Chettinad College of Engineering & Technology
Approved by AICTE-New Delhi and Affiliated to Anna University-Chennai.

# DEPARTMENT OF

# ELECTRONICS AND COMMUNICATION ENGINEERING

# EC3462 LINEAR INTEGRATED CIRCUITS LABORATORY

## 2021 REGULATION

## (As per ANNA University)

## OBSERVATION RECORD

**Name of the student :** ______________________

**Register Number :** ______________________

**Year/ Semester/ section:** ______________________

# CHETTINAD COLLEGE OF ENGINEERING AND TECHNOLOGY

**(Puliyur, Karur – 639114)**



## PROGRAM OUTCOMES:

**1.** Engineering knowledge: Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.

**2.** Problem analysis: Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.

**3.** Design/development of solutions: Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.

**4.** Conduct investigations of complex problems: Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.

**5.** Modern tool usage: Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modelling to complex engineering activities with an understanding of the limitations.

**6.** The engineer and society: Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.

**7.** Environment and sustainability: Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.

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

**9**. Individual and team work: Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.

**10.** Communication: Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.

**11**. Project management and finance: Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects and in multidisciplinary environments.

**12**. Life-long learning: Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change

## PROGRAM SPECIFIC OUTCOME (PSOs)

**PSO1:** Design, develop and analyze electronic systems through application of relevant electronics, mathematics and engineering principles

**PSO2:** Design, develop and analyze communication systems through application of fundamentals from communication principles, signal processing, and RF System Design & Electromagnetics.

**PSO3:** Adapt to emerging electronics and communication technologies and develop innovative solutions for existing and newer problems

**EC3462 LINEAR INTEGRATED CIRCUITS LABORATORY L T P C**
**0 0 3 1.5**

**COURSE OBJECTIVES:**

● To gain hands on experience in designing electronic circuits
● To learn simulation software used in circuit design
● To learn the fundamental principles of amplifier circuits
● To differentiate feedback amplifiers and oscillators.
● To differentiate the operation of various multivibrators

**LIST OF EXPERIMENTS:**

**DESIGN AND ANALYSIS OF THE FOLLOWING CIRCUITS**

1. Series and Shunt feedback amplifiers-Frequency response, Input and output impedance
2. RC Phase shift oscillator and Wien Bridge Oscillator
3. Hartley Oscillator and Colpitts Oscillator
4. RC Integrator and Differentiator circuits using Op-Amp
5. Clippers and Clampers
6. Instrumentation amplifier
7. Active low-pass, High pass & Band pass filters
8. PLL Characteristics and its use as frequency multiplier, clock synchronization
9. R-2R ladder type D-A converter using Op-Amp

**SIMULATION USING SPICE (Using Transistor):**
1. Tuned Collector Oscillator
2. Twin -T Oscillator / Wein Bridge Oscillator
3. Double and Stagger tuned Amplifiers
4. Bistable Multivibrator
5. Schmitt Trigger circuit with Predictable hysteresis
6. Analysis of power amplifier

**Components and Accessories:**

Transistors, Resistors, Capacitors, Inductors, diodes, Zener Diodes, Bread Boards, Transformers. SPICE Circuit Simulation Software: (any public domain or commercial software)

**Note:** Op-Amps uA741, LM 301, LM311, LM 324, LM317, LM723, 7805, 7812, 2N3524, 2N3525, 2N3391, AD 633, LM 555, LM 565 may be used

**TOTAL: 45 PERIODS**



## LIST OF EXPERIMENTS

| | | |
|---|---|---|
| | R-2R ladder type D-A converter using Op-Amp | |
| **SIMULATION USING SPICE (Using Transistor)** | | |
| | Tuned Collector Oscillator | |
| | Twin -T Oscillator / Wein Bridge Oscillator | |
| | Double and Stagger tuned Amplifiers | |
| | Bistable Multivibrator | |
| | Schmitt Trigger circuit with Predictable hysteresis | |
| | Analysis of power amplifier | |

| Ex. No: 1(a) | DESIGN AND ANALYSIS OF CURRENT SERIES FEEDBACK AMPLIFIER |
|---|---|
| Date: | |

**AIM:**

To design and analyze the current series feedback amplifier and calculate the Bandwidth, Input impedance and Output impedance for with & without feedback amplifier.

**EQUIPMENTS REQUIRED:**

| S.NO | EQUIPMENTS | RANGE | QUANTITY |
|---|---|---|---|
| 1 | Transistor | BC 107 | 1 |
| 2 | Resistor | 56KΩ,2.2KΩ,12KΩ,470Ω | Each one |
| 3 | Capacitor | 22µF ,2.2µF, 47µF | Each one |
| 4 | CRO | (0-30 )MHz | 1 |
| 5 | RPS | (0-30) V | 1 |
| 6 | Function Generator | (0 –3)MHz | 1 |
| 7 | Bread board | - | 1 |
| 8 | Connecting wires | - | As required |

**PRE-LAB QUESTIONS:**

1. What is feedback?
2. What are the types of feedback?
3. List the merits of negative feedback.
4. What is the difference between current and voltage feedback?
5. What is the effect of negative feedback on the gain in an amplifier?

**THEORY:**

A portion of the output signal is taken from the output of the amplifier and is combined with the normal input signal and thereby feedback is accomplished. There are two types of feedback, positive feedback and negative feedback. Feedback increases the stability of an amplifier, increases the bandwidth and reduces the distortion and noise. The property of positive feedback is utilized in oscillators. The series connection at the output increases the output resistance. Common emitter amplifier is an examplefor current series feedback amplifier.

