LAB MANUAL

ECUDLT1

Analog Circuit Lab

Bachelor of Technology

in

Electronics & Communication Engineering

Department of Electronics & Communication Engineering

School of Studies of Engineering & Technology

Guru Ghasidas Vishwavidyalaya

Bilaspur-495009 (C. G.)

Website: www.ggu.ac.in

SCHOOL OF STUDIES OF ENGINEERING & TECHNOLOGY
GURU GHASIDAS VISHWAVIDYALAYA, BILASPUR (C.G.)
(A CENTRAL UNIVERSITY)

CBCS-NEW SYLLABUS

B. TECH. SECOND YEAR (Electronics and Communication Engineering)

Vision and Mission of the Institute

Vision		To be a leading technological institute that imparts transformative education to create globally competent technologists, entrepreneurs, researchers and leaders for a sustainable society
Mission	1	To create an ambience of teaching learning through transformative education for future leaders with professional skills, ethics, and conduct.
	2	To identify and develop sustainable research solutions for the local and global needs.
	3	To build a bridge between the academia, industry and society to promote entrepreneurial skills and spirit

Vision and Mission of the Department

Vision		The Department endeavours for academic excellence in Electronics & Communication Engineering by imparting in depth knowledge to the students, facilitating research activities and cater to the ever-changing industrial demands, global and societal needs with leadership qualities.
Mission	1	To be the epitome of academic rigour, flexible to accommodate every student and faculty for basic, current and future technologies in Electronics and Communication Engineering with professional ethics.
	2	To develop an advanced research centre for local & global needs.
	3	To mitigate the gap between academia, industry & societal needs through entrepreneurial and leadership promotion.

Program Educational Objectives (PEOs)

The graduate of the Electronics and Communication Engineering Program will

PEO1: Have fundamental and progressive knowledge along with research initiatives in the field of Electronics & Communication Engineering.

PEO2: Be capable to contrive solutions for electronic & communication systems for real world applications which are technically achievable and economically feasible leading to academia, industry, government and social benefits.

PEO3: Have performed effectively in a multi-disciplinary environment and have self-learning & self-perceptive skills for higher studies, professional career or entrepreneurial endeavors to be confronted with a number of difficulties.

PEO4: Attain team spirit, communication skills, ethical and professional attitude for lifelong learning.

Programme Outcomes: Graduates will be able to:

PO1: Fundamentals: Apply knowledge of mathematics, science and engineering.

PO2: Problem analysis: Identify, formulate and solve real time engineering problems using first principles.

PO3: Design: Design engineering systems complying with public health, safety, cultural, societal and environmental considerations

PO4: Investigation: Investigate complex problems by analysis and interpreting the data to synthesize valid solution.

PO5: Tools: Predict and model by using creative techniques, skills and IT tools necessary for modern engineering practice.

PO6: Society: Apply the knowledge to assess societal, health, safety, legal and cultural issues for practicing engineering profession.

PO7: Environment: Understand the importance of the environment for sustainable development.

PO8: Ethics: Apply ethical principles and commit to professional ethics, and responsibilities and norms of the engineering practice.

PO9: Teamwork: Function effectively as an individual and as a member or leader in diverse teams and multidisciplinary settings.

PO10: Communication: Communicate effectively by presentations and writing reports.

PO11: Management: Manage projects in multidisciplinary environments as member or a team leader.

PO12: Life-long learning: Engage in independent lifelong learning in the broadest context of technological change.

Programme Specific Outcomes:

PSO1: Identify, formulate and apply concepts acquired through Electronics & Communication Engineering courses to the real-world applications.

PSO2: Design and implement products using the cutting-edge software and hardware tools to attain skills for analyzing and developing subsystem/processes.

PSO3: Ability to adapt and comprehend the technology advancement in research and contemporary industry demands with demonstration of leadership qualities and betterment of organization, environment and society.

Sub Code	L	T	P	Duratio n	IA	ESE	Total	Credits
ECUDLT1	-	-	2	2 Hours	25	25	50	1

ANALOG CIRCUITS LAB

Course Objectives:

- To identify and test various electronic components.
- To use DSO for various measurements.
- To plot the characteristics of diode and transistor.
- To design and implement feedback amplifier circuits.
- To measure the frequency of oscillators.

Course Outcomes:

At the end of the course, the students will able to:

CO1Implement the different configuration of BJT & FET amplifier at different frequencies.

CO2 Implement the RC coupled amplifier and analyze the characteristics at different frequencies.

CO3 Implement different types of feedback amplifier.

CO4 Implement different class of power amplifier with different loads and evaluate the efficiency.

CO5 Design audio & radio frequency oscillator and evaluate the frequency of oscillation.

Course Outcomes and their mapping with Program Outcomes & Program Specific Outcomes:

CO	PO												PSO		
	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12	PSO1	PSO2	PSO3
CO1	3	3	1	2					3			3	2	2	1
CO2	3	3	1	2					3			3	2	2	1
CO3	3	3	1	2					3			3	2	2	1
CO4	3	3	1	2					3			3	2	2	1
CO5	3	3	1	2					3			3	2	2	1

Weightage: **1-Sightly; 2-Moderately; 3-Strongly**

EXPERIMENT- 1

Objective: To study the single stage RC coupled CE, CB, CC amplifier.

Resources Required: Transistor, Resistance , Regulated Power supply, Capacitor, Signal Generator, CRO , Breadboard and Wires ,CRO Probes

Thoery:

The single stage common emitter amplifier circuit shown below uses what is commonly called "Voltage Divider Biasing" or “self biasing”. The Common Emitter Amplifier circuit has a resistor in its Collector circuit. The current flowing through this resistor produces the voltage output of the amplifier. The value of this resistor is chosen so that at the amplifiers quiescent operating point, Q-point this output voltage lies half way along the transistors load line. In Common Emitter Amplifier circuits, capacitors C1 and C2 are used as Coupling Capacitors to separate the AC signals from the DC biasing voltage. This ensures that the bias condition set up for the circuit to operate correctly is not affected by any additional amplifier stages, as the capacitors will only pass AC signals and block any DC component.

The output AC signal is then superimposed on the biasing of the following stages. Also a bypass capacitor, CE is included in the Emitter leg circuit. This capacitor is an open circuit component for DC bias meaning that the biasing currents and voltages are not affected by the addition of the capacitor maintaining a good Q-point stability. However, this bypass capacitor short circuits the Emitter resistor at high frequency signals and only RL plus a very small internal resistance acts as the transistors load increasing the voltage gain to its maximum.

Generally, the value of the bypass capacitor, CE is chosen to provide a reactance of at most, 1/10th the value of RE at the lowest operating signal frequency.

