UNIVERSITY INSTITUTE OF TECHNOLOGY
THE UNIVERSITY OF BURDWAN

A Major/Minor Project

on

TO DESIGN AN ARBITRARY WAVEFORM IN (12.5-17.5) KHz

Electronics & Communication Engineering (B.E)

<u>SUBMITTED BY:</u> Rupak Dutta

<u>UNIVERSITY ROLL NO:</u> L 2018-2071

<u>UNDER THE GUIDANCE OF:</u>

Ms. SMITA HAZRA

(Assistant Professor)

____

Signature of Mentor

TO DESIGN AN ARBITARY WAVEFORM IN (12.5-17.5) KHz

THESIS SUBMITTED TO

UNIVERSITY INSTITUTE OF TECHNOLOGY, BURDWAN

For the partial fulfilments for the award of degree

*This is to certify that the thesis entitled "**TO DESIGN AN ARBITRARY WAVEFORM IN** **(12.5-17.5) KHz**", submitted by **Rupak Dutta** of 4th year Bachelor of Engineering in Electronics and Communication Engineering students of University Institute of Technology for the award of project, that is a record of this research work carried out by us under my supervision and guidance. The thesis has fulfilled the requirements as per the regulations of the Institute and in my opinion has reached the standard needed for submission.*

Guided By

Ms. SMITA HAZRA
Assistant Professor,
Department of ECE, UIT Burdwan

Certified By

Ms. SMITA HAZRA
Assistant Professor,
Department of ECE, UIT Burdwan

<u>DECLARATION</u>

This is to certify that Report entitled "TO DESIGN AN ARBITRARY WAVEFORM IN (12.5-17.5) KHz" which is submitted by me for major project of the degree ***Bachelor of Engineering*** in Electronics & Communication Engineering to ***UNIVERSITY INSTITUTE OF TECHNOLOGY, University of Burdwan***.

Signature:
Name: Rupak Dutta
Roll no.: L 2018-2071
Date:

Date:
Approved by HOD
Electronics & Communication Engineering
UIT, BURDWAN

CERTIFICATE

This is to certify that the thesis entitled **"TO DESIGN AN ARBITRARY WAVEFORM IN (12.5-17.5) KHz"** is an authentic record of the work carried out by Rupak Dutta (L2018-2071) under my guidance and supervision for the partial fulfilment of the requirement for the degree of Bachelor of Engineering in Electronics and communication. In my opinion, this thesis satisfies the requirements for which it is submitted. To the best of my knowledge and belief, it has not been submitted to any other institute or university for the award of any degree.

Signature:

Date:

Guide: **Ms. SMITA HAZRA**

Assistant Professor
Department of Electronics and Communication Engineering
UNIVERSITY INSTITUTE OF TECHNOLOGY
The University of Burdwan

ACKNOWLEDGEMENT

I would like to express my deepest gratitude for the successful completion of my project to my lecturers, friends and family. I thank them from the bottom of my heart for their kind support and assistance to my never-ending questionnaires which have guided and enlightened me, making me even better person with full of confidence and perseverance to face future challenges. I feel extremely fortunate to be surrounded by great people providing me so much help during my final year project.

I would like to express my sincere thanks to my mentor Ms. SMITA HAZRA, Assistant Professor, Department of ECE, UIT Burdwan for her valuable advice and guidance. She has imparted new knowledges to me, sacrificing his precious time and giving me aspiring advice and constructive criticism. Her continuous support and comments inspired and motivated me to widen my research from various perspectives. I truly appreciate her for her painstakingly perusal of my project in ensuring correct input of materials, contents and connectivity.

I am thankful to my dear friends and course-mates for their participation and stimulating discussion relevant to my project. Also, I would like to thank the participants involved in my experiment, who have willingly shared their precious time with me.

Last but not the least, I would like to express my sincere appreciation to other faculty members of this department Dr. Partha Pratim Sarkar, Ms. Smita Hazra, Mr. Shantanu Mondal, Dr. Dibyendu Roy, Mr. Niladri Halder and Mr. Koushik Mathur for their guidance without which I would have been lost.

Thanking You,

Rupak Dutta

CONTENTS

Topic

1. Preface

This project represents a prototype experiment regarding the practical implementation of an affordable Arbitrary Waveform Generator. This project motive is to design an arbitrary waveform in frequency 12.5KHz to 17.5KHz which has the capability of producing two types of signal waveforms at its output signal. The output waveforms are sine wave and square wave. The frequencies of these waveforms may be adjusted from 12.5KHz to 17.5KHz.

2. Arbitrary Waveform Generator

Arbitrary Waveform Generators meet the demand of today's engineers offering a wide variety of signal types: analog, digital, mixed-signal and multi-channel applications can be covered by Active Technologies instruments.

The AWG (Arbitrary Waveform Generator) and AFG (Arbitrary Function Generator) family helps you to generate with confidence waveforms of almost any imaginable shape: complex signals like digital modulations, RF stimuli for functional and performance tests, radar simulation, optical communication and electronic warfare can be created through an easy-to-use interface.

Arbitrary waveforms generators can also be referred to by their initials, AWG, and sometimes they are even called and ARB - short for arbitrary.

The waveforms produced by arbitrary waveform generators, AWGs can be either repetitive or sometimes just a single-shot. If the AWG waveform is only a single shot, then a triggering mechanism is needed to trigger the AWG and possibly the measuring instrument.

The AWG is able to generate an arbitrary waveform defined by a set of values, i.e., "waypoints" entered to set the value of the waveform at specific times. They can make up a digital or even an analogue waveform.

As a result, an arbitrary waveform generator is a form of test equipment that is able to produce virtually any waveshape that is required.

Before investing in a separate arbitrary waveform generator test instrument, it is worth investigating whether one already exists and is available. Very many modern oscilloscopes contain an AWG which is integrated into the scope and can often integrate with the scope functions to capture waveforms.

Fig. No.- 01: Arbitrary Waveform Generator (P.I- Tektronix)

2.1. Arbitrary Waveform Generator techniques

There are a number of ways of designing arbitrary waveform generators. They are based around digital techniques, and their design falls into one of the main categories below:

- ***Direct Digital Synthesis, DDS:*** This type of arbitrary waveform generator is based around the DDS types of frequency synthesizer, and sometimes it may be referred to as an Arbitrary Function Generator, AFG.

- ***Variable-clock arbitrary waveform generator:*** The variable clock arbitrary function generator is the more flexible form of arbitrary waveform generator. These arbitrary waveform generators are generally more flexible, although they do have some limitations not possessed by the DDS versions. Sometimes these generators are referred to as just arbitrary waveform generators, AWGs rather than arbitrary function generators.

- ***Combined arbitrary waveform generator***: This format of AWG combines both of the other forms including the DDS and variable clock techniques. In this way the advantages of both systems can be realised within a single item of test equipment.

2.2. Arbitrary Waveform Generator Capabilities

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

- ***Sine wave:*** An Arbitrary Waveform Generator will normally be able to act as a sine wave generator. This is the standard waveform that oscillates between two levels with a standard sinusoidal shape. Using the function generator as a sine wave generator is one of the more commonly used applications. Sine waves are widely used in testing applications.

Fig. No.-02: Sine Wave

- ***Square wave:*** Another very widely used waveform is the square wave. It consists of a signal moving directly between high and low levels. Used as a square wave generator, this test instrument provides a very useful source of a basic digital waveform.

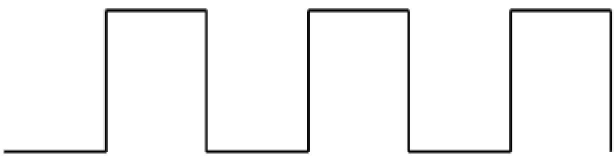


Fig. No.-03: Square wave

- ***Pulse:*** A pulse waveform is another type that can be produced by a function generator. It is effectively the same as a square wave, but with the mark space ratio very different to 1:1. This form of waveform is again often used within digital applications.

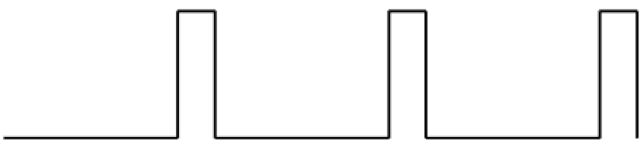


Fig. No.-04: Pulse

- ***Triangular wave:*** This form of signal produced by the function generator linearly moves between a high and low point. This form of waveform is often generated using an operational amplifier acting as an integrator. The triangular waveform generator typically also has a square wave output as well, and it is used as the basis for generating all the waveforms in a function generator test instrument.

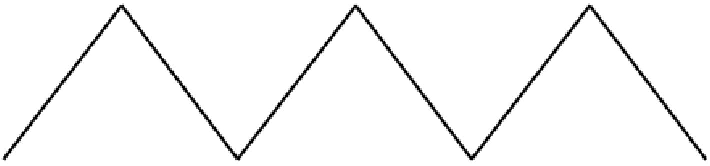


Fig. No.-05: Triangular wave

- The triangular waveform is often used in testing amplifiers – it is far easier to see distortion and clipping on a triangular waveform than it is on a sine waveform.