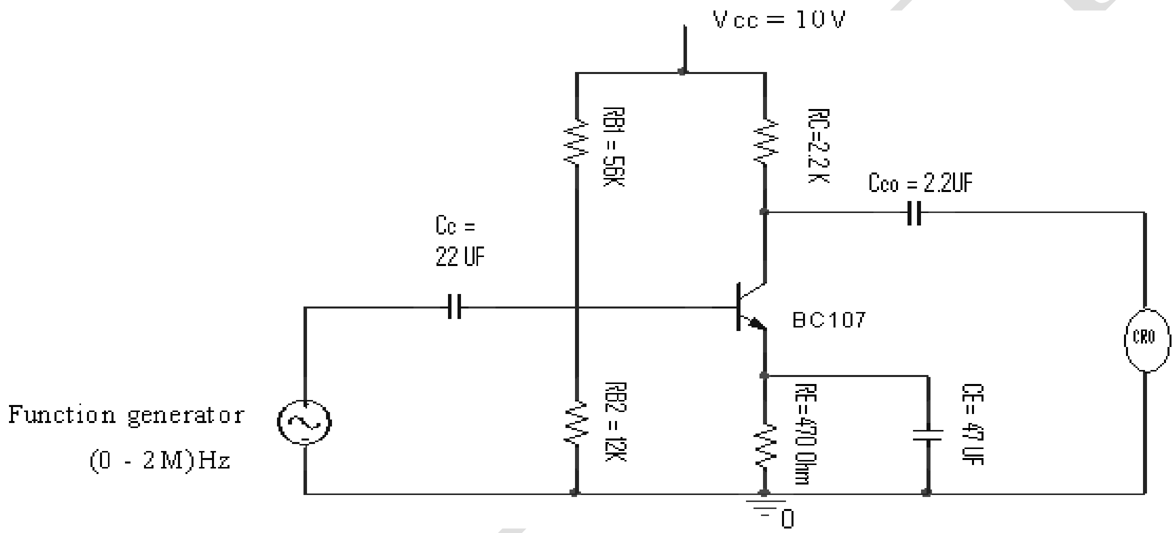
| Parameter | With feedback |
|---|---|
| Input Impedance | Increases |
| Output Impedance | Increases |
| Gain | Decreases |
| Bandwidth | Increases |

**PROCEDURE:**

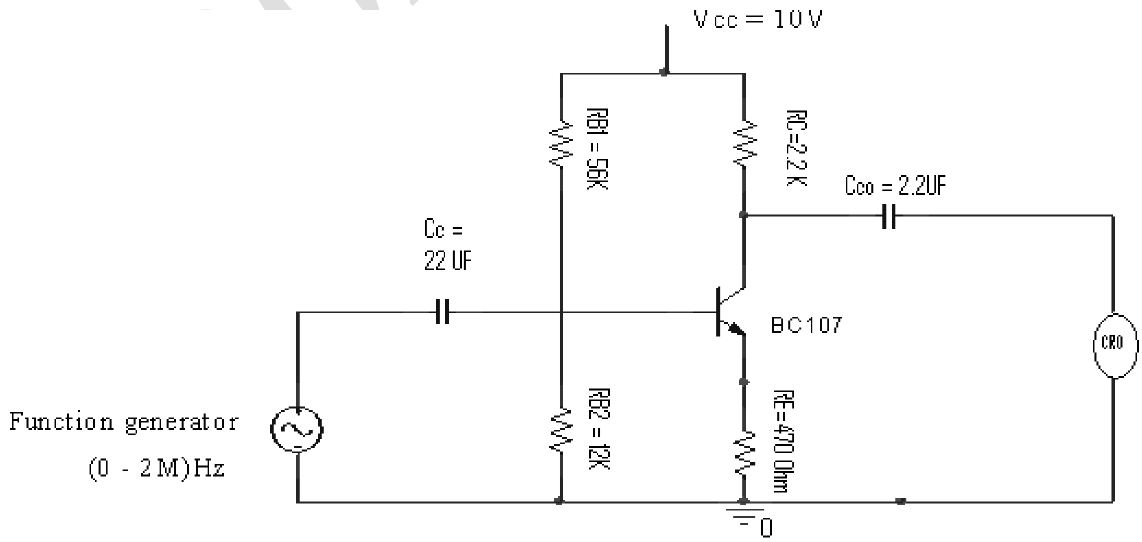
1. Design the current series feedback amplifier.
2. The connections are made as per the circuit diagram.
3. Set the input voltage to a fixed value. Measure the output voltages for the various input frequencies. Calculate the voltage gain in dB for each input frequency.
4. Plot the graph: Frequency in Hz Vs Gain in dB for the amplifier with feedback.
5. Then the capacitor $C_E$ is included in parallel with $R_E$ to get the amplifier without feedback.
6. The steps (2, 3, and 4) are repeated for an amplifier without feedback.
7. Calculate the input impedance and output impedance values.

**CIRCUIT DIAGRAM:**

**WITHOUT FEEDBACK**




**WITH FEEDBACK**



**DESIGN PROCEDURE:**

Assumptions: $V_{CC}$ = 10v, $V_E = V_{CC}/10$ = 1V, $V_{CE} = V_{CC}$ / 2 = 5; $R_L$ = 100 ohms ,$F_L$=1KHz Since $I_B$ is very small compare with Ic , $I_E$ ~ Ic = 2mA $A_V$ = 30; $h_{fe}$ = 120,S =5, $h_{ie}$ =1.56 KΩ, $R_{leff}$ = 234Ω, $R_{B1}$ = 56K Ω, $R_{B2}$ =12K Ω,D=29

