555 Timer

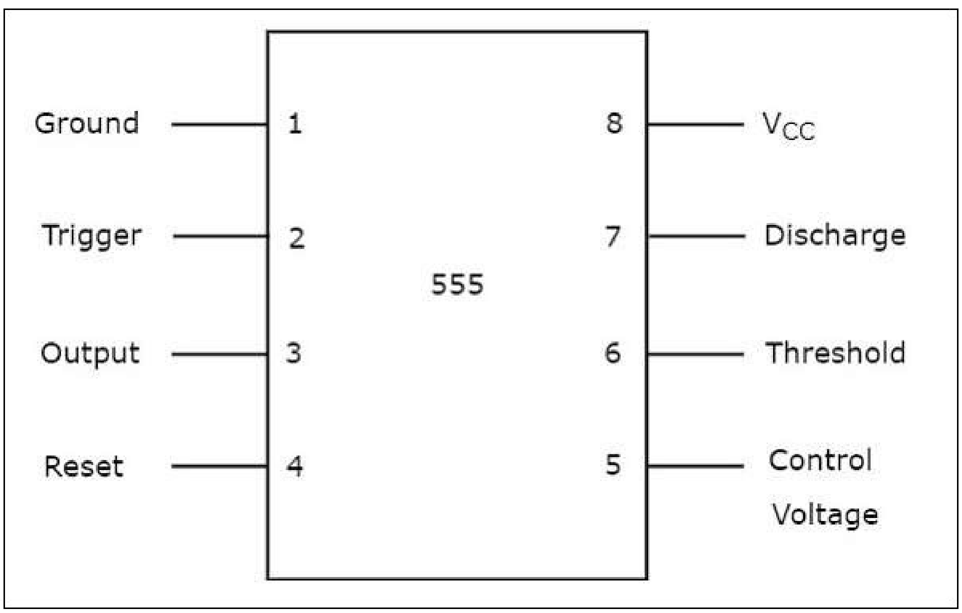
The **555 Timer** IC got its name from the three 5KΩ resistors that are used in its voltage divider network. This IC is useful for generating accurate time delays and oscillations. This chapter explains about 555 Timer in detail.

PIN DIAGRAM AND FUNCTIONAL DIAGRAM

In this section, first let us discuss about the pin diagram of 555 Timer IC and then its functional diagram.

PIN DIAGRAM

The 555 Timer IC is an 8-pin mini-Dual-Inline Package (DIP). The **pin diagram** of a 555 Timer IC is shown in the following figure –

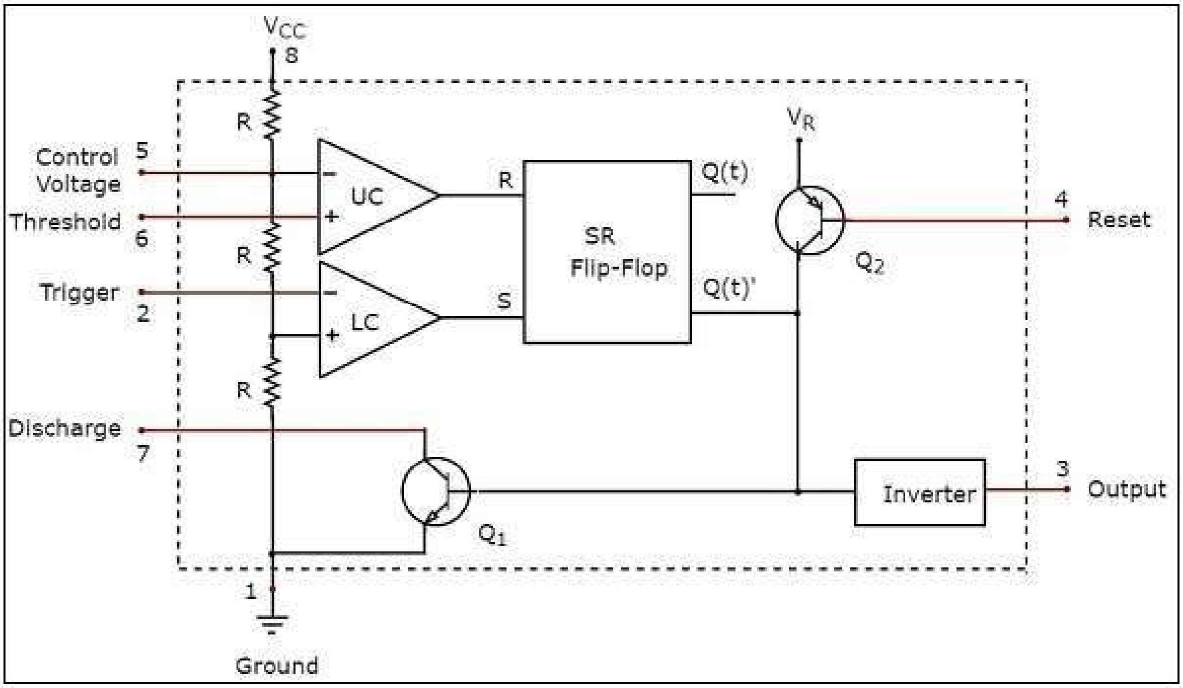


The significance of each pin is self-explanatory from the above diagram. This 555 Timer IC can be operated with a DC supply of +5V to +18V. It is mainly useful for generating **non-sinusoidal** wave forms like square, ramp, pulse & etc.

FUNCTIONAL DIAGRAM

The pictorial representation showing the internal details of a 555 Timer is known as functional diagram.

The **functional diagram** of 555 Timer IC is shown in the following figure –



Observe that the functional diagram of 555 Timer contains a voltage divider network, two comparators, one SR flip-flop, two transistors and an inverter. This section discusses about the purpose of each block or component in detail –

VOLTAGE DIVIDER NETWORK

- The voltage divider network consists of a three 5KΩ resistors that are connected in series between the supply voltage Vcc and ground.
- This network provides a voltage of Vcc / 3 between a point and ground, if there exists only one 5KΩ resistor. Similarly, it provides a voltage of 2Vcc / 3 between a point and ground, if there exists only two 5KΩ resistors.

COMPARATOR

- The functional diagram of a 555 Timer IC consists of two comparators: an Upper Comparator (UC) and a Lower Comparator (LC).
- Recall that a **comparator** compares the two inputs that are applied to it and produces an output.
- If the voltage present at the non-inverting terminal of an op-amp is greater than the voltage present at its inverting terminal, then the output of comparator will be +Vsat. This can be considered as **Logic High** ('1') in digital representation.
- If the voltage present at the non-inverting terminal of op-amp is less than or equal to the voltage at its inverting terminal, then the output of comparator will be −Vsat. This can be considered as **Logic Low** ('0') in digital representation.

SR Flip-Flop

- Recall that a **SR flip-flop** operates with either positive clock transitions or negative clock transitions. It has two inputs: S and R, and two outputs: Q(t) and Q(t)’. The outputs, Q(t) & Q(t)’ are complement to each other.
- The following table shows the **state table** of a SR flip-flop

S	R	Q(t+1)
0	0	Q(t)
0	1	0
1	0	1
1	1	-

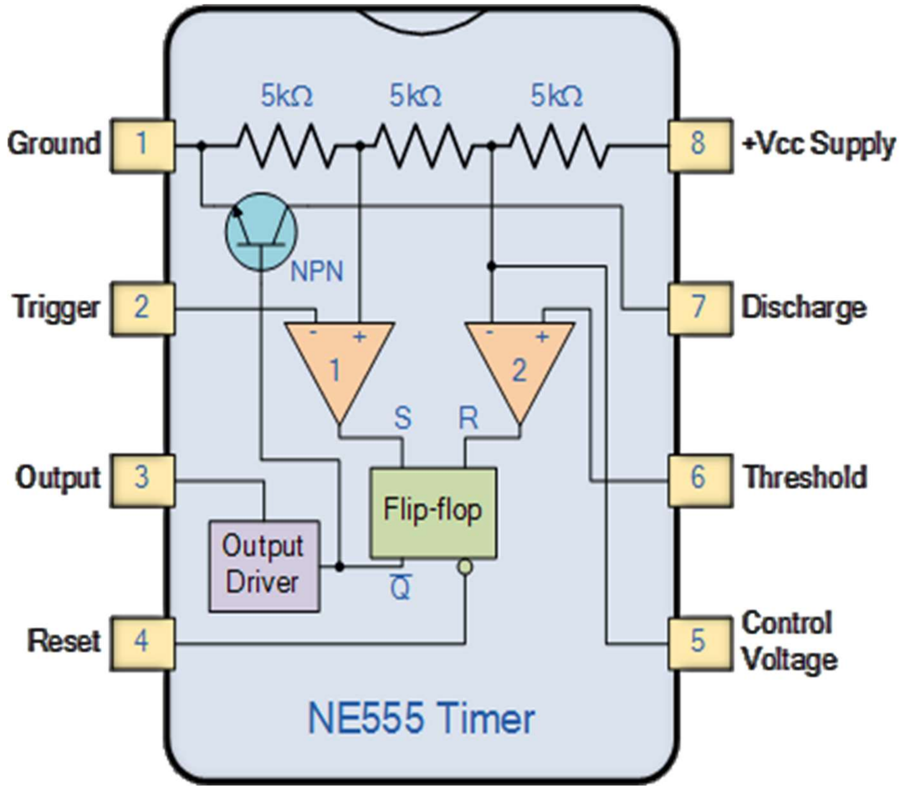
- Here, Q(t) & Q(t+1) are present state & next state respectively. So, SR flip-flop can be used for one of these three functions such as Hold, Reset & Set based on the input conditions, when positive (negative) transition of clock signal is applied.
- The outputs of Lower Comparator (LC) and Upper Comparator (UC) are applied as **inputs of SR flip-flop** as shown in the functional diagram of 555 Timer IC.

Transistors and Inverter

- The functional diagram of a 555 Timer IC consists of one NPN transistor Q1 and one PNP transistor Q2. The npn transistor Q1 will be turned ON if its base to emitter voltage is positive and greater than cut-in voltage. Otherwise, it will be turned-OFF.
- The PNP transistor Q2 is used as **buffer** in order to isolate the reset input from SR flip-flop and NPN transistor Q1.
- The **inverter** used in the functional diagram of a 555 Timer IC not only performs the inverting action but also amplifies the power level.

The 555 Timer IC can be used in mono stable operation in order to produce a pulse at the output. Similarly, it can be used in astable operation in order to produce a square wave at the output.

555 TIMER BLOCK DIAGRAM



1. • **Pin 1. – Ground,** the ground pin connects the 555 timers to the negative (0v) supply rail.
2. • **Pin 2. – Trigger,** The negative input to comparator No 1. A negative pulse on this pin "sets" the internal Flip-flop when the voltage drops below 1/3Vcc causing the output to switch from a "LOW" to a "HIGH" state.
3. • **Pin 3. – Output,** the output pin can drive any TTL circuit and is capable of sourcing or sinking up to 200mA of current at an output voltage equal to approximately Vcc – 1.5V so small speakers, LEDs or motors can be connected directly to the output.
4. • **Pin 4. – Reset**, this pin is used to "reset" the internal Flip-flop controlling the state of the output, pin 3. This is an active-low input and is generally connected to a logic "1" level when not used to prevent any unwanted resetting of the output.

5. • **Pin 5. – Control Voltage,** this pin controls the timing of the 555 by overriding the 2/3Vcc level of the voltage divider network. By applying a voltage to this pin, the width of the output signal can be varied independently of the RC timing network. When not used, it is connected to ground via a 10nF capacitor to eliminate any noise.
6. • **Pin 6. – Threshold,** The positive input to comparator No 2. This pin is used to reset the Flip-flop when the voltage applied to it exceeds 2/3Vcc causing the output to switch from "HIGH" to "LOW" state. This pin connects directly to the RC timing circuit.
7. • **Pin 7. – Discharge,** the discharge pin is connected directly to the Collector of an internal NPN transistor which is used to "discharge" the timing capacitor to ground when the output at pin 3 switches "LOW".
8. • **Pin 8. – Supply +Vcc,** this is the power supply pin and for general purpose TTL 555 timers is between 4.5V and 15V.

The **555 Timers** name comes from the fact that there are three 5kΩ resistors connected together internally producing a voltage divider network between the supply voltage at pin 8 and ground at pin 1. The voltage across this series resistive network holds the negative inverting input of comparator two at 2/3Vcc and the positive non-inverting input to comparator one at 1/3Vcc.

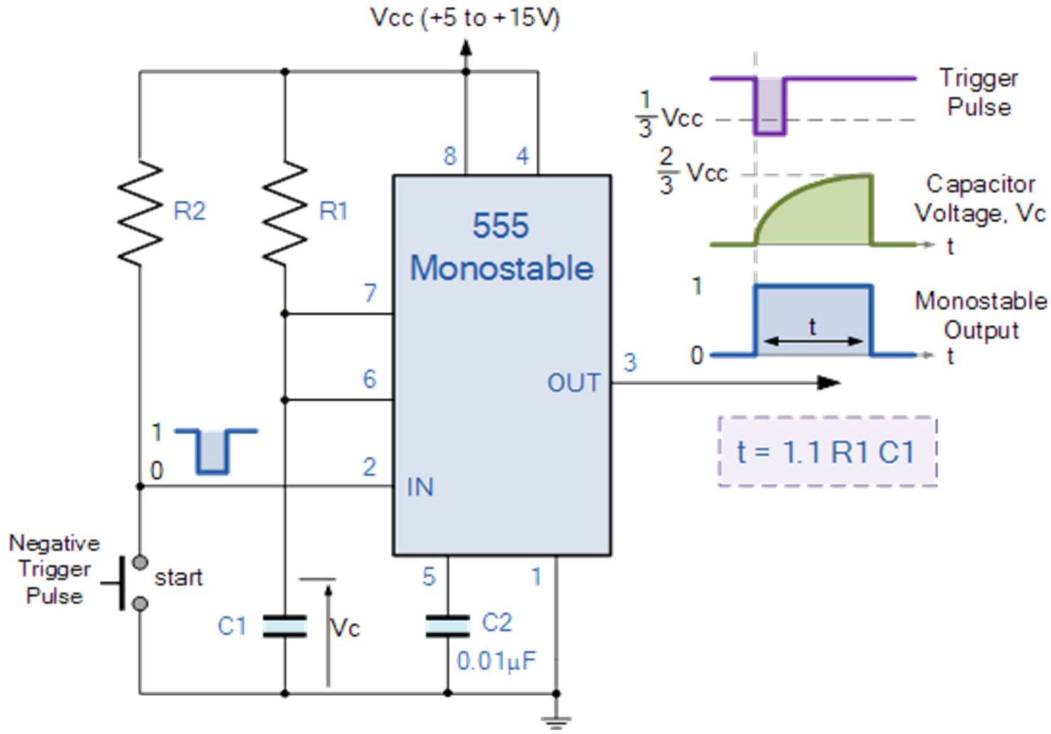
The two comparators produce an output voltage dependent upon the voltage difference at their inputs which is determined by the charging and discharging action of the externally connected RC network. The outputs from both comparators are connected to the two inputs of the flip-flop which in turn produces either a "HIGH" or "LOW" level output at Q based on the states of its inputs. The output from the flip-flop is used to control a high current output switching stage to drive the connected load producing either a "HIGH" or "LOW" voltage level at the output pin.

The most common use of the 555-timer oscillator is as a simple astable oscillator by connecting two resistors and a capacitor across its terminals to generate a fixed pulse train with a time period determined by the time constant of the RC network. But the 555-timer oscillator chip can also be connected in a variety of different ways to produce Monostable or Bistable multivibrators as well as the more common Astable Multivibrator.

THE MONOSTABLE 555 TIMER

The operation and output of the **555-timer monostable** is exactly the same as that for the transistorized one we look at previously in the Monostable Multivibrators tutorial. The difference this time is that the two transistors have been replaced by the 555-timer device. Consider the 555-timer monostable circuit below.

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.

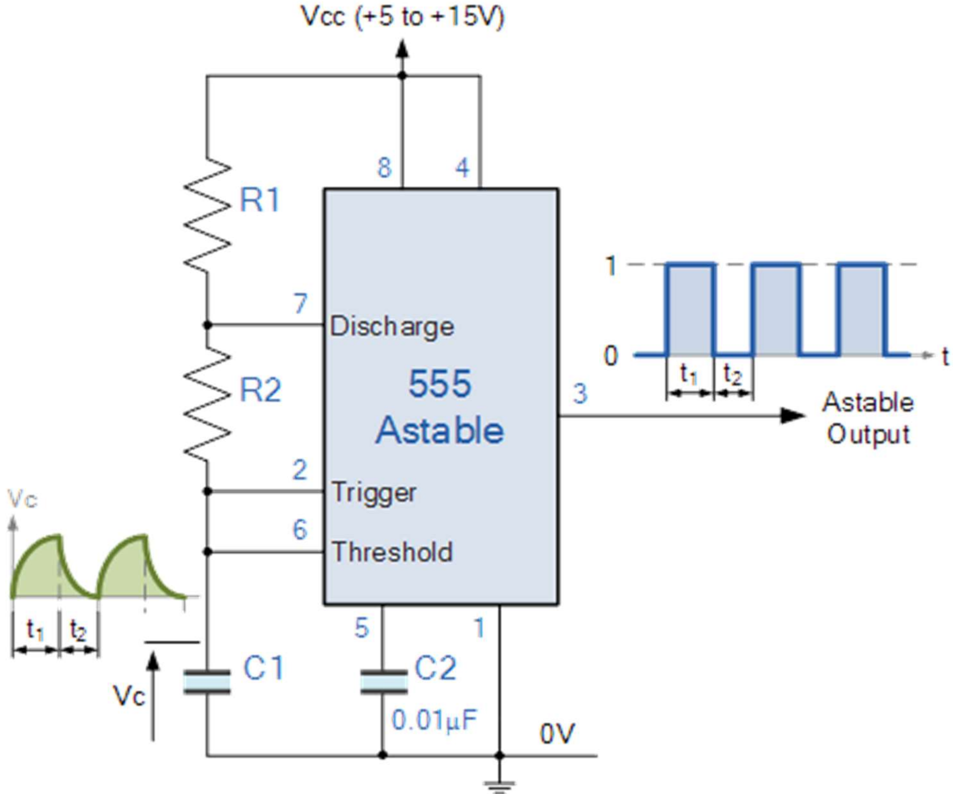
This action allows the timing capacitor to start to charge up through resistor, R1 until the voltage across the capacitor reaches the threshold (pin 6) voltage of 2/3Vcc set up by the internal voltage divider network. At this point the comparators output goes "HIGH" and "resets" the flip-flop back to 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.

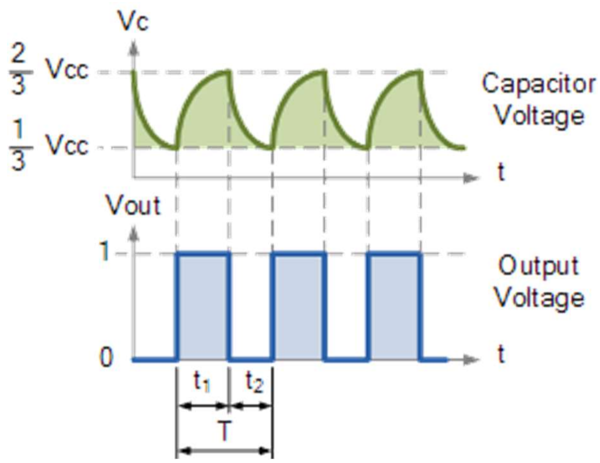
The **Monostable 555 Timer** circuit triggers on a negative-going pulse applied to pin 2 and this trigger pulse must be much shorter than the output pulse width allowing time for the timing capacitor to charge and then discharge fully. Once triggered, the 555 Monostable will remain in this "HIGH" unstable 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.

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 555 Oscillator Charge and Discharge Times

$$t_1 = 0.693(R_1 + R_2).C$$

and

$$t_2 = 0.693 \times R_2 \times C$$

Where, R is in Ω and C in Farads.

When connected as an astable multivibrator, the output from the **555 Oscillator** will continue indefinitely charging and discharging between 2/3Vcc and 1/3Vcc until the power supply is removed. As with the monostable multivibrator these charge and discharge times and therefore the frequency are independent on the supply voltage.

The duration of one full timing cycle is therefore equal to the sum of the two individual times that the capacitor charges and discharges added together and is given as:

555 Oscillator Cycle Time

$$T = t_1 + t_2 = 0.693(R_1 + 2R_2).C$$

The output frequency of oscillations can be found by inverting the equation above for the total cycle time giving a final equation for the output frequency of an Astable 555 Oscillator as:

555 Oscillator Frequency Equation

$$f = \frac{1}{T} = \frac{1.44}{(R_1 + 2R_2).C}$$

By altering the time constant of just one of the RC combinations, the **Duty Cycle** better known as the “Mark-to-Space” ratio of the output waveform can be accurately set and is given as the ratio of resistor R2 to resistor R1. The Duty Cycle for the 555 Oscillator, which is the ratio of the “ON” time divided by the “OFF” time is given by:

555 OSCILLATOR DUTY CYCLE

$$\text{Duty Cycle} = \frac{T_{ON}}{T_{OFF} + T_{ON}} = \frac{R_1 + R_2}{(R_1 + 2R_2)}\ \%$$

The duty cycle has no units as it is a ratio but can be expressed as a percentage (%). If both timing resistors, R1 and R2 are equal in value, then the output duty cycle will be 2:1 that is, 66% ON time and 33% OFF time with respect to the period.

555 OSCILLATOR EXAMPLE NO1

An **Astable 555 Oscillator** is constructed using the following components, R1 = 1kΩ, R2 = 2kΩ and capacitor C = 10uF. Calculate the output frequency from the 555 oscillator and the duty cycle of the output waveform.

t_1 – capacitor charge "ON" time is calculated as:

$$\begin{aligned} t_1 &= 0.693(R_1 + R_2).C \\ &= 0.693(1000 + 2000) \times 10 \times 10^{-6} \\ &= 0.021\text{s} = 21\text{ms} \end{aligned}$$

t_2 – capacitor discharge "OFF" time is calculated as:

$$\begin{aligned} t_2 &= 0.693\, R_2.C \\ &= 0.693 \times 2000 \times 10 \times 10^{-6} \\ &= 0.014\text{s} = 14\text{ms} \end{aligned}$$

Total periodic time (T) is therefore calculated as:

$$T = t_1 + t_2 = 21\text{ms} + 14\text{ms} = 35\text{ms}$$

The output frequency, ƒ is therefore given as:

$$f = \frac{1}{T} = \frac{1}{35\text{ms}} = 28.6\text{Hz}$$

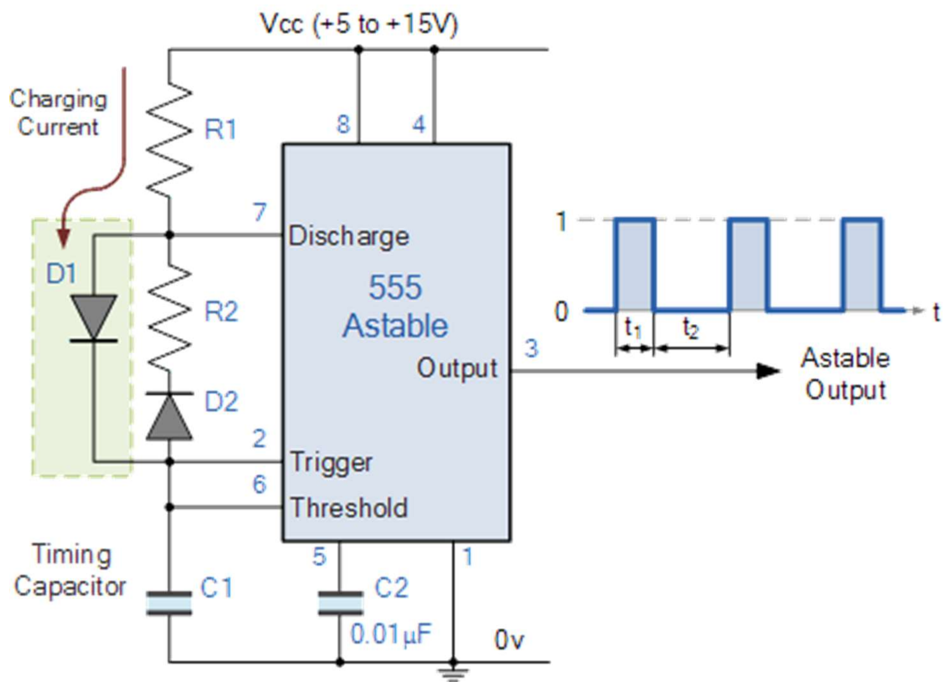
Giving a duty cycle value of:

$$\text{Duty Cycle} = \frac{R_1 + R_2}{(R_1 + 2R_2)} = \frac{1000 + 2000}{(1000 + 2 \times 2000)} = 0.6 \text{ or } 60\%$$

As the timing capacitor, C charges through resistors R1 and R2 but only discharges through resistor R2 the output duty cycle can be varied between 50 and 100% by changing the value of resistor R2. By decreasing the value of R2 the duty cycle increases towards 100% and by increasing R2 the duty cycle reduces towards 50%. If resistor, R2 is very large relative to resistor R1 the output frequency of the 555 astable circuit will determined by R2 x C only.

The problem with this basic astable 555 oscillator configuration is that the duty cycle, the “mark to-space” ratio will never go below 50% as the presence of resistor R2 prevents this. In other words, we cannot make the outputs “ON” time shorter than the “OFF” time, as (R1 + R2) C will always be greater than the value of R1 x C. One way to overcome this problem is to connect a signal bypassing diode in parallel with resistor R2 as shown below.

IMPROVED 555 OSCILLATOR DUTY CYCLE



By connecting this diode, D1 between the *trigger* input and the *discharge* input, the timing capacitor will now charge up directly through resistor R1 only, as resistor R2 is effectively shorted out by the diode. The capacitor discharges as normal through resistor, R2.

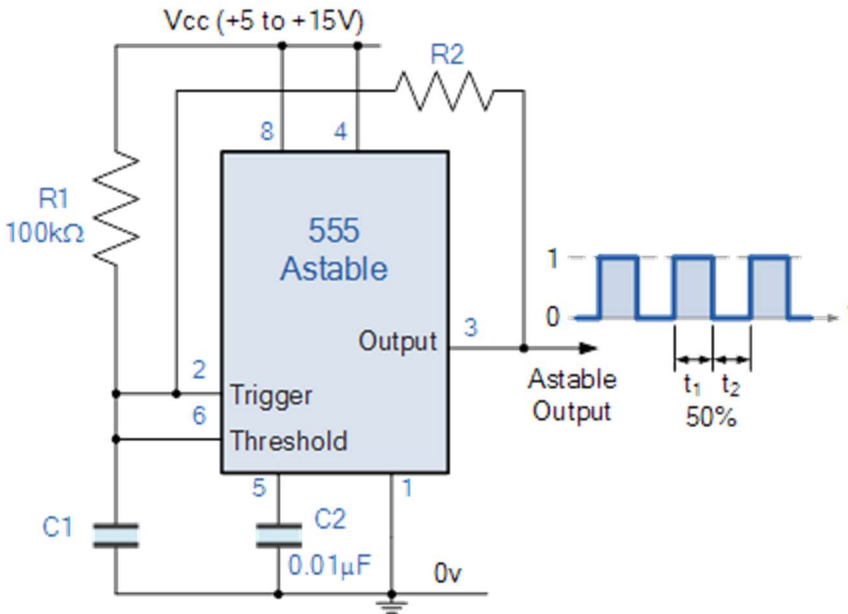
An additional diode, D2 can be connected in series with the discharge resistor, R2 if required to ensure that the timing capacitor will only charge up through D1 and not through the parallel path of R2. This is because during the charging process diode D2 is connected in reverse bias blocking the flow of current through itself.

Now the previous charging time of t_1 = 0.693(R1 + R2) C is modified to take account of this new charging circuit and is given as: 0.693(R1 x C). The duty cycle is therefore given as D = R1/ (R1 + R2). Then to generate a duty cycle of less than 50%, resistor R1 needs to be less than resistor R2.

Although the previous circuit improves the duty cycle of the output waveform by charging the timing capacitor, C1 through the R1 + D1 combination and then discharging it through the D2 + R2 combination, the problem with this circuit arrangement is that the 555-oscillator circuit uses additional components, i.e., two diodes.

We can improve on this idea and produce a fixed square wave output waveform with an exact 50% duty cycle very easily and without the need for any extra diodes by simply moving the position of the charging resistor, R2 to the output (pin 3) as shown.

50% Duty Cycle Astable Oscillator



The 555 oscillators now produce a 50% duty cycle as the timing capacitor, C1 is now charging and discharging through the same resistor, R2 rather than discharging through the timer's discharge pin 7 as before. When the output from the 555 oscillator is HIGH, the capacitor charges up through R2 and when the output is LOW, it discharges through R2. Resistor R1 is used to ensure that the capacitor charges up fully to the same value as the supply voltage.

However, as the capacitor charges and discharges through the same resistor, the above equation for the output frequency of oscillations has to be modified a little to reflect this circuit change. Then the new equation for the 50% Astable 555 Oscillator is given as:

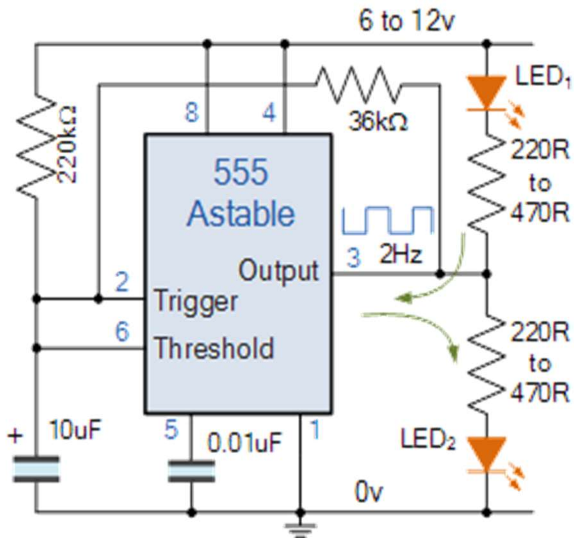
50% Duty Cycle Frequency Equation

$$f = \frac{1}{0.693(2R_2).C} \text{ Hz}$$

Note that resistor R1 needs to be sufficiently high enough to ensure it does not interfere with the charging of the capacitor to produce the required 50% duty cycle. Also changing the value of the timing capacitor, C1 changes the oscillation frequency of the astable circuit.

555 Oscillator Applications

We said previously that the maximum output to either sink or source the load current via pin 3 is about 200mA and this value is more than enough to drive or switch other logic IC's, a few LED's or a small lamp etc. and that we would need to use a bipolar transistor or MOSFET to amplify the 555's output to drive larger current loads such as motor or relays.



But the **555 Oscillator** can also be used in a wide range of waveform generator circuits and applications that require very little output current such as in electronic test equipment for producing a whole range of different output test frequencies.

The 555 can also be used to produce very accurate sine, square and pulse waveforms or as LED or lamp flashers and dimmers to simple noise making circuits such as metronomes, tone and sound effects generators and even musical toys for Christmas.

We could very easily build a simple 555 oscillator circuit to flash a few LED's "ON" and "OFF" similar to the one shown, or to produce a high frequency noise from a loudspeaker. But one very nice and simple to build science project using an astable based 555 oscillator is that of an Electronic Metronome.

Metronomes are devices used to mark time in pieces of music by producing a regular and recurring musical beat or click. A simple electronic metronome can be made using a 555 oscillator as the main timing device and by adjusting the output frequency of the oscillator the tempo or "Beats per Minute" can be set.

So, for example, a tempo of 60 beats per minute means that one beat will occur every second and in electronics terms that equates to 1Hz. So, by using some very common musical definitions we can easily build a table of the different frequencies required for our metronome circuit as shown below.

Metronome Frequency Table

Musical Definition	Rate	Beats per Minute	Cycle Time (T)	Frequency
Larghetto	**Very Slow**	**60**	**1sec**	**1.0Hz**
Andante	**Slow**	**90**	**666ms**	**1.5Hz**
Moderato	**Medium**	**120**	**500ms**	**2.0Hz**
Allegro	**Fast**	**150**	**400ms**	**2.5Hz**
Presto	**Very Fast**	**180**	**333ms**	**3.0Hz**

The output frequency range of the metronome was simply calculated as the reciprocal of 1 minute or 60 seconds divided by the number of beats per minute required, for example (1/(60 secs / 90 bpm) = 1.5Hz) and 120bpm is equivalent to 2Hz, and so on. So, by using our now familiar equation above for calculating the output frequency of an astable 555 oscillator circuit the individual values of R1, R2 and C can be found.

The time period of the output waveform for an astable 555 Oscillator is given as:

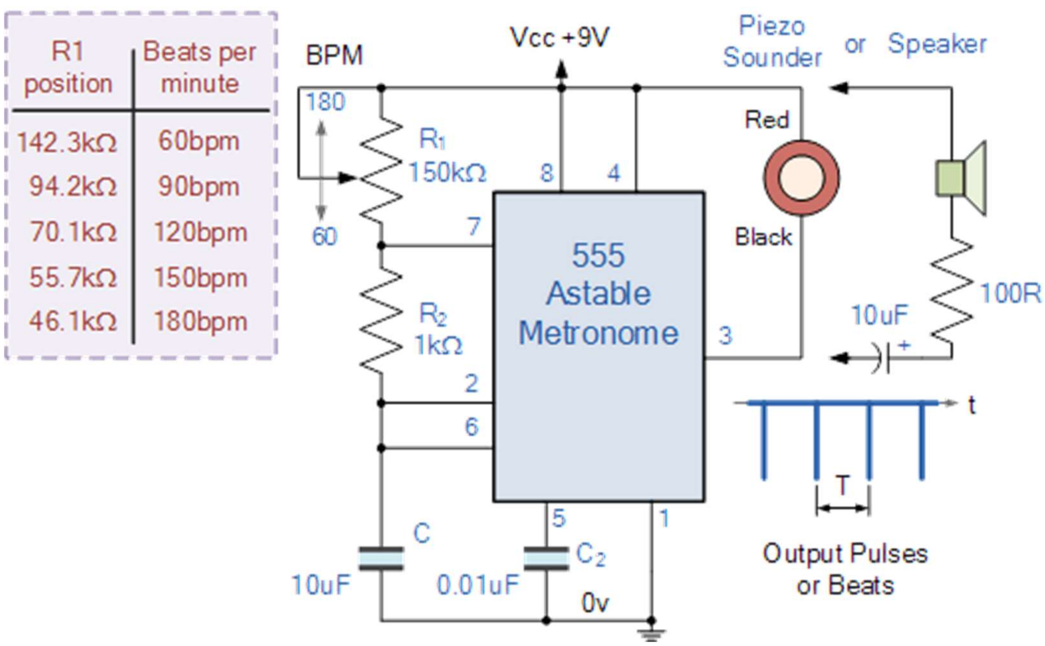
$$T = t_1 + t_2 = 0.693(R_1 + 2R_2).C$$

For our electronic metronome circuit, the value of the timing resistor R1 can be found by rearranging the equation above to give:

$$R_1 = \frac{T}{0.693 \times C} - 2R_2$$

Assuming a value for resistor R2 = 1kΩ and capacitor C = 10uF the value of the timing resistor R1 for our frequency range is given as 142k3Ω at 60 beats per minute to 46k1Ω at 180 beats per minute, so a variable resistor (potentiometer) of 150kΩ would be more than enough for the metronome circuit to produce the full range of beats required and some more. Then the final circuit for our electronic metronome example would be given as:

555 ELECTRONIC METRONOME



This simple metronome circuit demonstrates just one simple way of using a 555 oscillator to produce an audible sound or note. It uses a 150kΩ potentiometer to control the full range of output pulses or beats, and as it has a 150kΩ value it can be easily calibrated to give an equivalent percentage value corresponding to the position of the potentiometer. For example, 60 beats per minute equals 142.3kΩ or 95% rotation.