- ***Sawtooth wave:*** Again, this is a triangular waveform, but with the rise edge of the waveform faster or slower than the fall, making a form of shape similar to a sawtooth. It is generated by the same circuit as the triangular waveform, but with the different rise and fall times created by changing the charge rate for the rise and fall elements of the integrator.

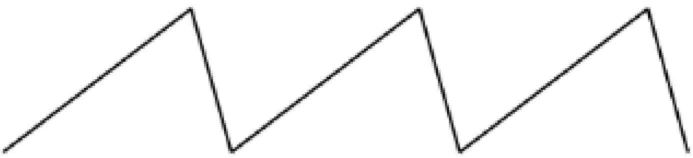


Fig. No.-06: Triangular wave

These are the basic waveforms that are produced within a function generator test instrument. These waveforms satisfy most of the needs for testing a number of items. Where specialised waveforms are required, then an arbitrary waveform generator is required.

3. Why Arbitrary Waveform Generator

Arbitrary Waveform Generator are often used in applying a known sequence of digital words to a digital system. The ability to define and generate custom waveforms is so useful that we will take a closer look at a typical arbitrary waveform generator (AWG).

4. How to use an Arbitrary Waveform Generator

This is how to use a function generator to test a circuit's behaviour:

1. Power on the generator and select the desired output signal: square wave, sine wave or triangle wave.
2. Connect the output leads to an oscilloscope to visualize the output signal and set its parameters using the amplitude and frequency controls.
3. Attach the output leads of the function generator to the input of the circuit you wish to test.
4. Attach the output of your circuit to a meter or oscilloscope to visualize the resulting change in signal.

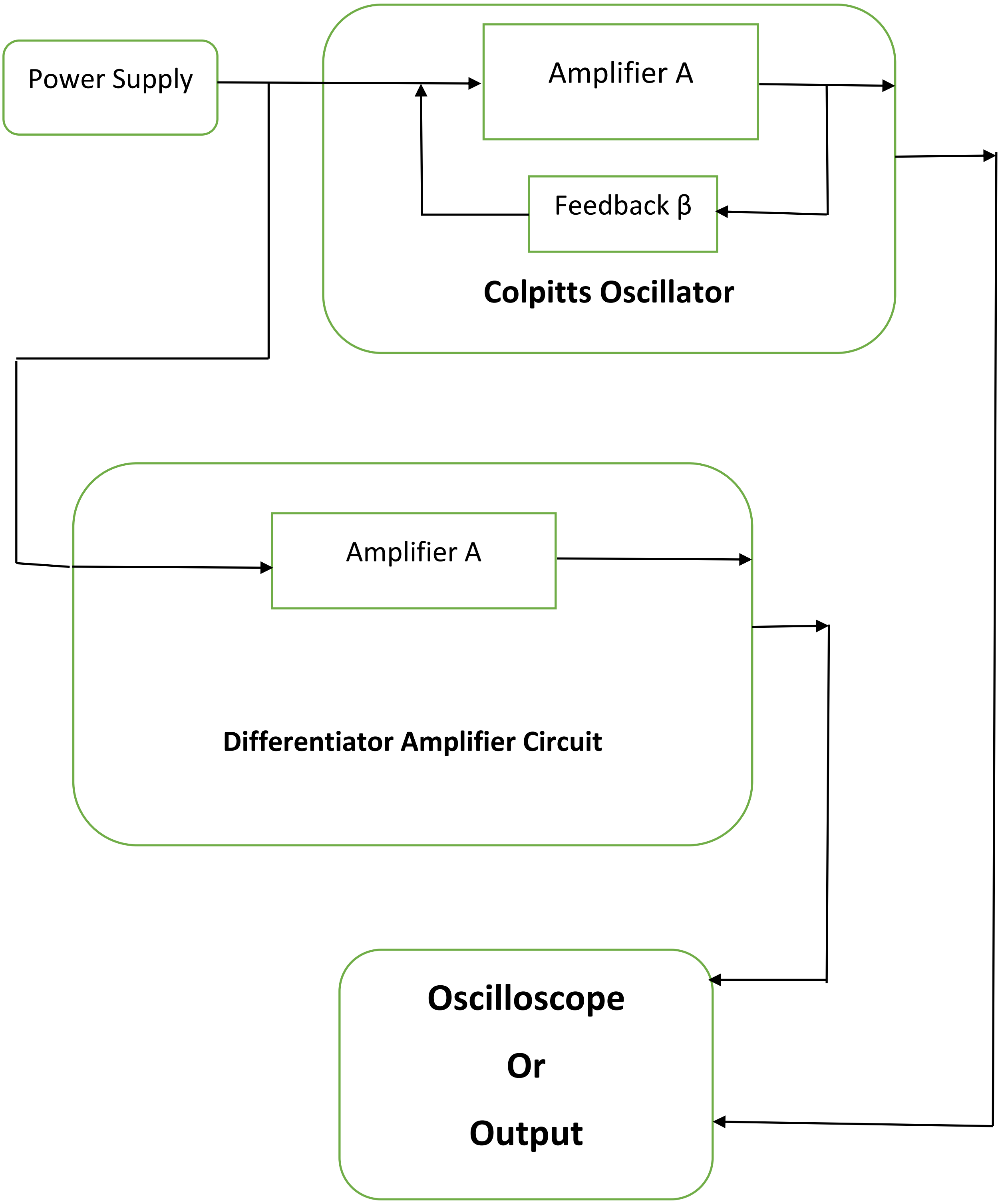
5. User requirement analysis

To perform this project first of all we have to know about the *Colpitts Oscillator* for the *Sine Wave* generation and for *Square Wave* generation we have to know about the *Band Pass Filter* and the *Narrow Band Pass Filter*.

6. Description of Hardware Used

Component Required	Description	Component's type
Resistor	In electronic circuits, resistors are used to reduce current flow, adjust signal levels, to divide voltages, bias active elements, and terminate transmission lines, among other uses.	200KΩ 220 KΩ 500 Ω 220 Ω
Capacitor	A capacitor is a device that stores electrical energy in an electric field. It is a passive electronic component with two terminals. The effect of a capacitor is known as capacitance	4 nF 10 nF 0.1 uF 0.9 nF 0.1 uF
Inductor	An inductor, also called a coil is a passive two-terminal electrical component that stores energy in a magnetic field when electric current flows through it.	101mH 35mH to 90mH (variable inductor)
Transistor	A transistor is a semiconductor device used to amplify or switch electronic signals and electrical power. Transistors are one of the basic building blocks of modern electronics.	BC-547
Amplifier	An operational amplifier is a DC-coupled high-gain electronic voltage amplifier with a differential input and, usually, a single-ended output	LM – 741
Transformer	The transformer is a device that steps up or steps down voltage and basically a voltage control device that is used widely in the distribution and transmission of alternating current power.	12V Step down transformer
Voltage Regulator	Voltage regulator is a type of self-contained fixed linear voltage regulator integrated circuit. The IC belongs to ic 78xx voltage regulator family.	LM-7812
Diode		IN4007
Connection wires		
Bread board		

7. Block Diagram



7.1. Power Supply

- A DC power supply, also known as a bench power supply, is a type of power supply that gives direct current (DC) voltage to power a device.
- A DC power supply management subsystem can use AC, DC, battery, or ultralow voltage as inputs.
- Step down transformation, rectification, DC filtration, and regulation are the four major steps in a regulated DC power supply block diagram.

7.1.1. Circuit

In this power supply we use LM7812 voltage regulator. main function of voltage regulator is given us exactly 12v output.

We use diode bridge because its convert AC volt to DC volt.

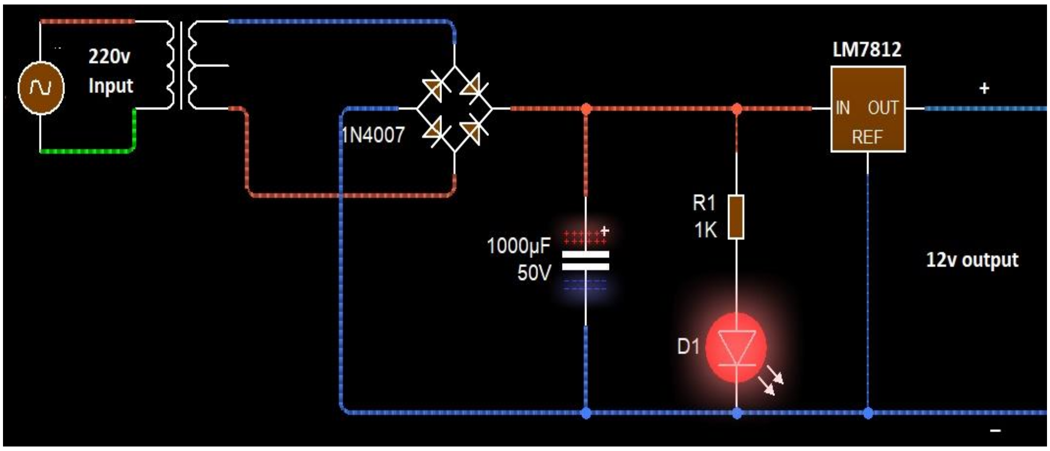


Fig. No.-07: power supply circuit (P.I- easy creative projects)

7.1.2. LM7812

LM7812 Voltage Regulator Pin out:

LM7812 voltage regulator has 3 pins.