**1. Calculation of $R_E$:**

$R_E = V_E / I_E$ = 470 Ω

$V_B = V_{CC.}R_{B1} / R_{B1} + R_{B2}V_B$

= 1.3V

**2. Calculation of $R_C$ :**

$V_{cc} = I_cR_c + V_{CE} + I_ER_E$

$I_C$ □ $I_E$

$R_c$ = 2.2K Ω

**3. Calculation of $R_B$:**

$R_B = R_{B1}$ □ □

$R_{B2}R_{B1}$ = 56K

Ω $R_{B2}$ =12K

Ω $R_B$ = 9.88K

Ω

**4. Calculation of Input impedance $Z_i$ :**

$Z_I = h_{ie}$ □ □

$R_B$ $Z_I$ =

1.34K Ω

**5. Calculation of Output impedance $Z_0$ :**

$Z_0 = R_c$ =2K Ω

| Parameter | Without feedback | With Feedback |
|---|---|---|
| Input Impedance | $Z_i = R_B$ // hie = 1.34K Ω | $Z_{if} = Z_i$ D = 25.23KΩ |
| Output Impedance | $Z_O = R_C$ // $R_L$= 2K Ω | $Z_{Of} = Z_O$ D = 58KΩ |

**TABULATION:**

**WITHOUTFEEDBACK :**

Vin =

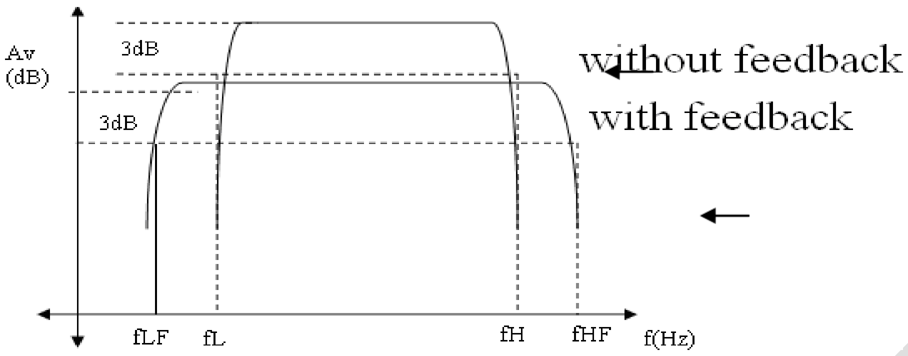
| S.NO | Frequency in Hz | Output voltage in Volts | Voltage gain | Voltage gain in dB |
|---|---|---|---|---|
| | | | | |

**WITH FEEDBACK:**

Vin =

| S.NO | Frequency in Hz | Output voltage in Volts | Voltage gain | Voltage gain in dB |
|---|---|---|---|---|
| | | | | |

**MODELGRAPH:**




**POST-LAB QUESTIONS:**

1. How do series feedback and shunt feedback differ from each other?
2. What is the effect of negative feedback on the input impedance of a voltage series feedback amplifier?
3. Why voltage series feedback is most commonly used in cascaded amplifiers?
4. What is the application of Current deries feedback amplifier?
5. Define transconductance and transresistance amplifiers.



**RESULT:**

Thus the current series feedback amplifier is designed for without and with feedback.

The circuit is analyzed to get the frequency response and measured the following parameters

Lower cutoff Frequency $f_L$ :

Upper cutoff Frequency $f_H$ :

Bandwidth :

Input impedance :

Output impedance :

| Ex. No: 1(b) | DESIGN AND ANALYSIS OF VOLTAGE SHUNT FEEDBACK AMPLIFIER |
|---|---|
| Date: | |

## AIM

To design and analyze the voltage shunt feedback amplifier and calculate the Bandwidth, Input impedance and Output impedance for with & without feedback amplifier.

## EQUIPMENTS REQUIRED:

| S.NO | EQUIPMENTS | RANGE | QUANTITY |
|---|---|---|---|
| 1 | Transistor | BC 107 | 1 |
| 2 | Resistor | 56KΩ,2.2KΩ,12KΩ,470Ω, 1 KΩ | Each one |
| 3 | Capacitor | 22µF ,2.2µF, 47µF, 0.1 µF | Each one |
| 4 | CRO | (0-30 )MHz | 1 |
| 5 | RPS | (0-30) V | 1 |
| 6 | Function Generator | (0 –3)MHz | 1 |
| 7 | Bread board | - | 1 |
| 8 | Connecting wires | - | As required |

## PRE-LAB QUESTIONS:

1. What is the another name for Voltage shunt feedback amplifiers?
2. What is the principle of voltage shunt feedback amplifier?
3. What is the inverting input of the voltage shunt feedback resistor commonly named as ?
4. Which is not a special case of voltage shut feedback amplifier?
5. Write the merits of voltage shunt feedback amplifier?

## THEORY:

A portion of the output signal is taken from the output of the amplifier and is combined with the normal input signal and thereby feedback is accomplished. There are two types of feedback, positive feedback and negative feedback. Feedback increases the stability of an amplifier, increases the bandwidth and reduces the distortion and noise. The property of positive feedback is utilized in oscillators. The shunt connection at the output decreases the output resistance. Common emitter amplifier with a resistive feedback from collector to base is an example for voltage shunt feedback amplifier.

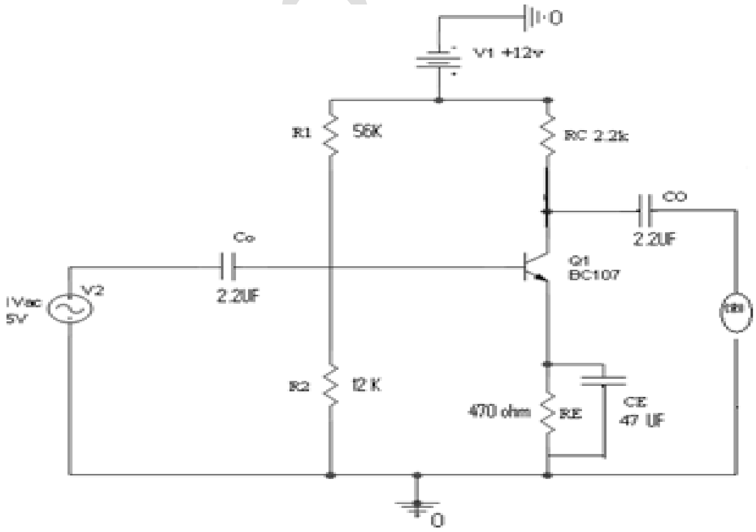
| Parameter | With feedback |
|---|---|
| Input Impedance | Decreases |
| Output Impedance | Decreases |
| Gain | Decreases |
| Bandwidth | Increases |