Likewise, 120 beats per minute equals 70.1kΩ or 47% rotation, etc. Additional resistors or trimmer's can be connected in series with the potentiometer to pre-set the outputs upper and lower limits to predefined values, but these additional components will need to be taken into account when calculating the output frequency or time period.

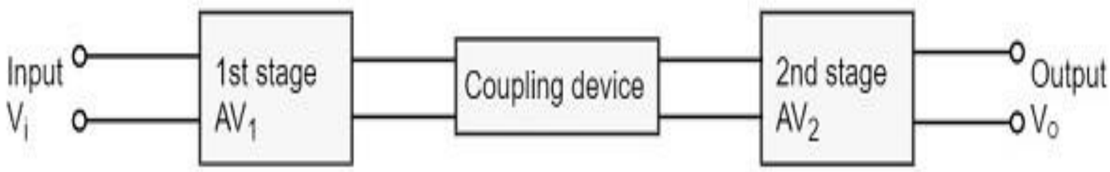
While the above circuit is a very simple and amusing example of sound generation, it is possible to use the **555 Oscillator** as a noise generator/synthesizer or to make musical sounds, tones and alarms by constructing a variable-frequency, variable-mark/space ratio waveform generator.

MULTI-STAGE TRANSISTOR AMPLIFIER

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

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

using a coupling device can be called as **Cascading**.



The 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** number 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, which means to isolate the DC conditions.

Types of Coupling

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

1 Resistance-Capacitance Coupling

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

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

2 Impedance Coupling

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

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

3 Transformer Coupling

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

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

4 Direct Coupling

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

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

Role of Capacitors in Amplifiers

Other than the coupling purpose, there are other purposes for which few capacitors are especially employed in amplifiers. To understand this, let us know about the role of capacitors in Amplifiers.

The Input Capacitor C_{in}

The input capacitor C_{in} present at the initial stage of the amplifier, couples AC signal to the base of the transistor. This capacitor C_{in} if not present, the signal source will be in parallel to resistor R_2 and the bias voltage of the transistor base will be changed.

Hence C_{in} allows, the AC signal from source to flow into input circuit, without affecting the bias conditions.

The Emitter By-pass Capacitor C_e

The emitter by-pass capacitor C_e is connected in parallel to the emitter resistor. It offers a low reactance path to the amplified AC signal.

In the absence of this capacitor, the voltage developed across R_E will feedback to the input side thereby reducing the output voltage. Thus, in the presence of C_e the amplified AC will pass through this.

Coupling Capacitor C_C

The capacitor C_C is the coupling capacitor that connects two stages and prevents DC interference between the stages and controls the operating point from shifting. This is also called as **blocking capacitor** because it does not allow the DC voltage to pass through it.

In the absence of this capacitor, R_C will come in parallel with the resistance R_1 of the biasing network of the next stage and thereby changing the biasing conditions of the next stage.

Amplifier Consideration

For an amplifier circuit, the overall gain of the amplifier is an important consideration. To achieve maximum voltage gain, let us find the most suitable transistor configuration for cascading.

CC Amplifier

- Its voltage gain is less than unity.
- It is not suitable for intermediate stages.

CB Amplifier

- Its voltage gain is less than unity.
- Hence not suitable for cascading.

CE Amplifier

- Its voltage gain is greater than unity.
- Voltage gain is further increased by cascading.

The characteristics of CE amplifier are such that, this configuration is very suitable for cascading in amplifier circuits. Hence most of the amplifier circuits use CE configuration.

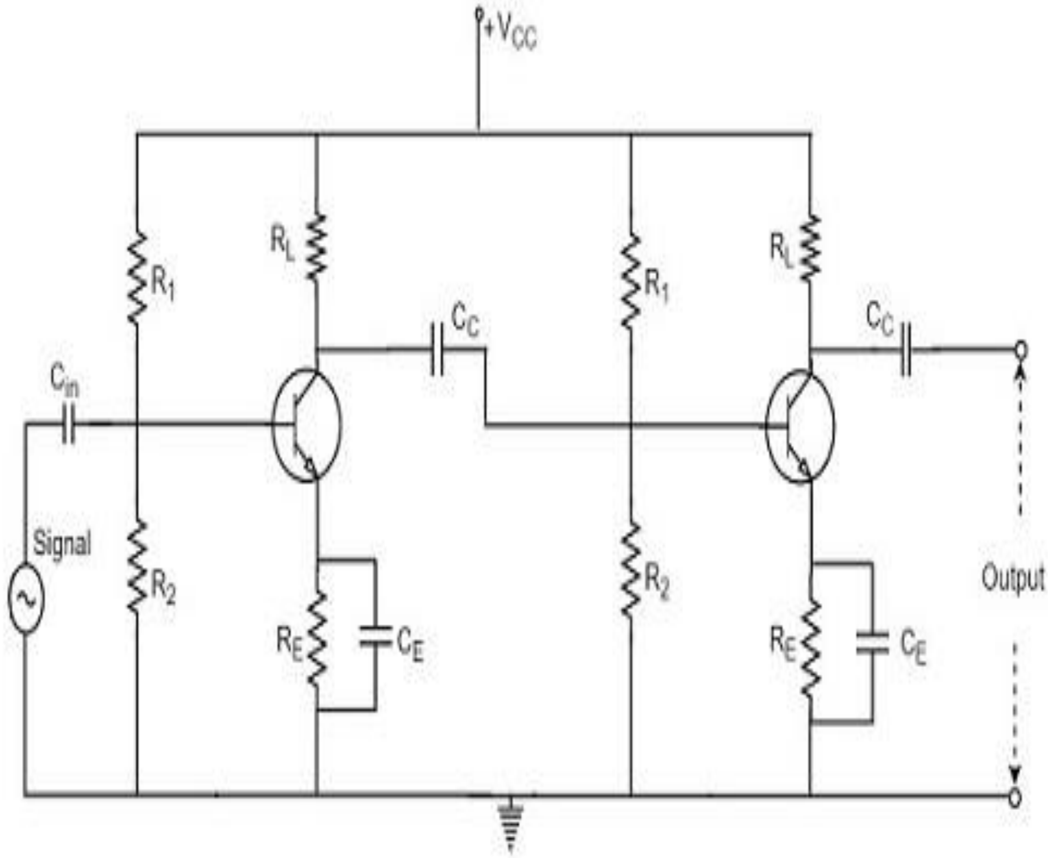
RC COUPLED AMPLIFIER

The resistance-capacitance coupling is, in short termed as RC coupling. This is the mostly used coupling technique in amplifiers.

Construction of a Two-stage RC Coupled Amplifier

The constructional details of a two-stage RC coupled transistor amplifier circuit are as follows. The two stage amplifier circuit has two transistors, connected in CE configuration and a common power supply V_{CC} is used. The potential divider network R_1 and R_2 and the resistor R_e form the biasing and stabilization network. The emitter by-pass capacitor C_e offers a low reactance path to the signal.

The resistor R_L is used as a load impedance. The input capacitor C_{in} present at the initial stage of the amplifier couples AC signal to the base of the transistor. The capacitor C_C is the coupling capacitor that connects two stages and prevents DC interference between the stages and controls the shift of operating point. The figure below shows the circuit diagram of RC coupled amplifier.



Operation of RC Coupled Amplifier

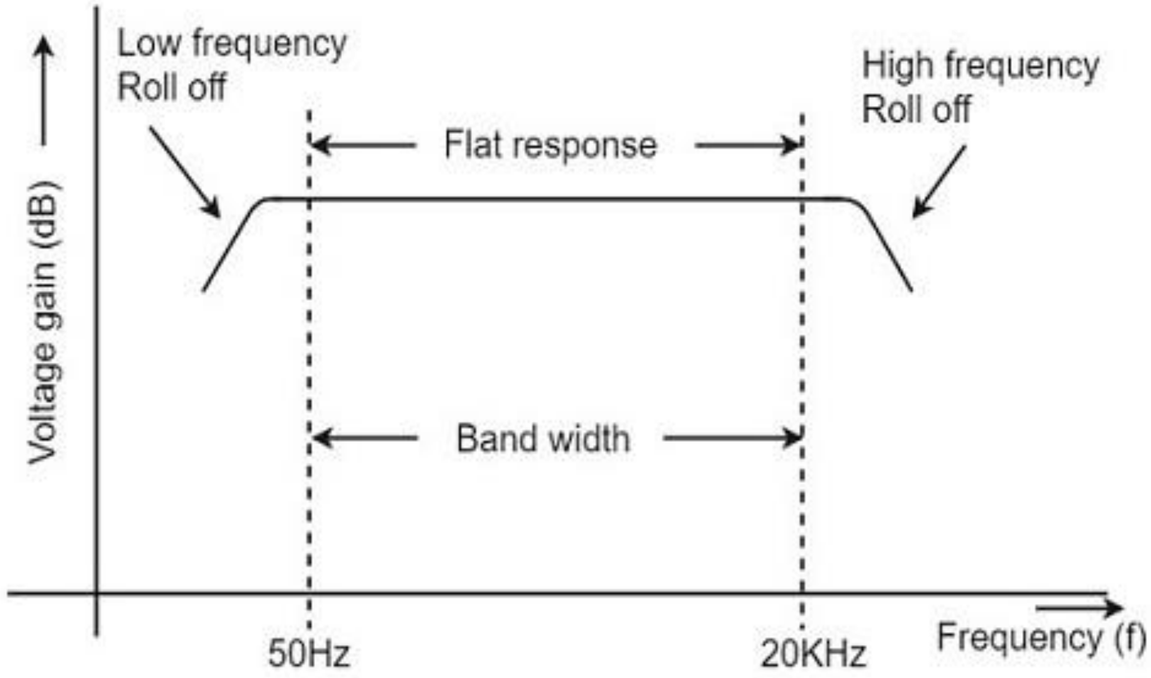
When an AC input signal is applied to the base of first transistor, it gets amplified and appears at the collector load R_L which is then passed through the coupling capacitor C_C to the next stage. This becomes the input of the next stage, whose amplified output again appears across its collector load.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

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

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

Frequency Response of RC Coupled Amplifier

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



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

We know that,

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

It means that the capacitive reactance is inversely proportional to the frequency.

At Low frequencies (i.e., below 50 Hz)

The capacitive reactance is inversely proportional to the frequency. At low frequencies, the reactance is quite high. The reactance of input capacitor C_{in} and the coupling capacitor C_C are so high that only small part of the input signal is allowed. The reactance of the emitter by pass capacitor C_E is also very high during low frequencies. Hence it cannot shunt the emitter resistance effectively. With all these factors, the voltage gain rolls off at low frequencies.

At High frequencies (i.e., above 20 KHz)

Again considering the same point, we know that the capacitive reactance is low at high frequencies. So, a capacitor behaves as a short circuit, at high frequencies. As a result of this, the loading effect of the next stage increases, which reduces the voltage gain. Along with this, as the capacitance of emitter diode decreases, it increases the base current of the transistor due to which the current gain (β) reduces. Hence the voltage gain rolls off at high frequencies.

At Mid-frequencies (i.e., 50 Hz to 20 KHz)

The voltage gain of the capacitors is maintained constant in this range of frequencies, as shown in figure. If the frequency increases, the reactance of the capacitor C_C decreases which tends to increase the gain. But this lower capacitance reactive increases the loading effect of the next stage by which there is a reduction in gain.

Due to these two factors, the gain is maintained constant.

Advantages of RC Coupled Amplifier

The following are the advantages of RC coupled amplifier.

- The frequency response of RC amplifier provides constant gain over a wide frequency range, hence most suitable for audio applications.
- The circuit is simple and has lower cost because it employs resistors and capacitors which are cheap.
- It becomes more compact with the upgrading technology.

Disadvantages of RC Coupled Amplifier

The following are the disadvantages of RC coupled amplifier.

- The voltage and power gain are low because of the effective load resistance.
- They become noisy with age.
- Due to poor impedance matching, power transfer will be low.

Applications of RC Coupled Amplifier

The following are the applications of RC coupled amplifier.

- They have excellent audio fidelity over a wide range of frequency.
- Widely used as Voltage amplifiers
- Due to poor impedance matching, RC coupling is rarely used in the final stages.

TRANSFORMER COUPLED AMPLIFIER

We have observed that the main drawback of RC coupled amplifier is that the effective load resistance gets reduced. This is because, the input impedance of an amplifier is low, while its output impedance is high.

When they are coupled to make a multistage amplifier, the high output impedance of one stage comes in parallel with the low input impedance of next stage. Hence, effective load resistance is decreased. This problem can be overcome by a **transformer coupled amplifier**.

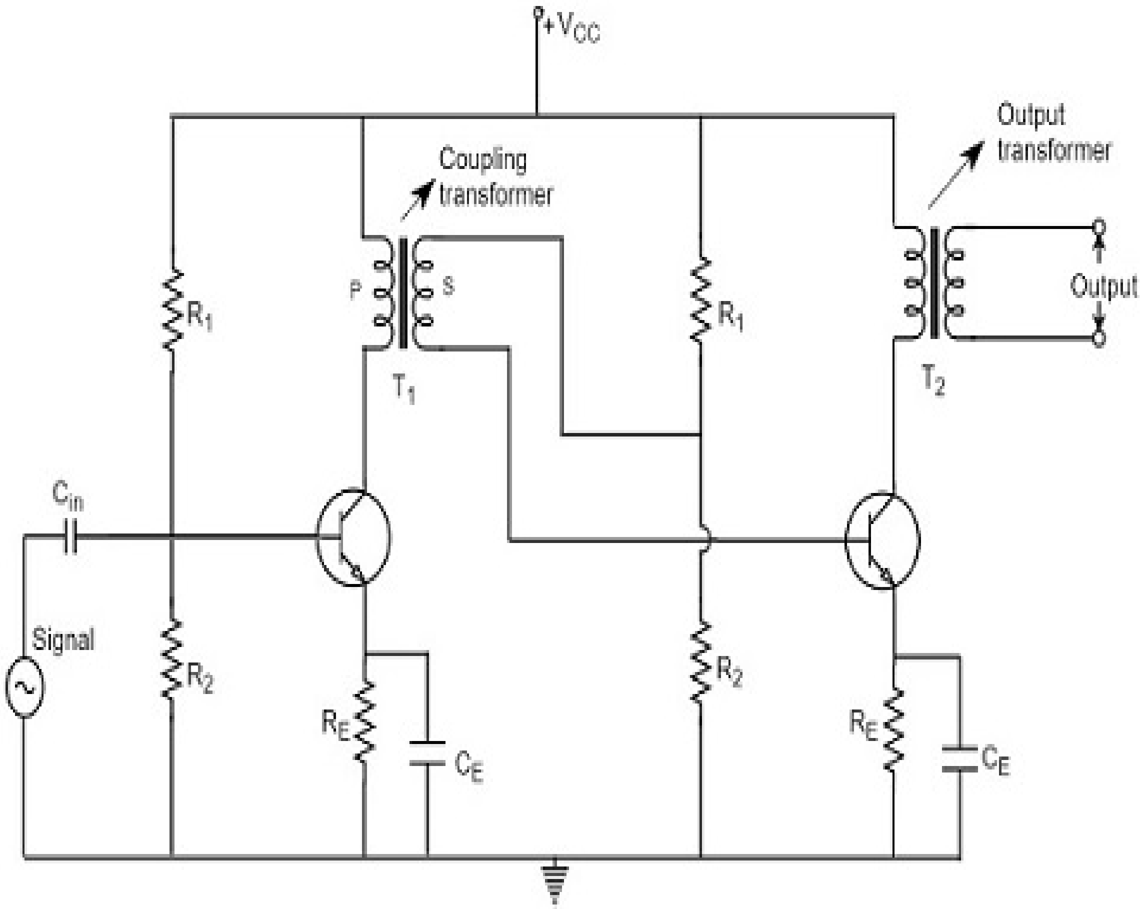
In a transformer-coupled amplifier, the stages of amplifier are coupled using a transformer. Let us go into the constructional and operational details of a transformer coupled amplifier.

Construction of Transformer Coupled Amplifier

The amplifier circuit in which, the previous stage is connected to the next stage using a coupling transformer, is called as Transformer coupled amplifier.

The coupling transformer T_1 is used to feed the output of 1st stage to the input of 2nd stage. The collector load is replaced by the primary winding of the transformer. The secondary winding is connected between the potential divider and the base of 2nd stage, which provides the input to the 2nd stage. Instead of coupling capacitor like in RC coupled amplifier, a transformer is used for coupling any two stages, in the transformer coupled amplifier circuit.

The figure below shows the circuit diagram of transformer coupled amplifi



The potential divider network R_1 and R_2 and the resistor R_e together form the biasing and stabilization network. The emitter by-pass capacitor C_e offers a low reactance path to the signal. The resistor R_L is used as a load impedance. The input capacitor C_{in} present at the initial stage of the amplifier couples AC signal to the base of the transistor. The capacitor C_C is the coupling capacitor that connects two stages and prevents DC interference between the stages and controls the shift of operating point.

Operation of Transformer Coupled Amplifier

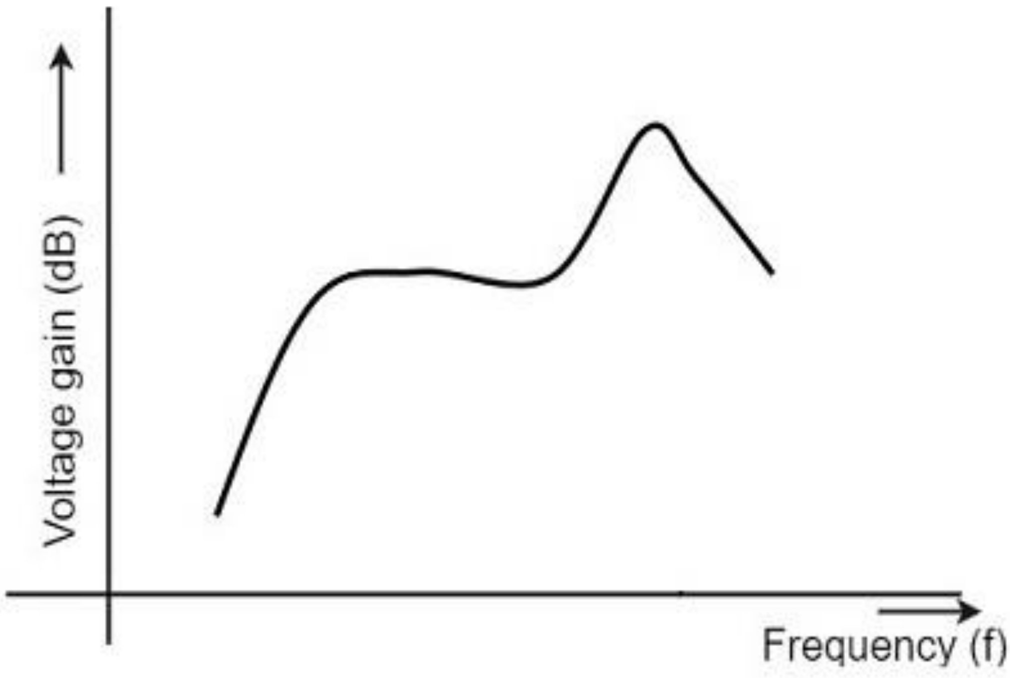
When an AC signal is applied to the input of the base of the first transistor then it gets amplified by the transistor and appears at the collector to which the primary of the transformer is connected.

The transformer which is used as a coupling device in this circuit has the property of impedance changing, which means the low resistance of a stage (or load) can be reflected as a high load resistance to the previous stage. Hence the voltage at the primary is transferred according to the turns ratio of the secondary winding of the transformer.

This transformer coupling provides good impedance matching between the stages of amplifier. The transformer coupled amplifier is generally used for power amplification.

Frequency Response of Transformer Coupled Amplifier

The figure below shows the frequency response of a transformer coupled amplifier. The gain of the amplifier is constant only for a small range of frequencies. The output voltage is equal to the collector current multiplied by the reactance of primary.



At low frequencies, the reactance of primary begins to fall, resulting in decreased gain. At high frequencies, the capacitance between turns of windings acts as a bypass condenser to reduce the output voltage and hence gain.

So, the amplification of audio signals will not be proportionate and some distortion will also get introduced, which is called as **Frequency distortion**.

Advantages of Transformer Coupled Amplifier

The following are the advantages of a transformer coupled amplifier −

- An excellent impedance matching is provided.
- Gain achieved is higher.
- There will be no power loss in collector and base resistors.
- Efficient in operation.

Disadvantages of Transformer Coupled Amplifier

The following are the disadvantages of a transformer coupled amplifier −

- Though the gain is high, it varies considerably with frequency. Hence a poor frequency response.
- Frequency distortion is higher.
- Transformers tend to produce hum noise.
- Transformers are bulky and costly.

Applications

The following are the applications of a transformer coupled amplifier −

- Mostly used for impedance matching purposes.
- Used for Power amplification.
- Used in applications where maximum power transfer is needed.

DIRECT COUPLED AMPLIFIER

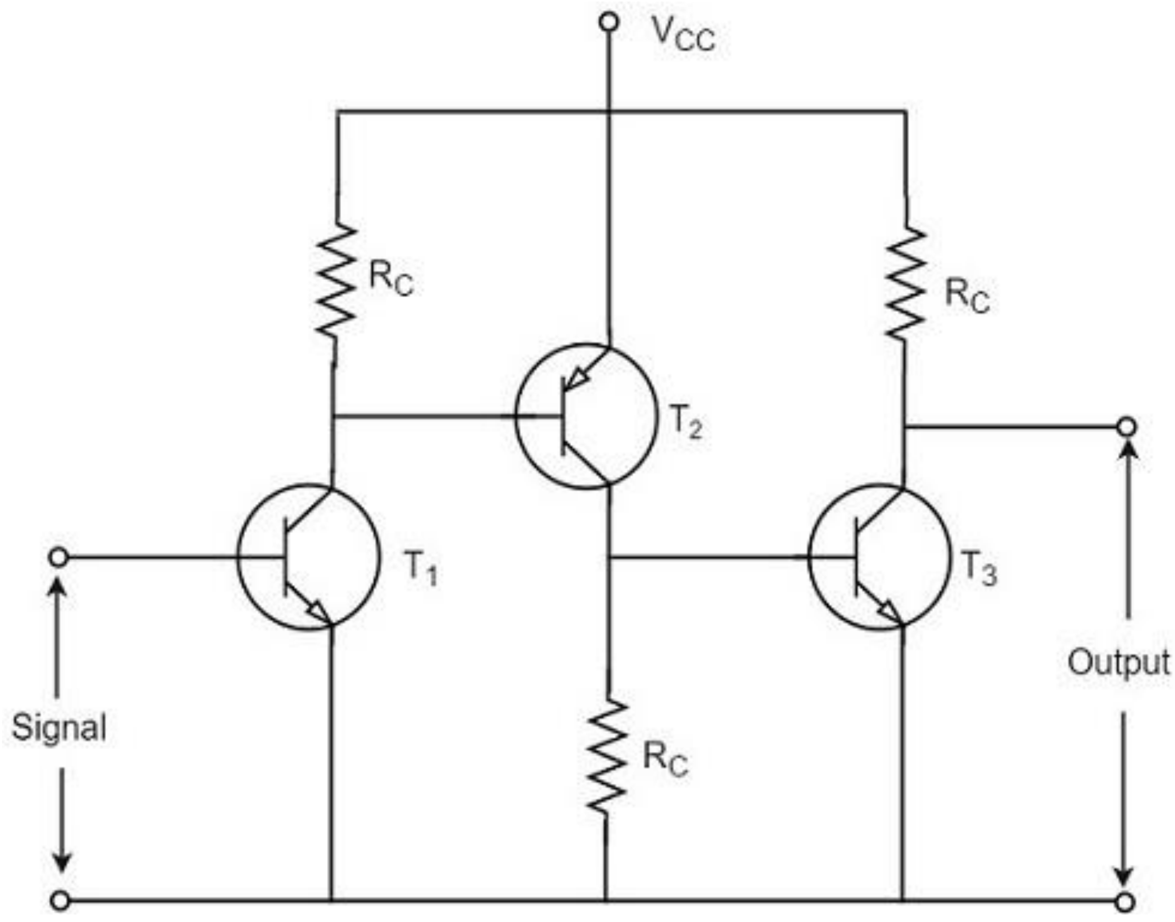
The other type of coupling amplifier is the direct coupled amplifier, which is especially used to amplify lower frequencies, such as amplifying photo-electric current or thermo-couple current or so.

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 T_1 is connected to the input of second stage transistor T_2.



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 T_1, it gets amplified due to the transistor action and the amplified output appears at the collector resistor R_c of transistor T_1. This output is applied to the base of transistor T_2 which further amplifies the signal. In this way, a signal is amplified in a direct coupled amplifier circuit.

Advantages

The advantages of direct coupled amplifier are as follows.

- 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

The disadvantages of direct coupled amplifier are as follows.

- It cannot be used for amplifying high frequencies.
- The operating point is shifted due to temperature variations.

Applications

The applications of direct coupled amplifier are as follows.

- Low frequency amplifications.
- Low current amplifications.

Comparisions

Let us try to compare the characteristics of different types of coupling methods discussed till now.

S.No	Particular	RC Coupling	Transformer Coupling	Direct Coupling
1	Frequency response	Excellent in audio frequency range	Poor	Best
2	Cost	Less	More	Least
3	Space and Weight	Less	More	Least
4	Impedance matching	Not good	Excellent	Good
5	Use	For voltage amplification	For Power amplification	For amplifying extremely low frequencies

Multivibrators

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

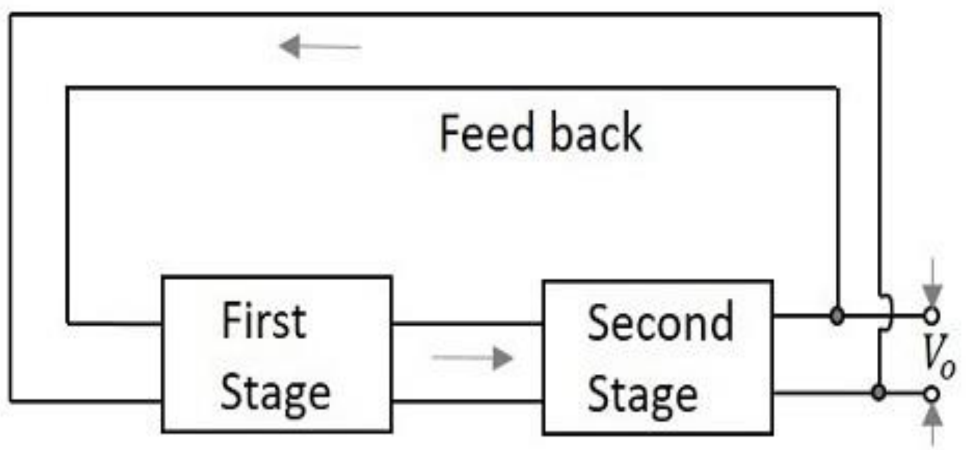
A Transistor basically functions as an amplifier in its linear region. If a transistor amplifier output stage is joined with the previous amplifier stage, such a connection is said to be coupled. If a resistor is used in coupling two stages of such an amplifier circuit, it is called as **Resistance coupled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

WHAT IS A MULTIVIBRATOR?

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

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

The following figure shows the block diagram of a Multivibrator.



TYPES OF MULTIVIBRATORS

There are two possible states of a Multivibrator. In first stage, the transistor Q_1 turns ON while the transistor Q_2 turns OFF. In second stage, the transistor Q_1 turns OFF while the transistor Q_2 turns ON. These two states are interchanged for certain time periods depending upon the circuit conditions.

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

ASTABLE MULTIVIBRATOR

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

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

MONOSTABLE MULTIVIBRATOR

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

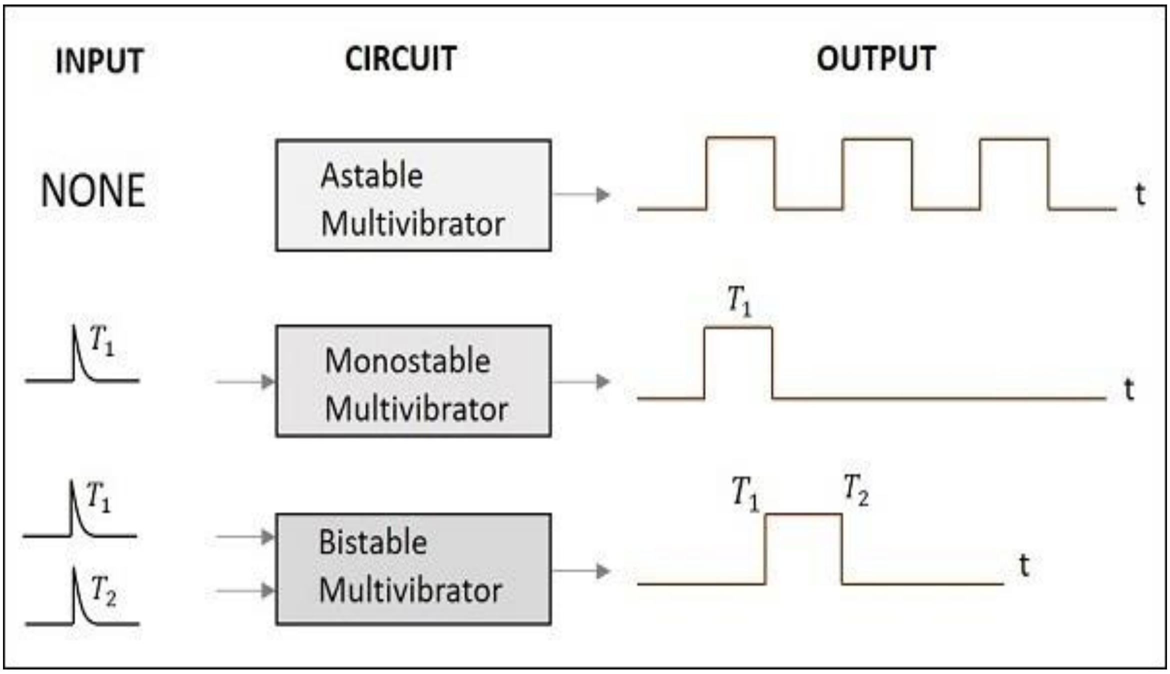
As this Multivibrator produces a single output for each trigger pulse, this is known as **One-shot Multivibrator**. This Multivibrator cannot stay in quasi-stable state for a longer period while it stays in stable state until the trigger pulse is received.

BISTABLE MULTIVIBRATOR

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

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

To get a clear idea on the above discussion, let us have a look at the following figure.



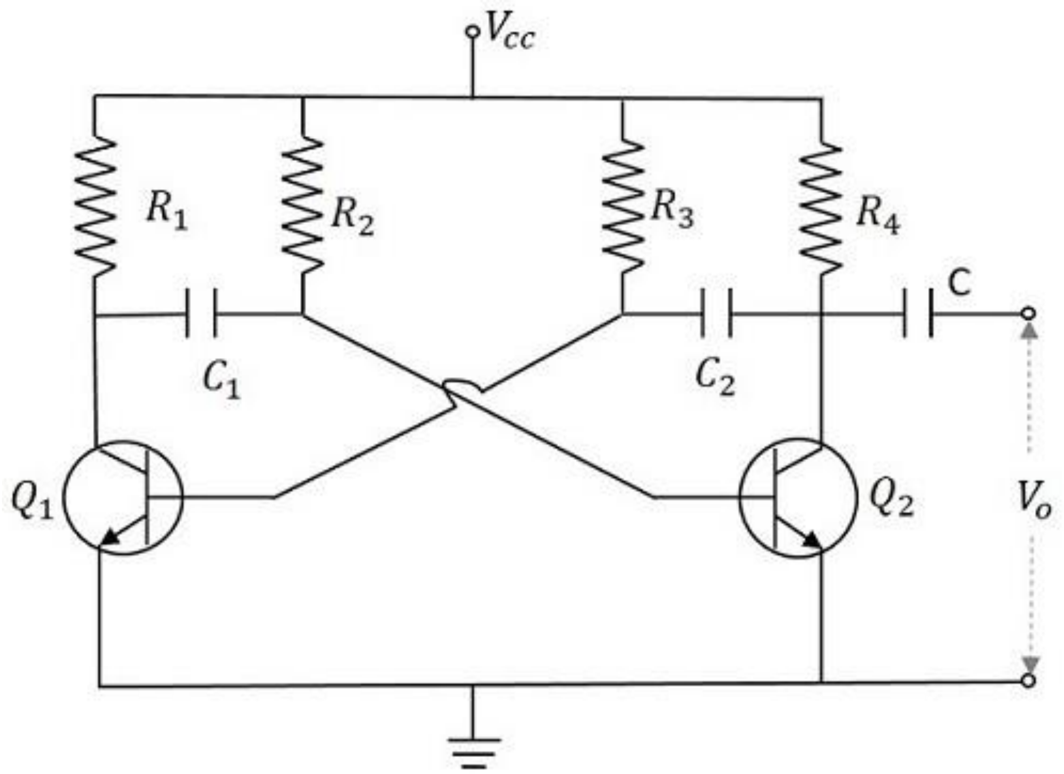
ASTABLE MULTIVIBRATOR

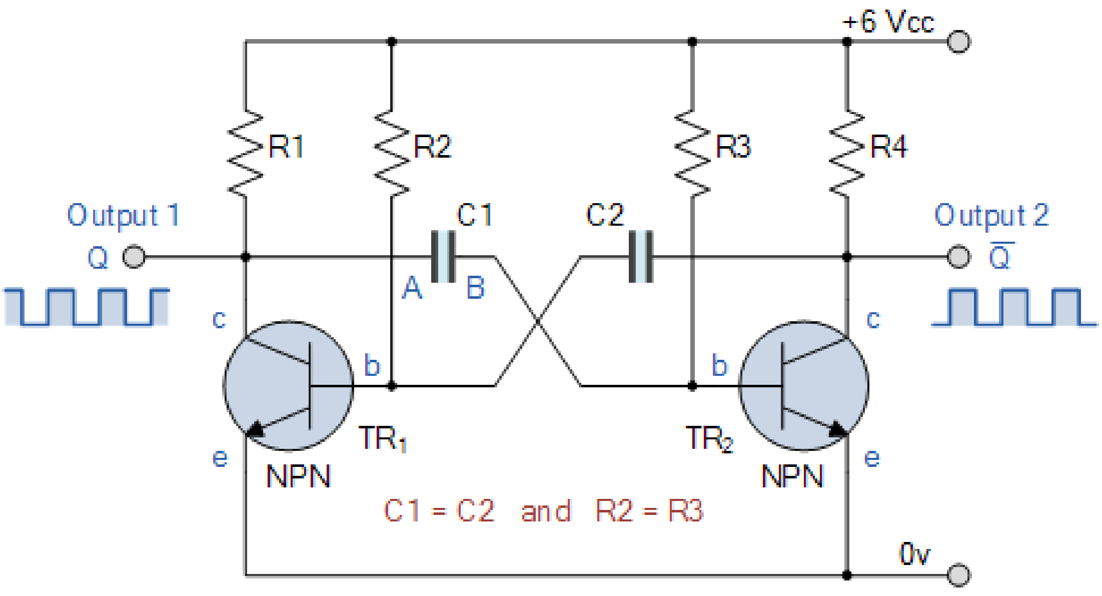
An astable multivibrator has **no stable states**. Once the Multivibrator is ON, it just changes its states on its own after a certain time period which is determined by the R_C time constants. A dc power supply or V_{cc} is given to the circuit for its operation.