- 1st Input
- 2nd Ground
- 3rd Output

main function of voltage regulator it provides exactly 12v output.

7.2. Colpitts Oscillator

In many ways, the Colpitts oscillator is the exact opposite of the **Hartley Oscillator**. Just like the Hartley oscillator, the tuned tank circuit consists of an LC resonance sub-circuit connected between the collector and the base of a single stage transistor amplifier producing a sinusoidal output waveform.

The basic configuration of the **Colpitts Oscillator** resembles that of the *Hartley Oscillator* but the difference this time is that the centre tapping of the tank sub-circuit is now made at the junction of a “capacitive voltage divider” network instead of a tapped autotransformer type inductor as in the Hartley oscillator.

7.2.1. Colpitts Oscillator Tank Circuit

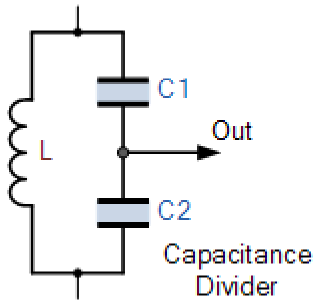


Fig. No.-08: Colpitts Oscillator Tank Circuit (P.I- electronics tutorial)

The Colpitts oscillator uses a capacitive voltage divider network as its feedback source. The two capacitors, C1 and C2 are placed across a single common inductor, L as shown. Then C1, C2 and L form the tuned tank circuit with the condition for oscillations being: $X_{C1} + X_{C2} = X_L$, the same as for the Hartley oscillator circuit.

The advantage of this type of capacitive circuit configuration is that with less self and mutual inductance within the tank circuit, frequency stability of the oscillator is improved along with a simpler design.

As with the Hartley oscillator, the Colpitts oscillator uses a single stage bipolar transistor amplifier as the gain element which produces a sinusoidal output. Consider the circuit below.

7.2.2. Basic Colpitts Oscillator Circuit

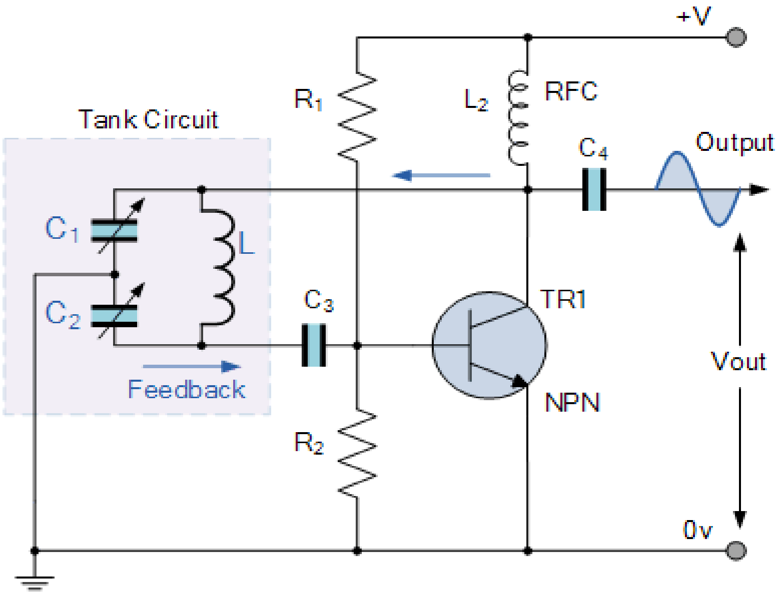


Fig. No.-09: Basic circuit (P.I- electronics tutorial)

The emitter terminal of the transistor is effectively connected to the junction of the two capacitors, C1 and C2 which are connected in series and act as a simple voltage divider. When the power supply is firstly applied, capacitors C1 and C2 charge up and then discharge through the coil L. The oscillations across the capacitors are applied to the base-emitter junction and appear in the amplified at the collector output.

Resistors, R1 and R2 provide the usual stabilizing DC bias for the transistor in the normal manner while the additional capacitors act as a DC-blocking bypass capacitors. A radio-frequency choke (RFC) is used in the collector circuit to provide a high reactance (ideally open circuit) at the frequency of oscillation, (fr) and a low resistance at DC to help start the oscillations.

The required external phase shift is obtained in a similar manner to that in the Hartley oscillator circuit with the required positive feedback obtained for sustained undamped oscillations. The amount of feedback is determined by the ratio of C1 and C2. These two capacitances are generally “ganged” together to provide a constant amount of feedback so that as one is adjusted the other automatically follows.

The frequency of oscillations for a Colpitts oscillator is determined by the resonant frequency of the LC tank circuit and is given as:

$$f_r = \frac{1}{2\pi\sqrt{L C_T}}$$

where C_T is the capacitance of C1 and C2 connected in series and is given as:

$$\frac{1}{C_T} = \frac{1}{C_1} + \frac{1}{C_2} \quad \text{or} \quad C_T = \frac{C_1 \times C_2}{C_1 + C_2}$$

The configuration of the transistor amplifier is of a *Common Emitter Amplifier* with the output signal 180^o out of phase with regards to the input signal. The additional 180^o phase shift require for oscillation is achieved by the fact that the two capacitors are connected together in series but in parallel with the inductive coil resulting in overall phase shift of the circuit being zero or 360^o.

The amount of feedback depends on the values of C1 and C2. We can see that the voltage across C1 is the same as the oscillators output voltage, V_{out} and that the voltage across C2 is the oscillators feedback voltage. Then the voltage across C1 will be much greater than that across C2.

Therefore, by changing the values of capacitors, C1 and C2 we can adjust the amount of feedback voltage returned to the tank circuit. However, large amounts of feedback may cause the output sine wave to become distorted, while small amounts of feedback may not allow the circuit to oscillate.

Then the amount of feedback developed by the Colpitts oscillator is based on the capacitance ratio of C1 and C2 and is what governs the the excitation of the oscillator. This ratio is called the “feedback fraction” and is given simply as:

$$\text{Feedback Fraction} = \frac{C_1}{C_2}\,\%$$

7.2.3. Colpitts Oscillator Circuit

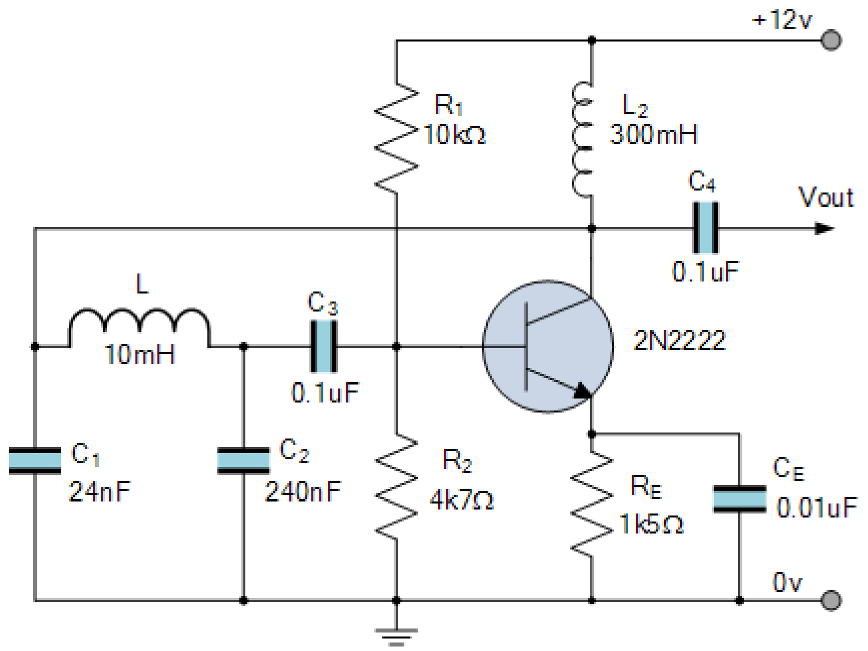


Fig. No.-10: Colpitts Oscillator circuit (P.I- electronics tutorial)

7.2.4. Colpitts Oscillator using an Op-amp

Just like the previous *Hartley Oscillator*, as well as using a bipolar junction transistor (BJT) as the oscillators active stage, we can also an operational amplifier, (op-amp). The operation of an **Op-amp Colpitts Oscillator** is exactly the same as for the transistorised version with the frequency of operation calculated in the same manner. Consider the circuit below.