**PROCEDURE:**

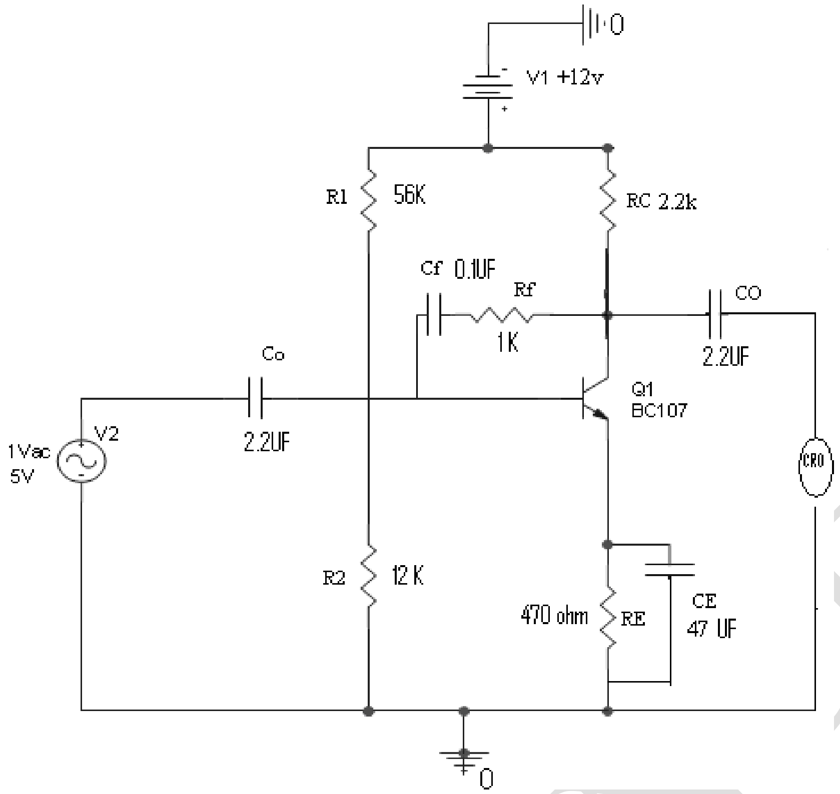
1. Design the voltage shunt feedback amplifier.
2. The connections are made as per the circuit diagram.
3. Set the input voltage to a fixed value. Measure the output voltages for the various input frequencies. Calculate the voltage gain in dB for each input frequency.
4. Plot the graph: Frequency in Hz Vs Gain in dB for the amplifier with feedback.
5. Then the capacitor $C_f$ and resistance $R_f$ is removed from the feedback to get the amplifier without feedback
6. The steps (2, 3, and 4) are repeated for an amplifier without feedback.
7. Calculate the input impedance and output impedance values.

**CIRCUIT DIAGRAM:**

` **WITHOUT FEEDBACK**



**WITH FEEDBACK:**




**DESIGN PROCEDURE:**

Assumptions: $V_{CC}$ = 12v, AV=30 $f_L$=1KHz,S=2, $I_E$=1.2mA,$R_L$=4.7K Ω, β = 0.4;

1. **Calculation of $h_{ie}$:**

   $r_e = 26\ mV / I_{E\ 2}$

   $h_{ie} = h_{fe}\ r_e$

2. **Caluclation of $A_{Vf}$**

   $A_v$= hfe $R_c$□□ $R_L$

   ―――――――

   hie

   $V_E = V_{CC}/10$ ,$V_{CE} = V_{CC}/ 2$

3. **Calculation of $R_C$ :**

   $V_{cc}=I_cR_c+V_{CE}+I_ER_E$

   $R_C = (V_{cc} - V_{CE} - I_ER_E)$

   /$I_cI_C$□$I_E$

   $V_B=V_{CC}.R_2/R_1+R_2$

4. **Calculation of $R_E$:**

   $R_E=V_E/I_E$

**5.** **Calculation of $R_B$:**

$R_B = R_1 \square\square R_2$

**6.** **Calculation of Input impedance $Z_i$ :**

$R_I = h_{ie} \square\square\ R_B\ \square\square R_F$

**7.** **Calculation of Input impedance $Z_0$ :**

$R_0 = R_c \square\square R_F$

**8.** **Calculation of Coupling Capacitors:**

$X_{ci} = R_i/10$

$X_{c0} = R_c \square\square\ R_L$

**WITHFEEDBACK**D

$=1+A\beta A_{VF} = A_v/D$

$R_{if} = R_i/D$

$R_{of} = R_0/D$

| Parameter | Without feedback | With Feedback |
|---|---|---|
| Input Impedance | $R_I = h_{ie} \square\square\ R_B\ \square\square R_F$ | $R_{if} = R_i/D$ |
| Output Impedance | $R_0 = R_c \square\square R_F$ | $R_{of} = R_0/D$ |
| Gain | $A_v = \frac{hfe\ R_c \square\square\ R_L}{Hie}$ | $A_{VF} = A_v/D$ |