Circuit Diagram:

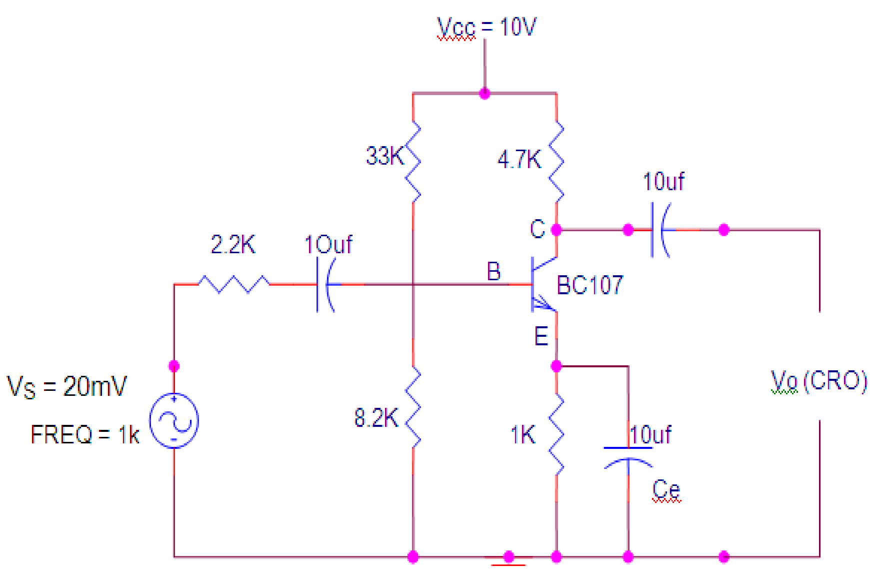


Fig: CE Amplifier Circuit

PROCEDURE:

1. Connections are made as per the circuit diagram.
2. A 10V supply is given to the circuit.
3. A certain amplitude of input signal (say 20mv at 1 kHz) is kept constant using signal generator and for different frequencies, the output voltage (V0) from CRO are noted.
4. Gain for with and without feedback is calculated using Gain (in dB)= 20 log (V_0/V_i) where Vo is output voltage, Vi is input voltage.
5. Plot the graph between Gain (in dB) and frequency
6. Repeat the circuit diagram shown above for CB and CC configuration.

Model graph:

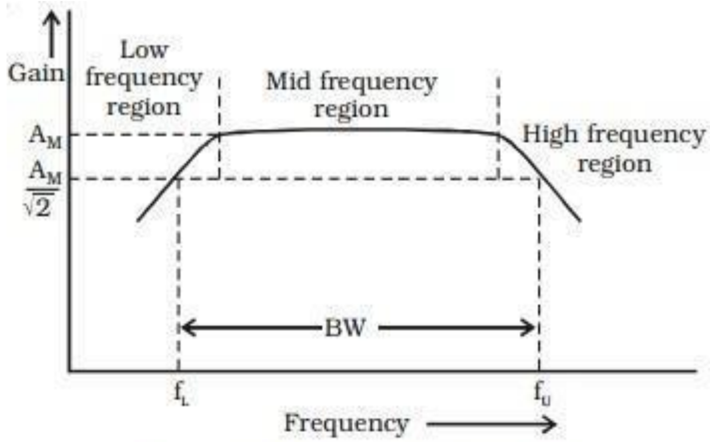


Fig Frequency response curve

OBSERVATION TABLE:

S.No	Frequency(hz)	Output voltage(vo)	Voltage gain (vo/vi)	Gain (db) Avf=20 log (vo/vi).

Bandwidth of the CE amplifier = fh-fl HZ

PRECAUTIONS:

1. While doing the experiment do not exceed the ratings of the transistor. This may lead to damage the transistor.
2. Do not switch **ON** the power supply unless you have checked the circuit connections as per the circuit diagram.
3. Make sure while selecting the emitter, base and collector terminals of the transistor.

RESULT: The Frequency response characteristics of Common emitter amplifier are obtained and the graph was plotted. From the graph, the Bandwidth was obtained as …………..

<u>**EXPERIMENT- 2**</u>

Objective: To study the two stage RC coupled amplifier.

Resources Required: Transistor, Resistance , Regulated Power supply, Capacitor, Signal Generator, CRO, Breadboard and Wires, CRO Probes

Theory:

A Resistance Capacitance (RC) Coupled Amplifier is basically a multi-stage amplifier circuit extensively used in electronic circuits. Here the individual stages of the amplifier are connected together using a resistor–capacitor combination due to which it bears its name as RC Coupled.

A two-stage amplifier whose individual stages are nothing but the common emitter amplifiers. Hence the design of individual stages of the RC coupled amplifiers is similar to that in the case of common emitter amplifiers in which the resistors R_1 and R_2 form the biasing network while the emitter resistor RE form the stabilization network. Here the C_E is also called bypass capacitor which passes only AC while restricting DC, which causes only DC voltage to drop across R_E while the entire AC voltage will be coupled to the next stage.Further, the coupling capacitor C_C also increases the stability of the network as it blocks the DC while offers a low resistance path to the AC signals, thereby preventing the DC bias conditions of one stage affecting the other. In addition, in this circuit, the voltage drop across the collector- emitter terminal is chosen to be 50% of the supply voltage V_{CC} in order to ensure appropriate biasing point.

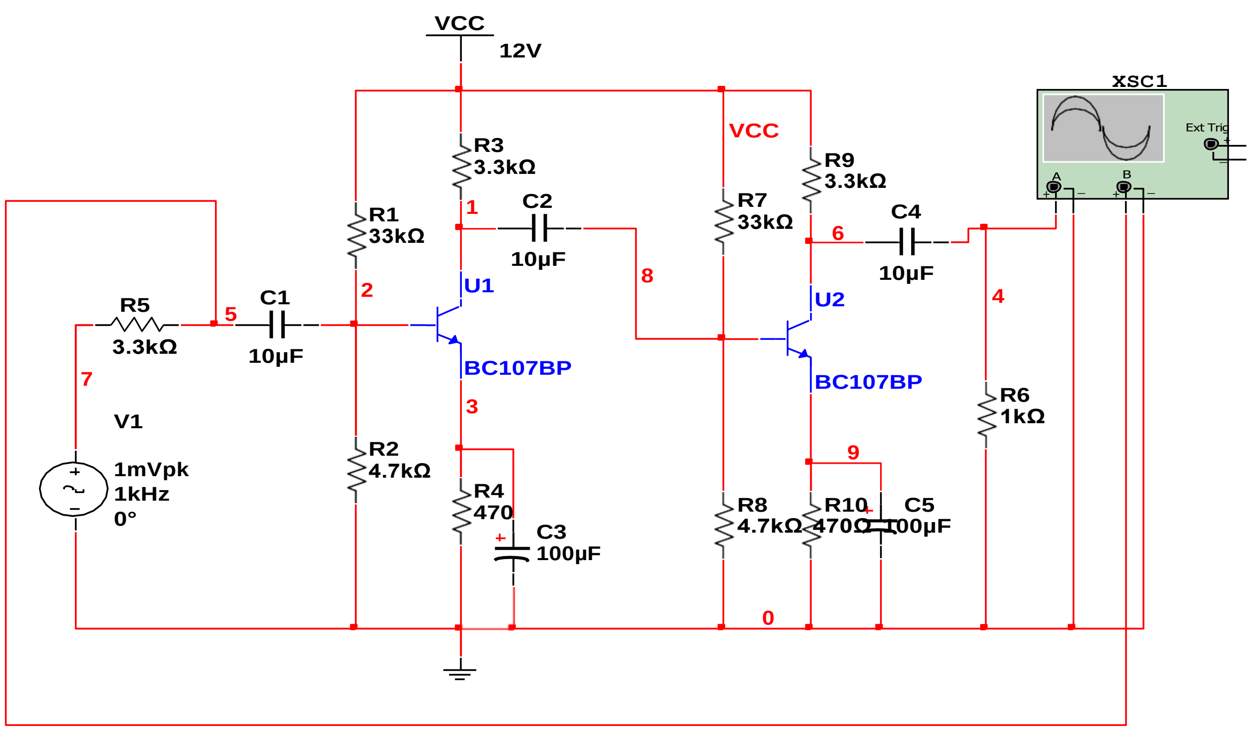


Fig 1 : R-C Coupled Amplifier Circuit

PROCEDURE:

1. Connect the circuit as shown in the figure.

2. A 10V supply is given to the circuit and a certain amplitude of input signal is kept constant using signal generator.
3. Measure the output voltage (say Vo2)and also output voltage at the output of 1st stage (say Vo1) from CRO.
4. Calculate total voltage gain and also individual voltage gain.
5. Now, by varying the input frequency note the output voltages from CRO and calculate the gain.
6. Calculate overall gain by multiplying gain of individual stage.

OBSERVATION TABLE:

S.No	Frequency(hz)	Output voltage(vo)	Voltage gain (vo/vi)	Gain (db) Avf=20 log (vo/vi).

Model Graph

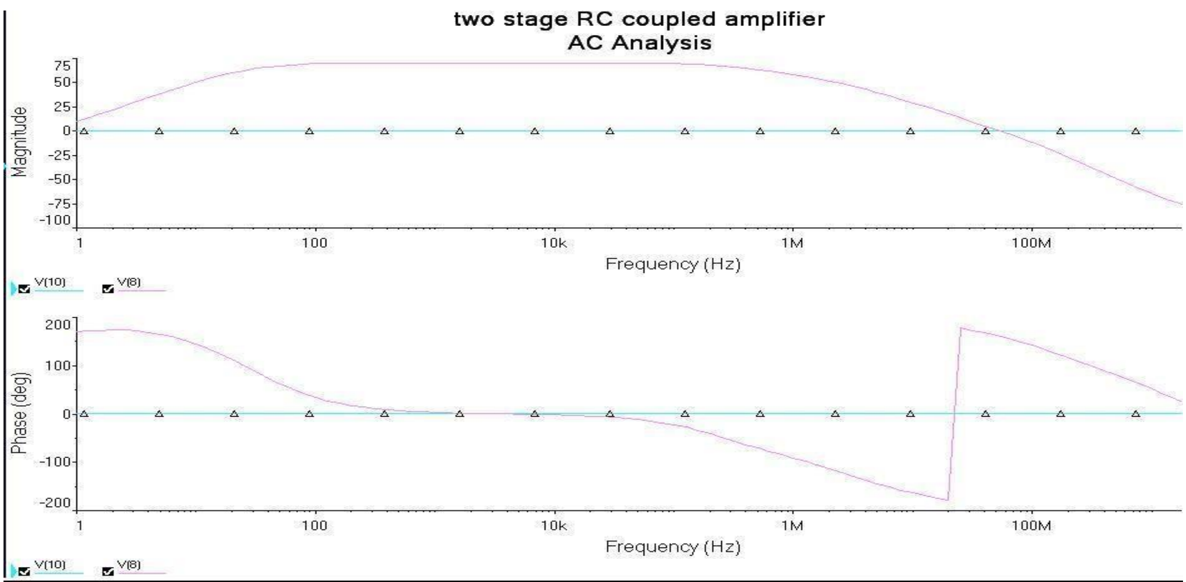


Fig 2: Frequency Response

Result:

The maximum gain is________________dB and bandwidth is_________Hz of the CE Amplifier.

EXPERIMENT- 3

Objective: To study different types of Feedback amplifier.

Resources Required: Transistor, Resistance, Regulated Power supply, Capacitor, Signal Generator, CRO , Breadboard and Wires ,CRO Probes

Thoery:

amplifier is a type of amplifier whose feedback exists between the output and input signal. The concept of feeding the output signal back to its input circuit is known as feedback and that is why it is known as a feedback amplifier. It is dependent between the output and input with effective control. Feedback is of two type

1.Poitive Feedback

2. Negative Feedback

In positive feedback, the feedback energy (voltage or currents), is in phase with the input signal and thus aids it. Positive feedback increases gain of the amplifier also increases distortion, noise and

instability. Because of these disadvantages, positive feedback is seldom employedin amplifiers. But the positive feedback is used in oscillators.

In negative feedback, the feedback energy (voltage or current), is out of phase with the input signal and thus opposes it. Negative feedback reduces gain of the amplifier. It also reduce distortion, noise and instability.This feedback increases bandwidth and improves input and output impedances. Due to these advantages, the negative feedback is frequently used in amplifiers.

There are four types of feedback amplifiers

- Voltage series feedback amplifiers
- Voltage shunt feedback amplifiers
- Current shunt feedback amplifiers
- Current series feedback amplifiers

CIRCUIT DIAGRAMS:-

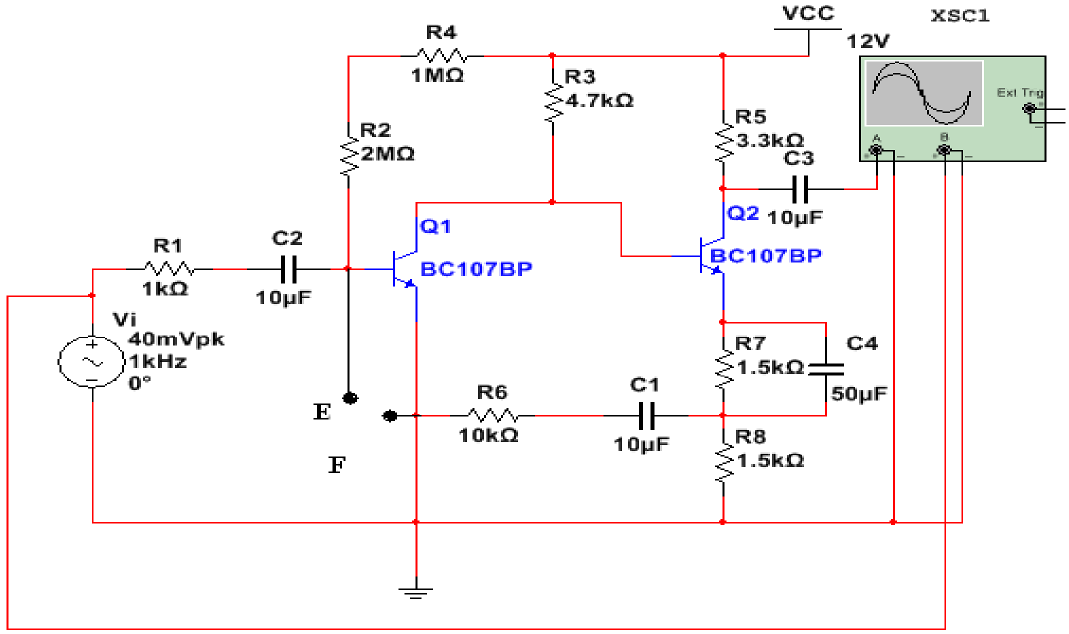


Fig: Current shunt feedback amplifiers

PROCEDURE:

1. Place all the necessary components required for the design of the current shunt feedback amplifier circuit i.e. Resistors, Capacitors, Diodes, Transistors, Voltage sources, Power sources, Ground etc on the design window.
2. Connect all the components by proper wiring and also assure that nodes are formed at the interconnection points
3. .Connect the channel of the Oscilloscope to the output of the circuit and by using the simulation switch and check output waveform.
4. Vary the input frequency from 10Hz to 1MHz with input voltage constant (40mvpp) and note down the output voltage.
5. Calculate the voltage gain in dB using the formula Av=20 log(Vo/Vi).