7.2.5. Colpitts Oscillator Op-amp Circuit

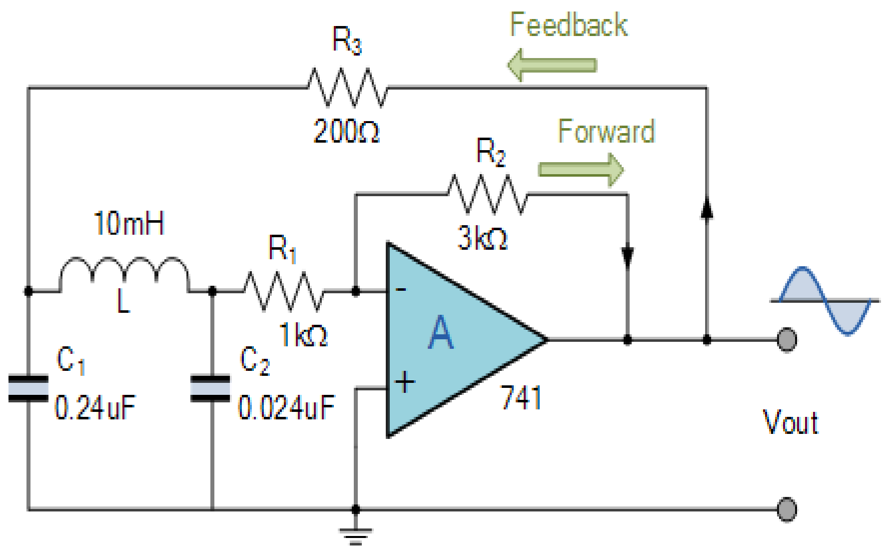


Fig. No.-11: Colpitts Oscillator Op-amp Circuit (P.I- electronics tutorial)

Note that being an inverting amplifier configuration, the ratio of R2/R1 sets the amplifiers gain. A minimum gain of 2.9 is required to start oscillations. Resistor R3 provides the required feedback to the LC tank circuit.

The advantages of the **Colpitts Oscillator** over the Hartley oscillators are that the Colpitts oscillator produces a purer sinusoidal waveform due to the low impedance paths of the capacitors at high frequencies. Also due to these capacitive reactance properties the FET based Colpitts oscillator can operate at very high frequencies. Of course, any op-amp or FET used as the amplifying device must be able to operate at the required high frequencies.

7.2.6. Colpitts Oscillator Summary

Then to summarise, the **Colpitts Oscillator** consists of a parallel LC resonator tank circuit whose feedback is achieved by way of a capacitive divider. Like most oscillator circuits, the Colpitts oscillator exists in several forms, with the most common form being similar to the transistor circuit above.

The centre tapping of the tank sub-circuit is made at the junction of a “capacitive voltage divider” network to feed a fraction of the output signal back to the emitter of the transistor. The two capacitors in series produce a 180^o phase shift which is inverted by another 180^o to produce the required positive feedback. The oscillating frequency which is a purer sine-wave voltage is determined by the resonance frequency of the tank circuit.

In the next tutorial about Oscillators, we will look at RC Oscillators which uses resistors and capacitors as its tank circuit to produce a sinusoidal waveform.

7.3. Differentiator Amplifier

The basic operational amplifier differentiator circuit produces an output signal which is the first derivative of the input signal.

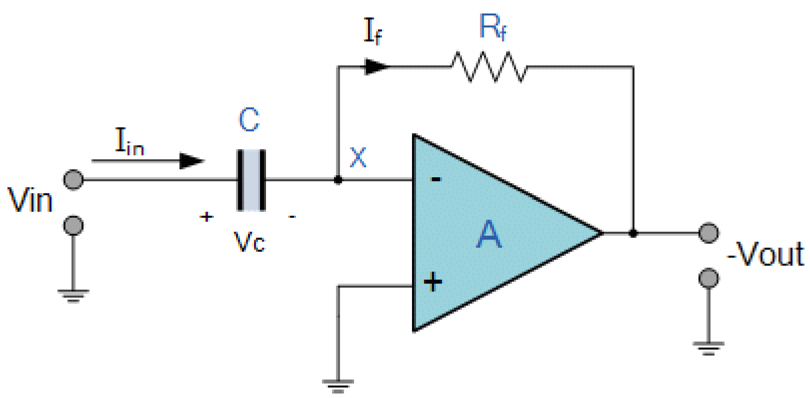


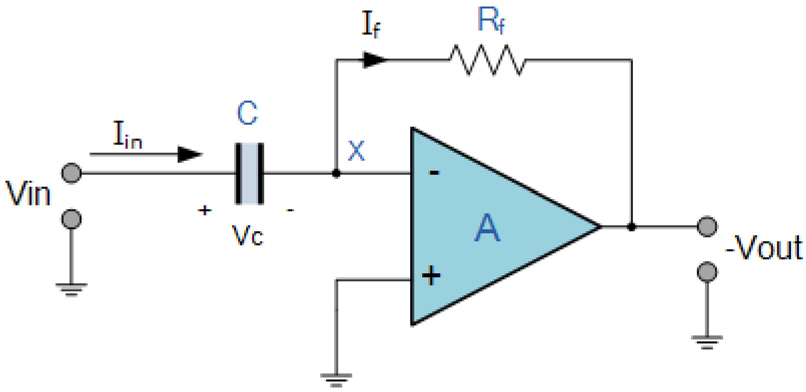
Fig. No.-12: Differentiator Amplifier Circuit (P.I- electronics tutorial)

Here, the position of the capacitor and resistor have been reversed and now the reactance, X_C is connected to the input terminal of the inverting amplifier while the resistor, $Rƒ$ forms the negative feedback element across the operational amplifier as normal.

This operational amplifier circuit performs the mathematical operation of **Differentiation**, that is it "*produces a voltage output which is directly proportional to the input voltage's rate-of-change with respect to time*". In other words the faster or larger the change to the input voltage signal, the greater the input current, the greater will be the output voltage change in response, becoming more of a "spike" in shape.

As with the integrator circuit, we have a resistor and capacitor forming an RC Network across the operational amplifier and the reactance (Xc) of the capacitor plays a major role in the performance of a **Op-amp Differentiator**.

7.3.1. Op-amp Differentiator Circuit



The input signal to the differentiator is applied to the capacitor. The capacitor blocks any DC content so there is no current flow to the amplifier summing point, X resulting in zero output voltage. The capacitor only allows AC type input voltage changes to pass through and whose frequency is dependant on the rate of change of the input signal.

At low frequencies the reactance of the capacitor is "High" resulting in a low gain (Rƒ/Xc) and low output voltage from the op-amp. At higher frequencies the reactance of the capacitor is much lower resulting in a higher gain and higher output voltage from the differentiator amplifier.

However, at high frequencies an op-amp differentiator circuit becomes unstable and will start to oscillate. This is due mainly to the first-order effect, which determines the frequency response of the op-amp circuit causing a second-order response which, at high frequencies gives an output voltage far higher than what would be expected. To avoid this the high frequency gain of the circuit needs to be reduced by adding an additional small value capacitor across the feedback resistor Rƒ.

Ok, some math's to explain what's going on!. Since the node voltage of the operational amplifier at its inverting input terminal is zero, the current, i flowing through the capacitor will be given as:

$$I_{IN} = I_F \text{ and } I_F = -\frac{V_{OUT}}{R_F}$$

The charge on the capacitor equals Capacitance times Voltage across the capacitor

$$Q = C \times V_{IN}$$

Thus the rate of change of this charge is:

$$\frac{dQ}{dt} = C\frac{dV_{IN}}{dt}$$

but dQ/dt is the capacitor current, *i*

$$I_{IN} = C\frac{dV_{IN}}{dt} = I_F$$

$$\therefore -\frac{V_{OUT}}{R_F} = C\frac{dV_{IN}}{dt}$$

from which we have an ideal voltage output for the op-amp differentiator is given as:

$$V_{OUT} = -R_F C\frac{dV_{IN}}{dt}$$

Therefore, the output voltage Vout is a constant –R*ƒ**C times the derivative of the input voltage Vin with respect to time. The minus sign (–) indicates a 180^{o} phase shift because the input signal is connected to the inverting input terminal of the operational amplifier.

One final point to mention, the **Op-amp Differentiator** circuit in its basic form has two main disadvantages compared to the previous operational amplifier integrator circuit. One is that it suffers from instability at high frequencies as mentioned above, and the other is that the capacitive input makes it very susceptible to random noise signals and any noise or harmonics present in the source circuit will be amplified more than the input signal itself. This is because the output is proportional to the slope of the input voltage so some means of limiting the bandwidth in order to achieve closed-loop stability is required.