**TABULATION:**

**WITHOUT FEEDBACK:**

Vin =

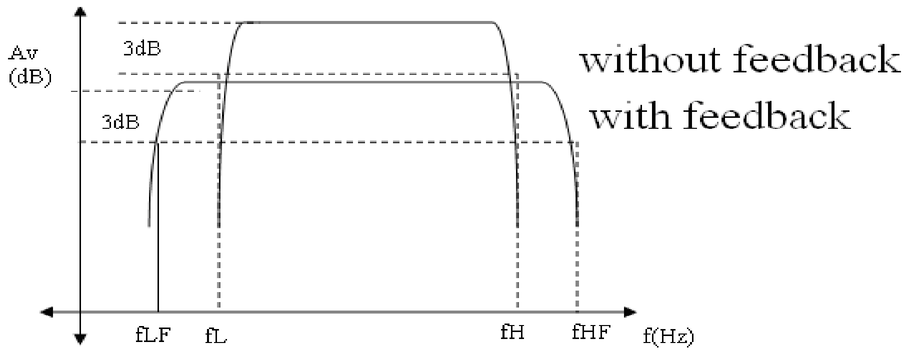
| S.NO | Frequency in Hz | Amplitude in Volts | Voltage gain | Voltage gain in dB |
|---|---|---|---|---|
| | | | | |

**WITH FEEDBACK:**

Vin =

| S.NO | Frequency in Hz | Amplitude in Volts | Voltage gain | Voltage gain in dB |
|---|---|---|---|---|
| | | | | |

**MODELGRAPH:**




**POST LAB QUESTIONS :**

1. What is the application of Voltage shunt feedback amplifiers?
2. How is the i/p impedance and o/p impedance of a voltage shunt feedback amplifier
3. What is the relationship between the transfer gain with feedback Af and that without feedback A
4. What is the effect of voltage shunt feedback amplifier?
5. What is Barkhusen criterion?

**RESULT:**

Thus the voltage shunt feedback amplifier is designed for without and with feedback. The circuit is analyzed to get the frequency response and measured the following parameters

Lower cutoff Frequency $f_L$ :

Upper cutoff Frequency $f_H$ :

Bandwidth :

Input impedance :

Output impedance :

| Ex. No: 2(a) | DESIGN AND ANALYSIS OF RC PHASE SHIFT OSCILLATOR |
|---|---|
| Date: | |

**AIM:**

To design and analyze a RC phase shift oscillator for a given frequency and test the result withthe design.

**EQUIPMENTS REQUIRED:**

| S.NO | EQUIPMENTS | RANGE | QUANTITY |
|---|---|---|---|
| 1 | Transistor | BC 107 | 1 |
| 2 | Resistor | 56K, 12KΩ, 470 Ω, 2.2 KΩ | Each 1 |
| 3 | Resistor | | Each 3 |
| 4 | Capacitor | 0.01µF | 3 |
| 5 | CRO | (0-30 )MHz | 1 |
| 6 | RPS | (0-30) V | 1 |
| 7 | Bread board | - | 1 |
| 8 | Connecting wires | - | As required |

**PRELAB QUESTIONS:**

1. What is oscillator?
2. What are the types of oscillator?
3. What is Phase shift oscillator?
4. What kind of amplifier used in RC phase shift oscillator?
5. As Per Barkhausen criterion how much should be phase shift?

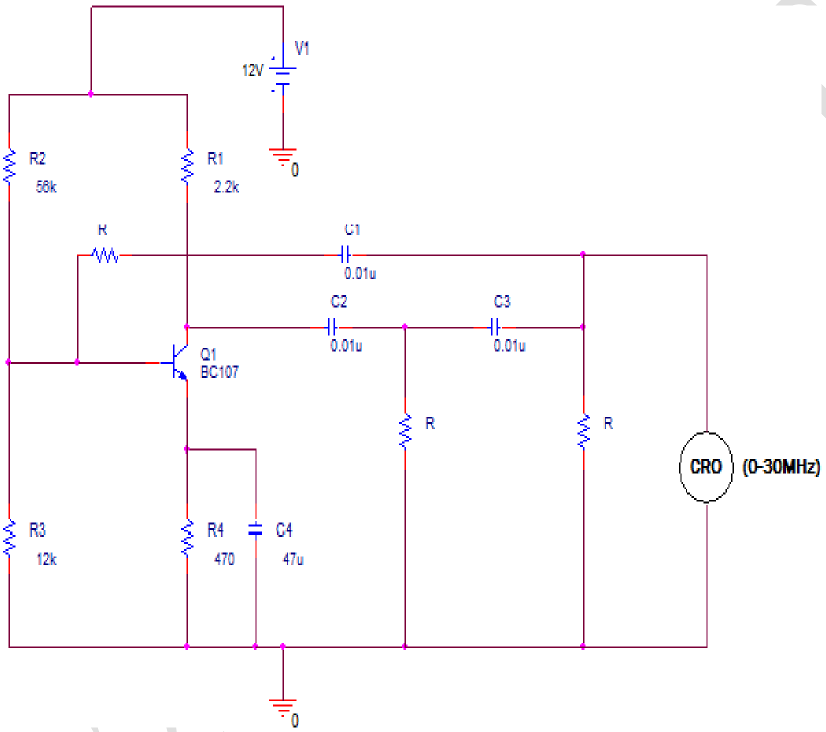
**THEORY:**

A common emitter amplifier followed by three sections of RC phase shift network, the output of the last section being returned to the input form a RC phase shift oscillator. In practice the value of R is adjusted such that the phase angle becomes 60. If the values of R and C are chosen so that for the given frequency the phase shift of each RC section is 60. Thus such a RC ladder network produces a total phase shift of 180 between its input and output voltages for only the given frequency the total phase shift from the base of the transistor around the circuit and back to the base will be exactly 360 or 0, thereby satisfying Barkhausan condition for oscillation. The RC phase shift oscillator is suitable for audio frequencies only. Its main drawbacks are that the three capacitors and resistors should be changed simultaneously to change the frequency of oscillation and it is difficult to control the amplitude of oscillation without affecting the frequency of oscillation. It is mainly used in audio frequency transmitter for generating carrier signals.

