is high if the input voltage is greater than the upper threshold value and the output of the Schmitt trigger is low if the input voltage is lower than the lower threshold value.

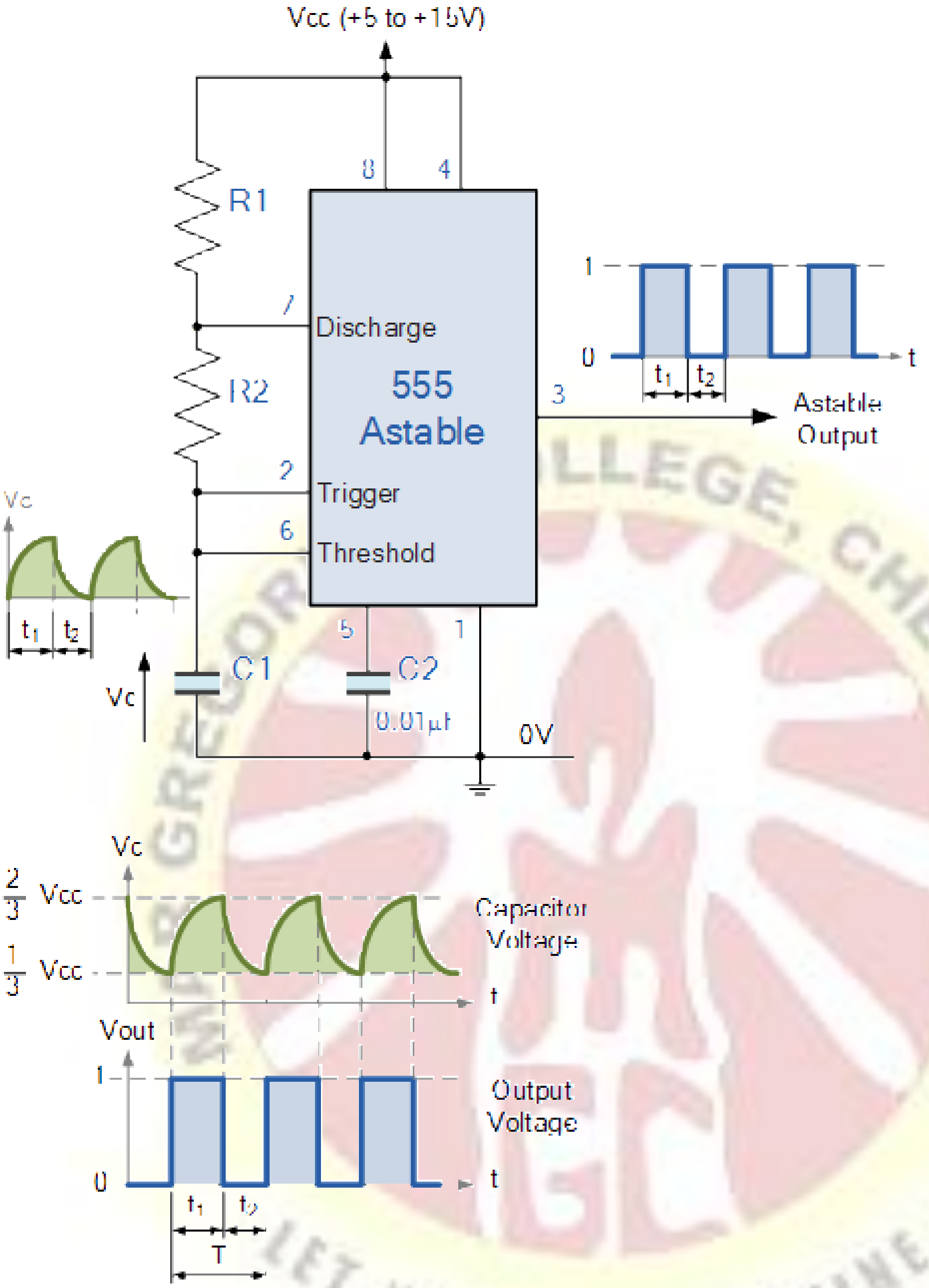
The output retains its value when the input is between the two threshold values. The usage of two threshold values is called Hysteresis and the Schmitt trigger acts as a memory element (a bistable multivibrator or a flip-flop).

The threshold values in this case are 2/3 VCC and 1/3 VCC i.e. the upper comparator trips at 2/3 VCC and the lower comparator trips at 1/3 VCC. The input voltage is compared to these threshold values by the individual comparators and the flip-flop is SET or RESET accordingly. Based on this the output becomes high or low.