7.3.2. Op-amp Differentiator Waveforms

If we apply a constantly changing signal such as a Square-wave, Triangular or Sine-wave type signal to the input of a differentiator amplifier circuit the resultant output signal will be changed and whose final shape is dependant upon the RC time constant of the Resistor/Capacitor combination.

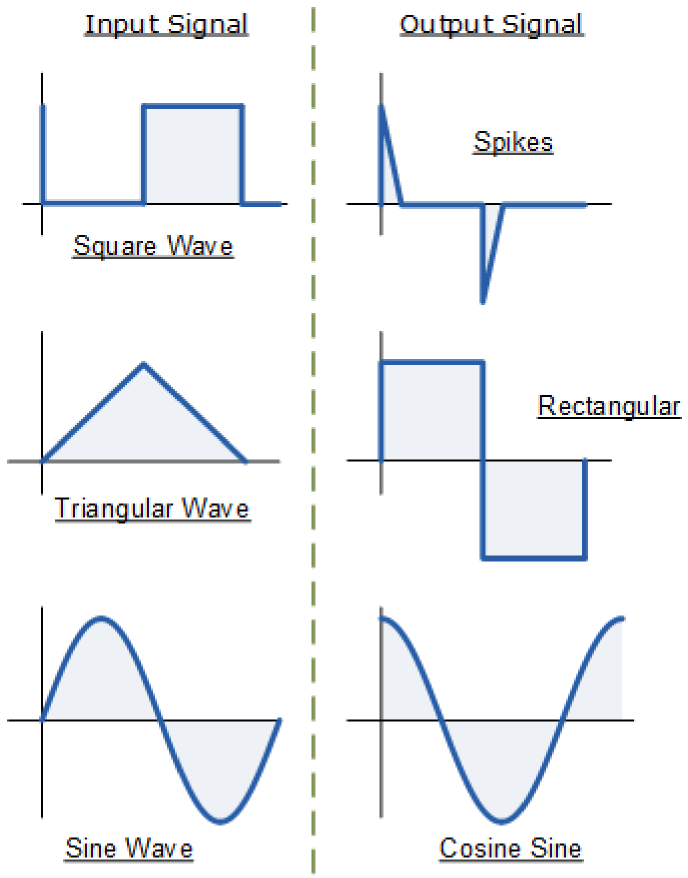


Fig. No.-18: waveforms of Differentiator Amplifier (P.I- electronics tutorial)

7.3.3. Applications Of Differentiator Circuit

- An op-amp differentiating amplifier is an inverting amplifier circuit configuration, which uses reactive components (usually a capacitor than inductor).
- The differentiator performs mathematical differentiation operation on the input signal with respect to time i.e., the instantaneous output voltage is proportional to the rate of change of the input signal.
- Differentiating circuits are commonly used to operate on triangular and rectangular signals. While operating on sine wave inputs, differentiating circuits have frequency limitations.

7.4. OSCILLOSCOPE

An oscilloscope, formerly known as an oscillograph, is an instrument that graphically displays electrical signals and shows how those signals change over time. It measures these signals by connecting with a sensor, which is a device that creates an electrical signal in response to physical stimuli like sound, light and heat. For instance, a microphone is a sensor that converts sound into an electrical signal.

Here we'll cover everything you need to know about an oscilloscope from how it works to how to find the right one.

Fig. No.-13: Oscilloscope (P.I- Tektronix)

7.4.1. History of the oscilloscope

In 1897, a German physicist named Karl Ferdinand Braun invented a cathode ray tube and, along with it, the first oscilloscope, which was expanded upon decades later by the company A. C. Cossor. In 1934, the first commercial oscilloscope was released by General Radio, and it became the first to be used outside of a laboratory. And in 1946, Howard Vollum and Melvin Jack Murdock founded Tektronix, which has gone on to become a world leader oscilloscope. Since then, Tek has continued releasing innovative new technologies, including the first digital oscilloscope in 1971 and the first oscilloscope-to-cloud software solution—Tek Drive—in 2020. Oscilloscopes are a staple of any engineer's bench and have even been featured in famous films throughout history. You can visit the Tek museum website to see a complete list of oscilloscopes in films.

7.4.2. What is an oscilloscope used for?

Oscilloscopes are often used when designing, manufacturing or repairing electronic equipment. Engineers use an oscilloscope to measure electrical phenomena and solve measurement challenges quickly and accurately to verify their designs or confirm that a sensor is working properly.

7.4.3. Who uses an oscilloscope?

Scientists, engineers, physicists, repair technicians and educators use oscilloscopes to see signals change over time. An automotive engineer might use an oscilloscope to correlate analog data from sensors with serial data from the engine control unit. Meanwhile, a medical researcher might use an oscilloscope to measure brain waves. There is no shortage of applications for this powerful instrument.

7.4.4. How does an oscilloscope work?

There are three primaries oscilloscope systems: vertical, horizontal and trigger systems. Together, these systems provide information about the electrical signal, so the oscilloscope can accurately reconstruct it. The picture below shows the block diagram of an oscilloscope.

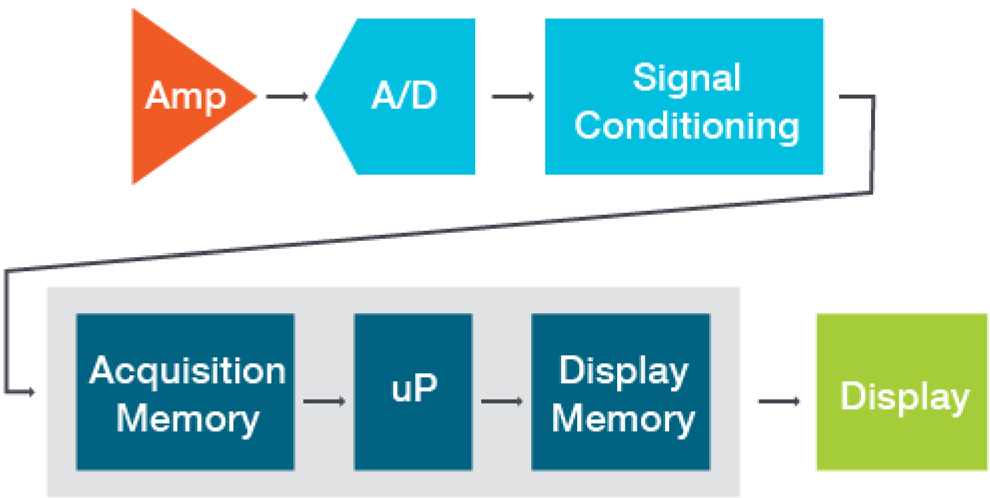


Fig. No.-14: Working of Oscilloscope (P.I- Tektronix)

The first stage attenuates or amplifies the signal voltage in order to optimize the amplitude of the signal; this is referred to as the vertical system since it depends on the vertical scale control. Then the signal reaches the acquisition block, where the analog-to-digital converter (ADC) is used to sample the signal voltage and convert it in a digital format value. The horizontal system, which contains a sample clock, gives each voltage sample a precise time (horizontal) coordinate. The sample clock drives the ADC and its digital output is stored in the acquisition memory as a record point. The trigger system detects a user-specified condition in the incoming signal stream and applies it as a time reference in the waveform record. The event that met the trigger criteria is displayed, as is the waveform data preceding or following the event.

7.4.5. Oscilloscope vs. digital multimeter vs. voltmeter

Oscilloscope, digital multimeter, voltmeter—what's the difference and are they interchangeable? A voltmeter measures the potential difference between two nodes on an electrical circuit. Though a digital multimeter also measures voltage, it can measure current and resistance as well. And an oscilloscope shows how the voltage changes over time. Typically, as the application becomes more advanced, so does the instrument.

7.4.6. What does an oscilloscope measure?

Simply put, an oscilloscope measures voltage waves. On an oscilloscope screen, voltage is displayed vertically on the Y axis and time is represented horizontally on the X axis. The intensity or brightness of the display is sometimes called the Z axis. The resulting graph can tell you many things about a signal, including:

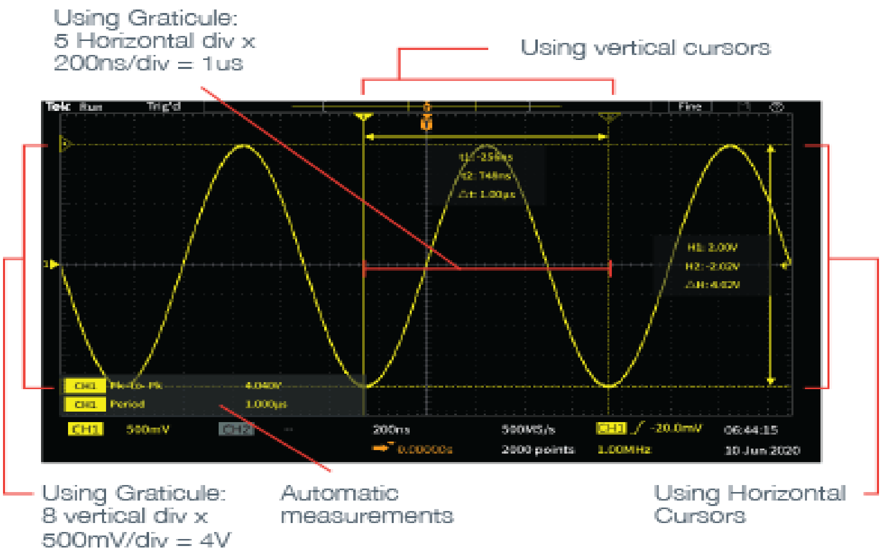


Fig. No.-15: Oscilloscope screen (P.I- Tektronix)

- Time and voltage values of a signal.
- Frequency of an oscillating signal.
- The “moving parts” of a circuit represented by the signal.
- Frequency with which a particular portion of the signal is occurring relative to other portions.
- Whether or not a malfunctioning component is distorting the signal.
- How much of a signal is direct current (DC) or alternating current (AC).
- The portion of the signal that is noise.
- Whether noise is changing over time.