**PROCEDURE:**

1. Design the RC phase shift oscillator.
2. The connections are made as per the circuit diagram.
3. Measure the output sine wave form using CRO.
4. The value of frequency is calculated and compared with the design.

**CIRCUIT DIAGRAM:**



**DESIGN PROCEDURE:**

**Design of a Transistor circuit:**

$V_{CC}$ = 10V, Ic = 2mA, β = 100

$I_B = I_C / \beta$ =0.02mA

$V_{CE} = V_{CC} / 2$ =5V

$V_{BE}$ = 10% of $V_{CC}$ = 1.0V

$V_B = V_{RE} + V_{BE}$ = 1.0 + 0.7 = 1.7V

$R_E = V_{RE} / I_E$ = 1.0 / (2.02*10^-3) = 495Ω = 470 Ω

R2 = $V_B$ / (10* $I_B$) = 8.5K Ω = 12K Ω

R1 = [$V_{CC}$ / (10* $I_B$)] – $R_2$ = 50K Ω = 56K Ω

RC = ($V_{CC} – V_{CE} – I_E R_E$) / $I_C$ = 2.2K Ω

$X_{CE}$ must be equal to one tenth of value of $R_E$ at the lowest operating frequency.

$X_{CE}$ = RE / 10 = 47, ω=2πf, f=75Hz

- ⇨ 1/(ω $C_E$) =47
- ⇨ $C_E$ = 56μF = 47μF

Choose coupling capacitor = 0.01μF

**Design of Tank circuit:**

Given frequency = 1KHZ and assume C = 0.01μF

Design Equation: Frequency f = 1 / (2πRC√4K+6)

=> R = 6.8K Ω

**Barkhausen Criterion:**

K = Rc / R

= 2.4K / 6.5K

= 0.32

$h_{fe}$ = 4K + 23 +29 / K

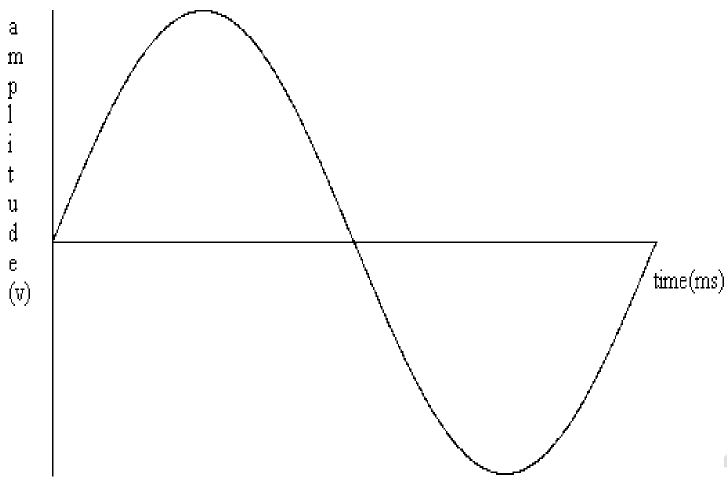
= 114.99

**TABULATION:**

| S.No | Capacitor | Resistor | Theoretical frequency in KHz | Amplitude in Volts | Time period in ms | Observed frequency in KHz |
|---|---|---|---|---|---|---|
| 1 | 0.01uF | | 1K | | | |
| 2 | 0.01uF | | 1.5K | | | |

| | | | | | | |
|---|---|---|---|---|---|---|
| 3 | 0.01uF | | 2K | | | |
| 4 | 0.01uF | | 2.5K | | | |

**MODEL GRAPH:**




**POST LAB QUESTIONS:**

1.What is the principle of RC phase shift oscillator?

2. What are the applications of RC phase shift oscillator?

3.Why an RC Phase shift oscillator not used in high frequency?

4. Why we need Phase shift between input and output?

5. How is the phase angle determined in RC Phase shift oscillator?



**RESULT:**

Thus the RC phase shift oscillator is designed and analyzed as per the design.

Designed frequency =

Obtained frequency =

| Ex. No: 2(b)<br>Date: | DESIGN AND ANALYSIS OF WEIN BRIDGE OSCILLATOR. |
|---|---|

**AIM:**

To design and analyze a wein bridge oscillator for a given frequency and test the result with the design.

**EQUIPMENTS REQUIRED:**

| S.NO | EQUIPMENTS | RANGE | QUANTITY |
|---|---|---|---|
| 1 | Transistor | BC 107 | 1 |
| 2 | Resistor | 470 Ω | 1 |
| 3 | Resistor | 56 KΩ, 12 KΩ, 1 KΩ, 1.5 KΩ<br>1 KΩ | Each 2 |
| 4 | Resistor | 4 KΩ | 1 |
| 5 | Capacitor | 0.1µF, 10 µF | Each 2 |
| 6 | CRO | (0-30 )MHz | 1 |
| 7 | RPS | (0-30) V | 1 |
| 8 | Bread board | - | 1 |
| 9 | Connecting wires | - | As required |

**PRE LAB QUESTIONS:**

1.Give the formula for frequency of oscillations in Wein Bridge Oscillator circuit?

2.What is the condition for Wien Bridge oscillator to generate oscillations?

3.What is the total phase shift provided by the Wein Bridge oscillator?

4.What is the function of lead-lag network in Wein Bridge oscillator?

5. Which type of feedback is used in Wein Bridge oscillator

**THEORY:**

Wein bridge oscillator is an audio frequency RC oscillator. The amplifier used in wein bridge oscillator is non inverting type. Here two common emitter amplifier is cascaded to produce a phase shiftof 0. Therefore the feedback network need not provide any phase shift. By changing the value of Rand C in frequency sensitive arm we can change the output signal frequency. To generate low frequency audiosignal wein bridge oscillators is used in signal generators.