CONSTRUCTION OF ASTABLE MULTIVIBRATOR

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

The following figure shows the circuit diagram for Astable Multivibrator.





(Practical circuit for Astable Multivibrator)

OPERATION OF ASTABLE MULTIVIBRATOR

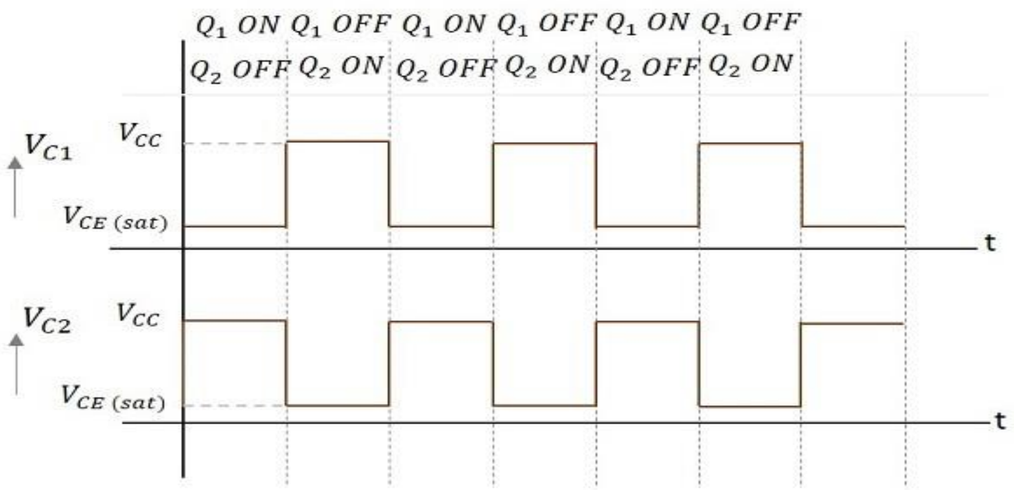
When V_{CC} is applied, the collector current of the transistors increases. As the collector current depends upon the base current and no transistor characteristics are alike, one of the two transistors say Q_1 has its collector current increase and thus conducts. The collector of Q_1 is applied to the base of Q_2 through C_1. This connection lets the increased negative voltage at the collector of Q_1 to get applied at the base of Q_2 and its collector current decreases. This continuous action makes the collector current of Q_2 to decrease further. This current when applied to the base of Q_1 makes it more negative and with the cumulative actions Q_1 gets into saturation and Q_2 to cut off. Thus, the output voltage of Q_1 will be $V_{CE\ (sat)}$ and Q_2 will be equal to V_{CC}.

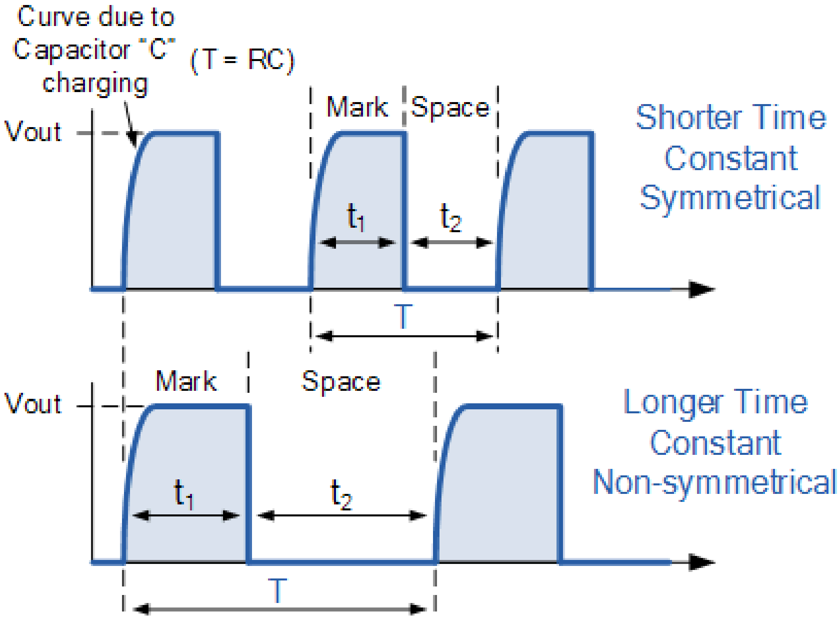
The capacitor C_1 charges through R_1 and when the voltage across C_1 reaches 0.7v, this is enough to turn the transistor Q_2 to saturation. As this voltage is applied to the base of Q_2, it gets into saturation, decreasing its collector current. This reduction of voltage at point B is applied to the base of transistor Q_1 through C_2 which makes the Q_1 reverse bias. A series of these actions turn the transistor Q_1 to cut off and transistor Q_2 to saturation. Now point A has the potential V_{CC}. The capacitor C_2 charges through R_2. The voltage across this capacitor C_2 when gets to 0.7v, turns on the transistor Q_1 to saturation.

Hence the output voltage and the output waveform are formed by the alternate switching of the transistors Q_1 and Q_2. The time period of these ON/OFF states depends upon the values of biasing resistors and capacitors used, i.e., on the R_C values used. As both the transistors are operated alternately, the output is a square waveform, with the peak amplitude of V_{CC}.

WAVEFORMS

The output waveforms at the collectors of Q_1 and Q_2 are shown in the following figures.





(Practical Waveform)

FREQUENCY OF OSCILLATIONS

The ON time of transistor Q_1 or the OFF time of transistor Q_2 is given by

$$t_1 = 0.69R_1C_1$$

Similarly, the OFF time of transistor Q_1 or ON time of transistor Q_2 is given by

$$t_2 = 0.69R_2C_2$$

Hence, total time period of square wave

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

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

$$f = \frac{1}{T} = \frac{1}{1.38RC}$$

ADVANTAGES

The advantages of using an astable multivibrator are as follows −

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

DISADVANTAGES

The drawbacks of using an astable multivibrator are as follows −

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

APPLICATIONS

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

MONOSTABLE MULTIVIBRATOR

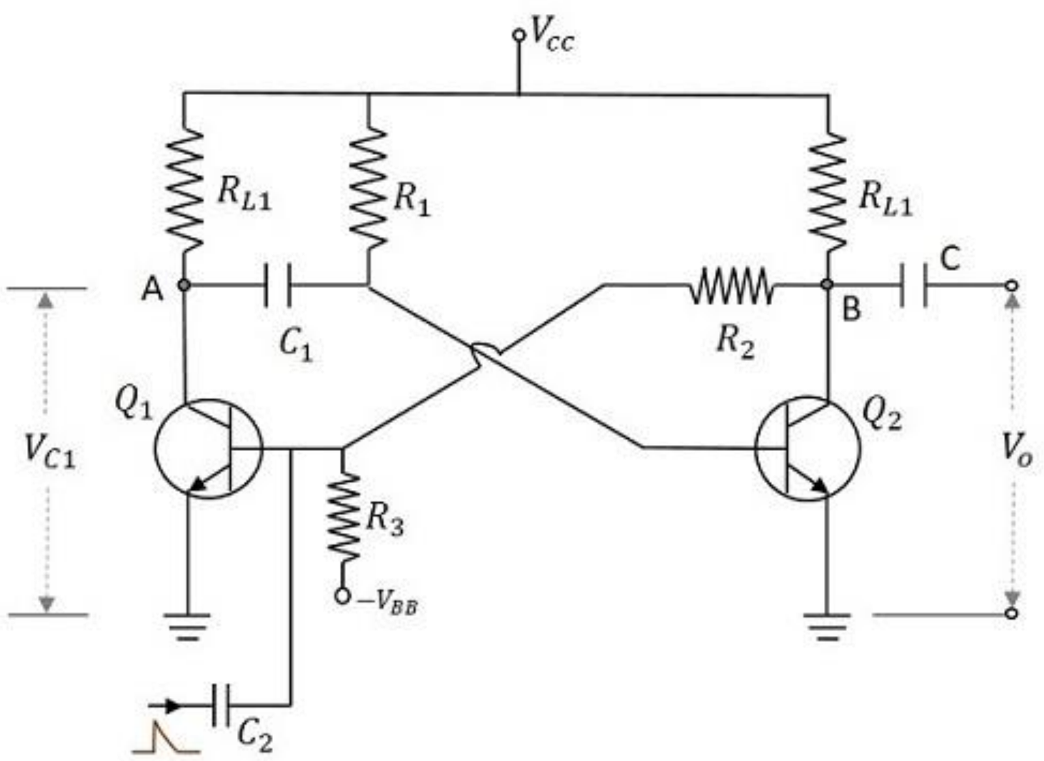
A monostable multivibrator, as the name implies, has only **one stable state**. When the transistor conducts, the other remains in non-conducting state. A stable state is such a state where the transistor remains without being altered, unless disturbed by some external trigger pulse. As Monostable works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

CONSTRUCTION OF MONOSTABLE MULTIVIBRATOR

Two transistors Q_1 and Q_2 are connected in feedback to one another. The collector of transistor Q_1 is connected to the base of transistor Q_2 through the capacitor C_1. The base Q_1 is connected to the collector of Q_2 through the resistor R_2 and capacitor C. Another dc supply voltage –V_{BB} is given to the base of transistor Q_1 through the resistor R_3. The trigger pulse is given to the base of Q_1 through the capacitor C_2 to change its state. R_{L1} and R_{L2} are the load resistors of Q_1 and Q_2.

One of the transistors, when gets into a stable state, an external trigger pulse is given to change its state. After changing its state, the transistor remains in this quasi-stable state or Meta-stable state for a specific time period, which is determined by the values of RC time constants and gets back to the previous stable state.

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



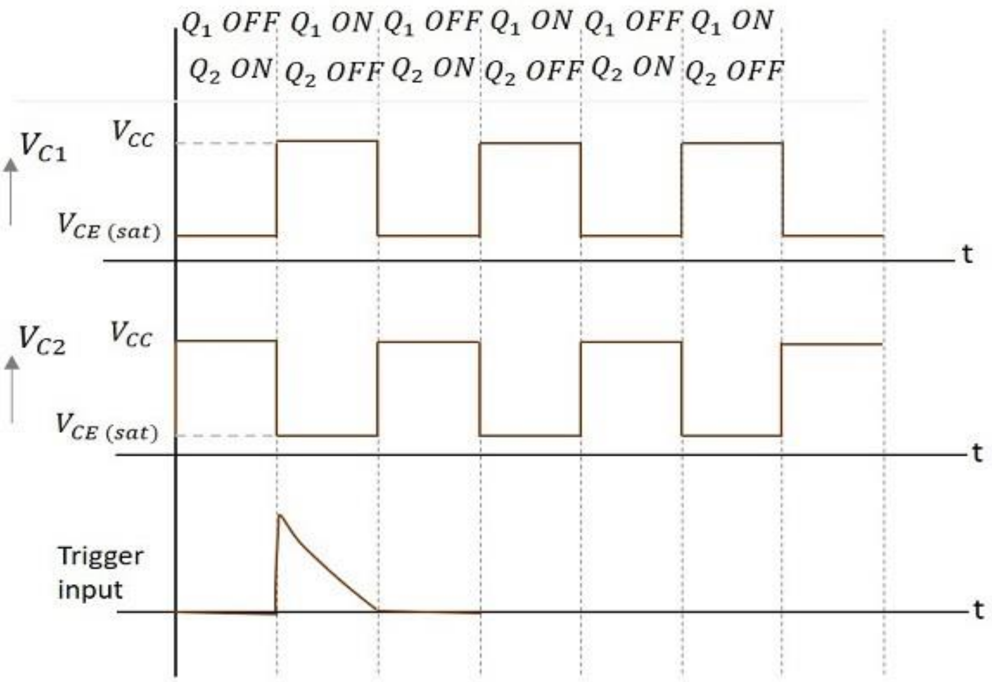
OPERATION OF MONOSTABLE MULTIVIBRATOR

Firstly, when the circuit is switched ON, transistor Q_1 will be in OFF state and Q_2 will be in ON state. This is the stable state. As Q_1 is OFF, the collector voltage will be V_{CC} at point A and hence C_1 gets charged. A positive trigger pulse applied at the base of the transistor Q_1 turns the transistor ON. This decreases the collector voltage, which turns OFF the transistor Q_2. The capacitor C_1 starts discharging at this point of time. As the positive voltage from the collector of transistor Q_2 gets applied to transistor Q_1, it remains in ON state. This is the quasi-stable state or Meta-stable state.

The transistor Q_2 remains in OFF state, until the capacitor C_1 discharges completely. After this, the transistor Q_2 turns ON with the voltage applied through the capacitor discharge. This turn ON the transistor Q_1, which is the previous stable state.

OUTPUT WAVEFORMS

The output waveforms at the collectors of Q_1 and Q_2 along with the trigger input given at the base of Q_1 are shown in the following figures.



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

$$T = 0.69\ R_1\ C_1$$

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

ADVANTAGES

The advantages of Monostable Multivibrator are as follows −

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

DISADVANTAGES

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

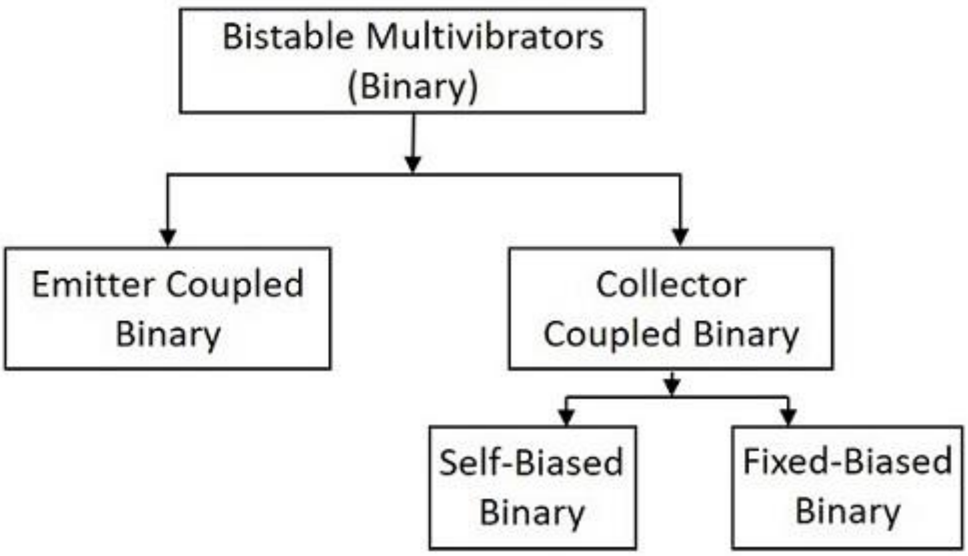
APPLICATIONS

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

BISTABLE MULTIVIBRATOR

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

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

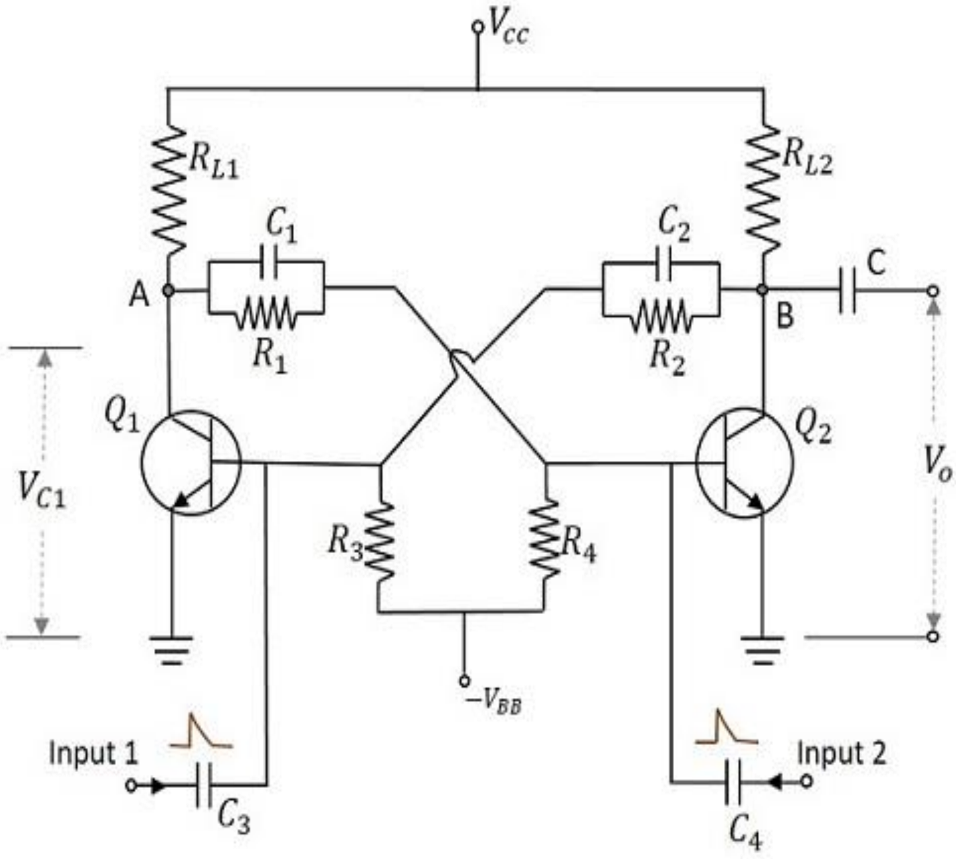


CONSTRUCTION OF BISTABLE MULTIVIBRATOR

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

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

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



OPERATION OF BISTABLE MULTIVIBRATOR

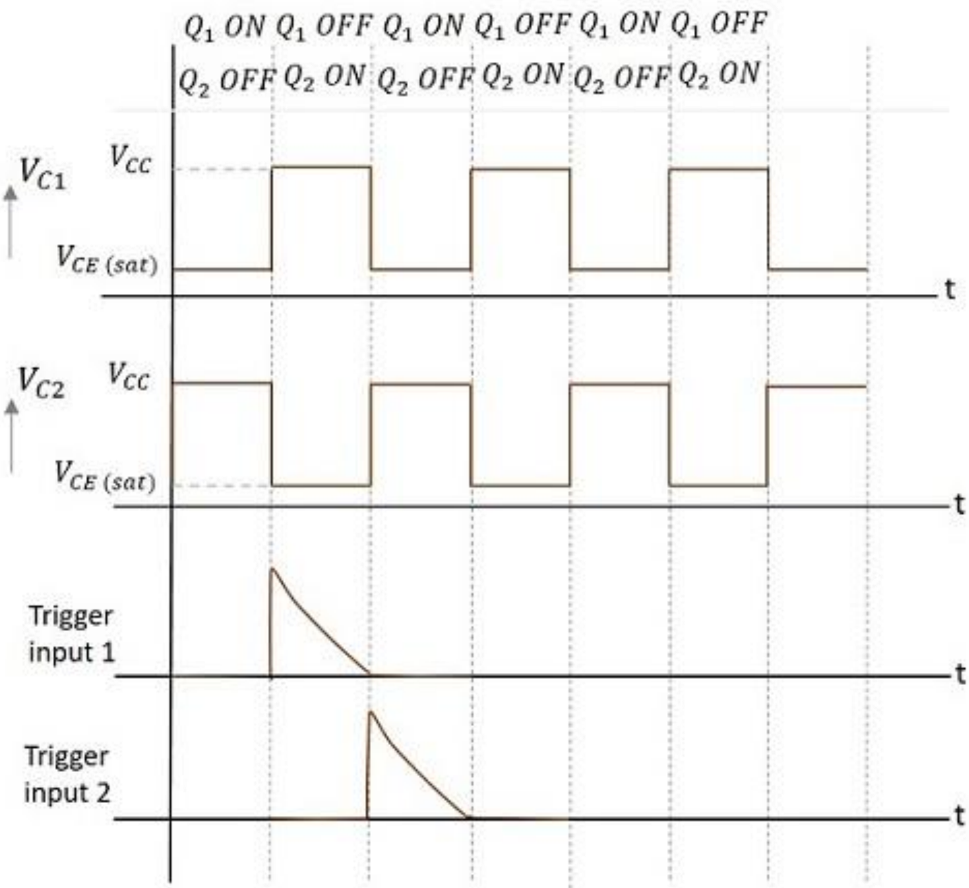
When the circuit is switched ON, due to some circuit imbalances as in Astable, one of the transistors, say Q_1 gets switched ON, while the transistor Q_2 gets switched OFF. This is a stable state of the Bistable Multivibrator.

By applying a negative trigger at the base of transistor Q_1 or by applying a positive trigger pulse at the base of transistor Q_2, this stable state is unaltered. So, let us understand this by considering a negative pulse at the base of transistor Q_1. As a result, the collector voltage increases, which forward biases the transistor Q_2. The collector current of Q_2 as applied at the base of Q_1, reverse biases Q_1 and this cumulative action, makes the transistor Q_1 OFF and transistor Q_2 ON. This is another stable state of the Multivibrator.

Now, if this stable state has to be changed again, then either a negative trigger pulse at transistor Q_2 or a positive trigger pulse at transistor Q_1 is applied.

OUTPUT WAVEFORMS

The output waveforms at the collectors of Q_1 and Q_2 along with the trigger inputs given at the bases of Q_W and Q_2 are shown in the following figures.



Advantages

The advantages of using a Bistable Multivibrator are as follows −

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

Disadvantages

The drawbacks of a Bistable Multivibrator are as follows −

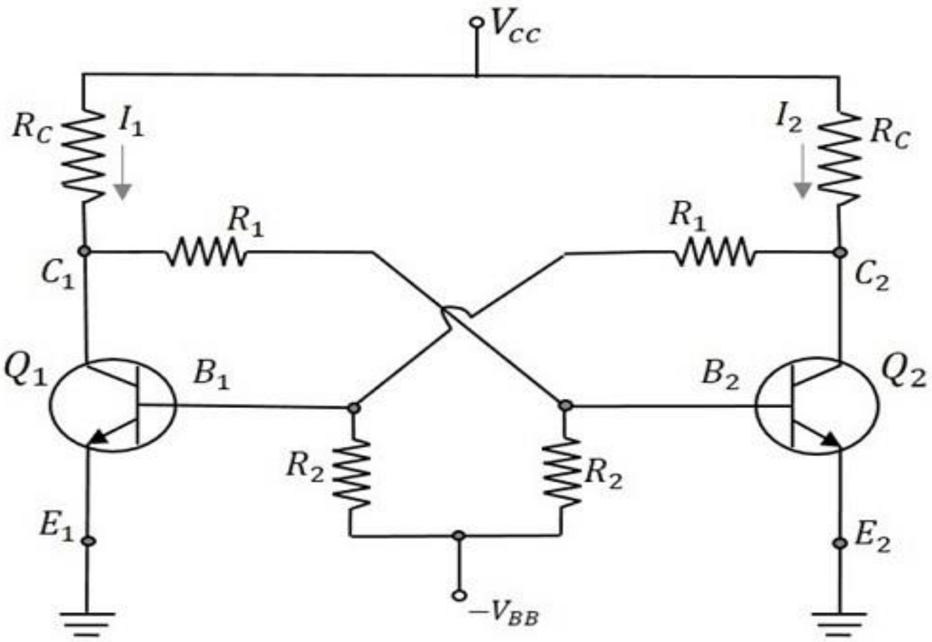
- Two kinds of trigger pulses are required.
- A bit costlier than other Multivibrators.

Applications

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

Fixed-Bias Binary

A fixed-bias binary circuit is similar to an Astable Multivibrator but with a simple SPDT switch. Two transistors are connected in feedback with two resistors, having one collector connected to the base of the other. The figure below shows the circuit diagram of a fixed-bias binary.



To understand the operation, let us consider the switch to be in position 1. Now the transistor Q_1 will be OFF as the base is grounded. The collector voltage at the output terminal V_{O1} will be equal to V_{CC} which turns the transistor Q_2 ON. The output at the terminal V_{O2} goes LOW. This is a stable state which can be altered only by an external trigger. The change of switch to position 2, works as a trigger.

When the switch is altered, the base of transistor Q_2 is grounded turning it to OFF state. The collector voltage at V_{O2} will be equal to V_{CC} which is applied to transistor Q_1 to turn it ON. This is the other stable state. The triggering is achieved in this circuit with the help of a SPDT Switch.

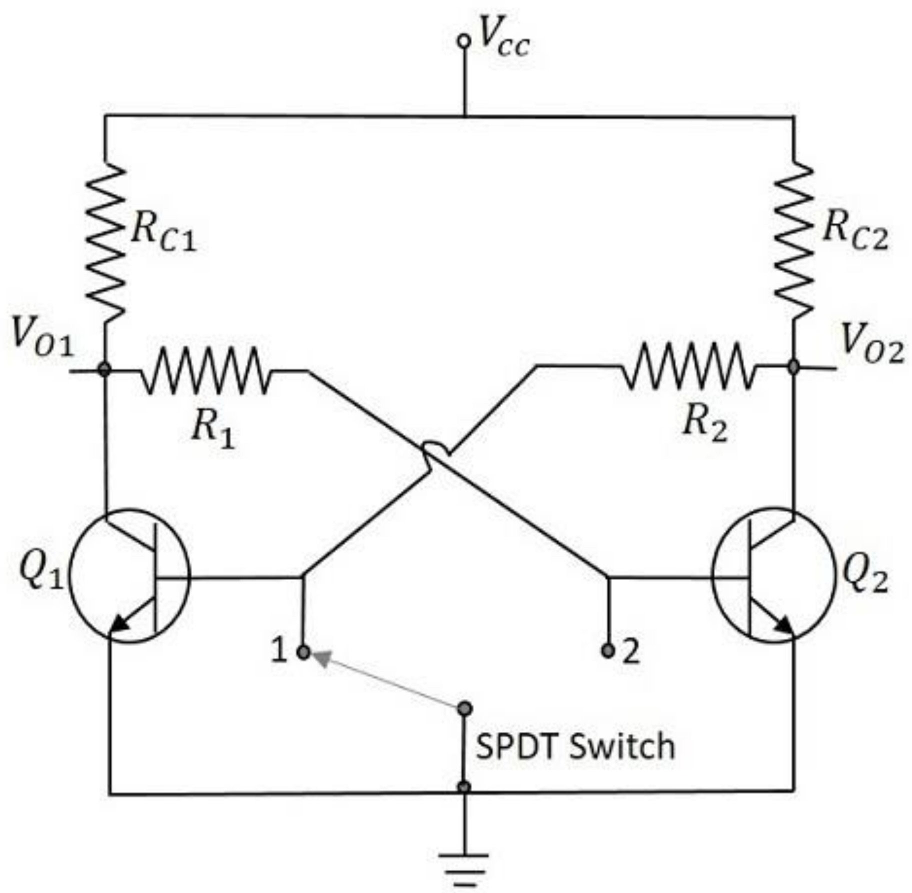
There are two main types of triggering given to the binary circuits. They are

- Symmetrical Triggering
- Asymmetrical Triggering

SCHMITT TRIGGER

Another type of binary circuit which is ought to be discussed is the **Emitter Coupled Binary** Circuit. This circuit is also called as **Schmitt Trigger** circuit. This circuit is considered as a special type of its kind for its applications.

The main difference in the construction of this circuit is that the coupling from the output C_2 of the second transistor to the base B1 of the first transistor is missing and that feedback is obtained now through the resistor R_e. This circuit is called as the **Regenerative circuit** for this has positive feedback and **no Phase inversion**. The circuit of Schmitt trigger using BJT is as shown below.



Initially we have Q_1 OFF and Q_2 ON. The voltage applied at the base of Q_2 is V_{CC} through R_{C1} and R_1. So, the output voltage will be

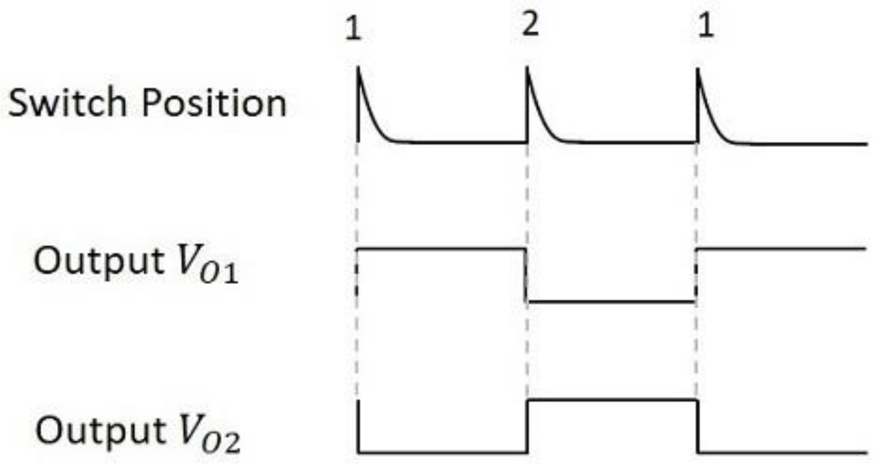
As Q_2 is ON, there will be a voltage drop across R_E, which will be $(I_{C2} + I_{B2}) R_E$. Now this voltage gets applied at the emitter of Q_1. The input voltage is increased and until Q_1 reaches cut-in voltage to turn ON, the output remains LOW. With Q_1 ON, the output will increase as Q_2 is also ON. As the input voltage continues to rise, the voltage at the points C_1 and B_2 continue to fall and E_2 continues to rise. At certain value of the input voltage, Q_2 turns OFF. The output voltage at this point will be V_{CC} and remains constant though the input voltage is further increased.

As the input voltage rises, the output remains LOW until the input voltage reaches V_1 were

The value where the input voltage equals V_1, lets the transistor Q_1 to enter into saturation, is called **UTP** (Upper Trigger Point). If the voltage is already greater than V_1, then it remains there until the input voltage reaches V_2, which is a low-level transition. Hence the value for which input voltage will be V_2 at which Q_2 gets into ON condition, is termed as **LTP** (Lower Trigger Point).

OUTPUT WAVEFORMS

The output waveforms are obtained as shown below.



The Schmitt trigger circuit works as a **Comparator** and hence compares the input voltage with two different voltage levels called as **UTP** (Upper Trigger Point) and **LTP** (Lower Trigger Point). If the input crosses this UTP, it is considered as a HIGH and if it gets below this LTP, it is taken as a LOW. The output will be a binary signal indicating 1 for HIGH and 0 for LOW. Hence an analog signal is converted into a digital signal. If the input is at intermediate value (between HIGH and LOW) then the previous value will be the output.

This concept depends upon the phenomenon called as **Hysteresis**. The transfer characteristics of electronic circuits exhibit a **loop** called as **Hysteresis**. It explains that the output values depends upon both the present and the past values of the input. This prevents unwanted frequency switching in Schmitt trigger circuits

ADVANTAGES

The advantages of Schmitt trigger circuit are

- Perfect logic levels are maintained.
- It helps avoiding Meta-stability.
- Preferred over normal comparators for its pulse conditioning.

Disadvantages

The main disadvantages of a Schmitt trigger are

- If the input is slow, the output will be slower.
- If the input is noisy, the output will be noisier.

Applications of Schmitt Trigger

Schmitt trigger circuits are used as Amplitude Comparator and Squaring Circuit. They are also used in Pulse conditioning and sharpening circuits.

These are the Multivibrator circuits using transistors. The same Multivibrators are designed using operational amplifiers and also IC 555 timer circuits, which are discussed in further tutorials.

OPERATIONAL AMPLIFIER

An **Operational Amplifier, or op-amp** for short, is fundamentally a voltage amplifying device designed to be used with external feedback components such as resistors and capacitors between its output and input terminals. These feedback components determine the resulting function or "operation" of the amplifier and by virtue of the different feedback configurations whether resistive, capacitive or both, the amplifier can perform a variety of different operations, giving rise to its name of **"Operational Amplifier".**

An *Operational Amplifier* is basically a three-terminal device which consists of two high impedance inputs. One of the inputs is called the Inverting Input, marked with a negative or "minus" sign, (−). The other input is called the Non-inverting Input, marked with a positive or "plus" sign (+).

A third terminal represents the operational amplifiers output port which can both sink and source either a voltage or a current. In a linear operational amplifier, the output signal is the amplification factor, known as the amplifiers gain (A) multiplied by the value of the input signal and depending on the nature of these input and output signals, there can be four different classifications of operational amplifier gain.

OPERATIONAL AMPLIFIER BASICS OF CLASSIFICATION

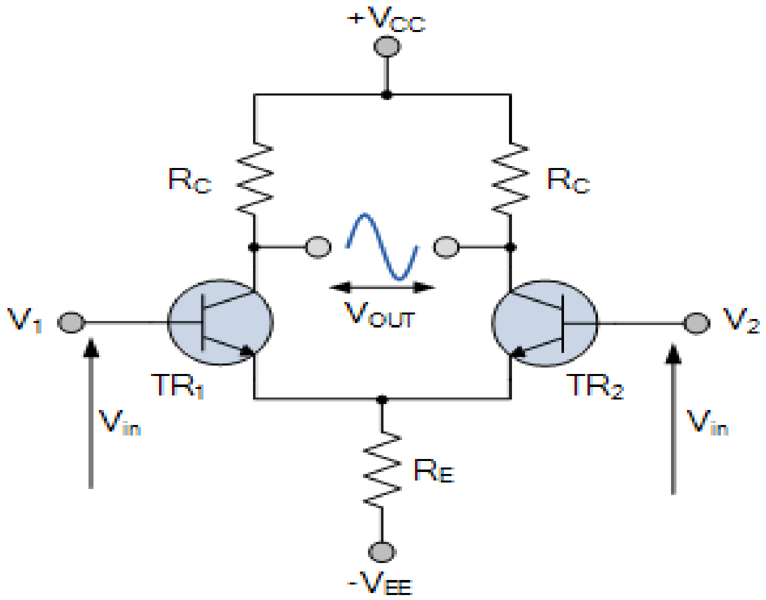
- Voltage – Voltage "in" and Voltage "out"
- Current – Current "in" and Current "out"
- Transconductance – Voltage "in" and Current "out"
- Trans resistance – Current "in" and Voltage "out"

Since most of the circuits dealing with operational amplifiers are voltage amplifiers, we will limit the tutorials in this section to voltage amplifiers only, (Vin and Vout).