OBSERVATIONS TABLE

1. **WITHOUT FEEDBACK**

Vi= 40mvp-p at 1kHz

S.No	Frequency(hz)	Output voltage(vo)	Voltage gain (vo/vi)	Gain (db) Avf=20 log (vo/vi).

Bandwidth without feedback=------------

2. **WITH FEEDBACK**

Vi= 40mvp-p at 1kHz

S.NO	Frequency(hz)	Output voltage(vo)	Voltage gain (avf=vo/vi)	Gain (db) Avf=20 log (vo/vi).

-- Bandwidth without feedback=------------

RESULT:

The Av of the current shunt feedback amplifier is____________and the bandwidth is ………without feedback and The Av of the current shunt feedback amplifier is ………and the bandwidth is ………..with feedback.

EXPERIMENT- 4

Objective: To study the Wien bridge oscillator.

Resources Required: Trainer kit of wien bridge oscillator.

Thoery :

A Wien-Bridge Oscillator is a type of phase-shift oscillator which is based upon a Wien-Bridge network (Figure 1a) comprising of four arms connected in a bridge fashion. Here two arms are purely resistive while the other two arms are a combination of resistors and capacitors. In particular, one arm has resistor and capacitor connected in series (R_1 and C_1) while the other hasthem in parallel (R_2 and C_2). This indicates that these two arms of the network behave identical to that of high pass filter or low pass filter.In this circuit, at high frequencies, the reactance of the capacitors C_1 and C_2 will be much less due to which the voltage V_0 will become zero as R_2 will be shorted. Next, at low frequencies, the reactance of the capacitors C_1 and C_2 will become very high.However even in this case, the output voltage V_0 will remain at zero only, as the capacitor C_1 would be acting as an open circuit. This kind of behavior exhibited by the Wien-Bridge network makes it a lead-lag circuit in the case of low and high frequencies

Circuit Diagram :

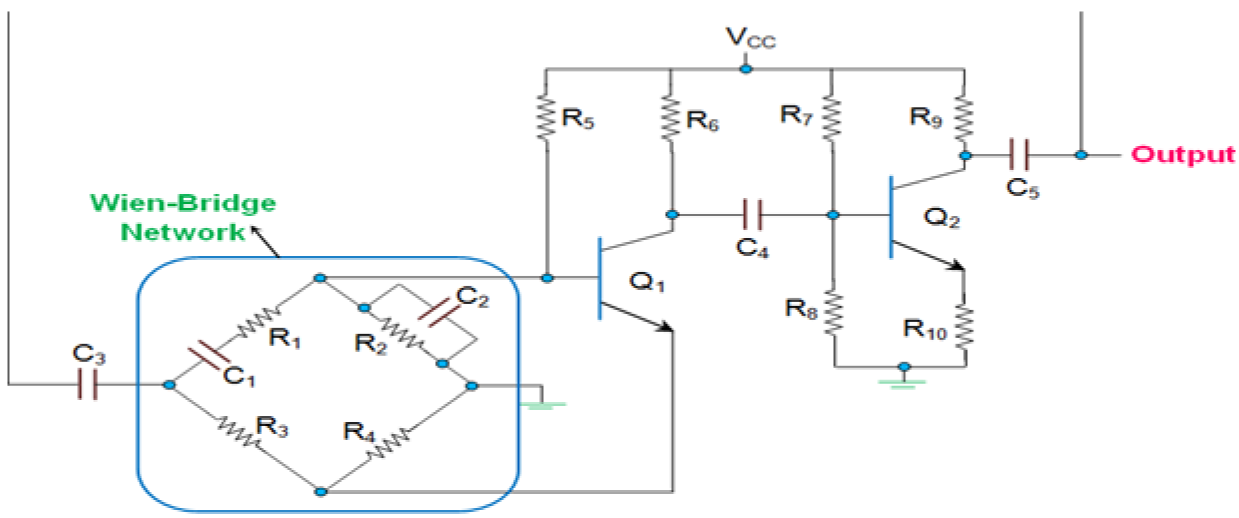


Fig: Wien bridge oscillator

Procedure:-

1. We should take all the components for this experiment.
2. Make the connection as per circuit diagram.
3. Switch ON the kit using ON/OFF toggle switch

4. The input signal is applied with the function generator.

5. Then observe the wave form.

6. Calculate the frequency using formulae

$$f = \frac{1}{2\pi\sqrt{R_1 R_2 C_1 C_2}}$$

RESULT:

Frequency of oscillation of wien bridge oscillator is calculated

EXPERIMENT- 5

Objective: To study the RC Phase Shift Oscillator.

Resources Required: Trainer kit of RC Phase Shift Oscillator.

Thoery :

RC phase-shift oscillators use resistor-capacitor (RC) network (Figure 1) to provide the phase-shift required by the feedback signal. They have excellent frequency stability and can yield a pure sine wave for a wide range of loads. Ideally a simple RC network is expected to have an output which leads the input by 90^o. However, in reality, the phase-difference will be less than this as the capacitor used in the circuit cannot be ideal. Mathematically the phase angle of the RC network is expressed as $\varphi = tan^{-1}(X_c/R)$.

Where, $X_C = 1/(2\pi fC)$ is the reactance of the capacitor C and R is the resistor. In oscillators, these kind of RC phase-shift networks, each offering a definite phase-shift can be cascaded so as to satisfy the phase-shift condition led by the Barkhausen Criterion. Here the collector resistor RC limits the collector current of the transistor, resistors R1 and R (nearest to the transistor) form the voltage divider network while the emitter resistor RE improves the stability. Next, the capacitors CE and Co are the emitter by-pass capacitor and the output DC decoupling capacitor, respectively. Further, the circuit also shows three RC networks employed in the feedback path.