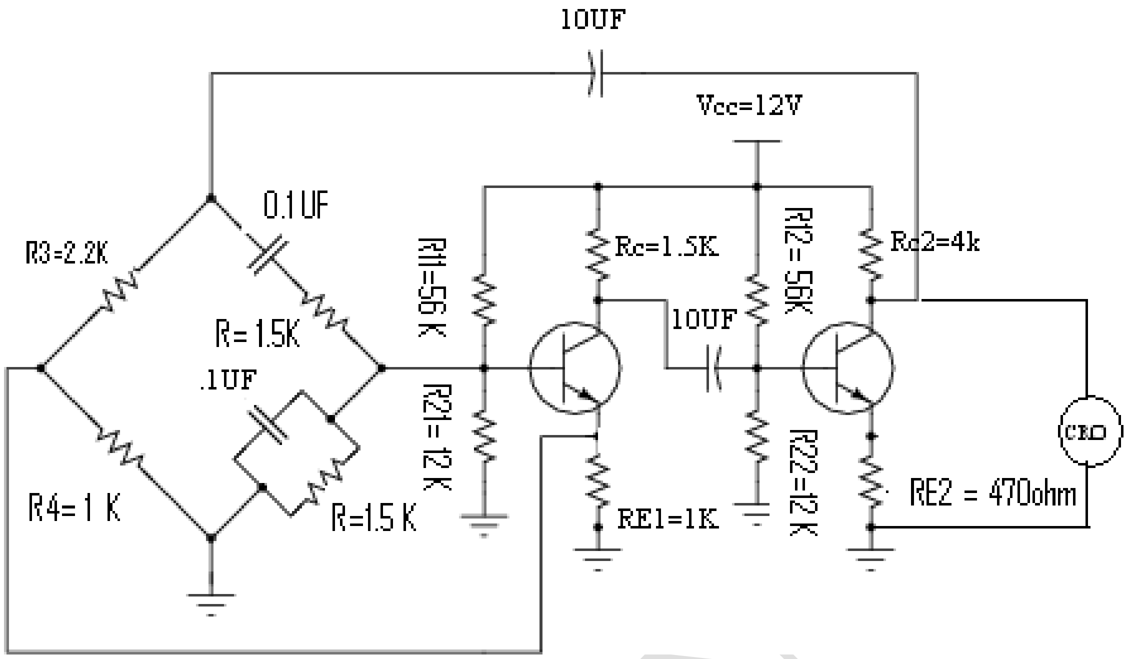
Frequency of oscillation = $1 / (2\pi RC\sqrt{6})$

**PROCEDURE:**

1. Design the wein bridge oscillator as per the design.
2. The connections are made as per the circuit diagram.

3. Measure the output sine wave form using CRO.
4. The value of frequency is calculated and compared with the design

**CIRCUIT DIAGRAM:**




**DESIGN PROCEDURE:**

**Design of a Transistor circuit 1:**

Assume $V_{cc}$= 12V,$I_c$ = 4mA = $I_E$,$V_{CE}$ = $V_{CC}/2$ = 6V, $V_E$ = $V_{CC}/10$, $R_{11}$ = 56KΩ

Z parallel =$R_C$ / $X_C$ = = 9.42Ω

Z series = R + $X_C$ == 1.6KΩ

$Z_f$ = 1.61KΩ

**To find $R_{E1}$:**

$R_{E1} = V_E / I_E$ = **1KΩ**

**To find $R_{C1}$**

$R_{C1} = V_{CC} / 2I_C$ = **1.5KΩ**

**To find $R_{12}$**

$R_{11} \| R_{12} > 10Z_F$ **$R_{12}$ = 56KΩ**

**Design of a Transistor circuit 2:**

Assume $V_{CC}$ = 12V,$I_C$ = 1.5mA = $I_E$,$V_{CE}$= $V_{CC}/ 2$ = 6V ,$V_E$ = VCC/10 =1.2V, $R_{22}$ = 12K

**To find $R_{E2}$**

$R_{E2} = V_E / I_E = 470\Omega$

**To find $R_{C2}$**

$R_{C2} = V_{CC} / 2I_C = 4K\Omega$

$V_B = V_{BE} + V_E = 1.9V$ $\quad$ $V_B = V_{CC} (R_{B2} / R_{22} + R_{21})$

**To find $R_{21}$**

$R_{21} + R_{22} > 10Z_F$ $\quad$ $R_{21} = 12K\Omega$

**Design of Tank circuit:**

Assume: $f = 1$ KHz, $C = 0.1\mu f$ $\quad$ $F = 1 /$

$2\pi RC R = 1.5K\Omega$

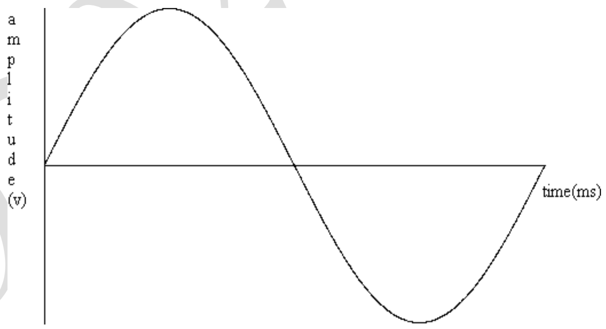
**Barkhausen Criterion:**

Assume $R_4 = 1K\Omega$ $\quad$ $R_3 / R_4 = 2$; $R_3 = 2K\Omega$

**TABULATION:**

| S.No | Amplitude in Volts | Time period in ms | Theoretical frequency in KHz | Observed frequency in KHz |
|---|---|---|---|---|
| | | | | |

**MODEL GRAPH:**




**POST LAB QUESTIONS:**

1. What are the applications of Wein Bridge oscillator
2. What is the condition for generating oscillations?
3. What is the difference between damped oscillations undamped oscillations?