The output voltage signal from an Operational Amplifier is the difference between the signals being applied to its two individual inputs. In other words, an op-amps output signal is the difference between the two input signals as the input stage of an Operational Amplifier is in fact a differential amplifier as shown below.

OPERATIONAL AMPLIFIER BASICS – THE DIFFERENTIAL AMPLIFIER

The circuit below shows a generalized form of a differential amplifier with two inputs marked V1 and V2. The two identical transistors TR1 and TR2 are both biased at the same operating point with their emitters connected together and returned to the common rail, -Vee by way of resistor Re.



Differential Amplifier

The circuit operates from a dual supply +Vcc and -Vee which ensures a constant supply. The voltage that appears at the output, Vout of the amplifier is the difference between the two input signals as the two base inputs are in **anti-phase** with each other.

So as the forward bias of transistor, TR1 is increased, the forward bias of transistor TR2 is reduced and vice versa. Then if the two transistors are perfectly matched, the current flowing through the common emitter resistor, Re will remain constant.

Like the input signal, the output signal is also balanced and since the collector voltages either swing in opposite directions (anti-phase) or in the same direction (in-phase) the output voltage signal, taken from between the two collectors is, assuming a perfectly balanced circuit the zero difference between the two collector voltages.

This is known as the ***Common Mode of Operation*** with the common mode gain of the amplifier being the output gain when the input is zero.

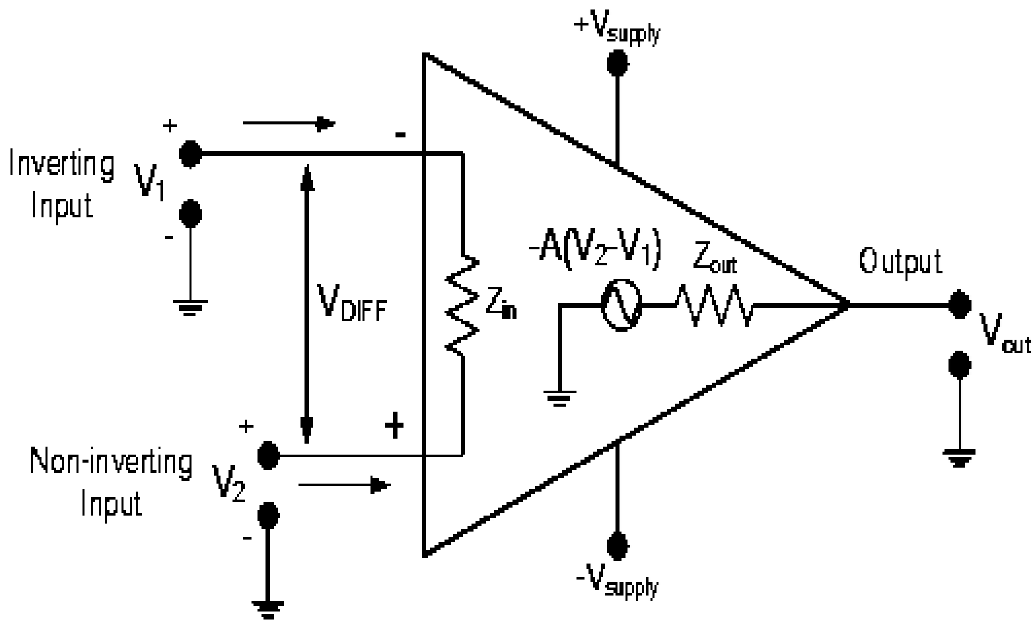
Operational Amplifiers also have one output (although there are ones with an additional differential output) of low impedance that is referenced to a common ground terminal and it should ignore any common mode signals that is, if an identical signal is applied to both the inverting and non-inverting inputs there should no change to the output.

However, in real amplifiers there is always some variation and the ratio of the change to the output voltage with regards to the change in the common mode input voltage is called the **Common Mode Rejection Ratio or CMRR** for short.

Operational Amplifiers on their own have a very high open loop DC gain and by applying some form of Negative Feedback we can produce an operational amplifier circuit that has a very precise gain characteristic that is dependant only on the feedback used. Note that the term "**open loop**" means that there are no feedback components used around the amplifier so the feedback path or loop is open.

An operational amplifier only responds to the difference between the voltages on its two input terminals, known commonly as the "***Differential Input Voltage***" and not to their common potential. Then if the same voltage potential is applied to both terminals the resultant output will be zero. An Operational Amplifiers gain is commonly known as the Open Loop Differential Gain, and is given the symbol (A_O).

EQUIVALENT CIRCUIT OF AN IDEAL OPERATIONAL AMPLIFIER



OP-AMP PARAMETER AND IDEALISED CHARACTERISTIC

- **Open Loop Gain, (Avo)**
  - **Infinite** – The main function of an operational amplifier is to amplify the input signal and the more open loop gain it has the better. Open-loop gain is the gain of the op-amp without positive or negative feedback and for such an amplifier the gain will be infinite but typical real values range from about 20,000 to 200,000.

- **Input impedance, (Z_{IN})**
  - **Infinite** – Input impedance is the ratio of input voltage to input current and is assumed to be infinite to prevent any current flowing from the source supply into the amplifiers input circuitry ($I_{IN} = 0$). Real op-amps have input leakage currents from a few pico-amps to a few milli-amps.

- **Output impedance, (Z_{OUT})**
  - **Zero** – The output impedance of the ideal operational amplifier is assumed to be zero acting as a perfect internal voltage source with no internal resistance so that it can supply as much current as necessary to the load. This internal resistance is effectively in series with the load thereby reducing the output voltage available to the load. Real op-amps have output impedances in the 100-20kΩ range.

Bandwidth, (BW)

- **Infinite** – An ideal operational amplifier has an infinite frequency response and can amplify any frequency signal from DC to the highest AC frequencies so it is therefore assumed to have an infinite bandwidth. With real op-amps, the bandwidth is limited by the Gain-Bandwidth product (GB), which is equal to the frequency where the amplifiers gain becomes unity.

- **Offset Voltage, (V_{IO})**

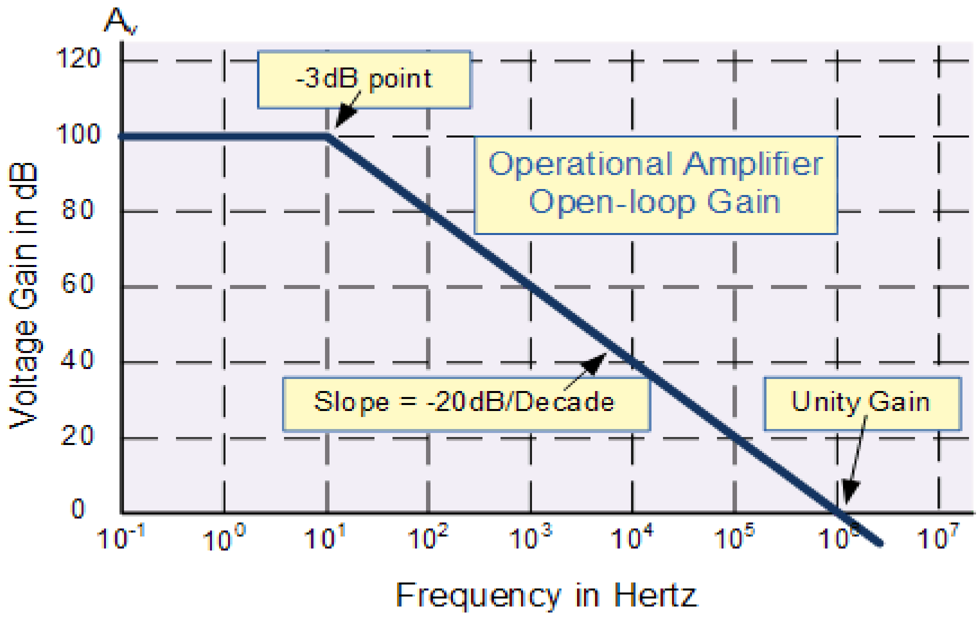
  - **Zero** – The amplifiers output will be zero when the voltage difference between the inverting and the non-inverting inputs is zero, the same or when both inputs are grounded. Real op-amps have some amount of output offset voltage.

From these **"idealized" characteristics** above, we can see that the input resistance is infinite, so no current flows into either input terminal (the "current rule") and that the differential input offset voltage is zero (the "voltage rule"). It is important to remember these two properties as they will help us understand the workings of the Operational Amplifier with regards to the analysis and design of op-amp circuits.

However, real Operational Amplifiers such as the commonly available uA741, for example do not have infinite gain or bandwidth but have a typical "Open Loop Gain" which is defined as the amplifiers output amplification without any external feedback signals connected to it.

For a typical operational amplifier, this open loop gain can be as high as 100dB at DC (zero Hz). Generally, an op-amps output gain decreases linearly as frequency increases down to "Unity Gain" or 1, at about 1MHz. This effect is shown in the following open loop gain response curve.

Operational Amplifier Basics – Open Loop Frequency Response



From this frequency response curve, we can see that the product of the gain against frequency is constant at any point along the curve. Also, that the unity gain (0dB) frequency also determines the gain of the amplifier at any point along the curve. This constant is generally known as the **Gain Bandwidth Product or GBP.** Therefore:

GBP = Gain x Bandwidth = A x BW

For example, from the graph above the gain of the amplifier at 100kHz is given as 20dB or 10, then the gain bandwidth product is calculated as:

GBP = A x BW = 10 x 100,000Hz = 1,000,000.

Similarly, the operational amplifiers gain at 1kHz = 60dB or 1000, therefore the GBP is given as:

GBP = A x BW = 1,000 x 1,000Hz = 1,000,000. The same.

The Voltage Gain (A_V) of the operational amplifier can be found using the following formula:

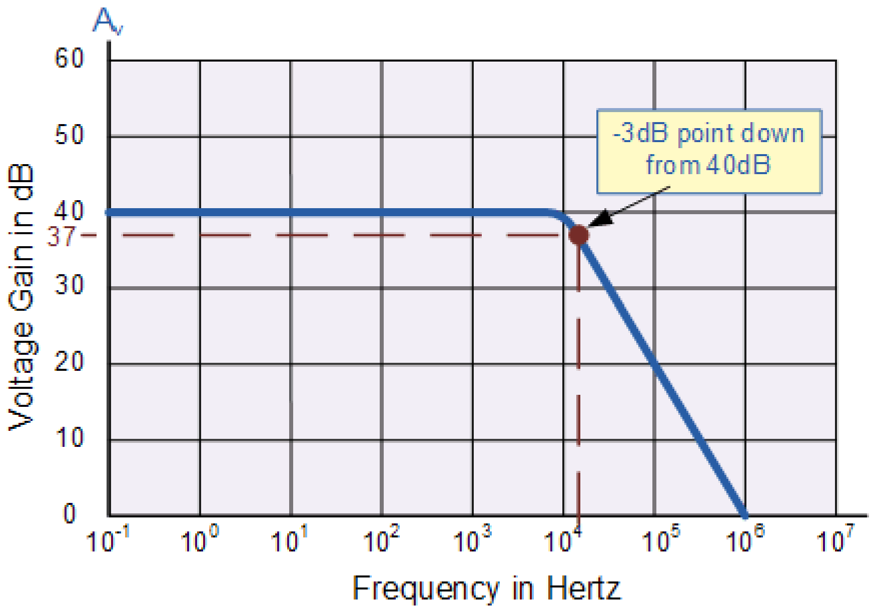
$$\textbf{Voltage Gain, (A)} = \frac{V_{out}}{V_{in}}$$

and in Decibels or (dB) is given as:

$$20\log(A) \text{ or } 20\log\frac{V_{out}}{V_{in}} \textit{ in dB}$$

An Operational Amplifiers Bandwidth

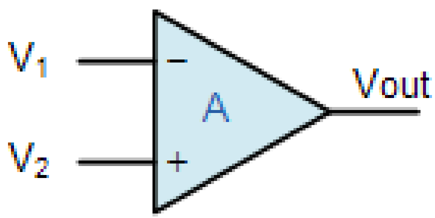
The operational amplifiers bandwidth is the frequency range over which the voltage gain of the amplifier is above 70.7% or -3dB (where 0dB is the maximum) of its maximum output value as shown below.



Here we have used the 40dB line as an example. The -3dB or 70.7% of Vmax down point from the frequency response curve is given as 37dB. Taking a line across until it intersects with the main GBP curve gives us a frequency point just above the 10kHz line at about 12 to 15kHz. We can now calculate this more accurately as we already know the GBP of the amplifier, in this particular case

Operational Amplifiers Summary

We know now that an Operational amplifier is a very high gain DC differential amplifier that uses one or more external feedback networks to control its response and characteristics. We can connect external resistors or capacitors to the op-amp in a number of different ways to form basic "building Block" circuits such as, Inverting, Non-Inverting, Voltage Follower, Summing, Differential, Integrator and Differentiator type amplifiers.

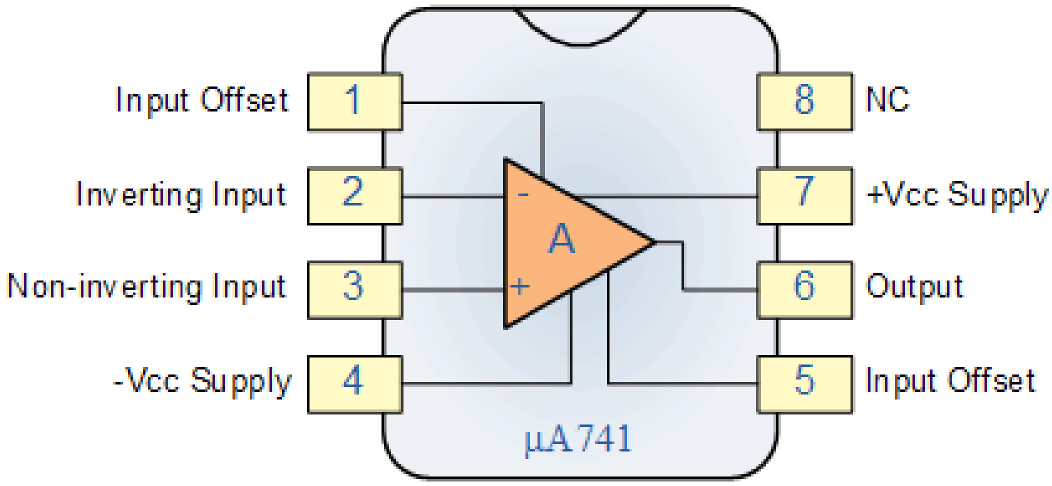


Op-amp Symbol

An **"ideal" or perfect operational amplifier** is a device with certain special characteristics such as i**nfinite open-loop gain A_O, infinite input resistance R_{IN}, zero output resistance R_{OUT}, infinite bandwidth 0 to ∞ and zero offset (the output is exactly zero when the input is zero).**

There are a very large number of operational amplifier IC's available to suit every possible application from standard bipolar, precision, high-speed, low-noise, high-voltage, etc, in either standard configuration or with internal Junction FET transistors.

Operational amplifiers are available in IC packages of either single, dual or quad op-amps within one single device. The most commonly available and used of all the operational amplifiers in basic electronic kits and projects is the industry standard μA-741 Learn more about operational amplifier basics and the different types of circuit configurations you can construct using the **μA-741.**



In the next tutorial about Operational Amplifier basics, we will use negative feedback connected around the op-amp to produce a standard closed-loop amplifier circuit called an Inverting Amplifier circuit that produces an output signal which is 180^o "out-of-phase" with the input.

Construction of Operational Amplifier

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

Characteristics of Operational Amplifier

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

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

This section discusses these characteristics in detail as given below −

Open loop voltage gain

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

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

$$A_v = V_0 / (V_1 - V_2)$$

Output offset voltage

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

Common Mode Rejection Ratio

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

Mathematically, CMRR can be represented as −

$$CMRR = Ad / Ac$$

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

Slew Rate

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

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

$$SR = \text{Maximum of } (dV_o / dt)$$

Where, V_0 is the output voltage. In general, slew rate is measured in either V / µ Sec or V/ m Sec.

Types of Operational Amplifiers

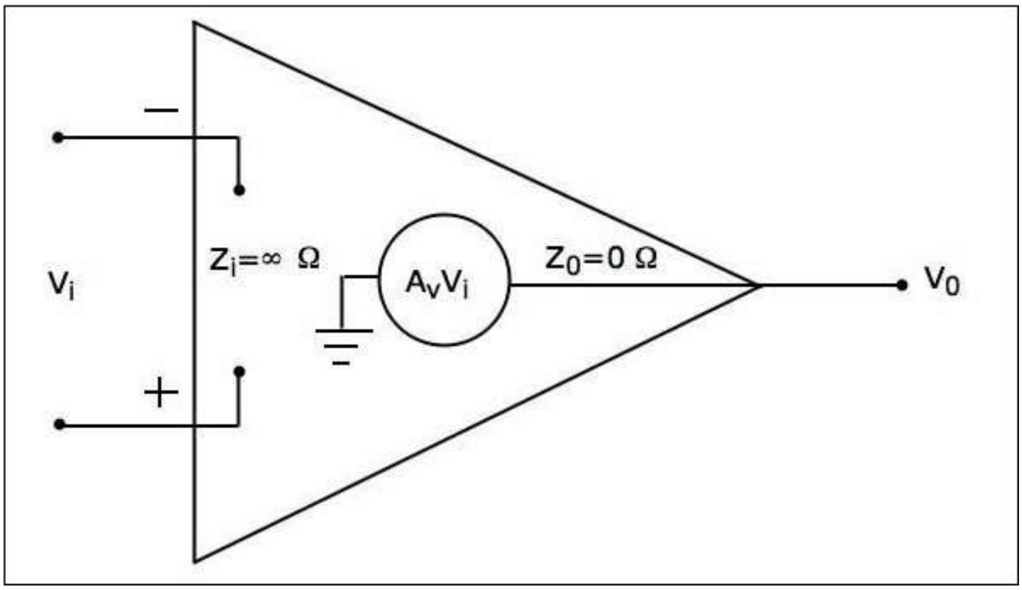
An op-amp is represented with a triangle symbol having two inputs and one output.

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

They are discussed in detail as given below –

Ideal Op-Amp

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

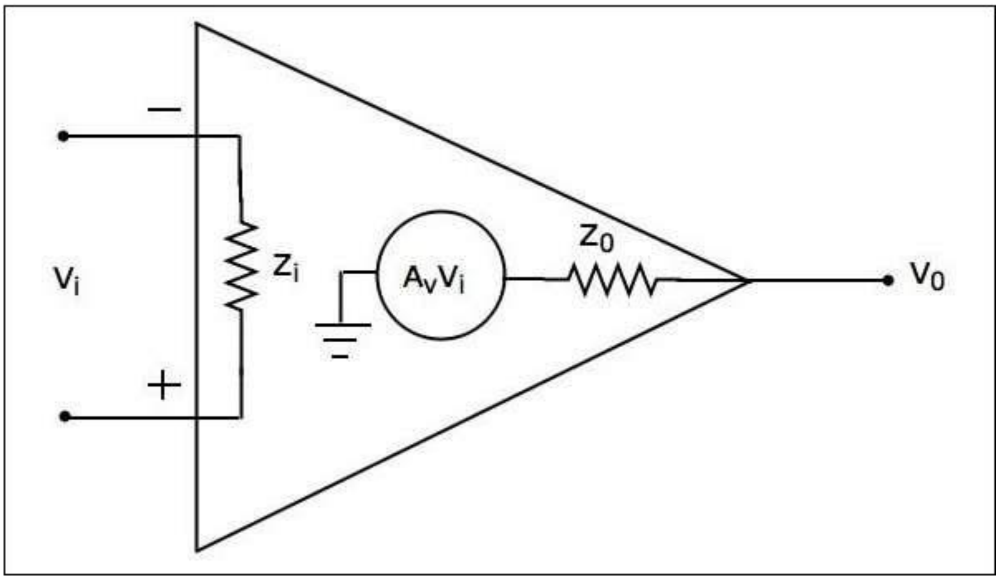


An ideal op-amp exhibits the following characteristics –

1) Input impedance $Z_i = \infty\ \Omega$
2) Output impedance $Z_o = 0\ \Omega$
3) Open loop voltage gain $A_v = \infty$
4) If (the differential) input voltage $V_i = 0$ V, then the output voltage will be $V_0 = 0$ V
5) Bandwidth is infinity. It means, an ideal op-amp will amplify the signals of any frequency without any attenuation.
6) Common Mode Rejection Ratio (CMRR) is infinity.
7) Slew Rate (SR) is infinity. It means, the ideal op-amp will produce a change in the output instantly in response to an input step voltage.

Practical Op-Amp

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



A practical op-amp exhibits the following characteristics –

a) Input impedance, Z_i in the order of Mega ohms.
b) Output impedance, Z_0 in the order of few ohms.
c) Open loop voltage gain, A_v will be high.

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

- ✓ Input impedance, Zi should be as high as possible.
- ✓ Output impedance, Z0 should be as low as possible.
- ✓ Open loop voltage gain, Av should be as high as possible.
- ✓ Output offset voltage should be as low as possible.
- ✓ The operating Bandwidth should be as high as possible.
- ✓ CMRR should be as high as possible.
- ✓ Slew rate should be as high as possible.

Note – IC 741 op-amp is the most popular and practical op-amp.

OP-AMP-APPLICATIONS

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

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

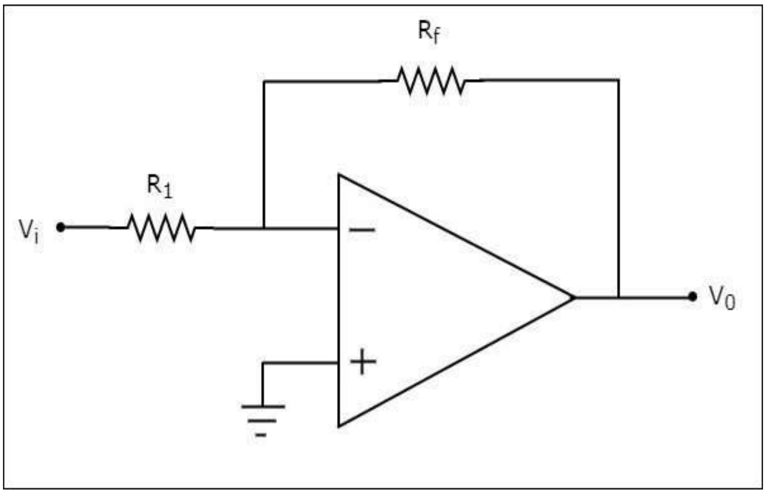
- **Inverting Amplifier**
- **Non-inverting Amplifier**
- **Voltage follower**

This article discusses these basic applications in detail.

Inverting Amplifier

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

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



Note that for an op-amp, the voltage at the inverting input terminal is equal to the voltage at its non-inverting input terminal. Physically, there is no short between those two terminals but **virtually**, they are in **short** with each other.

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

According to the **virtual short concept,** the voltage at the inverting input terminal of an op-amp will be zero volts.

The **nodal equation** at this terminal's node is as shown below −

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

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

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

$$\mathbf{=> V_0/Vi = -R_f/R_1}$$

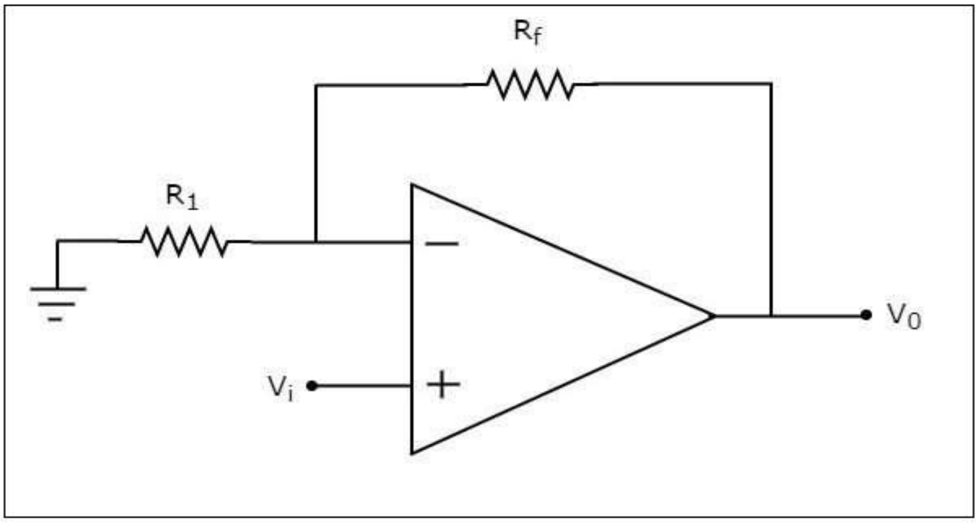
The ratio of the output voltage V_0 and the input voltage Vi is the voltage-gain or gain of the amplifier. Therefore, the gain of inverting amplifier is equal to $-R_f/R_1$.

Note that the gain of the inverting amplifier is having a negative sign. It indicates that there exists a 180^0 phase difference between the input and the output.

Non-Inverting Amplifier

A non-inverting amplifier takes the input through its non-inverting terminal, and produces its amplified version as the output. As the name suggests, this amplifier just amplifies the input, without inverting or changing the sign of the output.

The **circuit diagram of a non-inverting amplifier** is shown in the following figure −



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

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

$$=> V_1 = V_0[R_1/(R_1+R_f)\}$$

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

$$=> V_1 = V_i$$

$$=> V_0\{R_1/(R_1+R_f)\} = V_i$$

$$=> V_0/Vi = (R_1+R_f)/R_1$$

$$\mathbf{=> V_0/Vi = 1 + R_f/R_1}$$

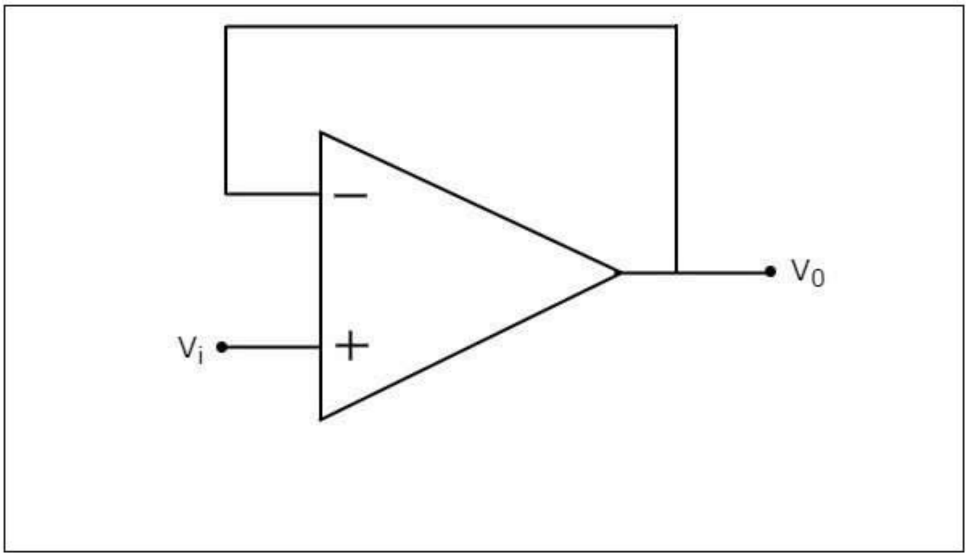
Now, the ratio of output voltage V_0 and input voltage Vi or the voltage-gain or **gain of the non-inverting amplifier** is equal to $1 + R_f/R_1$.

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

Voltage follower

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

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



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

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

$$=> V_0 = V_i$$

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

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

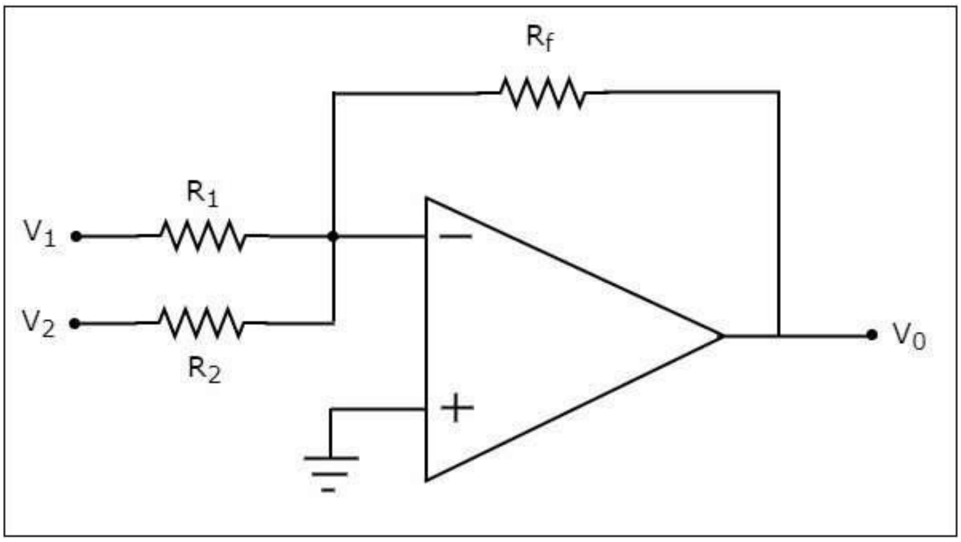
The electronic circuits, which perform arithmetic operations are called as arithmetic circuits. Using op-amps, you can build basic arithmetic circuits such as an adder and a subtractor. In this chapter, you will learn about each of them in detail.

Adder

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

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

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



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

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

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

$$(0-V_1) / R_1 + (0-V_2) / R_2 + (0-V_0) / R_f = 0$$

$$=> V_1/ R_1 - V_2/ R_2 = V_0/ R_f$$

$$=> V_0= R_f (V_1/ R_1 + V_2/ R_2)$$

If $R_f = R_1 = R_2 = R$, then the output voltage V_0will be −

$$V_0= -R (V_1/R + V_2/R)$$

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

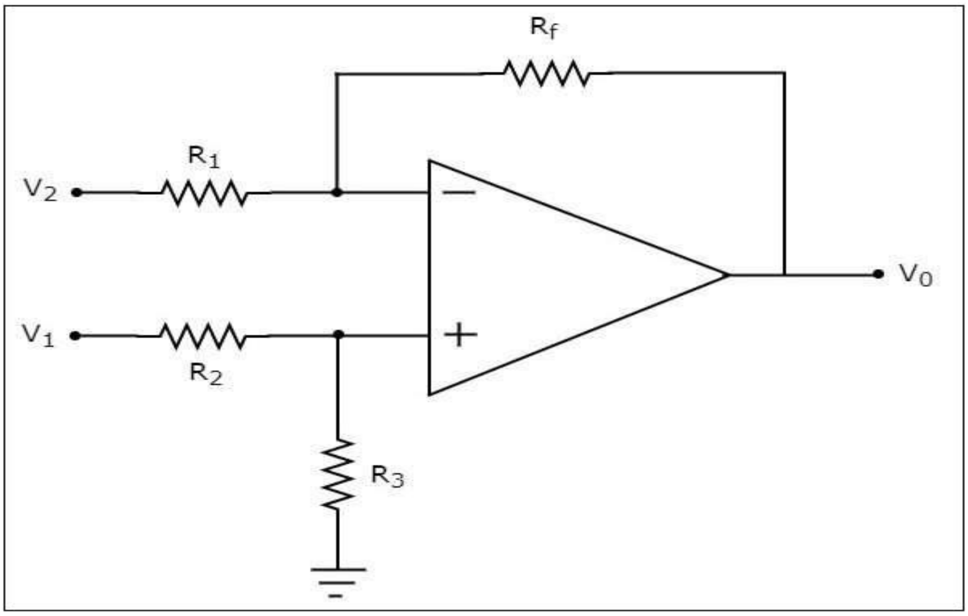
Therefore, the op-amp based adder circuit discussed above will produce the sum of the two input voltages $V_1$and V_2, as the output, when all the resistors present in the circuit are of same value. Note that the output voltage $V_0$of an adder circuit is having a **negative sign,** which indicates that there exists a 180^0 phase difference between the input and the output.

Subtractor

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

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

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

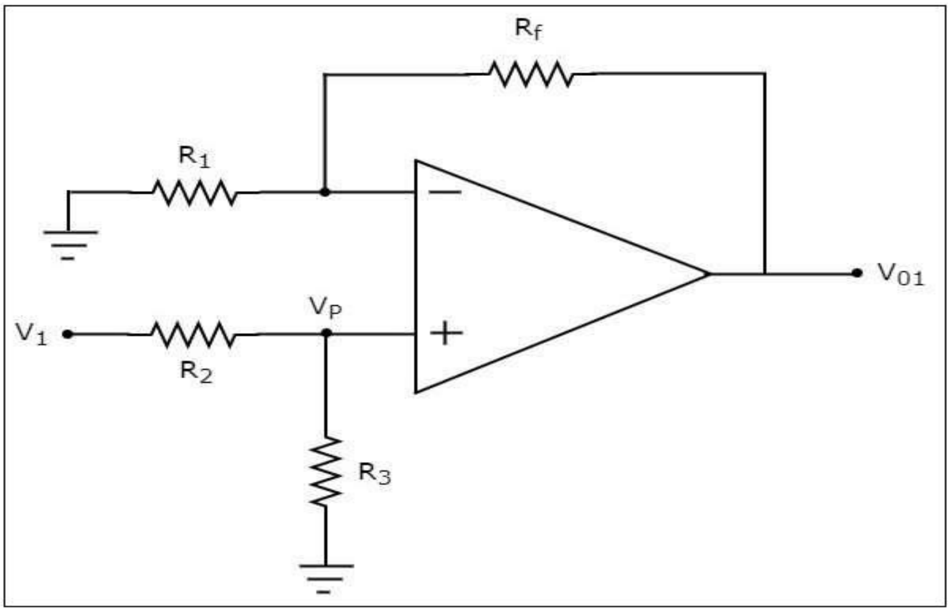


Now, let us find the expression for output voltage V_0 of the above circuit using superposition theorem using the following steps −

Step 1

Firstly, let us calculate the output voltage V_{01} by considering only V_1.