7.4.7. Types of oscilloscopes

There are two types of oscilloscopes: analog and digital. An analog oscilloscope captures and displays the voltage wave form in its original form, while a digital oscilloscope uses an analog-to-digital converter to capture and store information digitally. When it comes to debugging and design, most engineers today use digital oscilloscopes. Digital oscilloscopes generally fall into five categories, ranging from the less expensive general-purpose oscilloscopes to more complex oscilloscopes that, while more expensive, offer advanced functionality and greater accuracy than the more basic models.

- **Digital storage oscilloscope (DSO):** This is a conventional digital oscilloscope and is ideal for low repetition-rate or single-shot, high-speed, multichannel design applications.

- **Digital phosphor oscilloscope (DPO):** A DPO takes a new approach to oscilloscope architecture and, unlike DSOs, provides Z-axis (intensity) in real-time. DPOs are the best general-purpose design and troubleshooting tool for a wide range of applications and are often used for advanced analysis, communication mask testing, digital debug of intermittent signals, repetitive digital design and timing applications.

- **Mixed signal oscilloscope (MSO):** A type of DSO, MSOs are designed to display and compare both analog and digital signals. It is the instrument of choice for quickly debugging digital circuits using powerful digital triggering, high-resolution acquisition capability and analysis tools.

- **Mixed domain oscilloscope (MDO):** These oscilloscopes provide the same capabilities as mixed signal oscilloscopes, but also offer a built-in spectrum analyser, adding RF debugging to the analog and digital capabilities.

- **Digital sampling oscilloscope:** For very high-speed signal analysis, sampling oscilloscopes support jitter and noise analysis with ultra-low jitter acquisitions. It can achieve bandwidth and high-speed timing 10 times higher than other oscilloscopes for repetitive signals.

7.4.8. How to choose the best oscilloscope?

When it comes to choosing the right oscilloscope, there are a number of factors to consider, including bandwidth, waveform capture rate, sample rate, rise time, triggering capabilities and price. Much like shutter speed, lighting conditions and aperture of a camera all affect its ability to capture an image clearly and accurately, the performance considerations of an oscilloscope significantly affect its ability to achieve the required signal integrity.

8. Barkhausen criterion

The Barkhausen stability criterion is a mathematical condition to determine when a linear electronic circuit will oscillate. It was put forth in 1921 by German physicist Heinrich Georg Barkhausen (1881–1956). It is widely used in the design of electronic oscillators, and also in the design of general negative feedback circuits such as op amps, to prevent them from oscillating.

8.1. Limitations

Barkhausen's criterion applies to linear circuits with a feedback loop. It cannot be applied directly to active elements with negative resistance like tunnel diode oscillators.

The kernel of the criterion is that a complex pole pair must be placed on the imaginary axis of the complex frequency plane if steady state oscillations should take place. In the real world, it is impossible to balance on the imaginary axis, so in practice a steady-state oscillator is a non-linear circuit:

- It needs to have positive feedback.
- The loop gain is at unity ($|\beta A|=1$).

8.2. Criterion

It states that if *A* is the gain of the amplifying element in the circuit and $\beta(j\omega)$ is the transfer function of the feedback path, so βA is the loop gain around the feedback loop of the circuit, the circuit will sustain steady-state oscillations only at frequencies for which:

1. The loop gain is equal to unity in absolute magnitude, that is, $|\beta A|=1$.
2. The phase shift around the loop is zero or an integer multiple of 2π: $\angle\beta A=2\pi n$, n={1,2,3,…..}.

Barkhausen's criterion is a *necessary* condition for oscillation but not a *sufficient* condition: some circuits satisfy the criterion but do not oscillate. Similarly, the Nyquist stability criterion also indicates instability but is silent about oscillation. Apparently, there is not a compact formulation of an oscillation criterion that is both necessary and sufficient.

9. Observations

9.1. An Arbitrary Waveform Generator from (12.5-17.5) KHz:

As per our project we have successfully able to design a circuit which can give an output of sine wave and square wave with variable frequencies. These outputs should be taken at two separate outputs ports. So, that we can use two outputs at a time when we used the function generator.

To make function generator we have combined two circuit designs.

1. Colpitts Oscillator
2. Differentiator Amplifier.

9.1.1. Colpitts Oscillator

Generally, Class A amplifier is used in Colpitts Oscillator but we have used own modified Class A amplifier to generate Sine wave. In which we have used TANK circuit which can generate frequency of 10KHz. In this circuit we have used an input voltage of 12Volt DC. In this circuit design we can generate variable frequency for which we have used variable inductor (L1) which can vary frequency from 12.5KHz to 17.5KHz of voltage 12 V_{PP} (in Proteus Software) but when we did it practically, we get frequency of 22KHz with a voltage of 10 V_{PP}.

- **Colpitts Oscillator Circuit Diagram:**

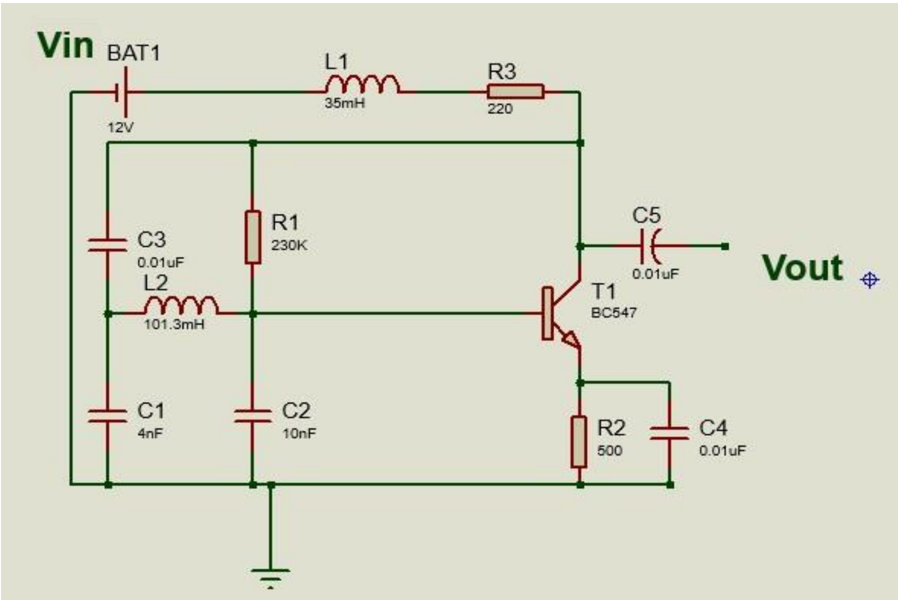


Fig. No.-16: Colpitts Oscillator (P.I- Self)

- **Practical Circuit:**

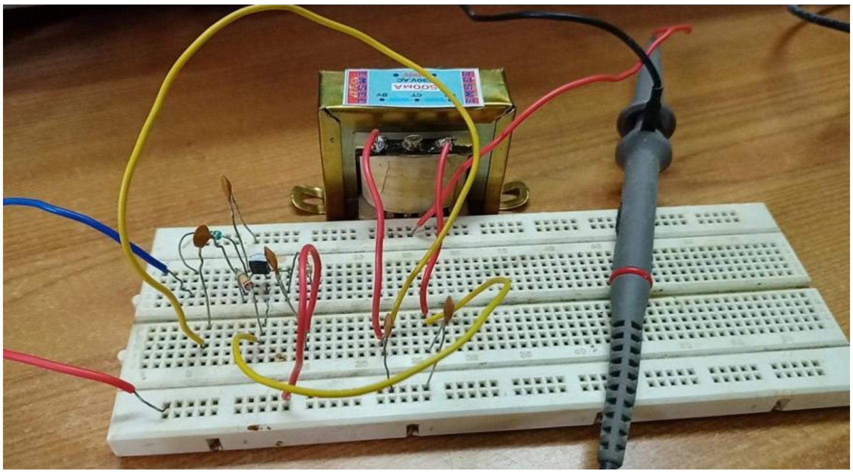


Fig. No.-17: circuit diagram (P.I- Self)

- **Colpitts Oscillator Calculations:**

$C_{equivalent} = \frac{C1*C2}{C1+C2}$

[C1=4nF, C2=10nF]

$C_{equivalent} = \frac{4*10}{4+10}$

$C_{equivalent}$ = 2.85nF

Frequency (f):

$$f = \frac{1}{2\pi\sqrt{(\mathrm{L} \times \mathrm{Cequivalent})}}$$

$$f = \frac{1}{2\pi\sqrt{101.3 \times (10^{-3}) \times 2.85 \times (10^{-9})}}$$

Frequency (f) = 9.36KHz

Gain (A_v):

$A_v = \frac{Vc}{Vb}$

$A_v = \frac{11.04}{2.94}$

Gain (A_v) = 3.75 Volts.