When a sine wave of amplitude greater than VCC / 6 is applied at the input, the flip-flop is set and reset alternately for the positive cycle and the negative cycle. The output is a square wave and the waveforms for input sine wave and output square wave are shown below.



Basic Astable 555 Oscillator Circuit



In the **555 Oscillator** circuit above, pin 2 and pin 6 are connected together allowing the circuit to re-trigger itself on each and every cycle allowing it to operate as a free running oscillator. During each cycle capacitor, C charges up through both timing resistors, R1 and R2 but discharges itself only through resistor, R2 as the other side of R2 is connected to the *discharge* terminal, pin 7.

Then the capacitor charges up to 2/3Vcc (the upper comparator limit) which is determined by the 0.693(R1+R2)C combination and discharges itself down to 1/3Vcc (the lower comparator limit) determined by the 0.693(R2*C) combination. This results in an output waveform whose voltage level is approximately equal to Vcc – 1.5V and whose output "ON" and "OFF" time periods are determined by the capacitor and resistors combinations. The individual times required to complete one charge and discharge cycle of the output is therefore given as:

Astable Multivibrator using 555 – Design Theory

The time during which the capacitor C charges from 1/**3 VCC to 2/3 VCC** is equal to the time the output is high and is given as **tc or THIGH = 0.693 (RA + RB) C**, which is proved below.

Voltage across the capacitor at any instant during charging period is given as **vc=VCC(1-et/RC)**

The time taken by the capacitor to charge from 0 to +1/3 V_{CC}

1/3 VCC = VCC (1-et/RC)

The time taken by the capacitor to charge from 0 to +2/3 V_{CC}

or **t2 = RC loge 3 = 1.0986 RC**

So the time taken by the capacitor to charge from +1/3 V_{CC} to +2/3 V_{CC}

tc = (t2 – t1) = (10986 – 0.405) RC = 0.693 RC

Substituting R = (R_A + R_B) in above equation we have

T HIGH = tc = 0.693 (RA + RB) C

where R_A and R_B are in ohms and C is in farads.

The time during which the capacitor discharges from +2/3 V_{CC} to +1/3 V_{CC} is equal tothe time the output is low and is given as **td or TL0W = 0.693 RB C** where R_B is in ohms and C is in farads The above equation is worked out as follows: Voltage across the capacitor at any instant during discharging period is given as**vc = 2/3 VCC e– td/ RBC**

Substituting v_c = 1/3 V_{CC} and t = t_d in above equation we have

+1/3 VCC = +2/3 VCC e– td/ RBCOr td = 0.693 RBC

Overall period of oscillations, **T = THIGH + TLOW = 0.693 (RA+ 2RB) C** , The frequency of oscillations being the reciprocal of the overall period of oscillations T is given as**f = 1/T = 1.44/ (RA+ 2RB)C**

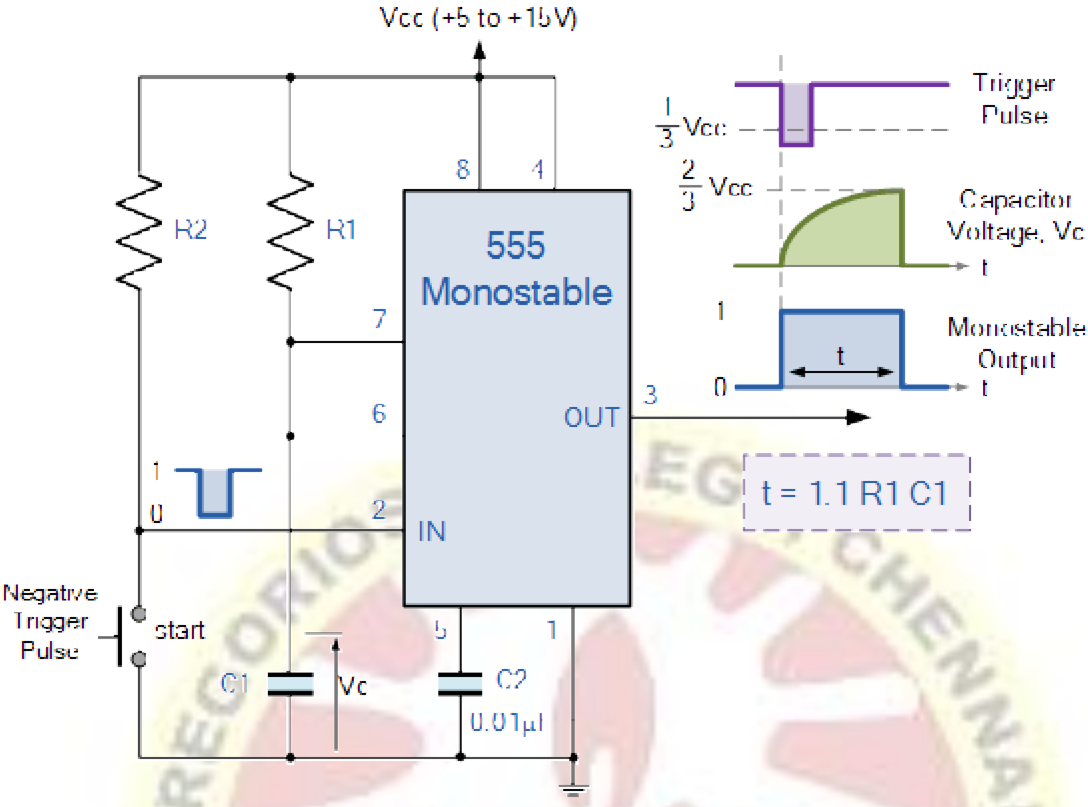
Equation indicates that the frequency of oscillation / is independent of the collector supply voltage +V_{CC}.