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



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

$$=> V_p = V_1\{R_3 / (R_2 + R_3)\}$$

Now, the above circuit looks like a non-inverting amplifier having input voltage Vp. Therefore, the output voltage V01 of above circuit will be

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

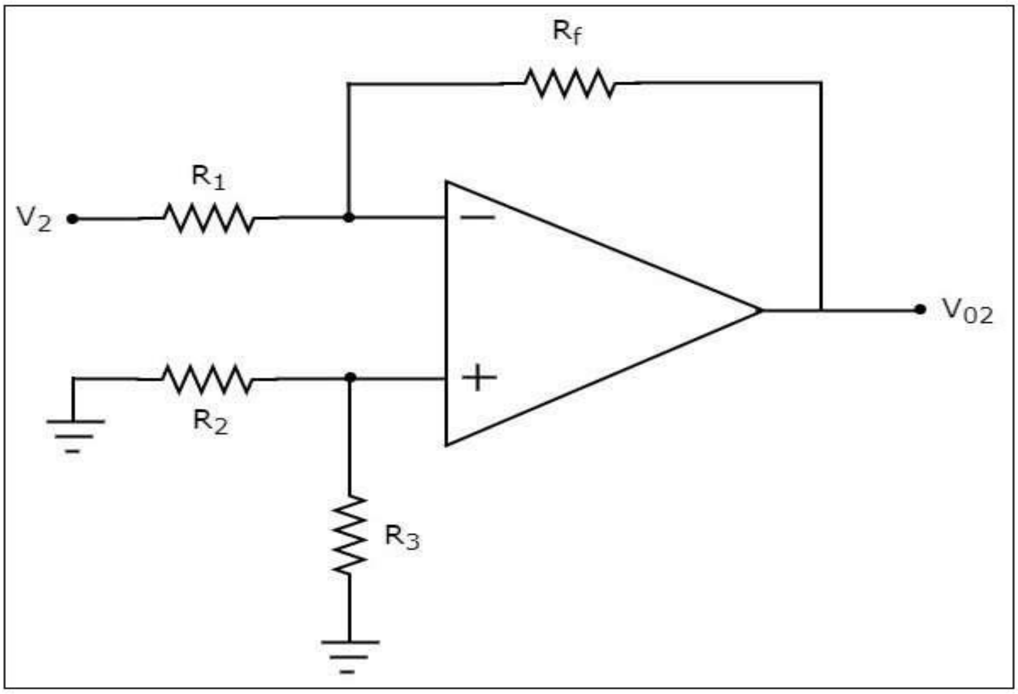
Substitute, the value of Vp in above equation, we obtain the output voltage V01 by considering only V1, as –

$$V_{01} = V_1\{R_3 / (R_2 + R_3)\} (1 + R_f / R_1)$$

Step 2

In this step, let us find the output voltage, V_{02} by considering only V_2.

Similar to that in the above step, eliminate $V_1$by making it short circuit. The **modified circuit diagram** is shown in the following figure.



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

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

Step 3

In this step, we will obtain the output voltage V0 of the subtractor circuit by **adding the output voltages** obtained in Step1 and Step2. Mathematically, it can be written as

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

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

$$V_0 = V_1\{R_3 / (R_2+ R_3)\} (1+ R_f/ R_1) + (-R_f/ R_1) V_2$$

$$=> V_0= V_1\{R_3 / (R_2+ R_3)\} (1+ R_f/ R_1) - (R_f/ R_1) V_2$$

If $R_f = R_1 = R_2 = R_3 = R$, then the output voltage V0 will be

$$V_0= V_1\{R/ (R+R)\} (1 + R/R) - (R/R) V_2$$

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

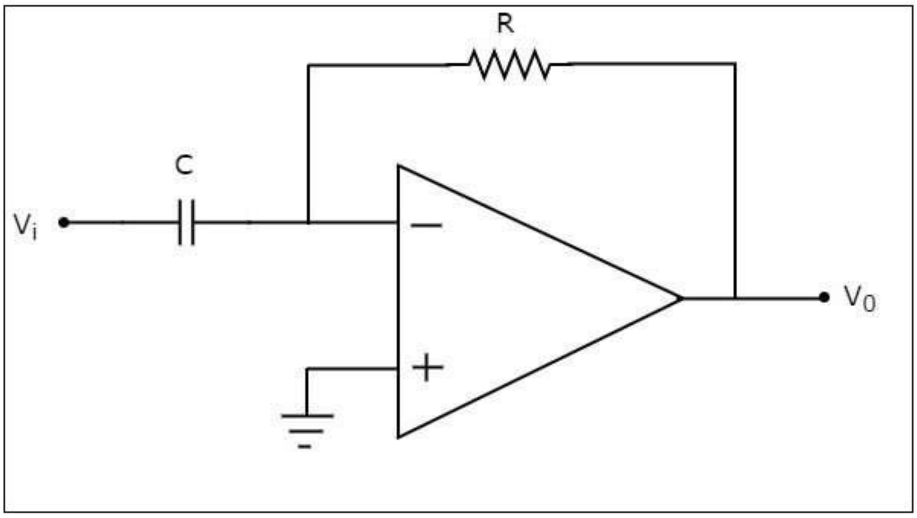
$$V_0 = V_1 - V_2$$

Thus, the op-amp based subtractor circuit discussed above will produce an output, which is the difference of two input voltages V1 and V_2, when all the resistors present in the circuit are of same value.

Differentiator

A differentiator is an electronic circuit that produces an output equal to the first derivative of its input. This section discusses about the op-amp based differentiator in detail.

An op-amp based differentiator produces an output, which is equal to the differential of input voltage that is applied to its inverting terminal.
The circuit diagram of an op-amp based differentiator is shown in the following figure –



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

According to the virtual short concept, the voltage at the inverting input terminal of op amp will be equal to the voltage present at its non-inverting input terminal. So, the voltage at the inverting input terminal of op-amp will be zero volts.

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

$$C\, d(0-Vi)/dt + (0 - V_0)/R = 0$$

$$= > -C \, dVi / dt = V_0 / R$$

$$= > V_0 = -RC \, dVi / dt$$

If RC = 1sec, then the output voltage V_0 will be –

$$\mathbf{V_0 = -dVi / dt}$$

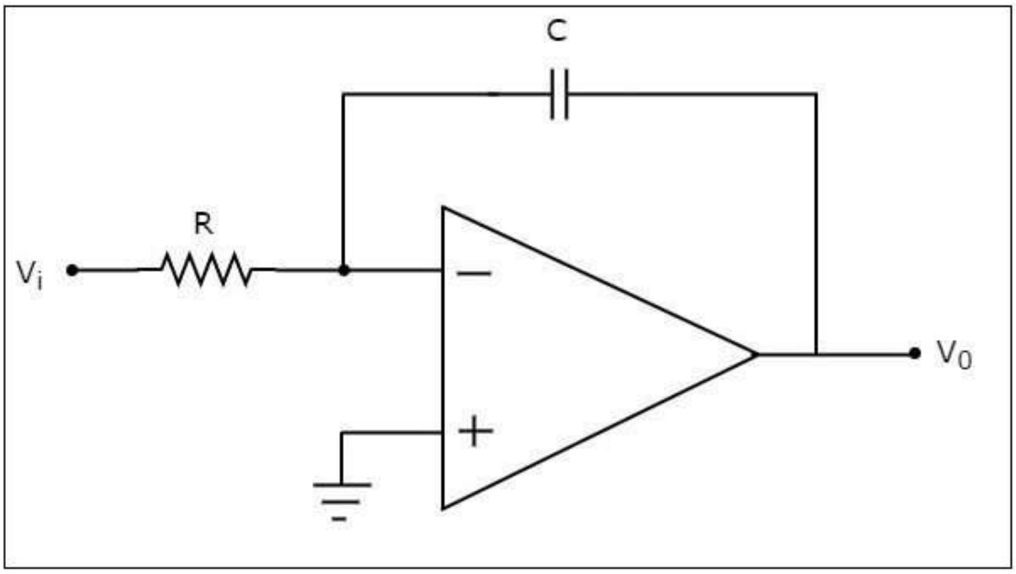
Thus, the op-amp based differentiator circuit shown above will produce an output, which is the differential of input voltage Vi, when the magnitudes of impedances of resistor and capacitor are reciprocal to each other.

Note that the output voltage V_0 is having a negative sign, which indicates that there exists a 180^0 phase difference between the input and the output.

Integrator

An integrator is an electronic circuit that produces an output that is the integration of the applied input. This section discusses about the op-amp based integrator.

An op-amp based integrator produces an output, which is an integral of the input voltage applied to its inverting terminal. The circuit diagram of an op-amp based integrator is shown in the following figure –



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

According to virtual short concept, the voltage at the inverting input terminal of op-amp will be equal to the voltage present at its non-inverting input terminal. So, the voltage at the inverting input terminal of op-amp will be zero volts.

The nodal equation at the inverting input terminal is −

$$(0-V_i) / R + C\, d(0-V_0)/dt = 0$$

$$=> -V_i / R = C\, dV_0/dt$$

$$=> d V_0\, dt = -V_i / RC$$

$$=> d V_0 = (-V_i / RC)\, dt$$

Integrating both sides of the equation shown above, we get −

$$\int d V_0 = \int (-V_i / RC)\, dt$$

$$=> V_0 = -1/RC \int V_t\, dt$$

If RC=1sec, then the output voltage, V_0 will be −

$$\mathbf{V_0 = -\int V_i dt}$$

So, the op-amp based integrator circuit discussed above will produce an output, which is the integral of input voltage Vi, when the magnitude of impedances of resistor and capacitor are reciprocal to each other.

Note − The output voltage, V_0 is having a negative sign, which indicates that there exists 180^0 phase difference between the input and the output.

OSCILLATORS

AnOscillator circuit is a complete set of all the parts of circuit which helps to produce the oscillations. These oscillations should sustain and should be Undamped as just discussed before. Let us try to analyze a practical Oscillator circuit to have a better understanding on how an Oscillator circuit works.

CLASSIFICATION OF OSCILLATORS

Electronic oscillators are classified mainly into the following two categories −

- **Sinusoidal Oscillators −** The oscillators that produce an output having a sine waveform are called **sinusoidal** or **harmonic oscillators**. Such oscillators can provide output at frequencies ranging from 20 Hz to 1 GHz.
- **Non-sinusoidal Oscillators −** The oscillators that produce an output having a square, rectangular or saw-tooth waveform are called **non-sinusoidal** or **relaxation oscillators**. Such oscillators can provide output at frequencies ranging from 0 Hz to 20 MHz

SINUSOIDAL OSCILLATORS

Sinusoidal oscillators can be classified in the following categories −

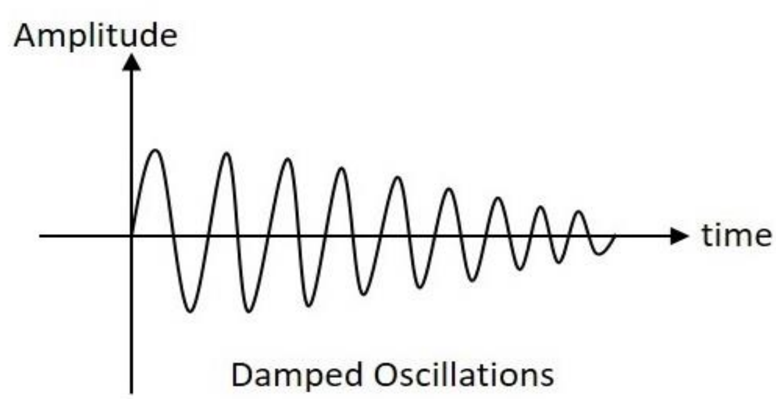
- **Tuned Circuit Oscillators −** These oscillators use a tuned-circuit consisting of inductors (L) and capacitors (C) and are used to generate high-frequency signals. Thus, they are also known as radio frequency R.F. oscillators. Such oscillators are Hartley, Colpitts, Clapp-oscillators etc.
- **RC Oscillators −** There oscillators use resistors and capacitors and are used to generate low or audio-frequency signals. Thus, they are also known as audio-frequency (A.F.) oscillators. Such oscillators are Phase – shift and Wein-bridge oscillators.
- **Crystal Oscillators −** These oscillators use quartz crystals and are used to generate highly stabilized output signal with frequencies up to 10 MHz. The Piezo oscillator is an example of a crystal oscillator.
- **Negative-resistance Oscillator** − These oscillators use negative-resistance characteristic of the devices such as tunnel devices. A tuned diode oscillator is an example of a negative-resistance oscillator.

NATURE OF SINUSOIDAL OSCILLATIONS

The nature of oscillations in a sinusoidal wave are generally of two types. They are **damped** and **undamped oscillations**.

DAMPED OSCILLATIONS

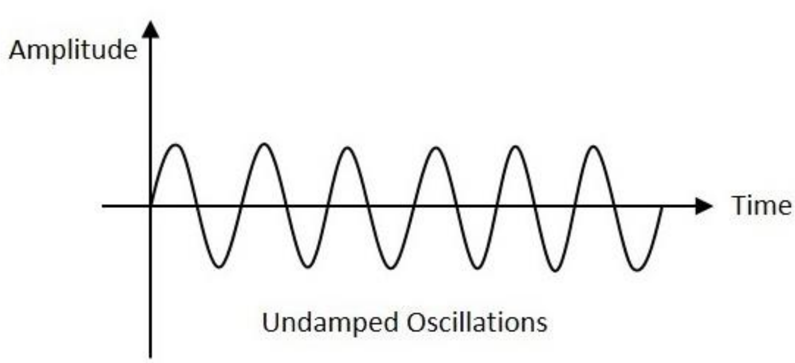
The electrical oscillations whose amplitude goes on decreasing with time are called as **Damped Oscillations**. The frequency of the damped oscillations may remain constant depending upon the circuit parameters.



Damped oscillations are generally produced by the oscillatory circuits that produce power losses and doesn't compensate if required.

UNDAMPED OSCILLATIONS

The electrical oscillations whose amplitude remains constant with time are called as **Undamped Oscillations**. The frequency of the Undamped oscillations remains constant.



Undamped oscillations are generally produced by the oscillatory circuits that produce no power losses and follow compensation techniques if any power losses occur.

Sinusoidal Oscillators - Basic Concepts

An amplifier with positive feedback produces its output to be in phase with the input and increases the strength of the signal. Positive feedback is also called as **degenerative feedback** or **direct feedback**. This kind of feedback makes a feedback amplifier, an oscillator.

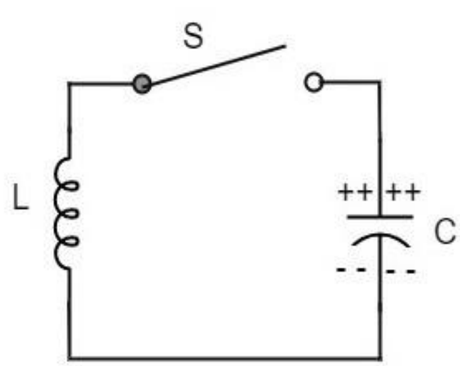
The use of positive feedback results in a feedback amplifier having closed-loop gain greater than the open-loop gain. It results in **instability** and operates as an oscillatory circuit. An oscillatory circuit provides a constantly varying amplified output signal of any desired frequency.

The Oscillatory Circuit

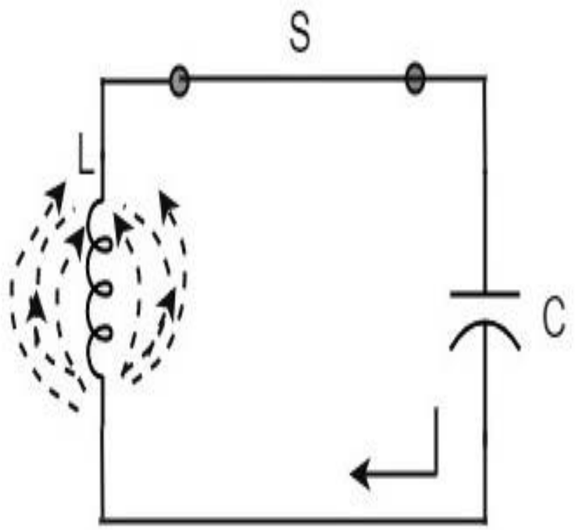
An oscillatory circuit produces electrical oscillations of a desired frequency. They are also known as **tank circuits**.

A simple tank circuit comprises of an inductor L and a capacitor C both of which together determine the oscillatory frequency of the circuit.

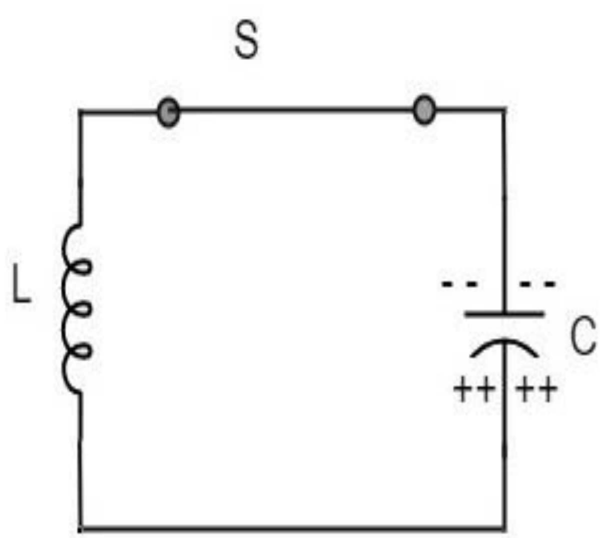
To understand the concept of oscillatory circuit, let us consider the following circuit. The capacitor in this circuit is already charged using a dc source. In this situation, the upper plate of the capacitor has excess of electrons whereas the lower plate has deficit of electrons. The capacitor holds some electrostatic energy and there is a voltage across the capacitor.



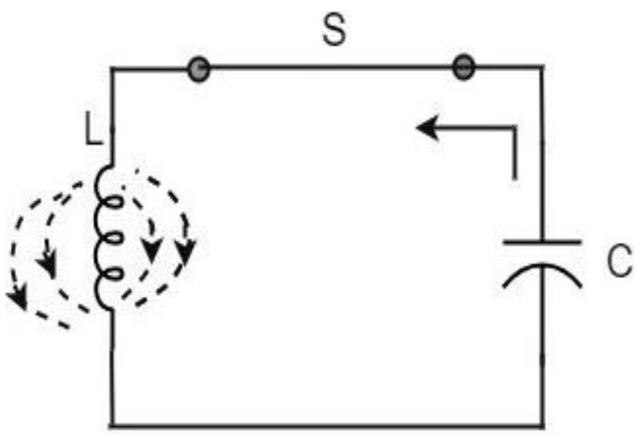
When the switch **S** is closed, the capacitor discharges and the current flows through the inductor. Due to the inductive effect, the current builds up slowly towards a maximum value. Once the capacitor discharges completely, the magnetic field around the coil is maximum.



Now, let us move on to the next stage. Once the capacitor is discharged completely, the magnetic field begins to collapse and produces a counter EMF according to Lenz’s law. The capacitor is now charged with positive charge on the upper plate and negative charge on the lower plate.



Once the capacitor is fully charged, it starts to discharge to build up a magnetic field around the coil, as shown in the following circuit diagram.



This continuation of charging and discharging results in alternating motion of electrons or an **oscillatory current**. The interchange of energy between L and C produces continuous **oscillations**.

In an ideal circuit, where there are no losses, the oscillations would continue indefinitely. In a practical tank circuit, there occur losses such as **resistive** and **radiation losses** in the coil and **dielectric losses** in the capacitor. These losses result in damped oscillations.

FREQUENCY OF OSCILLATIONS

The frequency of the oscillations produced by the tank circuit are determined by the components of the tank circuit, **the L** and **the C**. The actual frequency of oscillations is the **resonant frequency** (or natural frequency) of the tank circuit which is given by

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

CAPACITANCE OF THE CAPACITOR

The frequency of oscillation f_o is inversely proportional to the square root of the capacitance of a capacitor. So, if the value of the capacitor used is large, the charge and discharge time periods will be large. Hence the frequency will be lower.

Mathematically, the frequency,

$$f_0 \propto 1/\sqrt{C}$$

SELF-INDUCTANCE OF THE COIL

The frequency of the oscillation f_o is proportional to the square root of the self-inductance of the coil. If the value of the inductance is large, the opposition to change of current flow is greater and hence the time required to complete each cycle will be longer, which means time period will be longer and frequency will be lower.

Mathematically, the frequency,

$$f_0 \propto 1/\sqrt{L}$$

Combining both the above equations,

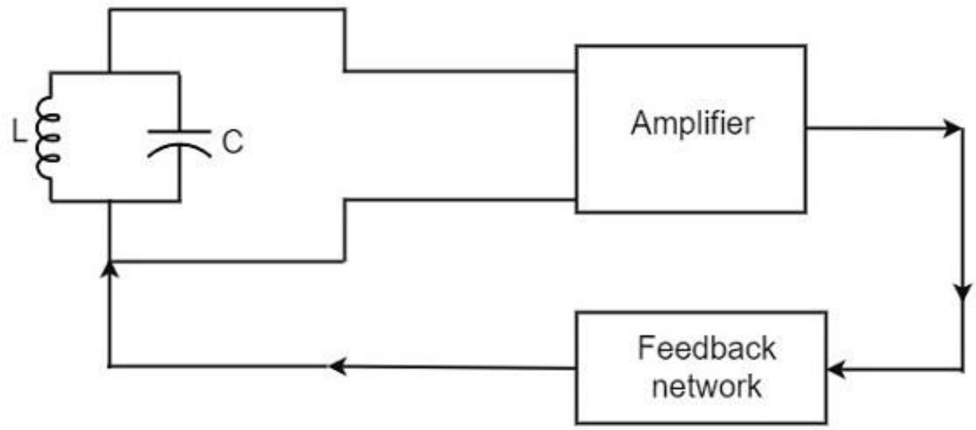
$$f_0 \propto 1/\sqrt{LC}$$

$$f_0 = 1/2\pi\sqrt{LC}$$

The above equation, though indicates the output frequency, matches the **natural frequency** or **resonance frequency** of the tank circuit.

PRACTICAL OSCILLATOR CIRCUIT

A Practical Oscillator circuit consists of a tank circuit, a transistor amplifier, and a feedback circuit. The following circuit diagram shows the arrangement of a practical oscillator.



Let us now discuss the parts of this practical oscillator circuit.

- **Tank Circuit –** The tank circuit consists of an inductance L connected in parallel with capacitor **C**. The values of these two components determine the frequency of the oscillator circuit and hence this is called as **Frequency determining circuit**.
- **Transistor Amplifier –** The output of the tank circuit is connected to the amplifier circuit so that the oscillations produced by the tank circuit are amplified here. Hence the output of these oscillations is increased by the amplifier.
- **Feedback Circuit –** The function of feedback circuit is to transfer a part of the output energy to LC circuit in proper phase. This feedback is positive in oscillators while negative in amplifiers.

Frequency Stability of an Oscillator

The frequency stability of an oscillator is a measure of its ability to maintain a constant frequency, over a long-time interval. When operated over a longer period of time, the oscillator frequency may have a drift from the previously set value either by increasing or by decreasing.

The change in oscillator frequency may arise due to the following factors –

- Operating point of the active device such as BJT or FET used should lie in the linear region of the amplifier. Its deviation will affect the oscillator frequency.
- The temperature dependency of the performance of circuit components affects the oscillator frequency.

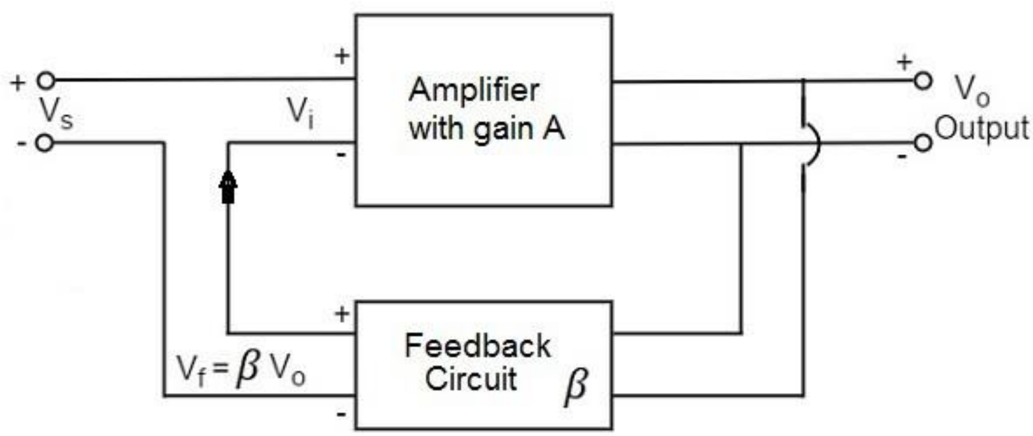
- The changes in d.c. supply voltage applied to the active device, shift the oscillator frequency. This can be avoided if a regulated power supply is used.
- A change in output load may cause a change in the Q-factor of the tank circuit, thereby causing a change in oscillator output frequency.
- The presence of inter element capacitances and stray capacitances affect the oscillator output frequency and thus frequency stability.

THE BARKHAUSEN CRITERION

With the knowledge we have till now, we understood that a practical oscillator circuit consists of a tank circuit, a transistor amplifier circuit and a feedback circuit. so, let us now try to brush up the concept of feedback amplifiers, to derive the gain of the feedback amplifiers.

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 concept of feedback amplifier can be understood from the following figure below.



From the above figure, the gain of the amplifier is represented as A. The gain of the amplifier is the ratio of output voltage Vo 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.

So, for positive feedback,

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

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

The output V_o must be equal to the input voltage ($V_s + \beta V_o$) multiplied by the gain A of the amplifier.

Hence,

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.,

Where **Aβ** is the **feedback factor** or the **loop gain**.

If $A\beta = 1$, $A_f = \infty$. Thus, the gain becomes infinity, i.e., there is output without any input. In another words, the amplifier works as an Oscillator.

The condition $A\beta = 1$ is called as **Barkhausen Criterion of oscillations**. This is a very important factor to be always kept in mind, in the concept of Oscillators.

TUNED CIRCUIT OSCILLATORS

Tuned circuit oscillators are the circuits that produce oscillations with the help of tuning circuits. The tuning circuits consists of an inductance L and a capacitor C. These are also known as **LC oscillators, resonant circuit oscillators** or **tank circuit oscillators**.

The tuned circuit oscillators are used to produce an output with frequencies ranging from 1 MHz to 500 MHz Hence these are also known as **R.F. Oscillators**. A BJT or a FET is used as an amplifier with tuned circuit oscillators. With an amplifier and an LC tank circuit, we can feedback a signal with right amplitude and phase to maintain oscillations.

Types of Tuned Circuit Oscillators

Most of the oscillators used in radio transmitters and receivers are of LC oscillators type. Depending upon the way the feedback is used in the circuit, the LC oscillators are divided as the following types.

- **Tuned-collector or Armstrong Oscillator –** It uses inductive feedback from the collector of a transistor to the base. The LC circuit is in the collector circuit of the transistor.
- **Tuned base Oscillator –** It uses inductive feedback. But the LC circuit is in the base circuit.
- **Hartley Oscillator –** It uses inductive feedback.
- **Colpitts Oscillator –** It uses capacitive feedback.
- **Clapp Oscillator –** It uses capacitive feedback.

We shall now discuss all the above-mentioned LC oscillators in detail.

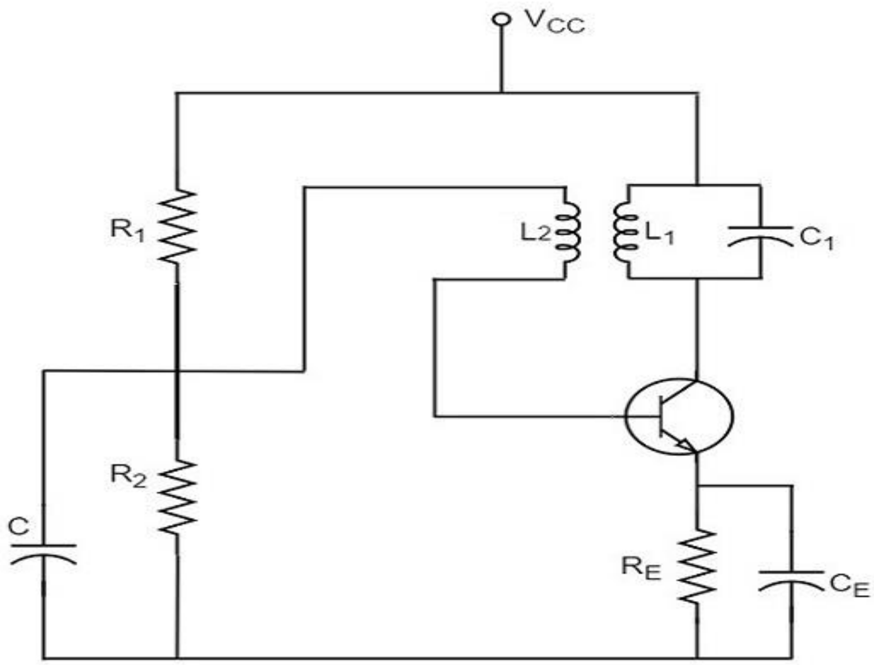
Tuned Collector Oscillator

Tuned collector oscillators are called so, because the tuned circuit is placed in the collector of the transistor amplifier. The combination of **L** and **C** form the tuned circuit or frequency determining circuit.

Construction

The resistors R_1, R_2 and R_E are used to provide d.c. bias to the transistor. The capacitors C_E and C are the by-pass capacitors. The secondary of the transformer provides a.c. feedback voltage that appears across the base-emitter junction of R_1 and R_2 is at a.c. ground due to by-pass capacitor C. In case, the capacitor was absent, a part of the voltage induced in the secondary of the transformer would drop across R_2 instead of completely going to the input of transistor.

As the CE configured transistor provides 180° phase shift, another 180° phase shift is provided by the transformer, which makes 360° phase shift between the input and output voltages. The following circuit diagram shows the arrangement of a tuned collector circuit.



OPERATION

Once the supply is given, the collector current starts increasing and charging of capacitor C takes place. When the capacitor is fully charged, it discharges through the inductance L_1. Now oscillations are produced. These oscillations induce some voltage in the secondary winding L_2. The frequency of voltage induced in the secondary winding is same as that of the tank circuit and its magnitude depends upon the number of turns in secondary winding and coupling between both the windings.

The voltage across L_2 is applied between base and emitter and appears in the amplified form in the collector circuit, thus overcoming the losses in the tank circuit. The number of turns of L_2 and coupling between L_1 and L_2 are so adjusted that oscillations across L_2 are amplified to a level just sufficient to supply losses to the tank circuit.

Tuned collector oscillators are widely used as the **local oscillator** in radio receivers.

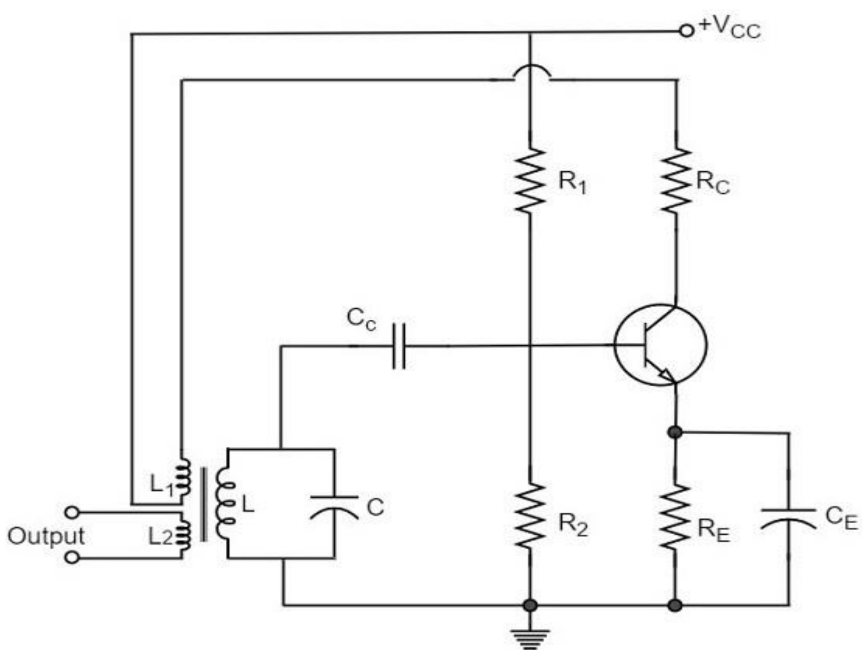
TUNED BASE OSCILLATOR

Tuned base oscillators are called so, because the tuned circuit is placed in the base of the transistor amplifier. The combination of **L** and **C** form the tuned circuit or frequency determining circuit.

CONSTRUCTION

The resistors R_1, R_2 and R_E are used to provide d.c. bias to the transistor. The parallel combination of R_e and C_e in the emitter circuit is the stabilizing circuit. C_C is the blocking capacitor. The capacitors C_E and C are the by-pass capacitors. The primary coil L and the secondary coil L_1 of RF transformer provides the required feedback to collector and base circuits.

As the CE configured transistor provides 180^o phase shift, another 180^o phase shift is provided by the transformer, which makes 360^o phase shift between the input and output voltages. The following circuit diagram shows the arrangement of a tuned base oscillator circuit.



OPERATION

When the circuit is switched on, the collector current starts rising. As the collector is connected to the coil L_1, that current creates some magnetic field around it. This induces a voltage in the tuned circuit coil L. The feedback voltage produces an increase in emitter base voltage and base current. A further increase in collector current is thus achieved and the cycle continues until the collector current becomes saturated. In the meanwhile, the capacitor is fully charged.

When the collector current reaches saturation level, there is no feedback voltage in L. As the capacitor has been charged fully, it starts discharging through L. This decreases the emitter base bias and hence I_B and the collector current also decreases. By the time the collector current reaches cutoff, the capacitor C is fully charged with opposite polarity. As the transistor now gets off, the condenser C begins to discharge through L. This increases the emitter-base bias. As a result, the collector current increases.

The cycle repeats so long as enough energy is supplied to **meet the losses** in L.C. circuit. The frequency of oscillation is equal to the resonant frequency of L.C. circuit.

DRAWBACK

The main **drawback** of tuned-base oscillator circuit is that, due to the low base-emitter resistance, which appears in shunt with the tuned circuit, the tank circuit gets loaded. This reduces its Q which in turn causes drift in oscillator frequency. Thus, stability becomes poorer. Due to this reason, the tuned circuit is **not** usually **connected in base** circuit.