Feedback Fraction (β):

$\beta = \frac{C1}{C2}\%$

$\beta = \frac{4}{10}$

Feedback Fraction (β) = 0.4%

Barkhausen's criterion (|βA|):

$|\beta * A_v| = 0.4*3.75$ [A=Av]

$|\beta * A| = 1.5$

We know $|\beta A|=1$ [as per Barkhausen's criterion]

Phase Shift (∠βA):

2π: $\angle\beta A = 2\pi n$ {n=1.5}

$= 2*\pi*1.5$

Phase Shift=135°

- **Frequency Response:**

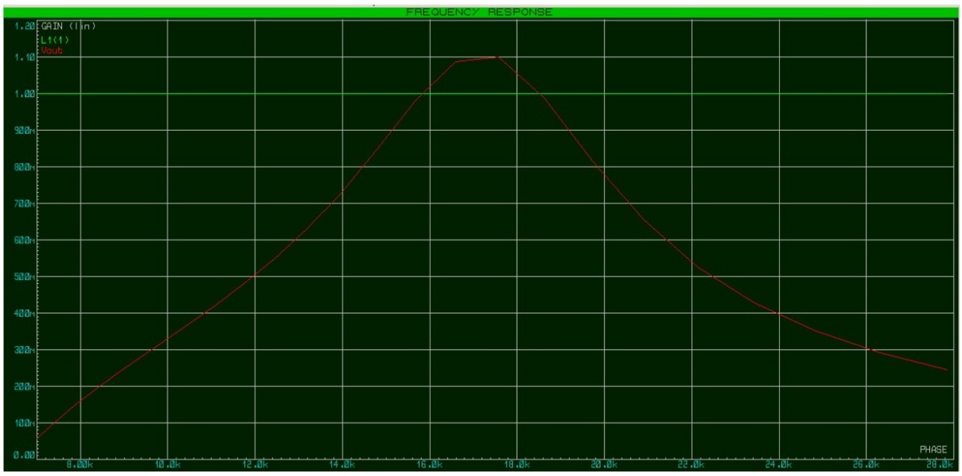


Fig. No.-18: Frequency response curve (P.I- Self)

- **Output Waveform by Proteus:**

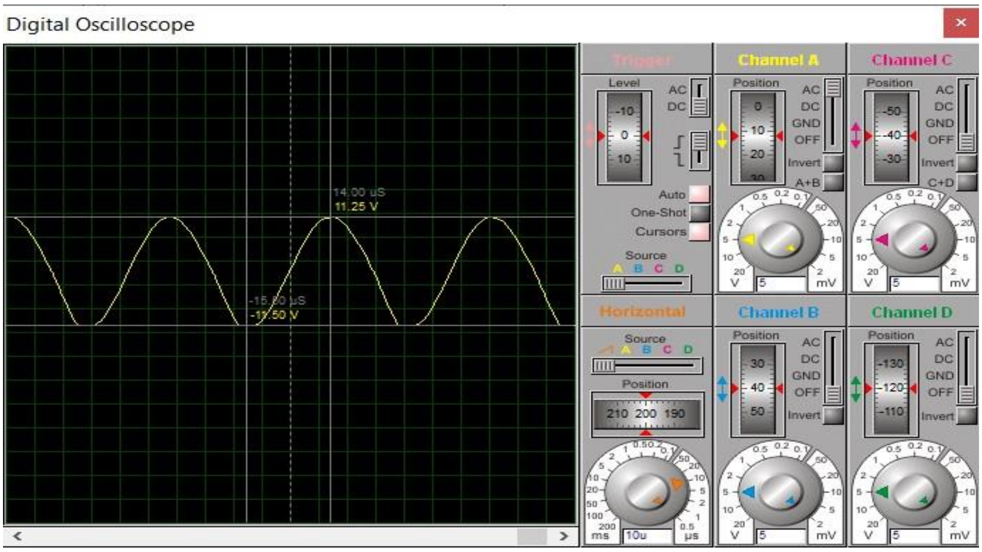


Fig. No.-19: Sine wave by proteus (P.I- Self)

- **Practical Output Waveform:**

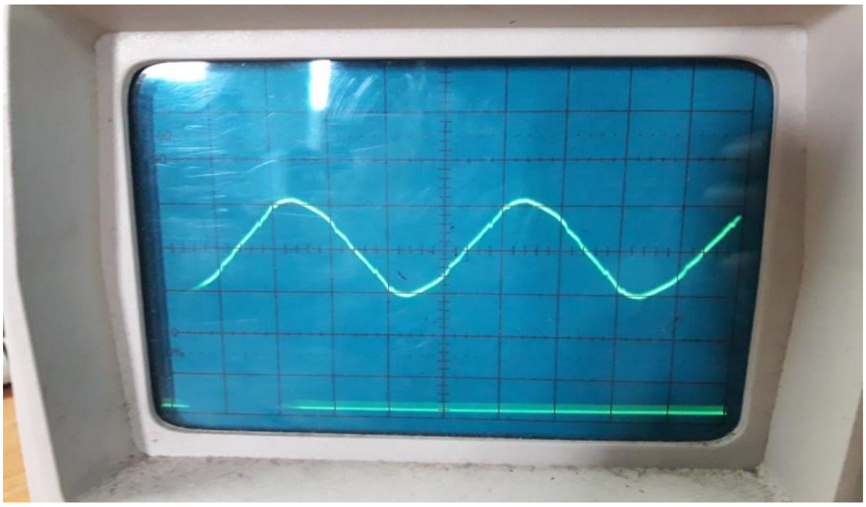


Fig. No.-20: Sine wave in lab (P.I- Self)

9.1.2. Differentiator Circuit :

To get square wave we use a Differentiator circuit. In which the input is 12 volts DC and, in the output, we got Square Wave of 15 V_{PP} with frequency of 70 KHz and it can variate to 71KHz and 75KHz (In Proteus) and in practically we got 15V_{pp} with frequency of 22KHz.

- **Differentiator Circuit Diagram:**

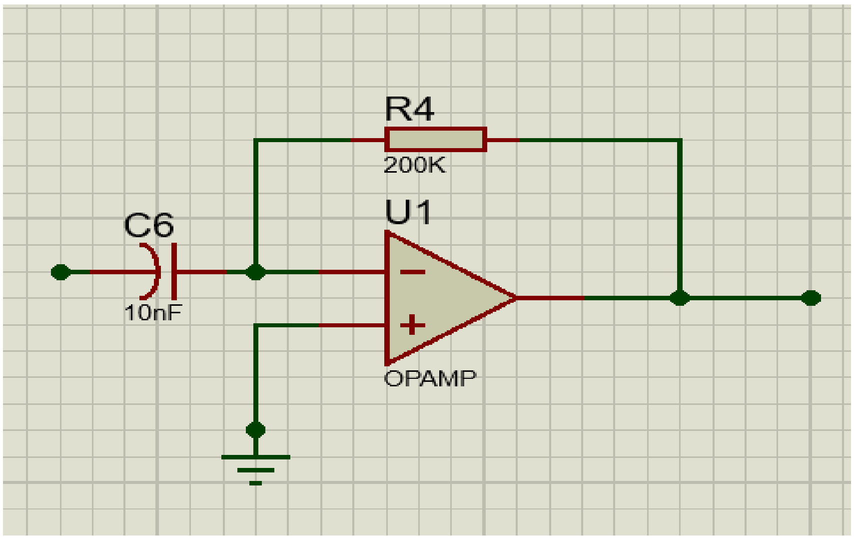


Fig. No.-21:Differentiator Circuit Diagram (P.I- Self)

- **Differential Amplifier Calculation**

Cycle Time $T_C = P_W + S_W$

$=32.5uS+32.5uS$

$=65$ uS

Frequency (f) $=1/T_C=1/65uS$

=10^6 / 65

=15.384 KHz

Duty Cycle = (Pw / Tc)*100

=(32.5/65)*100

=50 %

So the Output Waveform is a Square Wave.