Circuit Diagram :

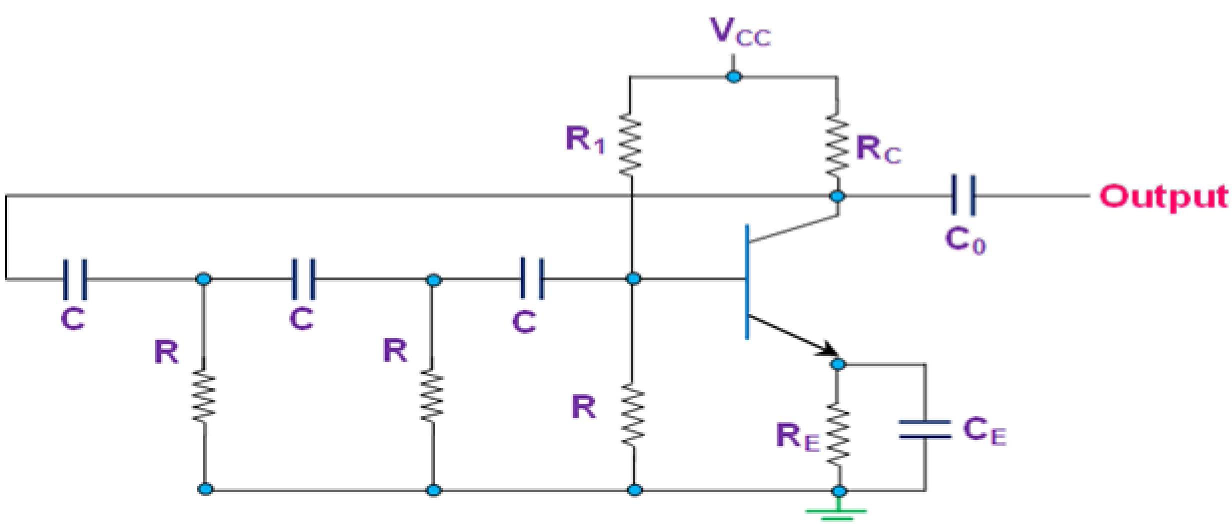


Fig: RC Phase Shift Oscillator

Procedure:-

1. We should take all the components for this experiment.
2. Make the connection as per circuit diagram.
3. Switch ON the kit using ON/OFF toggle switch
4. The input signal is applied with the function geneator
5. Then observe the wave form
6. Calculate the frequency using formulae $f = 1/2\pi RC$

RESULT:

Frequency of oscillation of wien bridge oscillator is calculated

EXPERIMENT- 6

Objective: To study Hartley & Collpit Oscillator.

.

Resources Required: Trainer kit of the Hartley & Collpit Oscillator

Thoery :

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

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

Hartley oscillator: The Hartley oscillator is an electronic oscillator. The frequency of this oscillation is determined by the tuned circuit. The tuned circuit consists of the capacitor and inductor; hence it is an LC oscillator. In 1915 by American engineer Ralph Hartley has invented this oscillator. The featuresof the Hartley circuit are the tuned circuit consists of a single capacitor in parallel with the two inductors which are in series. From the center connection of the two inductors for oscillation purpose, the feedback signal is taken.The Hartley oscillator is parallel to the Colpitts apart from that it uses a pair of tapping coils asan alternate of two tapped capacitors. From the below circuit the output voltage is developed across the inductor L1 and the feedback voltages are across the inductor L2. The feedback network is given in the mathematical expression which is given below

Colpitts Oscillator:

The Colpitts Oscillator was by American engineering by Edwin H. Colpitts in the year of 1918. This oscillator is a combination of both inductors and capacitor. The features of the Colpitts Oscillator are the feedback for the active devices and they are taken from the voltage divider and made up of two capacitors which are in series across the inductor. The Colpitts circuits consist of gain devices such as the bipolar junction, field effect transistor, operational amplifier and vacuum tubes. The output is connected to an input in a feedback loop it has a parallel tuned circuit and it functioned as a band-pass filter is used as a frequency of the oscillator. This oscillator is an electrically dual of the Hartley oscillator hence the feedback signal is taken from the inductive voltage divider it has two coils in the series.The following circuit diagram shows the common base Colpitts circuit. The inductor L and the both the capacitors C1 & C2 are in series with the parallel resonant tank circuit and it gives the frequency of the oscillator. The voltage across the C2 terminal is applied to the base-emitter junction of the transistor to create the feedback oscillations.

<u>Circuit Diagram :</u>

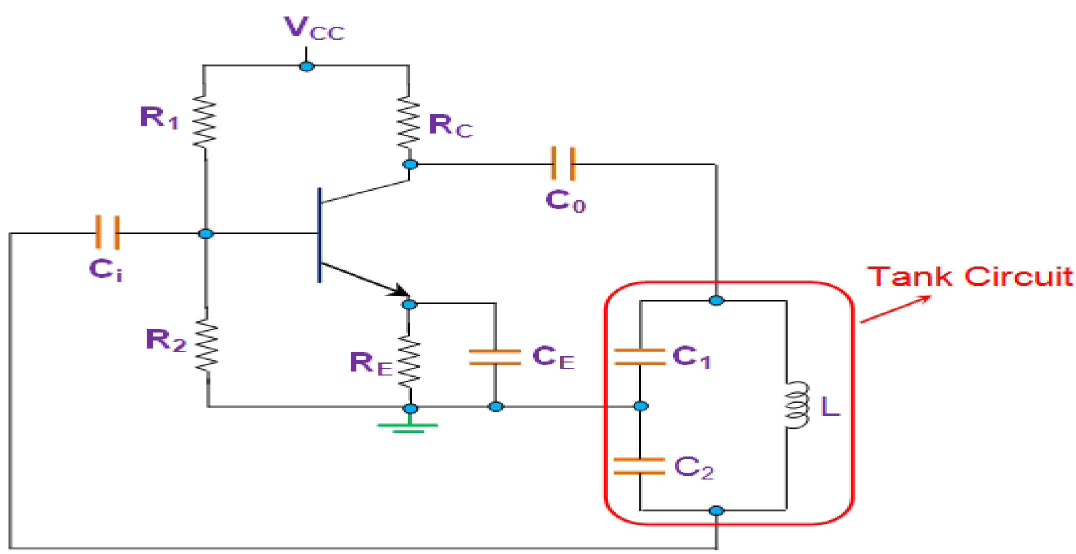


Figure 1 Colpitts Oscillator

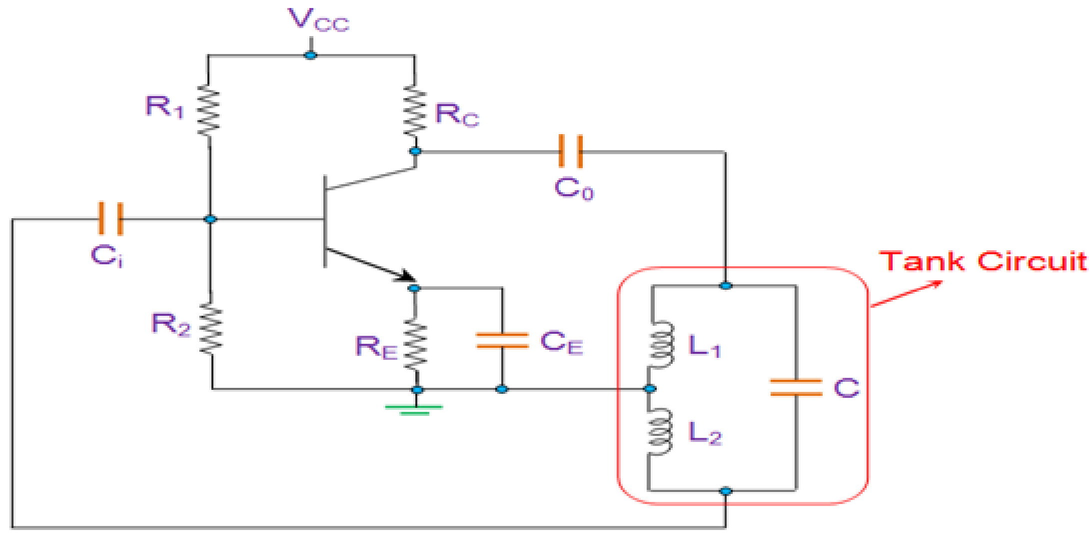


Figure 2: Hartley Oscillator

Procedure

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

RESULT:

Frequency of oscillation of Hartley & Collpit Oscillator is calculated

EXPERIMENT- 7

Objective: To study the JFET as an amplifier.

Resources Required: Trainer kit of JFET Amplifier

Theory:

A field-effect transistor (FET) is a type of transistor commonly used for weak-signal amplification (for example, for amplifying wireless (signals). The device can amplify analog or digital signals. It can also switch DC or function as an oscillator. In the FET, current flows along a semiconductor path called the channel. At one end of the channel, there is an electrode called the source. At the other end of the channel, there is an electrode called the drain. The physical diameter of the channel is fixed, but its effective electrical diameter can be varied by the application of a voltage to a control electrode called the gate. Field-effect transistors exist in two major classifications. These are known as the junction FET (JFET) and the metal-oxide-semiconductor FET (MOSFET). The junction FET has a channel consisting of N-type semiconductor (N-channel) or P-type semiconductor (P-channel) material; the gate is made of the opposite semiconductor type. In P-type material, electric charges are carried mainly in the form of electron deficiencies called holes.

The FET has some advantages and some disadvantages relative to the bipolar transistor. Field- effect transistors are preferred for weak-signal work, for example in wireless, communications and broadcast receivers. They are also preferred in circuits and systems requiring high impedance.

Circuit Diagram

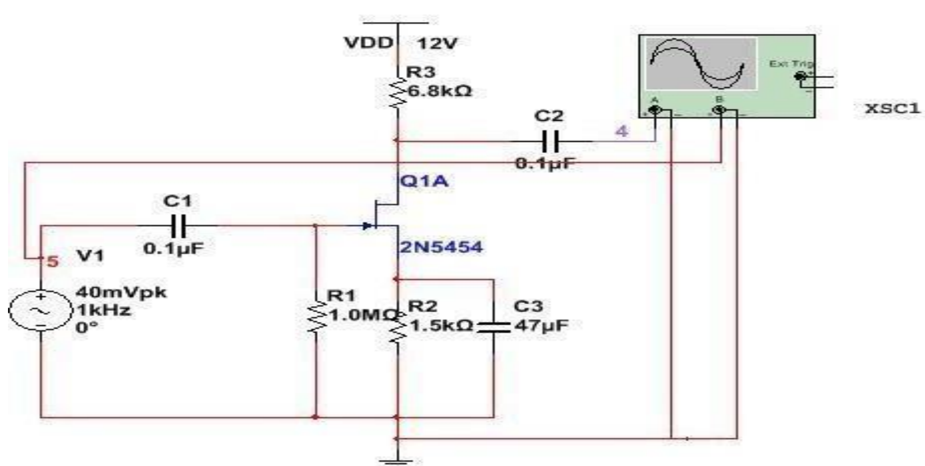


Fig: FET Amplifier

The FET is not, in general, used for high-power amplification, such as is required in large wireless communications and broadcast transmitters.Field-effect transistors are fabricated onto silicon integrated circuit (IC) chips. A single IC can contain many thousands of FETs, along with other components such as resistors, capacitors, and diodes. A common source amplifier FET amplifier has high input impedance and a moderate voltage gain. Also, the input and output voltages are 180 degrees out of Phase.

Procedure:

1. Place all the necessary components required for the design of the CS FET amplifier circuit i.e. Resistors, Capacitors, Transistors, Voltage sources, Power sources, Ground etc on the design window.
2. Connect all the components by proper wiring and also assure that nodes are formed at the interconnection points.
3. Connect the two channels of the Oscilloscope to input and output of the circuit.
4. From the frequency response, calculate the bandwidth of the Amplifier.

OBSERVATION TABLE:

S.No	Frequency(hz)	Output voltage(vo)	Voltage gain (vo/vi)	Gain (db) Avf=20 log (vo/vi).
1				

RESULT: We have obtained the frequency response of the common Source FET Amplifier and also found its Bandwidth to be___________________________Hz.

Objective: To study class-A power amplifier

.

Resources Required: Trainer kit of class-A power amplifier

Theory:

Class A Amplifiers are the most common type of amplifier topology as they use just one output switching transistor (Bipolar, FET, IGBT, etc) within their amplifier design. This single output transistor is biased around the Q-point within the middle of its load line and so is never driven into its cut-off or saturation regions thus allowing it to conduct current over the full 360 degreesof the input cycle. Then the output transistor of a class-A topology never turns "OFF" which is one of its main disadvantages.

Class "A" amplifiers are considered the best class of amplifier design due mainly to their excellent linearity, high gain and low signal distortion levels when designed correctly. Although seldom used in high power amplifier applications due to thermal power supply considerations, class-A amplifiers are probably the best sounding of all the amplifier classes mentioned here and as such are used in high-fidelity audio amplifier designs.

Circuit Diagram

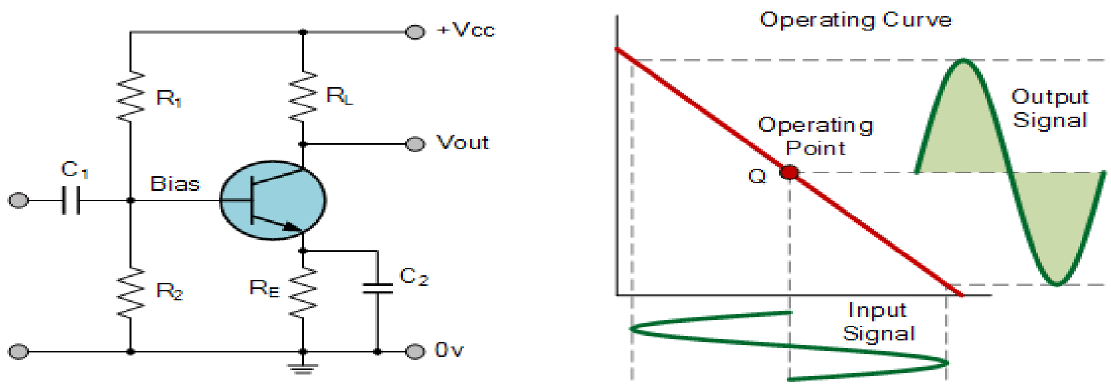


Fig: Class A Amplifier

PROCEDURE:

1. We should take all the components for this experiment.
2. Make the connection as per circuit diagram.
3. Switch ON the kit using ON/OFF toggle switch

4. The input signal is applied with the function generator.
5. The values of the voltage are observed and noted.
6. Calculate Gain.