DIFFERENCE BETWEEN AMPLIFIER AND OSCILLATOR

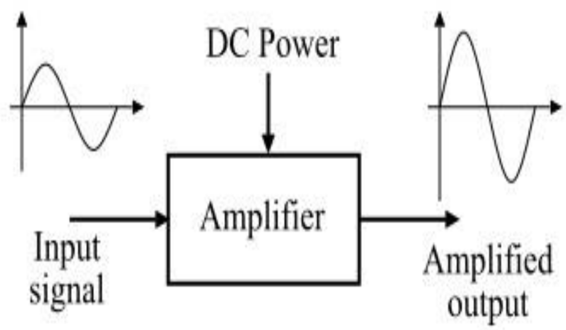


Figure - Amplifier

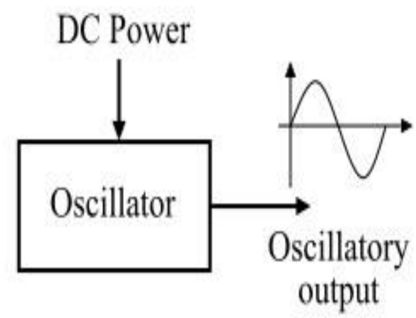


Figure - Oscillator

The following table highlights all the noticeable differences between an amplifier and an oscillator:

Basis of Difference	Amplifier	Oscillator
Definition	An electronic circuit that increases the magnitude of a weak signal is called an amplifier.	An electronic circuit that generates an AC signal of definite frequency having either sinusoidal or non-sinusoidal waveform is called an oscillator.
Primary function & location in circuit	The main function of an amplifier is to increase the intensity of a signal. Therefore, the amplifiers are repetitively used in a circuit because the signal losses its energy while travelling over long distances.	The main function of an amplifier is to increase the intensity of a signal. Therefore, the amplifiers are repetitively used in a circuit because the signal losses its energy while travelling over long distances.
Acts as	Amplifier acts as a multiplier.	Oscillator acts as a source.

Output signal	The output of an amplifier is just an amplified signal of same nature as the input. It may be period or aperiodic.	Oscillator always generates an oscillatory, i.e., periodic signal.
Presence of input & output	An amplifier necessarily has both input and output.	An oscillator has only an output.
Substituting	An amplifier can never perform the function of an oscillator.	We may use an oscillator itself in place of amplifier to produce a strong signal.
Feedback	In amplifiers, the negative feedback is used.	Oscillators use positive feedback.
Need of input signal	For the operation, an amplifier necessarily requires an input signal to generate an amplified output signal. Without input signal it does nothing.	Oscillator does not require input signal to generate an oscillatory output signal.
Applications	Amplifiers are widely used in audio systems to increase the intensity of audio signals.	Oscillators are extensively used in computers, laptops, and many other electronic devices and systems to generate the clock pulses for synchronization.

HARTLEY OSCILLATOR

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

CONSTRUCTION

In the circuit diagram of a Hartley oscillator shown below, the resistors R_1, R_2 and R_e provide necessary bias condition for the circuit. The capacitor

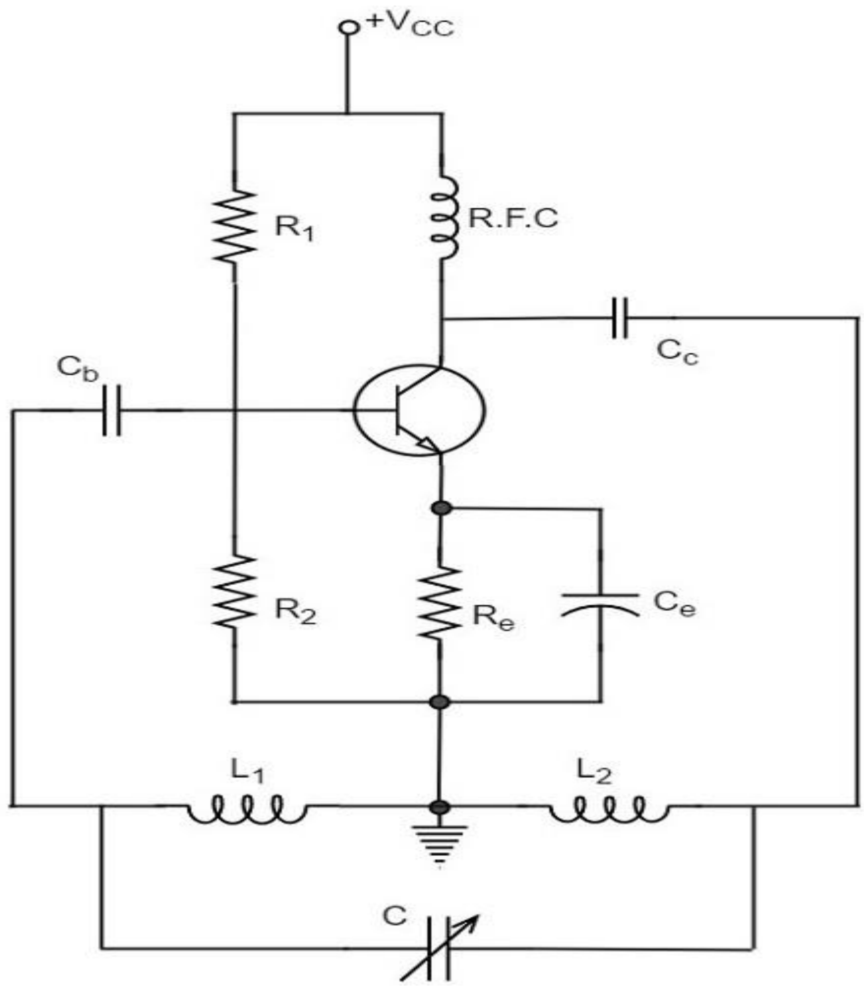
C_e provides a.c. ground thereby providing any signal degeneration. This also provides temperature stabilization.

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

TANK CIRCUIT

The frequency determining network is a parallel resonant circuit which consists of the inductors L_1 and L_2 along with a variable capacitor C. The junction of L_1 and L_2 are earthed. The coil L_1 has its one end connected to base via C_c and the other to emitter via C_e. So, L_2 is in the output circuit. Both the coils L_1 and L_2 are inductively coupled and together form an **Auto-transformer**.

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



OPERATION

When the collector supply is given, a transient current is produced in the oscillatory or tank circuit. The oscillatory current in the tank circuit produces a.c. voltage across L_1.

The **auto-transformer** made by the inductive coupling of L_1 and L_2 helps in determining the frequency and establishes the feedback. As the CE configured transistor provides 180^o phase shift, another 180^o phase shift is provided by the transformer, which makes 360^o phase shift between the input and output voltages.

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

FREQUENCY

The equation for **frequency of Hartley oscillator** is given as

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

$$L_T=L_1+L_2+2M$$

Here, $\mathbf{L_T}$ is the total cumulatively coupled inductance; $\mathbf{L_1}$ and $\mathbf{L_2}$ represent inductances of 1st and 2nd coils; and **M** represents mutual inductance.

Mutual inductance is calculated when two windings are considered.

ADVANTAGES

The advantages of Hartley oscillator are

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

DISADVANTAGES

The disadvantages of Hartley oscillator are

- It cannot be a low frequency oscillator.
- Harmonic distortions are present.

APPLICATIONS

The applications of Hartley oscillator are

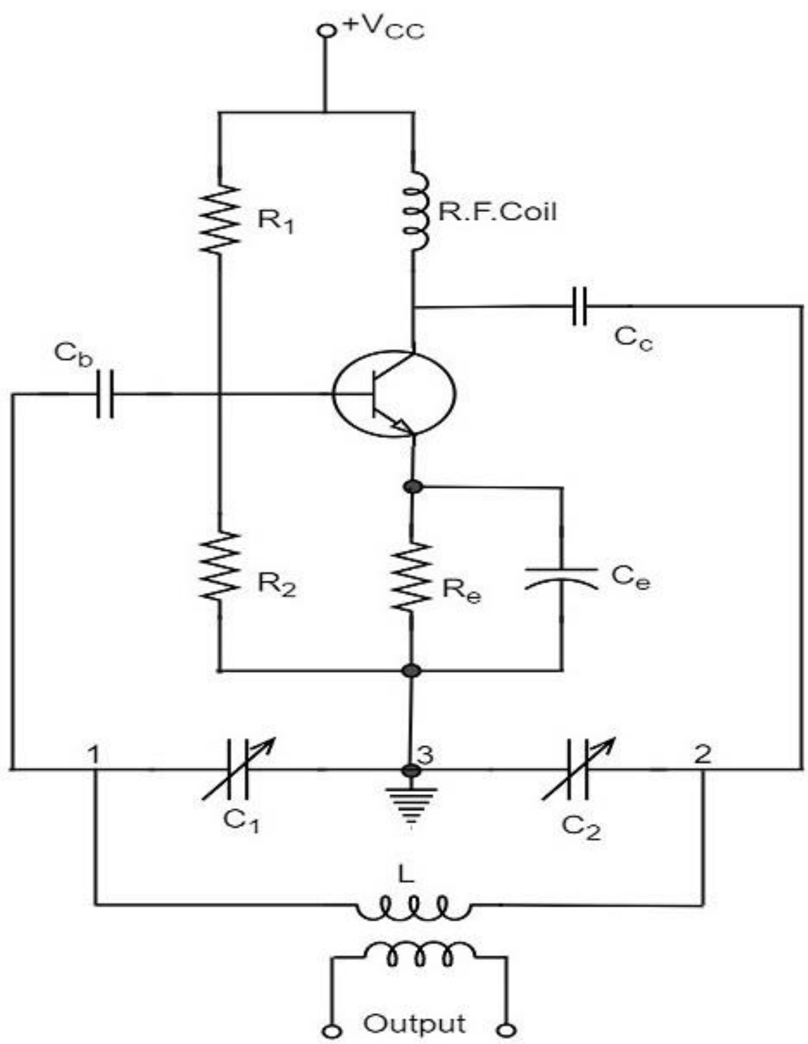
- It is used to produce a sinewave of desired frequency.
- Mostly used as a local oscillator in radio receivers.
- It is also used as R.F. Oscillator.

COLPITTS OSCILLATOR

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

CONSTRUCTION

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



The resistors R_1, R_2 and R_e provide necessary bias condition for the circuit. The capacitor C_e provides a.c. ground thereby providing any signal degeneration. This also provides temperature stabilization.

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

TANK CIRCUIT

The frequency determining network is a parallel resonant circuit which consists of variable capacitors C_1 and C_2 along with an inductor L. The junction of C_1 and C_2 are earthed. The capacitor C_1 has its one end connected to base via C_c and the other to emitter via C_e. the voltage developed across C_1 provides the regenerative feedback required for the sustained oscillations.

OPERATION

When the collector supply is given, a transient current is produced in the oscillatory or tank circuit. The oscillatory current in the tank circuit produces a.c. voltage across C_1 which are applied to the base emitter junction and appear in the amplified form in the collector circuit and supply losses to the tank circuit.

If terminal 1 is at positive potential with respect to terminal 3 at any instant, then terminal 2 will be at negative potential with respect to 3 at that instant because terminal 3 is grounded. Therefore, points 1 and 2 are out of phase by 180^o.

As the CE configured transistor provides 180^o phase shift, it makes 360^o phase shift between the input and output voltages. Hence, feedback is properly phased to produce continuous Undamped oscillations. When the **loop gain |βA| of the amplifier is greater than one, oscillations are sustained** in the circuit.

FREQUENCY

The equation for **frequency of Colpitts oscillator** is given as

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

C_T is the total capacitance of C_1 and C_2 connected in series.

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

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

ADVANTAGES

The advantages of Colpitts oscillator are as follows −

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

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

APPLICATIONS

The applications of Colpitts oscillator are as follows −

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

Phase Shift Oscillators

One of the important features of an oscillator is that the feedback energy applied should be in correct phase to the tank circuit. The oscillator circuits discussed so far has employed inductor (L) and capacitor (C) combination, in the tank circuit or frequency determining circuit.

We have observed that the LC combination in oscillators provide 180^o phase shift and transistor in CE configuration provide 180° phase shift to make a total of 360^o phase shift so that it would make a zero difference in phase.

Drawbacks of LC circuits

Though they have few applications, the **LC** circuits have few **drawbacks** such as

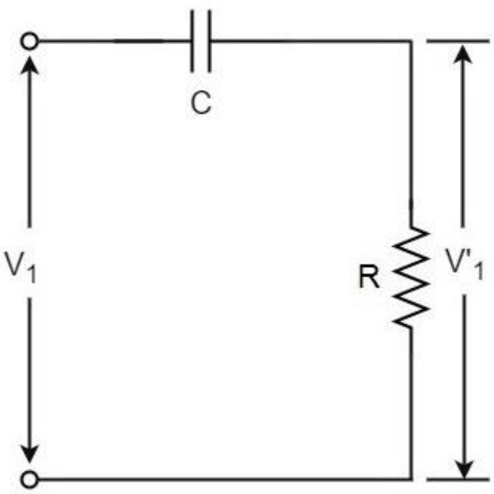
- Frequency instability
- Waveform is poor
- Cannot be used for low frequencies
- Inductors are bulky and expensive

We have another type of oscillator circuits, which are made by replacing the inductors with resistors. By doing so, the frequency stability is improved and a good quality waveform is obtained. These oscillators can also produce lower frequencies. As well, the circuit becomes neither bulky nor expensive.

All the drawbacks of **LC** oscillator circuits are thus eliminated in **RC** oscillator circuits. Hence the need for RC oscillator circuits arise. These are also called as **Phase–shift Oscillators**.

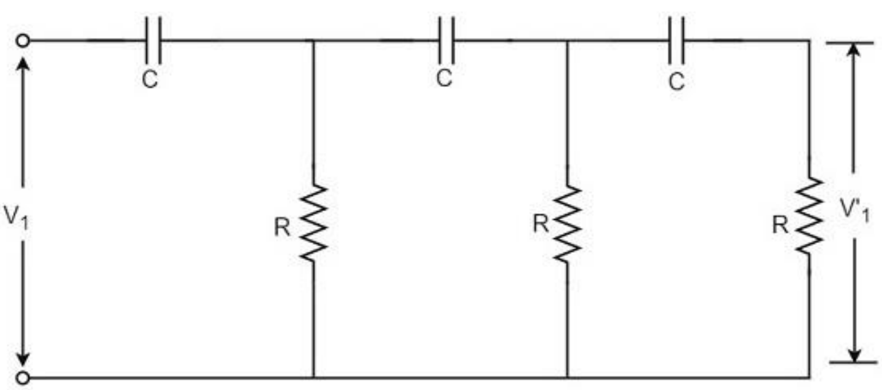
Principle of Phase-shift oscillators

We know that the output voltage of an RC circuit for a sinewave input leads the input voltage. The phase angle by which it leads is determined by the value of RC components used in the circuit. The following circuit diagram shows a single section of an RC network.



The output voltage V_1' across the resistor R leads the input voltage applied input V_1 by some phase angle ϕ^o. If R were reduced to zero, V_1' will lead the V_1 by 90^o i.e., $\phi^o = 90^o$.

However, adjusting R to zero would be impracticable, because it would lead to no voltage across R. Therefore, in practice, R is varied to such a value that makes V_1' to lead V_1 by 60^o. The following circuit diagram shows the three sections of the RC network.



Each section produces a phase shift of 60^o. Consequently, a total phase shift of 180^o is produced, i.e., voltage V_2 leads the voltage V_1 by 180^o.

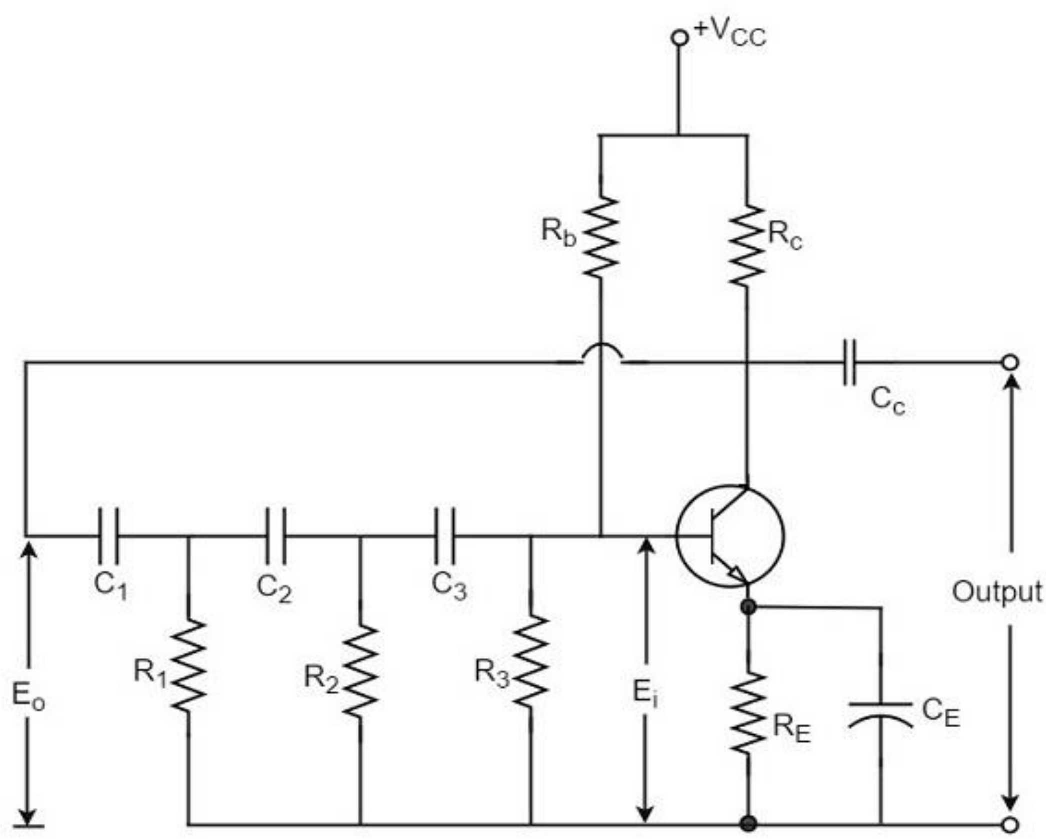
PHASE-SHIFT OSCILLATOR CIRCUIT

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

CONSTRUCTION

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

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



The frequency of oscillations is given by

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

Where

$$R_1 = R_2 = R_3 = R$$

$$C_1 = C_2 = C_3 = C$$

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 feedback applied will be

$$M = E_i/E_o$$

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

ADVANTAGES

The advantages of RC phase shift oscillator are as follows –

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

Disadvantages

The disadvantages of RC phase shift oscillator are as follows −

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

Wien Bridge Oscillator

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

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

Construction

The circuit construction of Wien bridge oscillator can be explained as below. It is a two-stage amplifier with RC bridge circuit. The bridge circuit has the arms R_1C_1, R_3, R_2C_2 and the tungsten lamp L_p. Resistance R_3 and the lamp L_p are used to stabilize the amplitude of the output.

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

The transistor T_1 serves as an oscillator and an amplifier while the other transistor T_2 serves as an inverter. The inverter operation provides a phase shift of 180^o. This circuit provides positive feedback through R_1C_1, C_2R_2 to the transistor T_1 and negative feedback through the voltage divider to the input of transistor T_2.

The frequency of oscillations is determined by the series element R_1C_1 and parallel element R_2C_2 of the bridge.

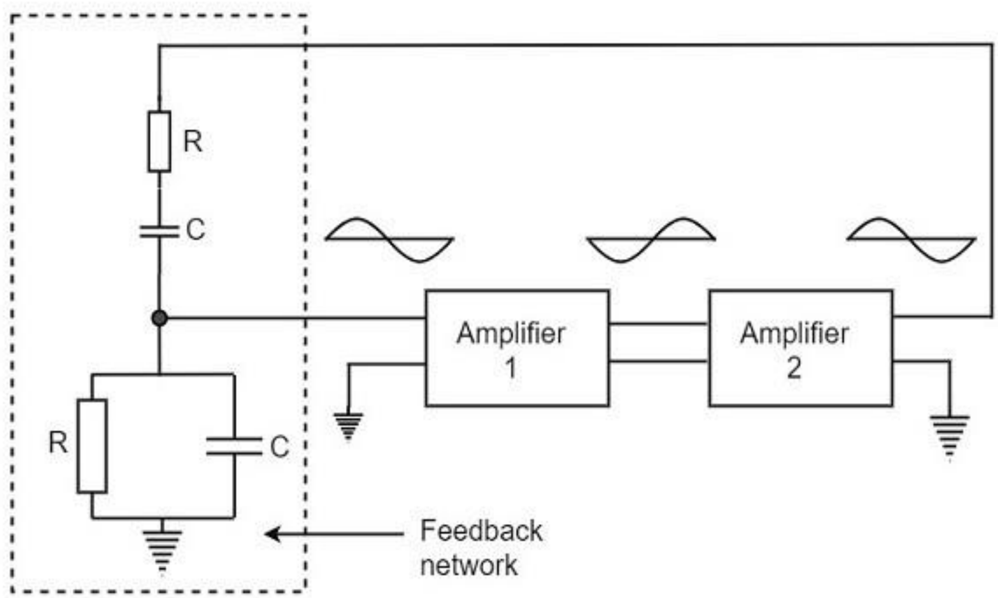
$$f = 1/2\pi\sqrt{R1C1R2C2}$$

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

Then,

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

Now, we can simplify the above circuit as follows –



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

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

Operation

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

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

Advantages

The advantages of Wien bridge oscillator are as follows −

- The circuit provides good frequency stability.
- It provides constant output.
- The operation of circuit is quite easy.
- The overall gain is high because of two transistors.
- The frequency of oscillations can be changed easily.
- The amplitude stability of the output voltage can be maintained more accurately, by replacing R_2 with a thermistor.

Disadvantages

The disadvantages of Wien bridge oscillator are as follows −

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

Crystal Oscillators

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

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

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

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

Crystal Oscillators

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

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

Piezo Electric Effect

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

Certain crystalline materials like Rochelle salt, quartz and tourmaline exhibit piezo electric effect and such materials are called as **Piezo electric crystals**.

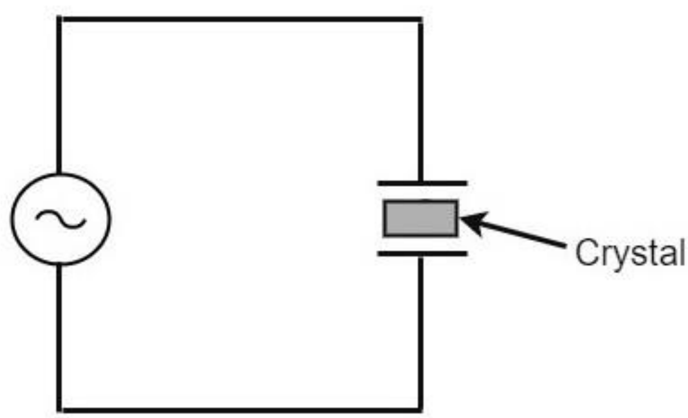
Quartz is the most commonly used piezo electric crystal because it is inexpensive and readily available in nature.

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

WORKING OF A QUARTZ CRYSTAL

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

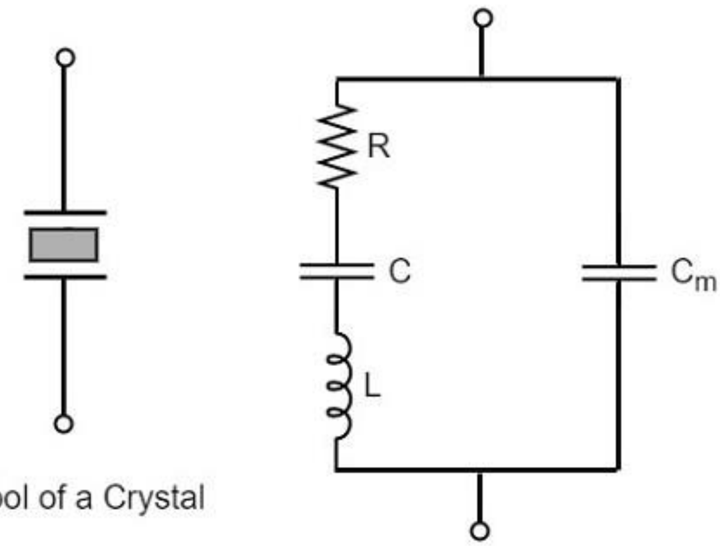
The circuit arrangement of a Quartz Crystal will be as shown below –



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

EQUIVALENT CIRCUIT OF A CRYSTAL

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



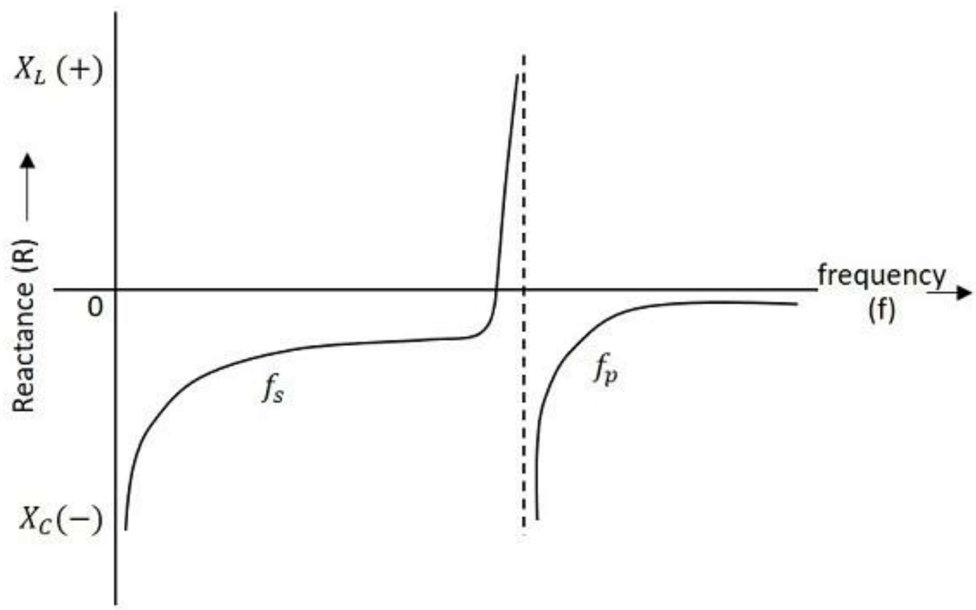
Symbol of a Crystal

Equivalent circuit of a crystal

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

FREQUENCY RESPONSE

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



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

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

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

$$f_p=12\pi\sqrt{L.C_T}$$

Where

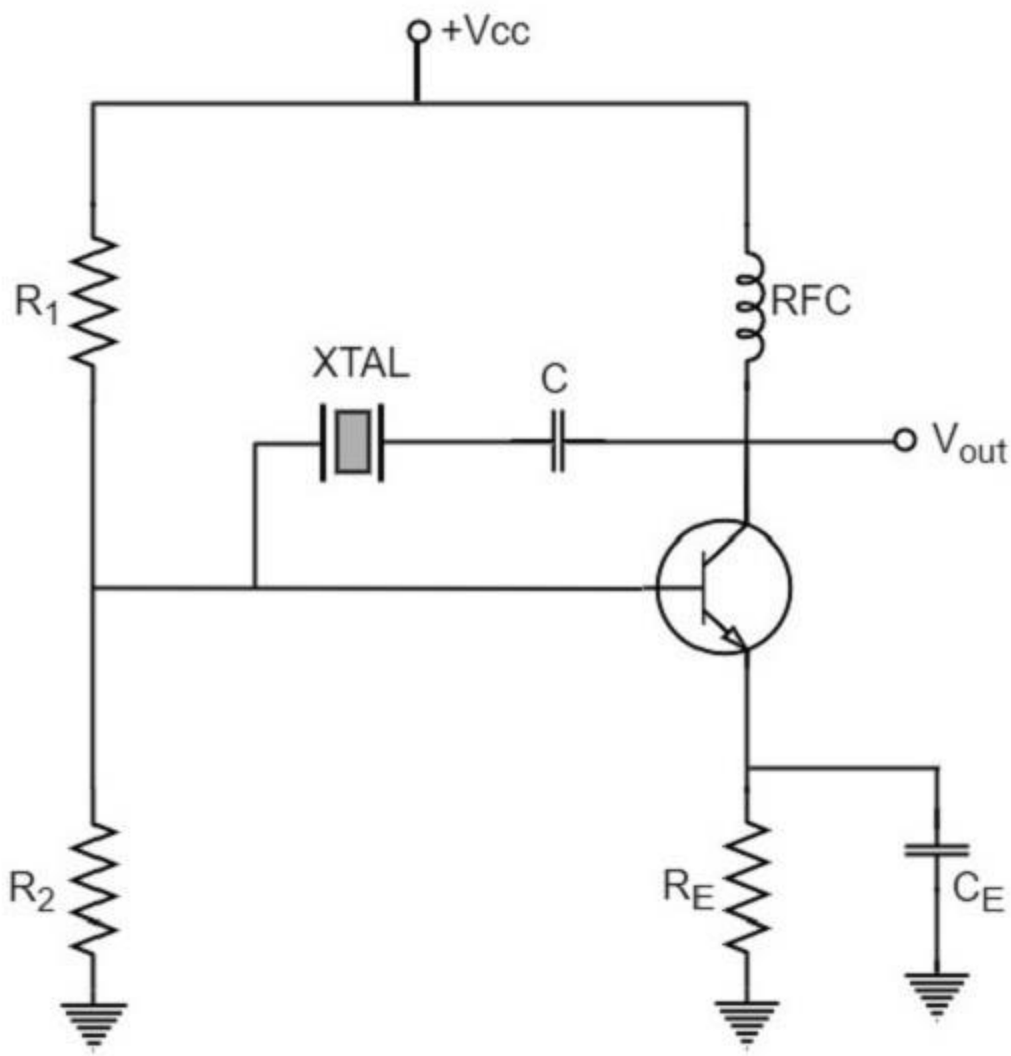
$$C_T=CCm/(C+Cm)$$

The value of C_m is usually very large as compared to C. Therefore, the value of C_T is approximately equal to C and hence the series resonant frequency is approximately equal to the parallel resonant frequency (i.e., $f_s = f_p$).

CRYSTAL OSCILLATOR CIRCUIT

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

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



In this circuit, the crystal is connected as a series element in the feedback path from collector to the base. The resistors R_1, R_2 and R_E provide a voltage-divider stabilized d.c. bias circuit. The capacitor C_E provides a.c. bypass of the emitter resistor and RFC (radio frequency choke) coil provides for d.c. bias while decoupling any a.c. signal on the power lines from affecting the output signal. The coupling capacitor C has negligible impedance at the circuit operating frequency. But it blocks any d.c. between collector and base.

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

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

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

Advantages

The advantages of crystal oscillator are as follows –

- They have a high order of frequency stability.
- The quality factor (Q) of the crystal is very high.

Disadvantages

The disadvantages of crystal oscillator are as follows –

- They are fragile and can be used in low power circuits.
- The frequency of oscillations cannot be changed appreciably.

Frequency Stability of an Oscillator

An Oscillator is expected to maintain its frequency for a longer duration without any variations, so as to have a smoother clear sinewave output for the circuit operation. Hence the term frequency stability really matters a lot, when it comes to oscillators, whether sinusoidal or non-sinusoidal.

The frequency stability of an oscillator is defined as the ability of the oscillator to maintain the required frequency constant over a long time interval as possible. Let us try to discuss the factors that affect this frequency stability.

Change in Operating Point

We have already come across the transistor parameters and learnt how important an operating point is. The stability of this operating point for the transistor being used in the circuit for amplification (BJT or FET), is of higher consideration.

The operating of the active device used is adjusted to be in the linear portion of its characteristics. This point is shifted due to temperature variations and hence the stability is affected.

Variation in Temperature

The tank circuit in the oscillator circuit, contains various frequency determining components such as resistors, capacitors and inductors. All of their parameters are temperature dependent. Due to the change in temperature, their values get affected. This brings the change in frequency of the oscillator circuit.

Due to Power Supply

The variations in the supplied power will also affect the frequency. The power supply variations lead to the variations in V_{cc}. This will affect the frequency of the oscillations produced.

In order to avoid this, the regulated power supply system is implemented. This is in short called as RPS. The details of regulated power supply were clearly discussed in the power supply section of ELECTRONIC CIRCUITS tutorial.

Change in Output Load

The variations in output resistance or output load also affects the frequency of the oscillator. When a load is connected, the effective resistance of the tank circuit is changed. As a result, the Q-factor of LC tuned circuit is changed. This results a change in output frequency of oscillator.

Changes in Inter-Element Capacitances

Inter-element capacitances are the capacitances that develop in PN junction materials such as diodes and transistors. These are developed due to the charge present in them during their operation.

The inter element capacitors undergo change due to various reasons as temperature, voltage etc. This problem can be solved by connecting swamping capacitor across offending inter-element capacitor.

Value of Q

The value of Q (Quality factor) must be high in oscillators. The value of Q in tuned oscillators determines the selectivity. As this Q is directly proportional to the frequency stability of a tuned circuit, the value of Q should be maintained high.

Frequency stability can be mathematically represented as,

$S_w = d\theta/dw$��=��/��

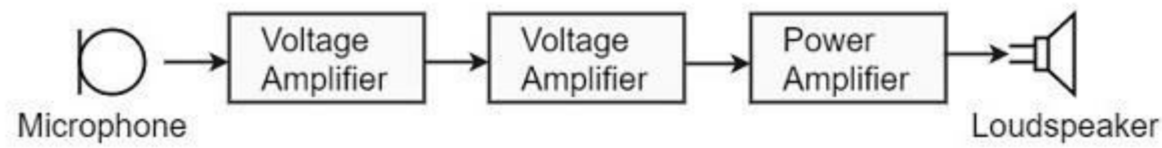
Where dθ is the phase shift introduced for a small frequency change in nominal frequency f_r. The circuit giving the larger value of (dθ/dw) has more stable oscillatory frequency.

POWER AMPLIFIERS

In practice, any amplifier consists of few stages of amplification. If we consider audio amplification, it has several stages of amplification, depending upon our requirement.

POWER AMPLIFIER

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



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

The DC power is distributed according to the relation,

DC power input = AC power output + losses

POWER TRANSISTOR

For such Power amplification, a normal transistor would not do. 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.

Difference between Voltage and Power Amplifiers

Let us try to differentiate between voltage and power amplifier.

Voltage Amplifier

The function of a voltage amplifier is to raise the voltage level of the signal. A voltage amplifier is designed to achieve maximum voltage amplification.

The voltage gain of an amplifier is given by

$$Av = \beta(Rc / Rin)$$

The characteristics of a voltage amplifier are as follows −

- The base of the transistor should be thin and hence the value of β should be greater than 100.
- The resistance of the input resistor R_{in} should be low when compared to collector load R_C.
- The collector load R_C should be relatively high. To permit high collector load, the voltage amplifiers are always operated at low collector current.
- The voltage amplifiers are used for small signal voltages.

Power Amplifier

The function of a power amplifier is to raise the power level of input signal. It is required to deliver a large amount of power and has to handle large current.

The characteristics of a power amplifier are as follows −

- The base of transistor is made thicken to handle large currents. The value of β being (β > 100) high.
- The size of the transistor is made larger, in order to dissipate more heat, which is produced during transistor operation.
- Transformer coupling is used for impedance matching.
- Collector resistance is made low.