$I_{IN} = I_F = 112$ uA [The Value is get from Protious Software]

From the Equation we Know

$$I_F = V_{OUT} / R_f$$

$$V_{OUT} = I_F * R_F$$

$$= 112uA * 150K\Omega$$

$$= 16.80\ V$$

- **Output Waveform by Proteus:**

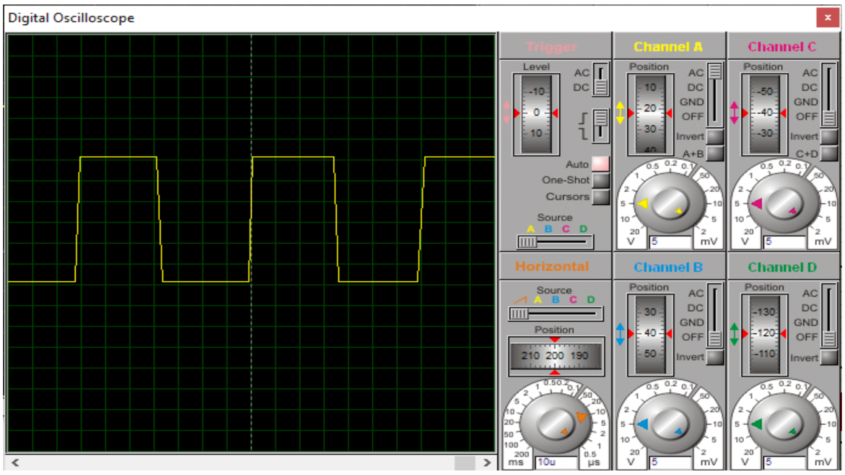


Fig. No.-22: Square wave by Proteus (P.I- Self)

- **Practical Output Waveform:**

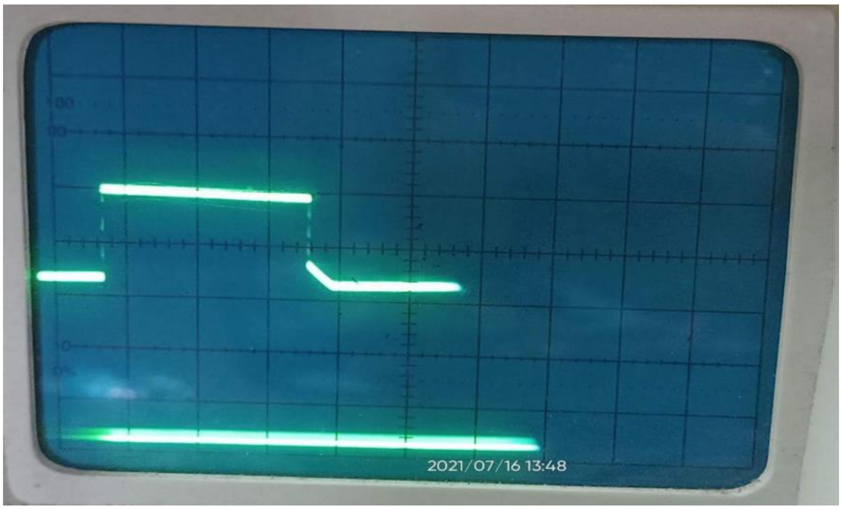


Fig. No.-23: Square wave in lab (P.I- Self)

9.2. Circuit of Arbitrary Waveform Generator:

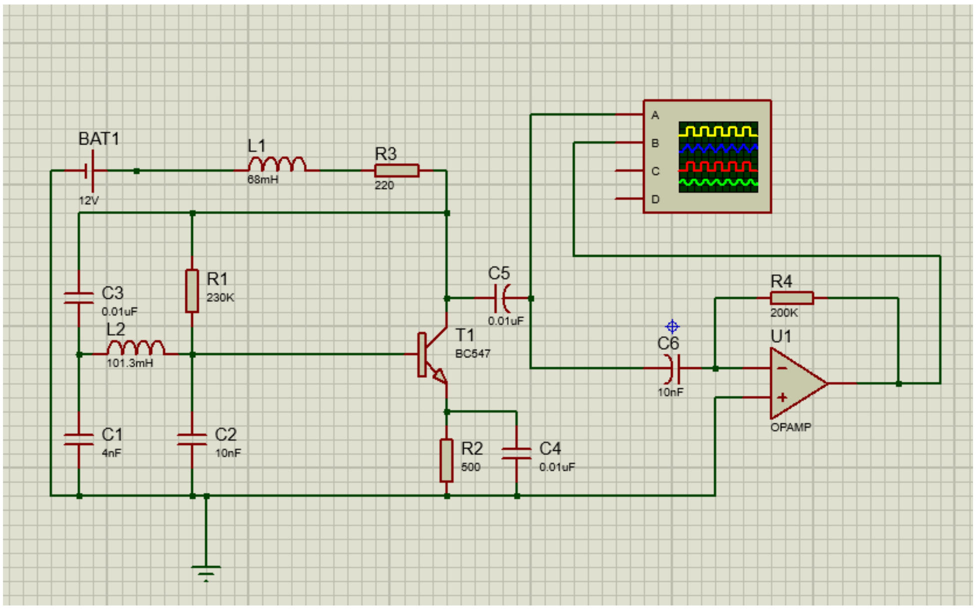


Fig. No.-24: Circuit Diagram of AWG (P.I- Self)

9.3. Output Waveform of Arbitrary Waveform Generator

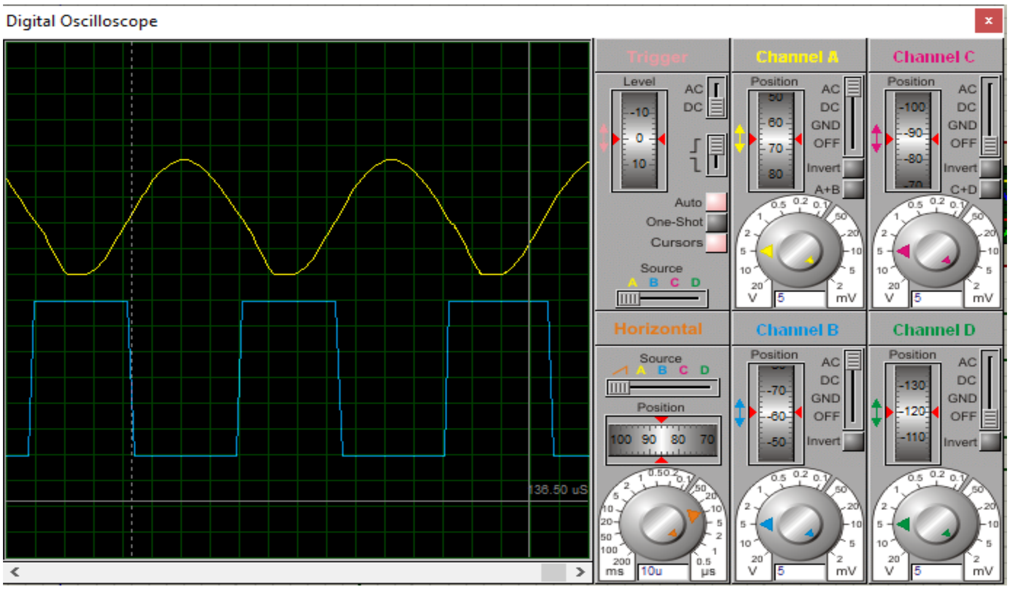


Fig. No.-25: Output of AWG (P.I- Self)

10. Conclusion

We are successfully completed our arbitrary waveform generator with very cheap cost with respect to the market price of a waveform function generation. But yes, very a smaller number of functions are made by us. In future we are definitely trying to pursue our project to add other function like sawtooth wave, triangle wave, frequency level on and also trying to add amplitude function because in our project the waveform amplitude was text. For upgrading these things, we are trying our best to reducing the cost as much as we can.

The Arbitrary Waveform Generator that already have in market was heavy weight and large but we are trying to reduce the size and also reducing the weight, that was also achievement for us.

The practical outputs are not as per calculated:

- The capacitors we used for practical are ceramic capacitors and these capacitors have their own leakage in between them.
- The inductor we used which have copper wire for inductance that should be zero but it was not zero.

11. Bibliography

To make the project I have Taken Source from following books: -

- Electronics Fundamental & Application by D Chattopadhyay and P C Rakshit.
- Narrow Band Pass Filter For Low Frequency Application by Dr Raman K. Attri.
- RF and Microwave Transistor Oscillator Design by Andrei Grebennikov.
- Microelectronics circuit by Adel S. Sedra, Kenneth C Smith.

I have taken source from internet also

1. https://en.wikipedia.org/wiki/Band-pass_filter
2. https://en.wikipedia.org/wiki/Colpitts_oscillator
3. https://www.electronics-notes.com/articles/test-methods/oscilloscope/scope-basics.php
4. https://en.wikipedia.org/wiki/Arbitrary_waveform_generator
5. https://www.youtube.com/

Our Mentor help us in this project Ms. Smita Hazra.