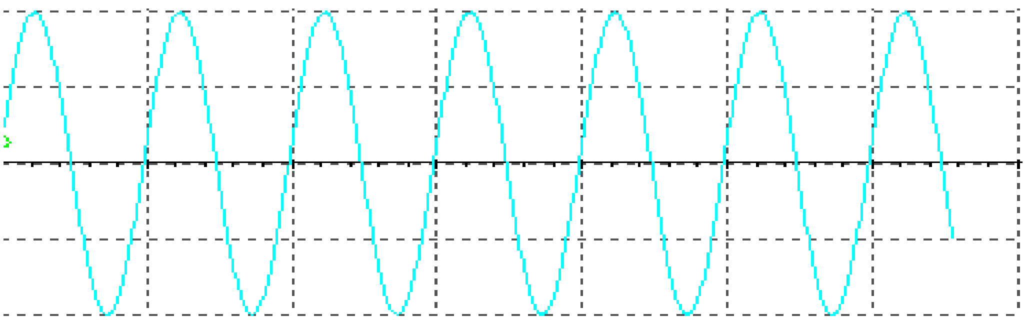
Calculations in a Class A Amplifier

The biasing base current is given by the formulaI(B) = (DC Input Voltage – V(BE))/R(b)

I(C) = I(B) × DC current gain

V(CE) = DC Input Voltage – (I(C) × R(c))Voltage gain = V(out)/V(in)

Model Graph:



Result: - From the above experiment, we learnt about the class of amplifier.

EXPERIMENT- 9

Objective: To study class-B power amplifier

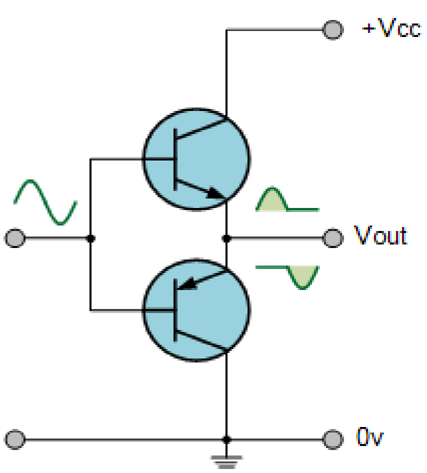
.

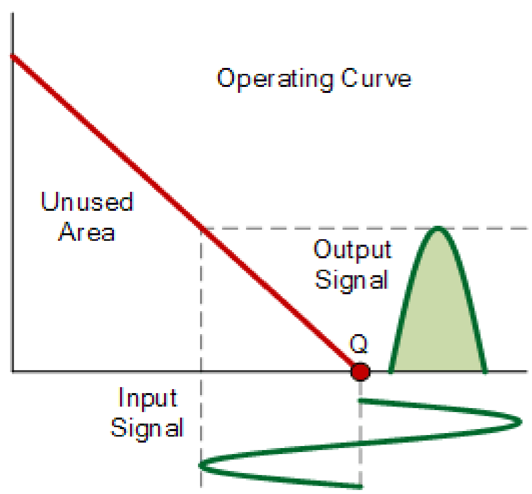
Resources Required: Trainer kit of class-B power amplifier

Theory:

Class B amplifiers were invented as a solution to the efficiency and heating problems associated with the previous class A amplifier. The basic class B amplifier uses two complimentary transistors either bipolar of FET for each half of the waveform with its output stage configured ina "push-pull" type arrangement, so that each transistor device amplifies only half of the output waveform.

In the class B amplifier, there is no DC base bias current as its quiescent current is zero, so that the dc power is small and therefore its efficiency is much higher than that of the class A amplifier. However, the price paid for the improvement in the efficiency is in the linearity of theswitching device.





When the input signal goes positive, the positive biased transistor conducts while the negative transistor is switched "OFF". Likewise, when the input signal goes negative, the positive transistor switches "OFF" while the negative biased transistor turns "ON" and conducts the negative portion of the signal. Thus the transistor conducts only half of the time, either on positive or negative half cycle of the input signal.

Then we can see that each transistor device of the class B amplifier only conducts through one half or 180 degrees of the output waveform in strict time alternation, but as the output stage has devices for both halves of the signal waveform the two halves are combined together to produce the full linear output waveform.

This push-pull design of amplifier is obviously more efficient than Class A, at about 50%, but theproblem with

the class B amplifier design is that it can create distortion at the zero-crossing point of the waveform due to the transistors dead band of input base voltages from -0.7V to
+0.7.

We remember from the transistor tutorial that it takes a base-emitter voltage of about 0.7 voltsto get a bipolar transistor to start conducting. Then in a class B amplifier, the output transistor isnot "biased" to an "ON" state of operation until this voltage is exceeded.

This means that the the part of the waveform which falls within this 0.7 volt window will not bereproduced accurately making the class B amplifier unsuitable for precision audio amplifier applications.

Circuit Diagram:

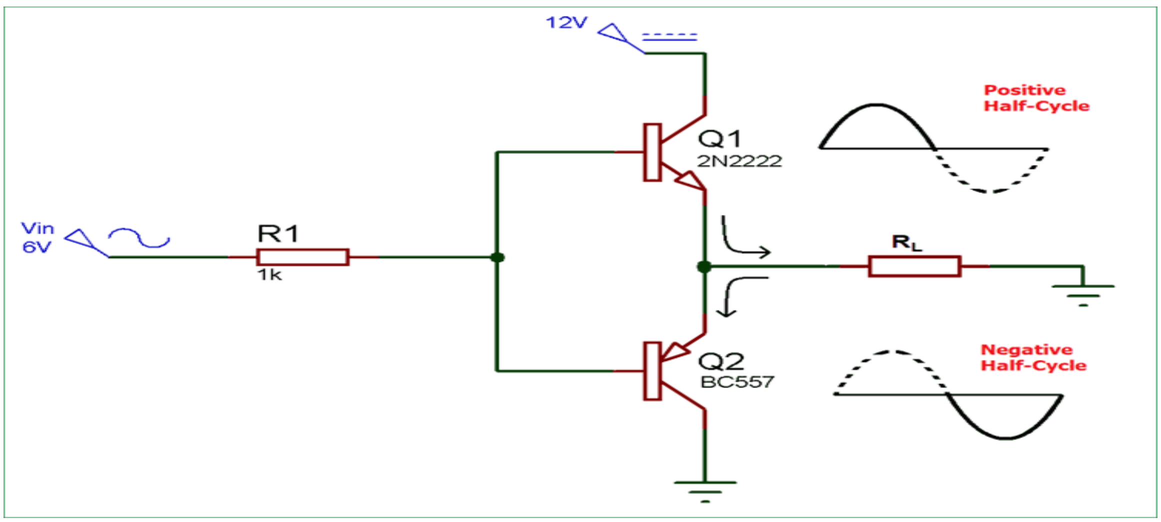


Fig: Class B Amplifier

Procedure:

1. We should take all the components for this experiment.
2. Make the connection as per circuit diagram.
3. Switch ON the kit using ON/OFF toggle switch
4. The input signal is applied with the function generator.
5. Connect the output to the CRO.
6. Now very the amplitude and frequency of the signal by their respective points.
7. See the output on CRO and verify the difference between input and output.
8. Then observe the wave form.