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

S.No	Particular	Voltage Amplifier	Power Amplifier
1	β	High (>100)	Low (5 to 20)
2	R_C	High (4-10 KΩ)	Low (5 to 20 Ω)
3	Coupling	Usually R-C coupling	Invariably transformer coupling
4	Input voltage	Low (a few m V)	High (2-4 V)
5	Collector current	Low (≈ 1 mA)	High (> 100 mA)
6	Power output	Low	High
7	Output impendence	High (≈ 12 K Ω)	Low (200 Ω)

CLASSIFICATION OF POWER AMPLIFIERS

The Power amplifiers amplify the power level of the signal. This amplification is done in the last stage in audio applications. The applications related to radio frequencies employ radio power amplifiers. But the **operating point** of a transistor, plays a very important role in determining the efficiency of the amplifier. The **main classification** is done based on this mode of operation.

The classification is done based on their frequencies and also based on their mode of operation.

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.

Before going into the details of these amplifiers, let us have a look at the important terms that have to be considered to determine the efficiency of an amplifier.

Terms Considering Performance

The primary objective of a power amplifier is to obtain maximum output power. In order to achieve this, the important factors to be considered are collector efficiency, power dissipation capability and distortion. Let us go through them in detail.

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 ac power output / average dc power input to transistor

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

The main aim of a power amplifier is to obtain maximum collector efficiency. Hence the higher the value of collector efficiency, the efficient the amplifier will be.

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

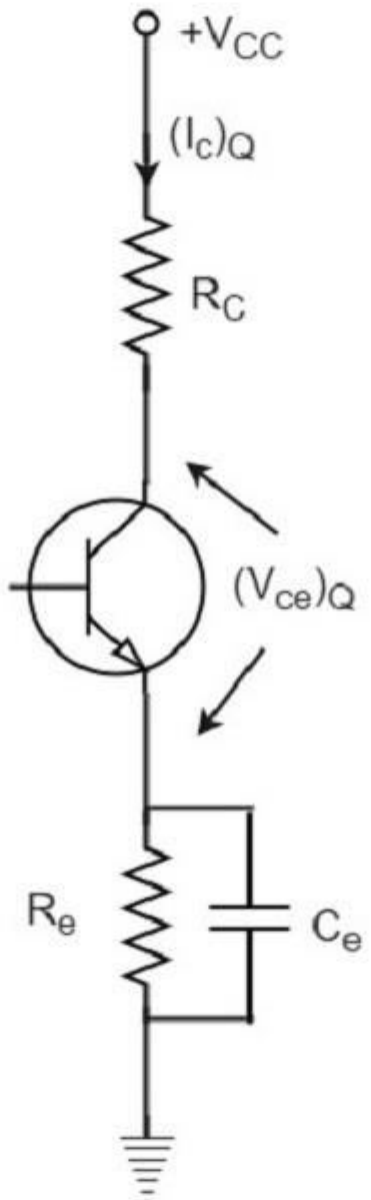
A transistor is a non-linear device. When compared with the input, there occur few variations in the output. In voltage amplifiers, this problem is not pre-dominant as small currents are used. But in power amplifiers, as large currents are in use, the problem of distortion certainly arises.

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

CLASS A POWER AMPLIFIERS

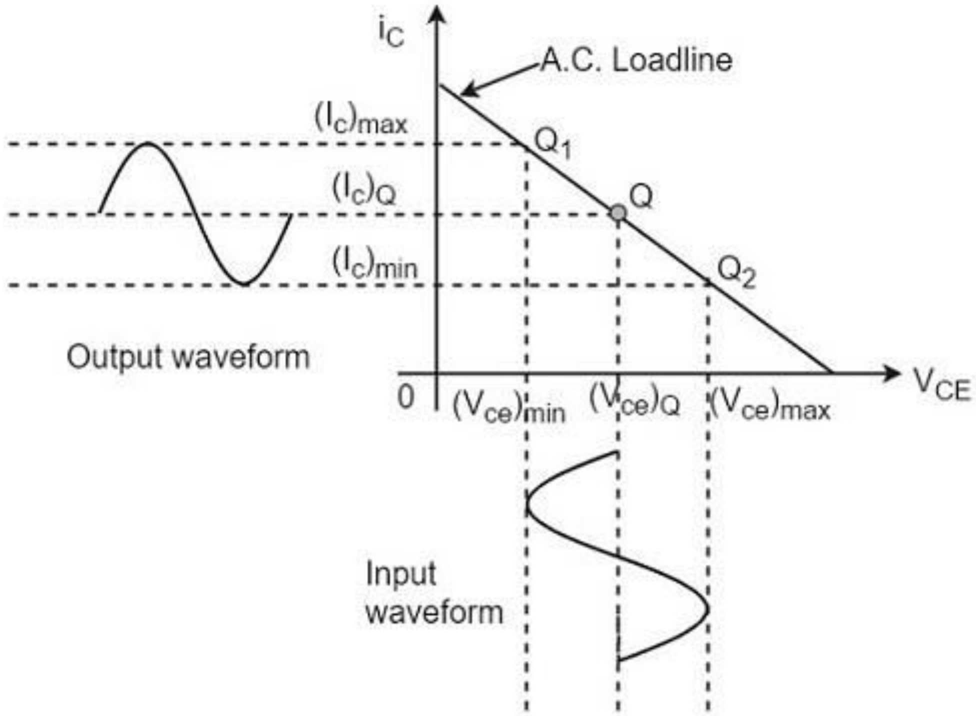
We have already come across the details of transistor biasing, which is very important for the operation of a transistor as an amplifier. Hence to achieve faithful amplification, the biasing of the transistor has to be done such that the amplifier operates over the linear region.

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



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

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



/

The output characteristics with operating point Q is shown in the figure above. Here $(I_c)_Q$ and $(V_{ce})_Q$ represent no signal collector current and voltage between collector and emitter respectively. When signal is applied, the Q-point shifts to Q_1 and Q_2. The output current increases to $(I_c)_{max}$ and decreases to $(I_c)_{min}$. Similarly, the collector-emitter voltage increases to $(V_{ce})_{max}$ and decreases to $(V_{ce})_{min}$.

D.C. Power drawn from collector battery V_{cc} is given

$$P_{in} = \text{voltage} \times \text{current} = V_{CC} (I_C)_Q$$

This power is used in the following two parts –

- Power dissipated in the collector load as heat is given by

$$P_{RC} = (\text{current})^2 \times \text{resistance} = (I_C)^2 Q_{RC}$$

- Power given to transistor is given by

$$P_{tr} = P_{in} - P_{RC} = V_{CC} - (I_C)^2_Q / R_C$$

When signal is applied, the power given to transistor is used in the following two parts –

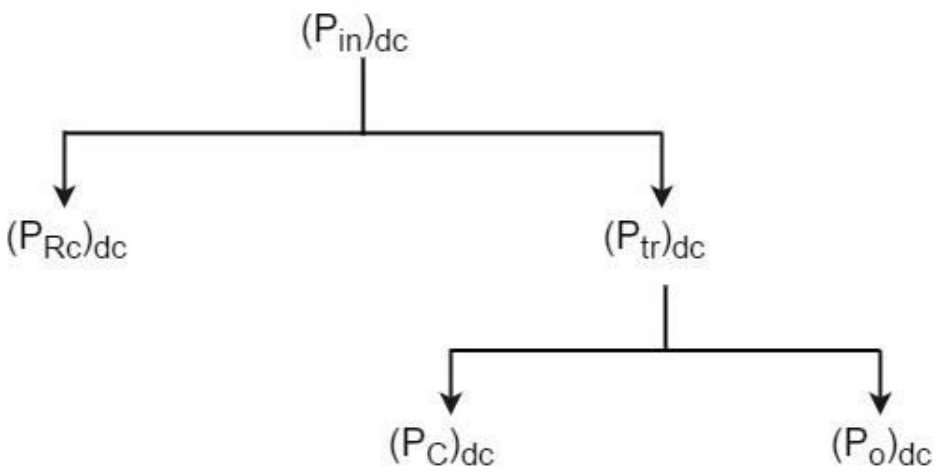
- A.C. Power developed across load resistors R_C which constitutes the a c power output.

$$(P_O)ac = I^2_{RC} = V^2/ R_C = (Vm/\sqrt{2})^2 \times 1/R_C = V^2_m / 2R_C$$

Where **I** is the R.M.S. value of ac output current through load, **V** is the R.M.S. value of ac voltage, and $\mathbf{V_m}$ is the maximum value of V.

- The D.C. power dissipated by the transistor (collector region) in the form of heat, i.e., $(P_C)_{dc}$

We have represented the whole power flow in the following diagram.



This class A power amplifier can amplify small signals with least distortion and the output will be an exact replica of the input with increased strength.

Let us now try to draw some expressions to represent efficiencies.

OVERALL EFFICIENCY

The overall efficiency of the amplifier circuit is given by

(η)overall = (ac power delivered to the load) / (total power delivered by dc supply = $(P_O)ac / (P_{in})dc$

COLLECTOR EFFICIENCY

The collector efficiency of the transistor is defined as

(η)collector = (average ac power output) / (average dc power input to transistor)

$$=(P_O)ac / (P_{tr})dc$$

EXPRESSION FOR OVERALL EFFICIENCY

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

$$= 1/\sqrt{2}[(Vce)max - (Vce)min] / 2 \times 1/\sqrt{2}[(IC)max - (IC)min]/2$$

$$= [(Vce)max - (Vce)min] \times [(IC)max - (IC)min] / 8$$

Therefore

$(\eta)overall = [(Vce)max - (Vce)min] \times [(IC)max - (IC)min] / 8 \times VCC(IC)Q$

ADVANTAGES OF CLASS A AMPLIFIERS

The advantages of Class A power amplifier are as follows −

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

DISADVANTAGES OF CLASS A AMPLIFIERS

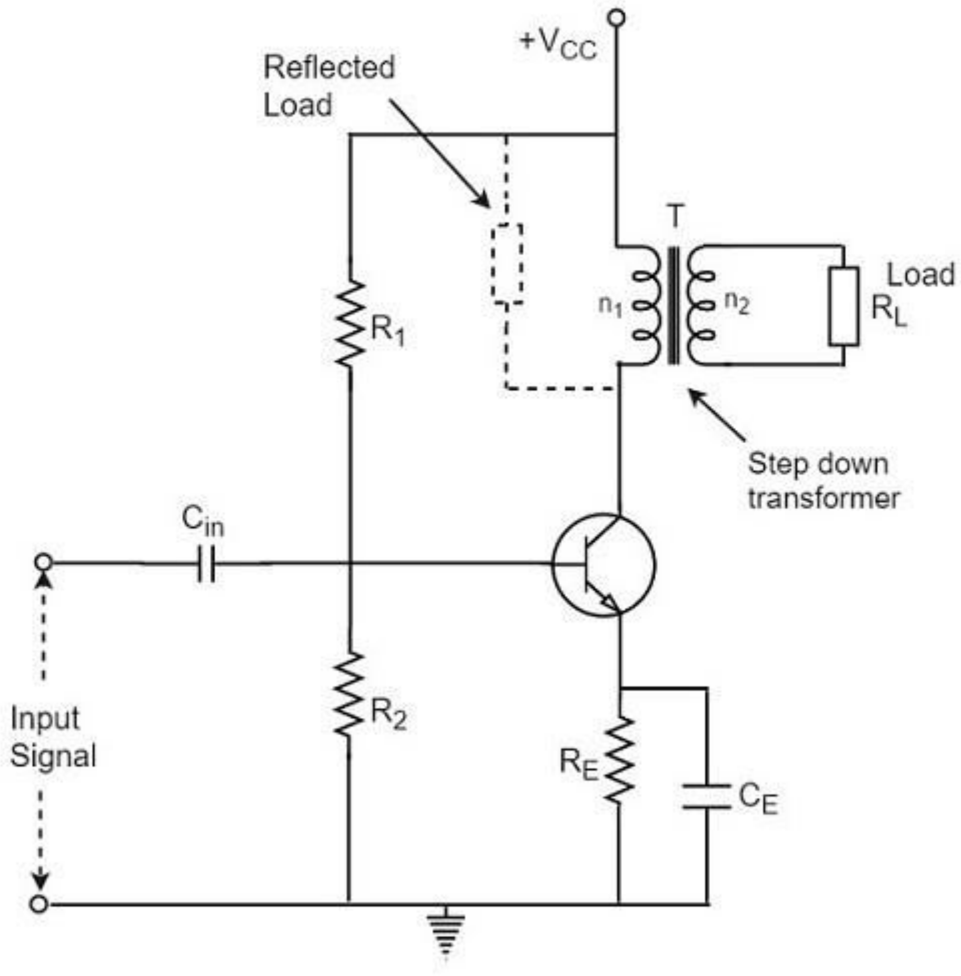
The advantages of Class A power amplifier are as follows −

- Low power output
- Low collector efficiency

TRANSFORMER COUPLED CLASS A POWER AMPLIFIER

The class A power amplifier as discussed in the previous chapter, is the circuit in which the output current flows for the entire cycle of the AC input supply. We also have learnt about the disadvantages it has such as low output power and efficiency. In order to minimize those effects, the transformer coupled class A power amplifier has been introduced.

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



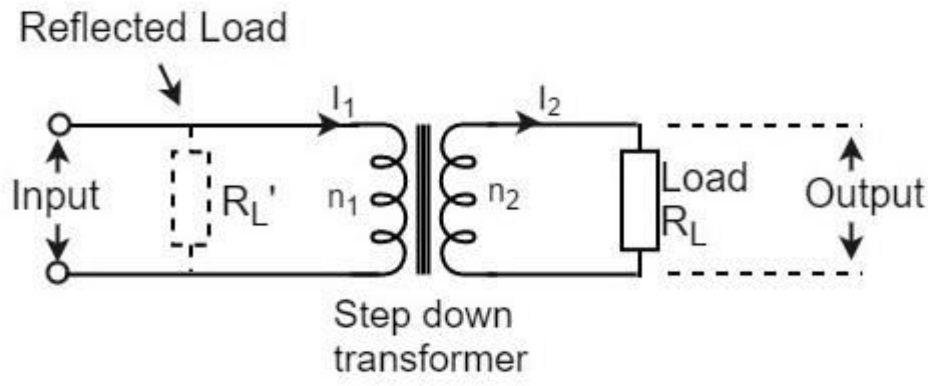
Here R_1 and R_2 provide potential divider arrangement. The resistor Re provides stabilization, C_e is the bypass capacitor and R_e to prevent a.c. voltage. The transformer used here is a step-down transformer.

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).

TRANSFORMER ACTION

The transformer used in the collector circuit is for impedance matching. R_L is the load connected in the secondary of a transformer. R_L' is the reflected load in the primary of the transformer.

The number of turns in the primary are n_1 and the secondary are n_2. Let V_1 and V_2 be the primary and secondary voltages and I_1 and I_2 be the primary and secondary currents respectively. The below figure shows the transformer clearly.



We know that

$$V1/V2 = n1/n2 \text{ and } I1/I2 = n1/n2$$

Or

$$V1=(n1/n2) \times V2 \text{ and } I1 = (n1/n2) \times I2$$

Hence

$$V1I1=(n1/n2)2 \,/\, (V2/I2)$$

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

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

Therefore,

$$R'L=(n1/n2)2RL = n2RL$$

Where

n = number of turns in primary /number of turns in secondary = n1/n2

A power amplifier may be matched by taking proper turn ratio in step down transformer.

Circuit Operation

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

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

Circuit Analysis

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

The input power under dc condition will be

$$(Pin)dc = (Ptr)dc = VCC \times (IC)Q$$

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

Hence

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

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

Therefore,

$$(PO)ac=Vrms\times Irms=VCC/\sqrt{2} \times (IC)Q / \sqrt{2} = [VCC \times (IC)Q] / 2$$

Therefore,

$$\text{Collector Efficiency} = (PO)ac / (Ptr)dc$$

Or,

$$(\eta)collector = VCC \times (IC)Q/2 \times VCC \times (IC)Q = 1/2$$

$$=1/2\times 100 = 50\%$$

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

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

- Low frequency signals are less amplified comparatively.
- Hum noise is introduced by transformers.
- Transformers are bulky and costly.
- Poor frequency response.

Applications

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

- This circuit is where impedance matching is the main criterion.
- These are used as driver amplifiers and sometimes as output amplifiers.

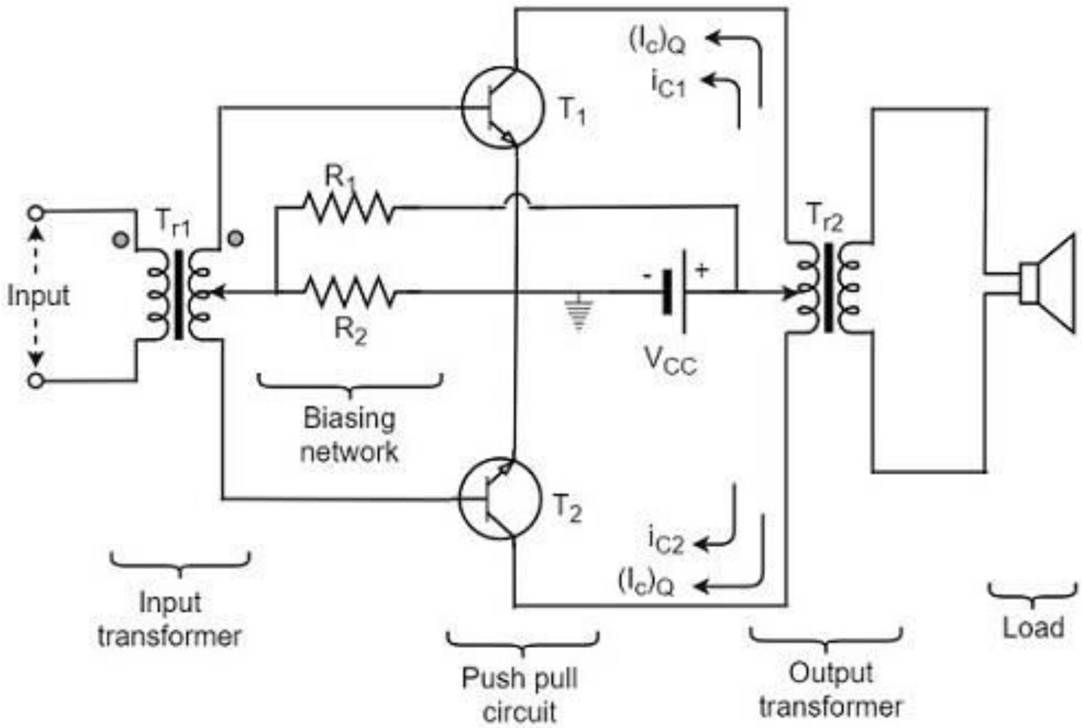
PUSH-PULL CLASS A POWER AMPLIFIER

So far, we have seen two types of class A power amplifiers. The main problems that should be dealt with are low power output and efficiency. It is possible to obtain greater power output and efficiency than that of the Class A amplifier by using a combinational transistor pair called as **Push-Pull** configuration.

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

CONSTRUCTION OF PUSH-PULL CLASS A POWER AMPLIFIER

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



In Push-pull arrangement, the two identical transistors T_1 and T_2 have their emitter terminals shorted. The input signal is applied to the transistors through the transformer T_{r1} which provides opposite polarity signals to both the transistor bases. The collectors of both the transistors are connected to the primary of output transformer T_{r2}. Both the transformers are center tapped. The V_{CC} supply is provided to the collectors of both the transistors through the primary of the output transformer.

The resistors R_1 and R_2 provide the biasing arrangement. The load is generally a loudspeaker which is connected across the secondary of the output transformer. The turns ratio of the output transformer is chosen in such a way that the load is well matched with the output impedance of the transistor. So maximum power is delivered to the load by the amplifier.

CIRCUIT OPERATION

The output is collected from the output transformer T_{r2}. The primary of this transformer T_{r2} has practically no dc component through it. The transistors T_1 and T_2 have their collectors connected to the primary of transformer T_{r2} so that their currents are equal in magnitude and flow in opposite directions through the primary of transformer T_{r2}.

When the a.c. input signal is applied, the base of transistor T_1 is more positive while the base of transistor T_2 is less positive. Hence the collector current i_{c1} of transistor T_1 increases while the collector current i_{c2} of transistor T_2 decreases. These currents flow in opposite directions in two halves of the primary of output transformer. Moreover, the flux produced by these currents will also be in opposite directions.

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

$$(ic_1 - ic_2)$$

Similarly, for the negative input signal, the collector current i_{c2} will be more than i_{c1}. In this case, the voltage developed across the load will again be due to the difference

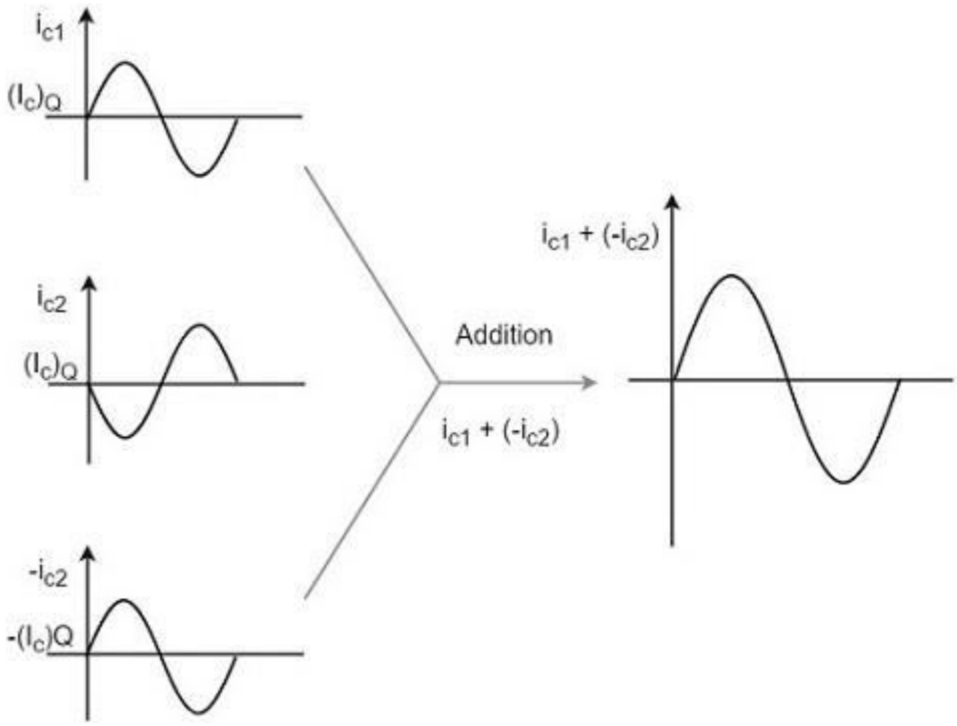
$$(ic_1 - ic_2)$$

$$\text{As } ic_2 > ic_1$$

The polarity of voltage induced across load will be reversed.

$$ic_1 - ic_2 = ic_1 + (-ic_2)$$

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



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

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

Advantages

The advantages of class A Push-pull amplifier are as follows

- High a.c. output is obtained.
- The output is free from even harmonics.
- The effect of ripple voltages are balanced out. These are present in the power supply due to inadequate filtering.

Disadvantages

The disadvantages of class A Push-pull amplifier are as follows

- The transistors are to be identical, to produce equal amplification.
- Center-tapping is required for the transformers.
- The transformers are bulky and co

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 powe**

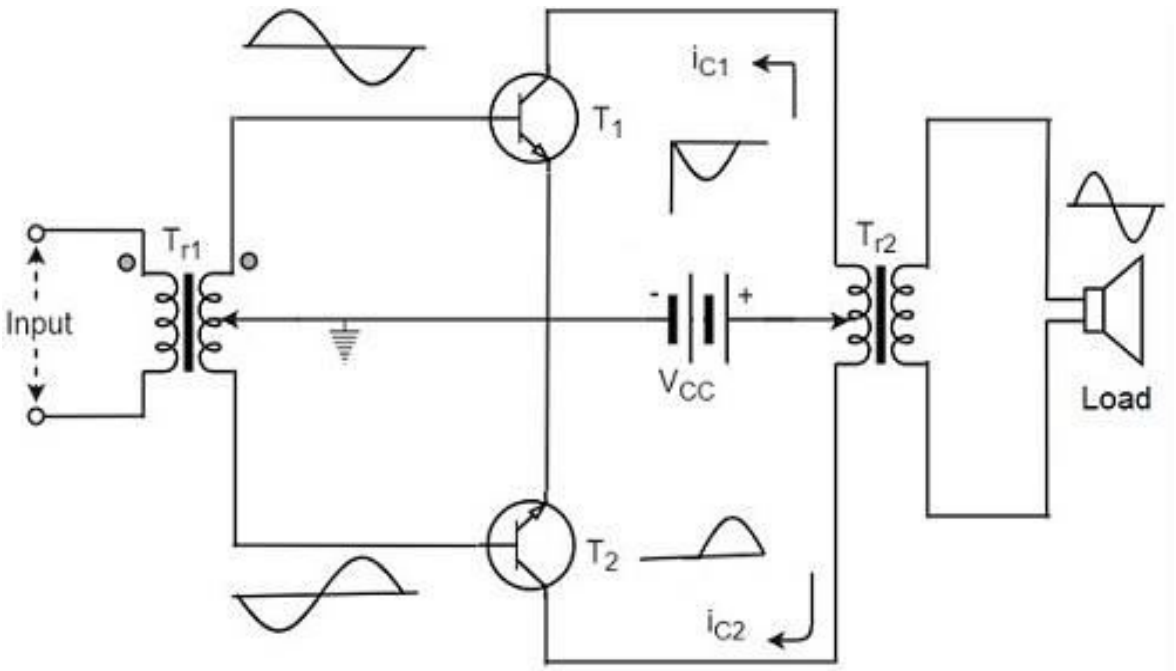
CLASS B PUSH-PULL AMPLIFIER

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

CONSTRUCTION

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

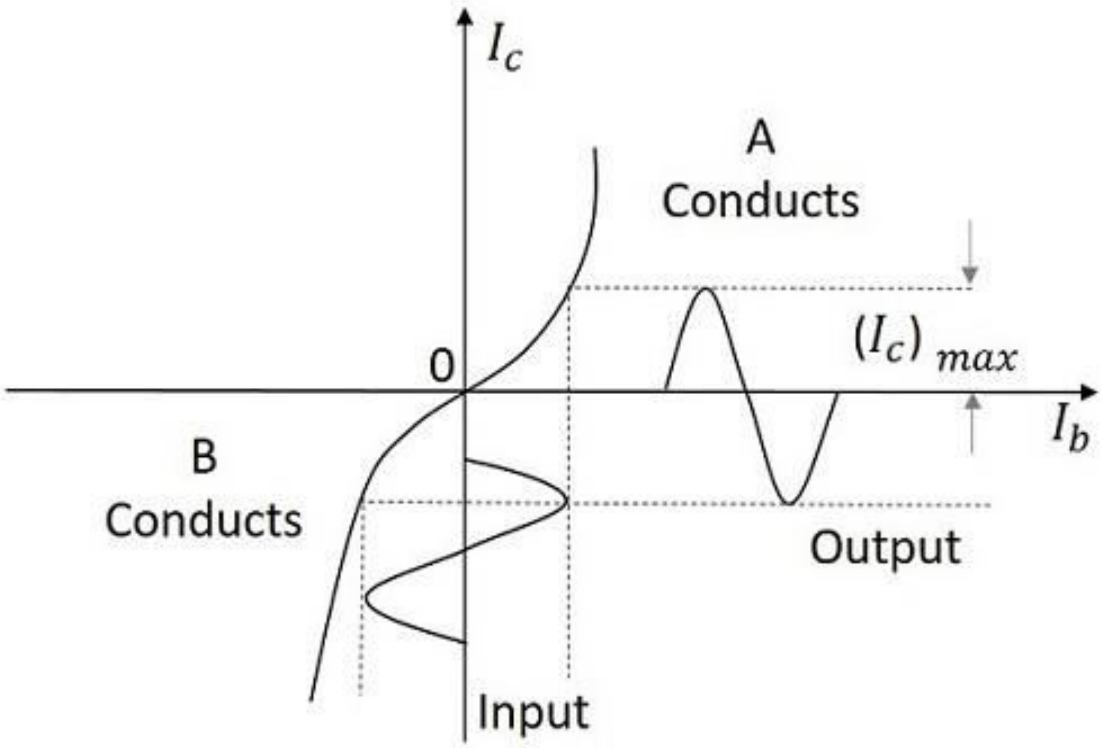


The circuit operation of class B push pull amplifier is detailed below.

OPERATION

The circuit of class B push-pull amplifier shown in the above figure clears that both the transformers are center-tapped. When no signal is applied at the input, the transistors 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° 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. The waveform is produced as shown in the following figure.



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

$$Idc = (IC)max / \pi$$

Therefore,

$$(P_{in})dc = 2 \times [\{(IC)max / \pi\} \times VCC]$$

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

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

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

Under ideal conditions of maximum power

Therefore,

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

Now overall maximum efficiency

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

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

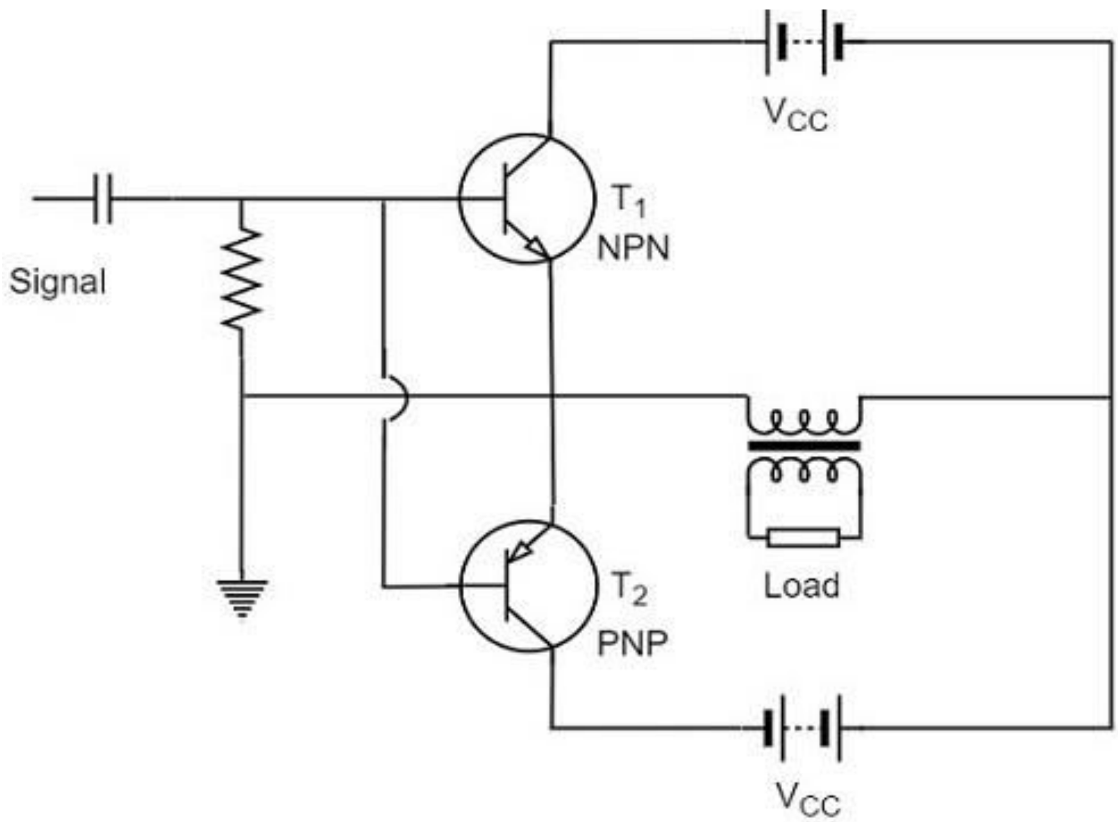
$$= \pi / 4 = 0.785 = 78.5\%$$

The collector efficiency would be the same.

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

COMPLEMENTARY SYMMETRY PUSH-PULL CLASS B AMPLIFIER

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



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

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

Advantages

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

- As there is no need of centre tapped transformers, the weight and cost are reduced.
- Equal and opposite input signal voltages are not required.

Disadvantages

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

- It is difficult to get a pair of transistors (NPN and PNP) that have similar characteristics.
- We require both positive and negative supply voltages.

Class AB and Class C Power Amplifiers

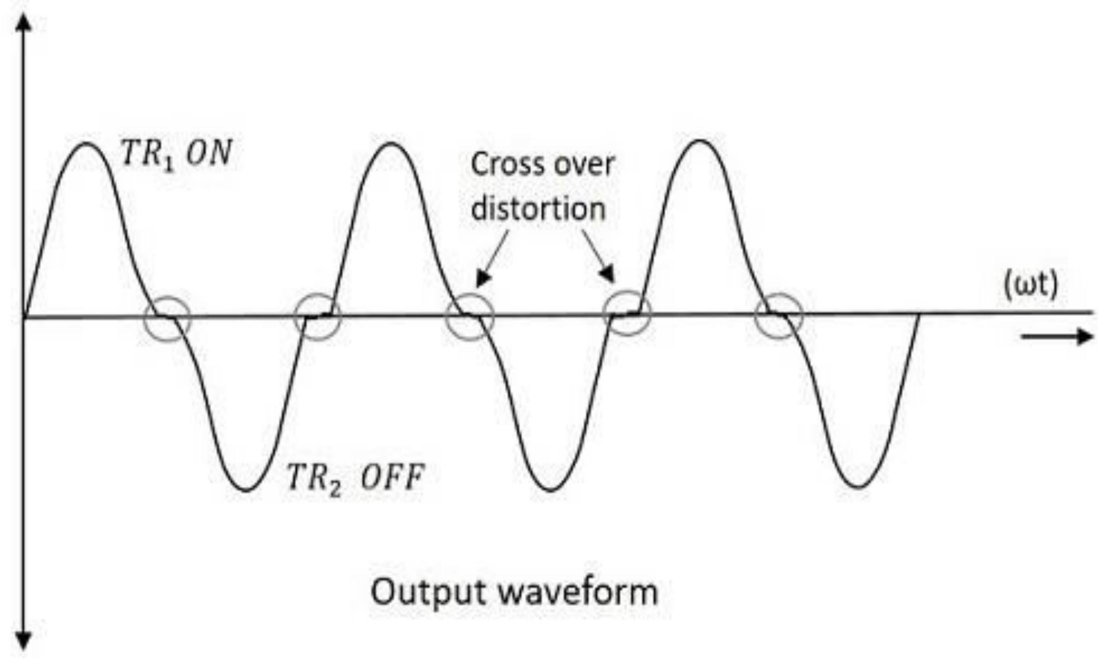
The class A and class B amplifier so far discussed has got few limitations. Let us now try to combine these two to get a new circuit which would have all the advantages of both class A and class B amplifier without their inefficiencies. Before that, let us also go through another important problem, called as **Cross over distortion**, the output of class B encounters with.