Often the term duty cycle is used in conjunction with the astable multivibrator.

The duty cycle, the ratio of the time tc during which the output is high to the total time period T is given as

***%* duty cycle, D = tc / T * 100 = (RA + RB) / (RA + 2RB) * 100**

Monostable 555 Timer



When a negative (0V) pulse is applied to the trigger input (pin 2) of the Monostable configured 555 Timer oscillator, the internal comparator, (comparator No1) detects this input and "sets" the state of the flip-flop, changing the output from a "LOW" state to a "HIGH" state. This action in turn turns "OFF" the discharge transistor connected to pin 7, thereby removing the short circuit across the external timing capacitor, C1.

This action allows the timing capacitor to start to charge up through resistor, R1 until the voltage across the capacitor reaches the threshold (pin 6) voltage of 2/3Vcc set up by the internal voltage divider network. At this point the comparators output goes "HIGH" and "resets" the flip-flop back to its original state which in turn turns "ON" the transistor and discharges the capacitor to ground through pin 7. This causes the output to change its state back to the original stable "LOW" value awaiting another trigger pulse to start the timing process over again. Then as before, the Monostable Multivibrator has only "ONE" stable state.

The **Monostable 555 Timer** circuit triggers on a negative-going pulse applied to pin 2 and this trigger pulse must be much shorter than the output pulse width allowing time for the timing capacitor to charge and then discharge fully. Once triggered, the 555 Monostable will remain in this "HIGH" unstable output state until the time period set up by the R_1 x C_1 network has elapsed. The amount of time that the output voltage remains "HIGH" or at a logic "1" level, is given by the following time constant equation.

$$\tau = 1.1 R_1 C_1$$

Where, t is in seconds, R is in Ω and C in Farads.

The capacitor C has to charge through resistance RA. The larger the time constant RAC, the longer it takes for the capacitor voltage to reach +2/3VCC. In other words, the RC time constant controls the width of the output pulse. The time during which the timer output remains high is given as tp = 1.0986 RAC where RA is in ohms and C is in farads. The above relation is derived as below. Voltage across the capacitor at any instant during charging period is given as vc = VCC (1- e -t/R AC) Substituting vc = 2/3 VCC in above equation we get the time taken by the capacitor to charge from 0 to +2/3VCC. So +2/3VCC. = VCC. (1 – e -t/RAC) or t – RAC loge 3 = 1.0986 RAC So pulse width, tP = 1.0986 RAC s 1.1 RAC The pulse width of the circuit may range from micro-seconds to many seconds. This circuit is widely used in industry for many different timing applications.

Derivation of Pulse Width:

The voltage across capacitor increases exponentially and is given by

$$V_C = V\,(1 - e^{-t/CR})$$

If $$V_C = 2/3\ V_{CC}$$

then $$\frac{2}{3} V_{CC} = V_{CC}\,(1 - e^{-t/CR})$$

$$\frac{2}{3} - 1 = -e^{t/CR}$$

$$\frac{1}{3} = e^{-t/CR}$$

$$-\frac{t}{CR} = -1.0986$$

$$t = +1.0986\ CR$$

$$t \approx 1.1\ CR$$

where C in farads, R in ohms, t in seconds.

Thus, we can say that voltage across capacitor will reach 2/3 Vcc in approximately 1.1 times, time constant i.e. 1.1 RC

Thus the pulse width denoted as W is given by,

W = 1.1 RC

*****