Result: From the above experiment, we learnt about the class of amplifier

EXPERIMENT- 10

Objective: To study complementary symmetry push pull amplifier

.

Resources Required: Trainer kit of complementary symmetry push pull amplifier

Theory:

The Class B amplifier circuit above uses complimentary transistors for each half of the waveform and while Class B amplifiers have a much high efficiency than the Class A types, one of the main disadvantages of class B type push-pull amplifiers is that they suffer froman effect known commonly as Crossover Distortion. It takes approximately 0.7 volts (measured from base to emitter) to get a bipolar transistor to start conducting. In a class B amplifier, the output transistors are not "pre -biased" to an "ON" state of operation. This means that the part of the output waveform which falls below this 0.7 volt window will not be reproduced accurately as the transition between the two transistors (when they are switching over from one to the other), the transistors do not stop or start conducting exactly at the zero crossover point even if they are specially matched pairs. The output transistors for each half of the waveform (positive and negative) will each have a 0.7 volt area in which they will not be conducting resulting in both transistors being "OFF" at the same time. A simple way to eliminate crossover distortion in a Class B amplifier is to add two small voltage sources to the circuit to bias both the transistors at a point slightly above their cut- off point.. However, it is impractical to add additional voltage sources to the amplifier circuit so pn-junctions are used to provide the additional bias in the form of silicon diodes.

We know that we need the base-emitter voltage to be greater than 0.7v for a silicon bipolar transistor to start conducting, so if we were to replace the two voltage divider biasing resistors connected to the base terminals of the transistors with two silicon Diodes, the biasing voltage applied to the transistors would now be equal to the forward voltage drop of the diode. These two diodes are generally called Biasing Diodes or Compensating Diodes and are chosen to match the characteristics of the matching transistor

Circuit Diagram

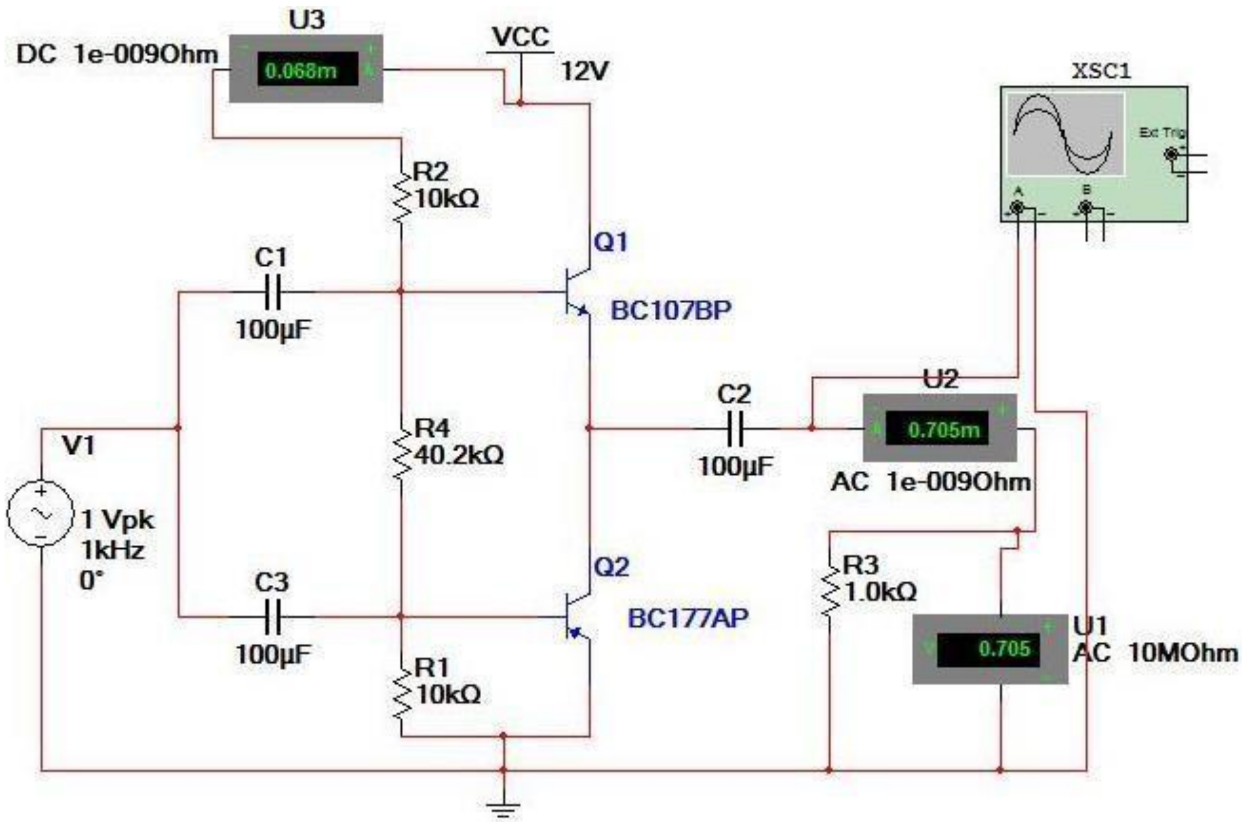
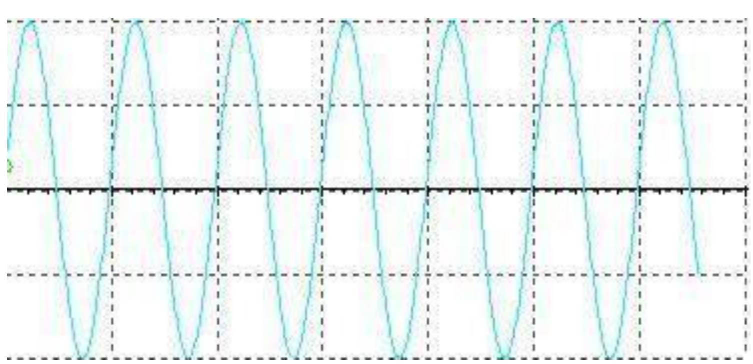


Fig : complementary symmetry push pull amplifier

Model Graph:



Procedure:

1. We should take all the components for this experiment.

2. Make the connection as per circuit diagram.
3. Switch ON the kit using ON/OFF toggle switch
4. The input signal is applied with the function generator.
5. Connect the output to the CRO.
6. Now very the amplitude and frequency of the signal by their respective points.
7. See the output on CRO and verify the difference between input and output.
8. Then observe the wave form.

Result: From the above experiment, we learnt about the complementary symmetry push pull amplifier

.