Cross-over Distortion

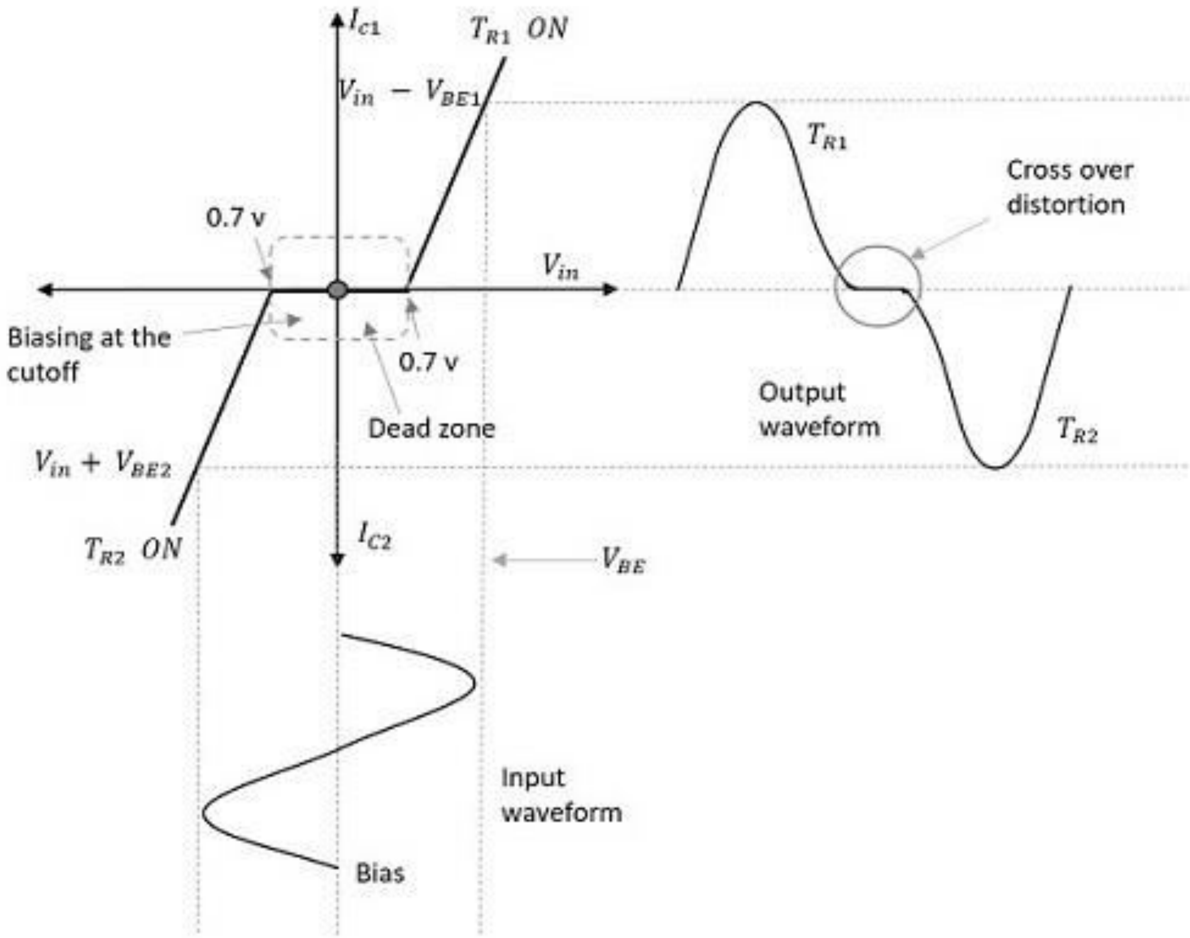
In the push-pull configuration, the two identical transistors get into conduction, one after the other and the output produced will be the combination of both.

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 can be more clearly understood through the non-linear characteristic of the waveform as shown below.



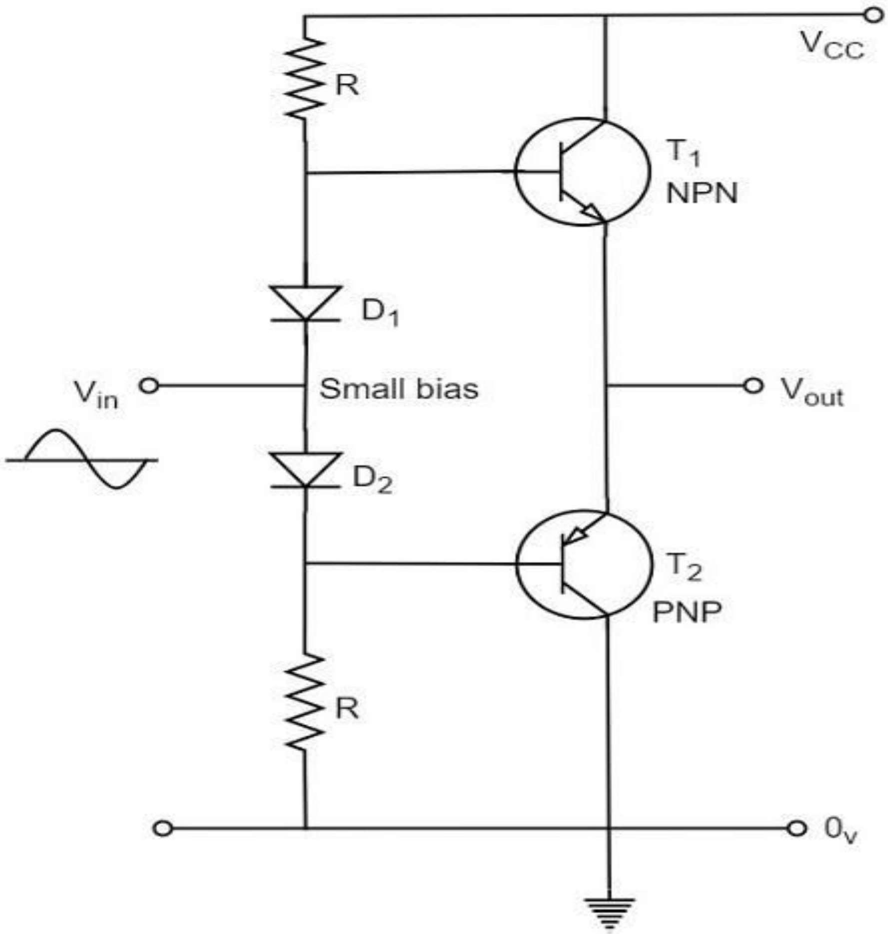
It is understood that this cross-over distortion is less pronounced for large input signals, where as it causes severe disturbance for small input signals. This cross over distortion can be eliminated if the conduction of the amplifier is more than one half cycle, so that both the transistors won't be OFF at the same time.

This idea leads to the invention of class AB amplifier, which is the combination of both class A and class B amplifiers, as discussed below.

Class AB Power Amplifier

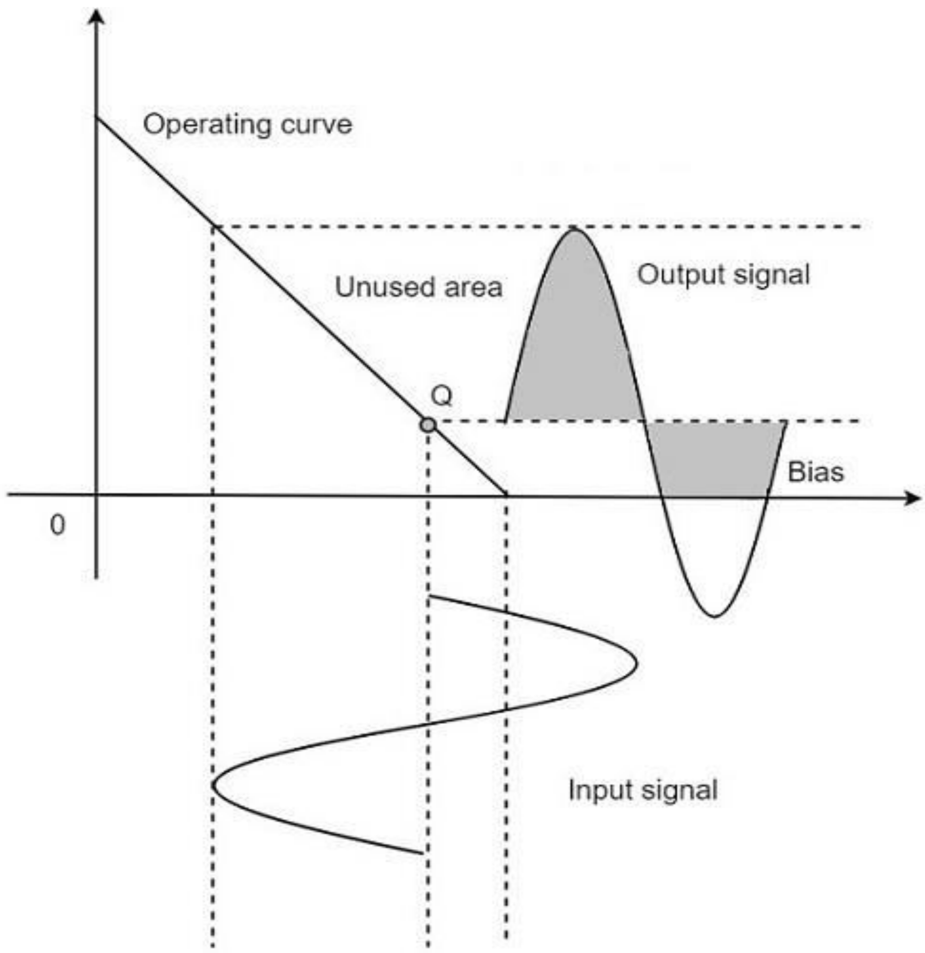
As the name implies, class AB is a combination of class A and class B type of amplifiers. As class A has the problem of low efficiency and class B has distortion problem, this class AB is emerged to eliminate these two problems, by utilizing the advantages of both the classes.

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



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

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



The small bias voltage given using diodes D_1 and D_2, as shown in the above figure, helps the operating point to be above the cutoff point. Hence the output waveform of class AB results as seen in the above figure. The crossover distortion created by class B is overcome by this class AB, as well the inefficiencies of class A and B don't affect the circuit.

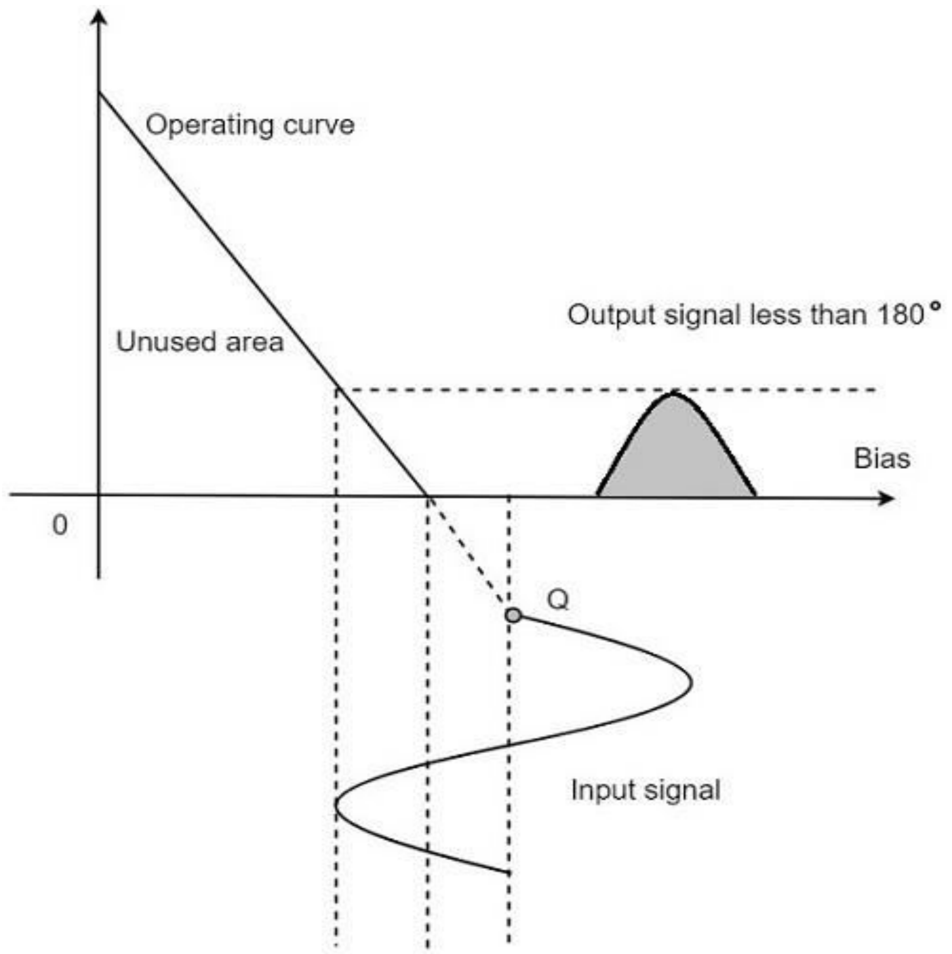
So, the class AB is a good compromise between class A and class B in terms of efficiency and linearity having the efficiency reaching about 50% to 60%. The class A, B and AB amplifiers are called as **linear amplifiers** because the output signal amplitude and phase are linearly related to the input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

Class C Power Amplifier

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

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

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



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

TUNED AMPLIFIERS

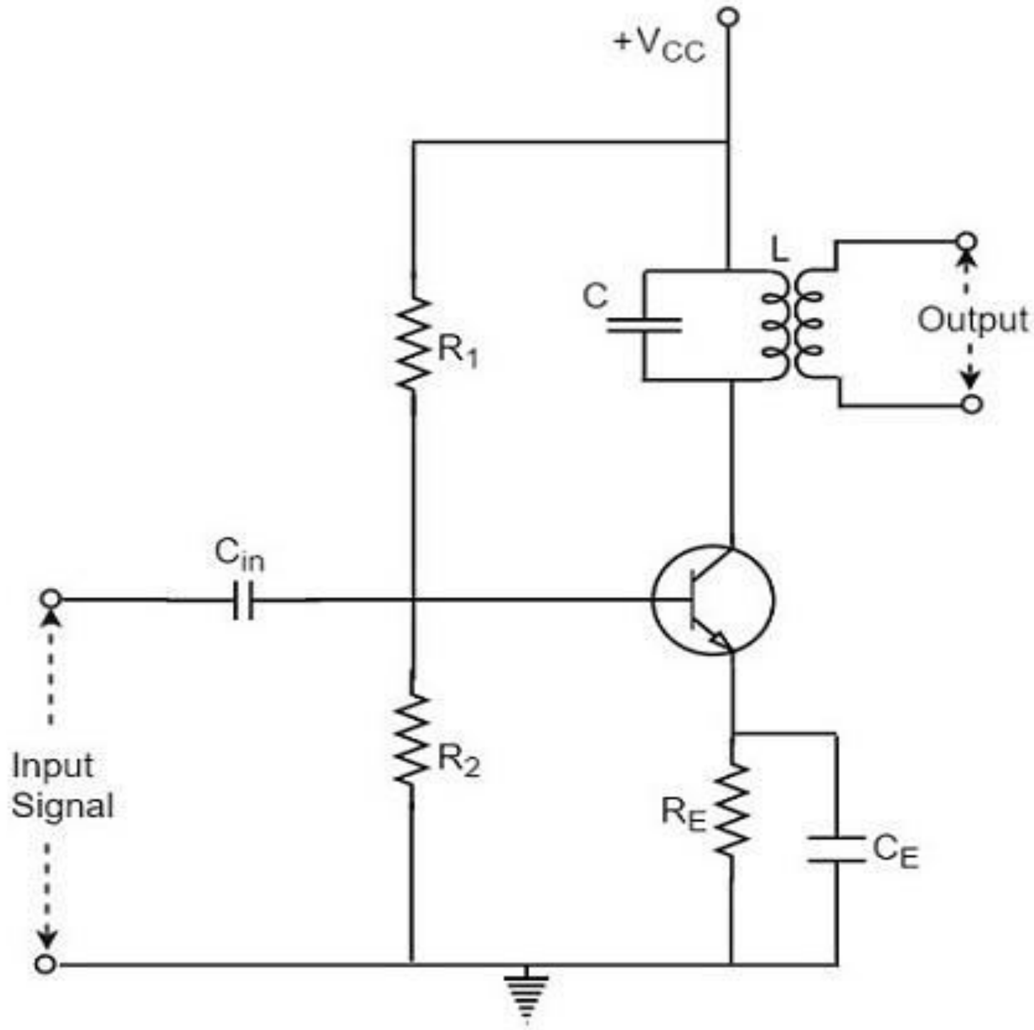
The types of amplifiers that we have discussed so far cannot work effectively at radio frequencies, even though they are good at audio frequencies. Also, the gain of these amplifiers is such that it will not vary according to the frequency of the signal, over a wide range. This allows the amplification of the signal equally well over a range of frequencies and does not permit the selection of particular desired frequency while rejecting the other frequencies.

So, there occurs a need for a circuit which can select as well as amplify. So, an amplifier circuit along with a selection, such as a tuned circuit makes a **Tuned amplifier**.

WHAT IS A TUNED AMPLIFIER?

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

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



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

When the reactance of the inductor balances the reactance of the capacitor, in the tuned circuit at some frequency, such a frequency can be called as **resonant frequency**. It is denoted by $\mathbf{f_r}$.

The formula for resonance is

$$2\pi f_L = 1/2\pi f_c$$

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

TYPES OF TUNED CIRCUITS

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

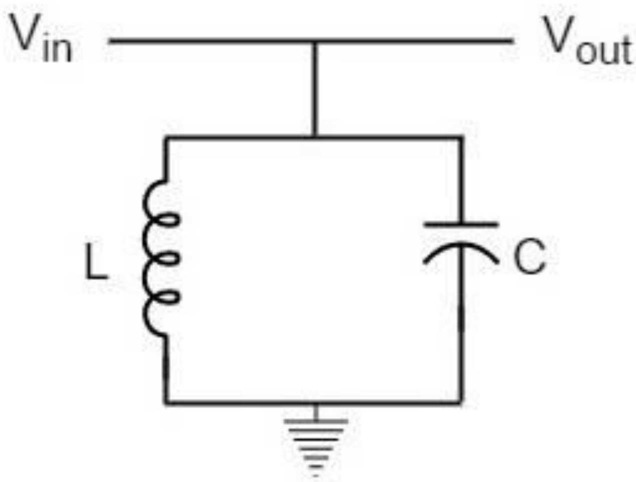
SERIES TUNED CIRCUIT

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

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

PARALLEL TUNED CIRCUIT

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



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

CHARACTERISTICS OF A PARALLEL TUNED CIRCUIT

The frequency at which parallel resonance occurs (i.e., reactive component of circuit current becomes zero) is called the resonant frequency $\mathbf{f_r}$. The main characteristics of a tuned circuit are as follows.

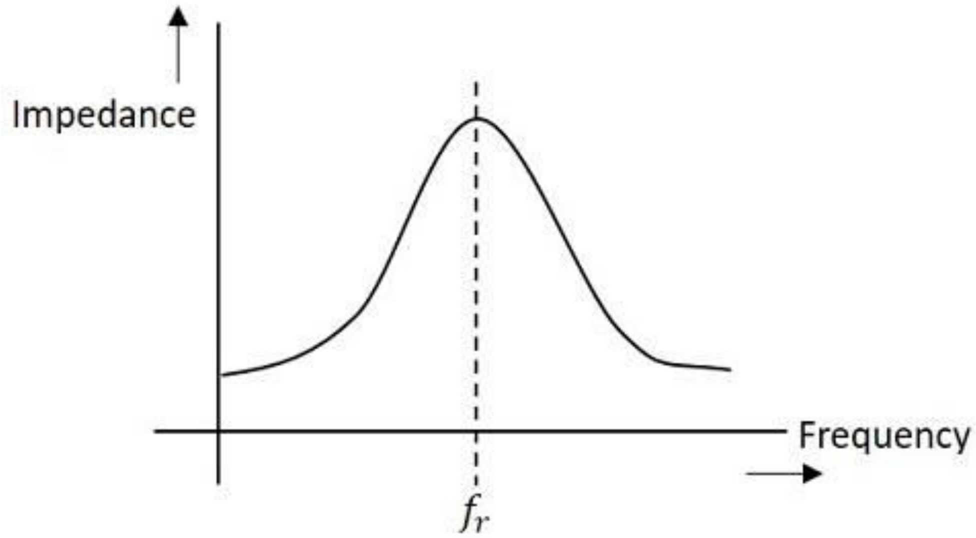
IMPEDANCE

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

$$\text{Supply voltage / Line current} = V / I$$

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

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



Impedance of the circuit decreases for the values above and below the resonant frequency $\mathbf{f_r}$. Hence the selection of a particular frequency and rejection of other frequencies is possible.

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

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

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

$$1 / Zr = R / Z^2_L$$

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

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

Therefore, circuit impedance Z_r is obtained as

$$Z_R = L / CR$$

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

CIRCUIT CURRENT

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

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

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

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

QUALITY FACTOR

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

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

$$Q = X_L / R = 2\pi frL / R$$

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

ADVANTAGES OF TUNED AMPLIFIERS

The following are the advantages of tuned amplifiers.

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

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

FREQUENCY RESPONSE OF TUNED AMPLIFIER

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

The voltage gain of such an amplifier is given by

$$\text{Voltage gain} = \beta Z_C \,/\, Z_{in}$$

Where Z_C = effective collector load and Z_{in} = input impedance of the amplifier.

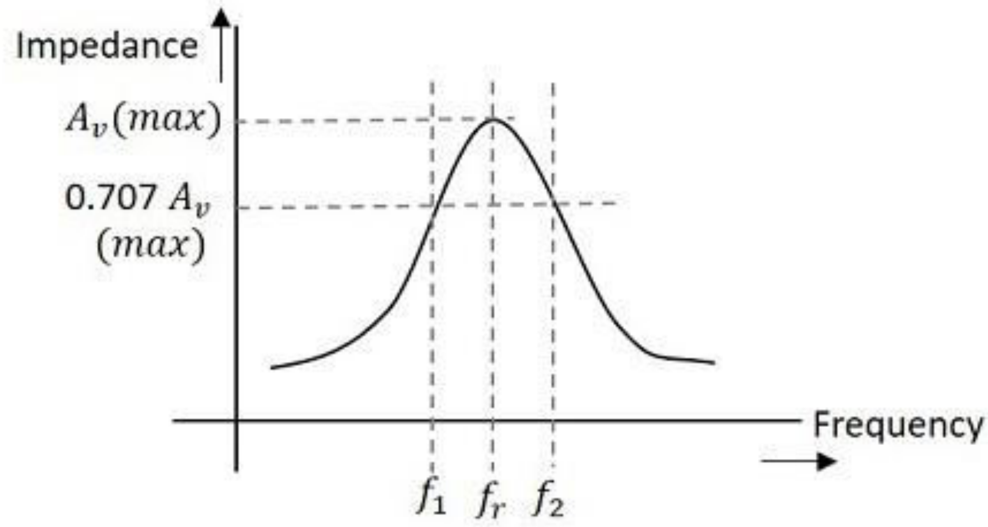
The value of Z_C depends upon the frequency of the tuned amplifier. As Z_C is maximum at resonant frequency, the gain of the amplifier is maximum at this resonant frequency.

BANDWIDTH

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

The range of frequencies between f_1 and f_2 is called as bandwidth of the tuned amplifier. The bandwidth of a tuned amplifier depends upon the Q of the LC circuit i.e., upon the sharpness of the frequency response. The value of Q and the bandwidth are inversely proportional.

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



RELATION BETWEEN Q AND BANDWIDTH

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

$$Q = fr / BW$$

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

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

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

TRANSISTOR AS A SWITCH

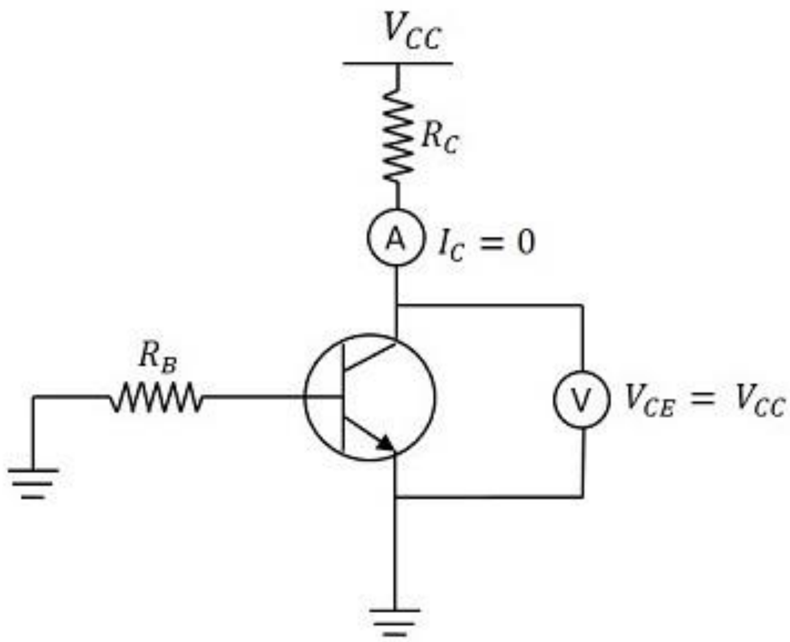
A **transistor** is used as an electronic switch by driving it either in **saturation** or in **cut off**. The region between these two is the linear region. A transistor works as a linear amplifier in this region. The Saturation and Cut **off** states are important consideration in this regard.

ON & OFF STATES OF A TRANSISTOR

There are two main regions in the operation of a transistor which we can consider as **ON** and **OFF** states. They are saturation and cut **off** states. Let us have a look at the behavior of a transistor in those two states.

OPERATION IN CUT-OFF CONDITION

The following figure shows a transistor in cut-off region.



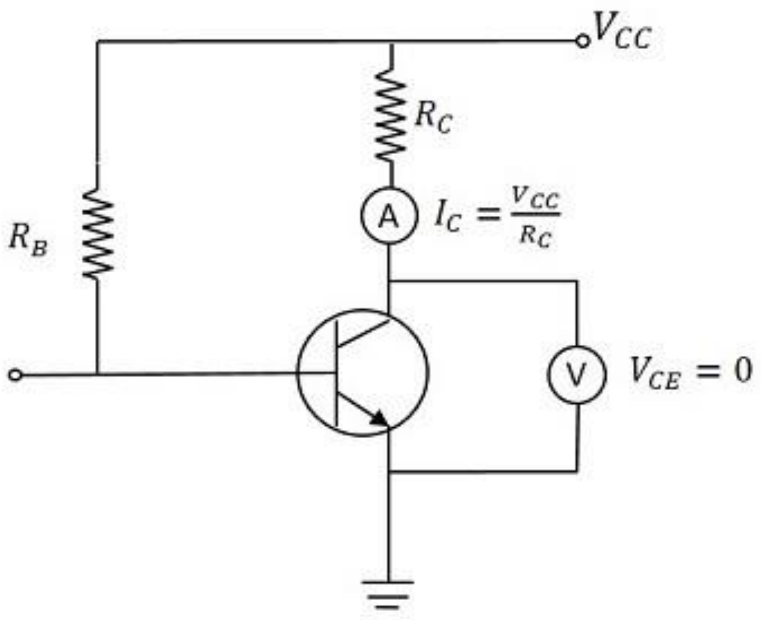
When the base of the transistor is given negative, the transistor goes to cut off state. There is no collector current. Hence $I_C = 0$.

The voltage V_{CC} applied at the collector, appears across the collector resistor R_C. Therefore,

$$V_{CE} = V_{CC}$$

OPERATION IN SATURATION REGION

The following figure shows a transistor in saturation region.



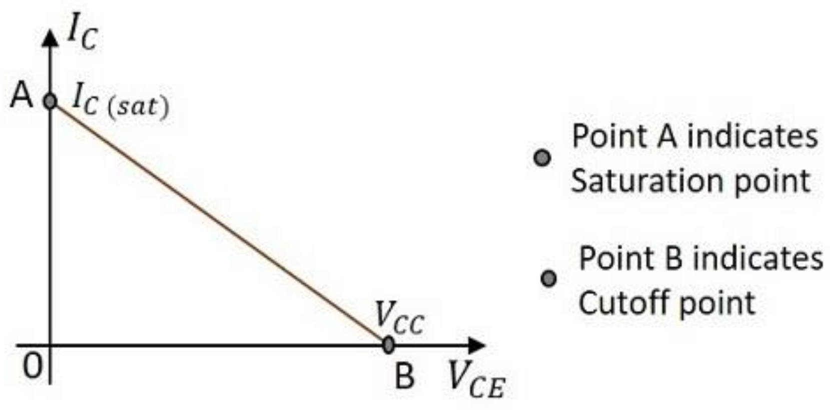
When the base voltage is positive and transistor goes into saturation, I_C flows through R_C.

Then V_{CC} drops across R_C. The output will be zero.

$$IC = IC\ (sat) = VCC\ /RC \text{ and } VCE = 0$$

Actually, this is the ideal condition. Practically, some leakage current flows. Hence, we can understand that a transistor works as a switch when driven into saturation and cut off regions by applying positive and negative voltages to the base.

The following figure gives a better explanation.



Graph indicating DC Load line

Observe the dc load line that connects the I_C and V_{CC}. If the transistor is driven into saturation, I_C flows completely and $V_{CE} = 0$ which is indicated by the point **A**.

If the transistor is driven into cut off, I_C will be zero and $V_{CE} = V_{CC}$ which is indicated by the point B. the line joining the saturation point A and cut off B is called as **Load line**. As the voltage applied here is dc, it is called as **DC Load line**.

PRACTICAL CONSIDERATIONS

Though the above-mentioned conditions are all convincing, there are a few practical limitations for such results to occur.

During the Cut off state

An ideal transistor has $V_{CE} = V_{CC}$ and $I_C = 0$.

But in practice, a smaller leakage current flows through the collector.

Hence I_C will be a few μA.

This is called as **Collector Leakage Current** which is of course, negligible.

DURING THE SATURATION STATE

An ideal transistor has $V_{CE} = 0$ and $I_C = I_{C\,(sat)}$.

But in practice, V_{CE} decreases to some value called **knee voltage**.

When V_{CE} decreases more than knee voltage, β decreases sharply.

As $I_C = \beta I_B$ this decreases the collector current.

Hence that maximum current I_C which maintains V_{CE} at knee voltage, is known as **Saturation Collector Current**.

$$\text{Saturation Collector Current} = IC_{(sat)} = (VCC - V_{knee}) / R_C$$

A Transistor which is fabricated only to make it work for switching purposes is called as **Switching Transistor**. This works either in Saturation or in Cut off region. While in saturation state, the **collector saturation current** flows through the load and while in cut off state, the **collector leakage current** flows through the load.

SWITCHING ACTION OF A TRANSISTOR

A Transistor has three regions of operation. To understand the efficiency of operation, the practical losses are to be considered. So let us try to get an idea on how efficiently a transistor works as a switch.

During Cut off (OFF) state

$$\text{The Base current } I_B = 0$$

$$\text{The Collector current } I_C = I_{CEO} \text{ (collector leakage current)}$$

$$\text{Power Loss} = \text{Output Voltage} \times \text{Output Current}$$

$$= V_{CC} \times I_{CEO}$$

As I_{CEO} is very small and V_{CC} is also low, the loss will be of very low value. Hence, a transistor works as an efficient switch in OFF state.

During Saturation (ON) State

As discussed earlier,

$$I_{C\,(sat)} = (V_{CC} - V_{knee}) / R_C$$

The output voltage is $\mathbf{V_{knee}}$.

$$\text{Power loss} = \text{Output Voltage} \times \text{Output Current}$$

$$= V_{knee} \times I_{C(sat)}$$

As V_{knee} will be of small value, the loss is low. Hence, a transistor works as an efficient switch in ON state.

During Active Region

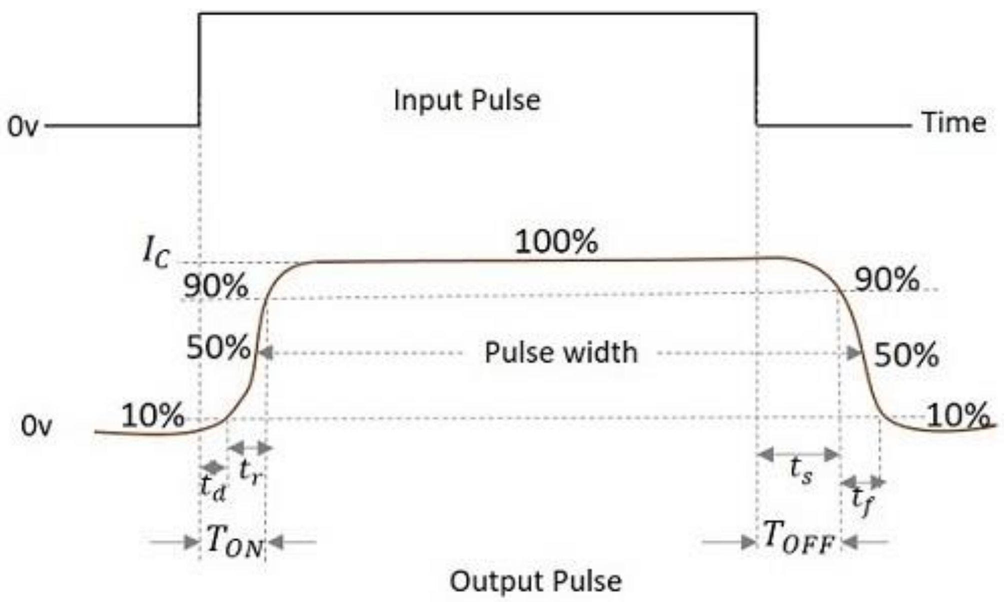
The transistor lies between ON & OFF states. The transistor operates as a linear amplifier where small changes in input current cause large changes in the output current (ΔI_C).

Switching Times

The Switching transistor has a pulse as an input and a pulse with few variations will be the output. There are a few terms that you should know regarding the timings of the switching output pulse. Let us go through them.

Let the input pulse duration = **T**

When the input pulse is applied the collector current takes some time to reach the steady state value, due to the stray capacitances. The following figure explains this concept.



From the figure above,

- **Time delay(t_d)** – The time taken by the collector current to reach from its initial value to 10% of its final value is called as the **Time Delay**.
- **Rise time(t_r)** – The time taken for the collector current to reach from 10% of its initial value to 90% of its final value is called as the **Rise Time**.
- **Turn-on time (T_{ON})** – The sum of time delay (t_d) and rise time (t_r) is called as **Turn-on time**.

$$T_{ON} = t_d + t_r$$

- **Storage time (t_s)** – The time interval between the trailing edge of the input pulse to the 90% of the maximum value of the output, is called as the **Storage time**.
- **Fall time (t_f)** – The time taken for the collector current to reach from 90% of its maximum value to 10% of its initial value is called as the **Fall Time**.
- **Turn-off time (T_{OFF})** – The sum of storage time (t_s) and fall time (t_f) is defined as the **Turn-off time**.

$$T_{OFF} = t_s + t_f$$

- **Pulse Width(W)** – The time duration of the output pulse measured between two 50% levels of rising and falling waveform is defined as **Pulse Width**.