4. Wein Bridge oscillator is either LC or RC oscillator.
5. What are the drawbacks in using Wein Bridge Oscillators?



**RESULT:**

Thus the Wein bridge oscillator is designed and analyzed as per the design.

Designed frequency =

Obtained frequency =

| Ex. No: 3(a) | DESIGN AND ANALYSIS OF HARTLEY OSCILLATOR |
|---|---|
| Date: | |

**AIM:**

To design and analyze a Hartley oscillator for a given frequency and test the result with the design.

**EQUIPMENTS REQUIRED:**

| S.NO | EQUIPMENTS | RANGE | QUANTITY |
|---|---|---|---|
| 1 | Transistor | BC 107 | 1 |
| 2 | Resistor | 56KΩ,2.2KΩ,12KΩ,470Ω | Each one |
| 3 | Capacitor | 22 µF, 2.2µF, 47µF,0.01 µF | Each one |
| 4 | Inductor | mH | 2 |
| 5 | CRO | (0-30 )MHz | 1 |
| 6 | RPS | (0-30) V | 1 |
| 7 | Bread board | - | 1 |
| 8 | Connecting wires | - | As required |

**PRE LAB QUESTIONS:**

1. Which type of feedback is used by Hartley oscillator?
2. Which component of Hartley oscillator is used in feedback system?
3. Which network is used to give feedback to transistor?
4. How many capacitors are there in a tank of Hartley oscillator?
5. Which configuration of the transistor amplifier is used for Hartley oscillator?

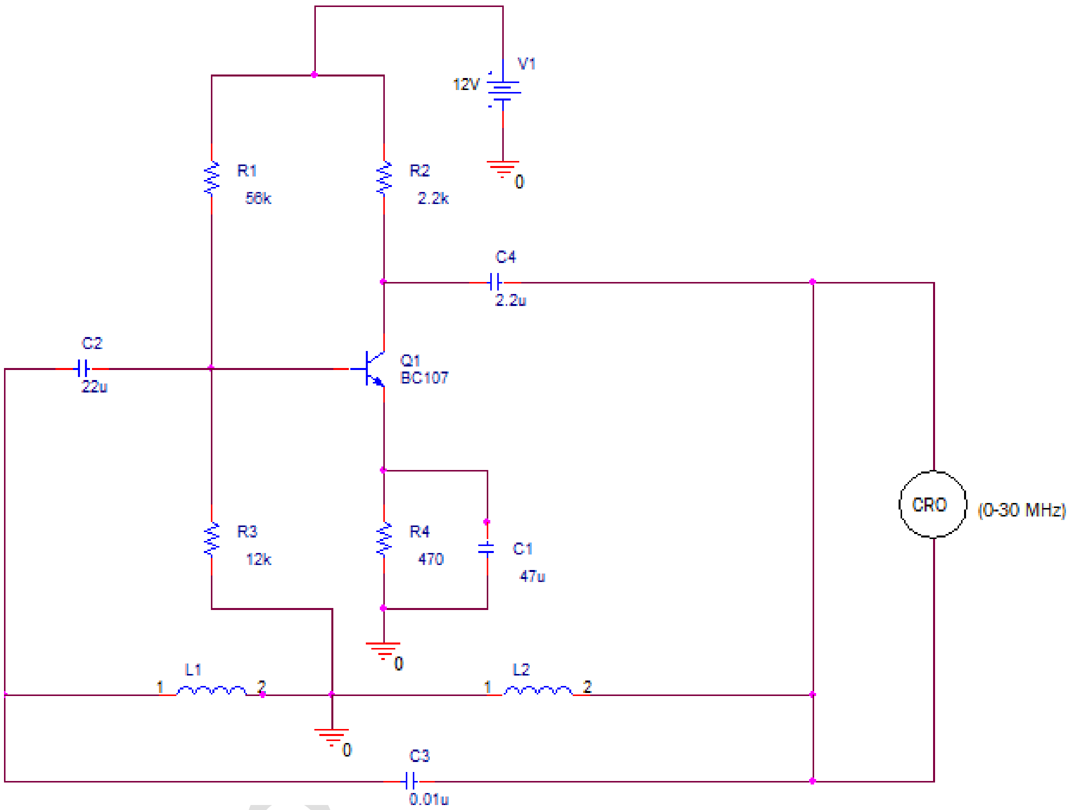
**THEORY:**

When the supply voltage +Vcc is switched on, a transient current is produced in the tank circuit and consequently damped harmonic oscillations are set up in the circuit. The oscillatory current in the tank circuit produces ac voltages across L1 and L2. As terminal 3 is earthed it will be at zero potential. If terminal 1 is at positive potential with respect to 3 at any instant, terminal 2 will be a negative potential with respect to 3 at the same instant. Thus the phase difference between the terminals1 and 2 is always 180. In the CE mode, the transistor provides the phase difference of 180 between the input and output. Therefore the total phase shift is 360. Thus at the frequency determinant for the tank circuit, the necessary condition for sustained oscillations is satisfied. If the feedback is adjusted so that the loop gain $A\beta = 1$, the circuit acts as an oscillator. In radio frequency transmitter to generate high frequency carriersignals LC oscillators are used.

**PROCEDURE:**

1. Design the Hartley oscillator as per the design procedure.
2. The connections are made as per the circuit diagram.
3. Measure the output sine wave form using CRO.
4. The value of frequency is calculated and compared with the design

**CIRCUIT DIAGRAM:**




**DESIGN PROCEDURE:**

**Design of Transistor circuit:**

Assume $V_{cc}$= 10V,$I_c$ = 4mA = $I_E$,$V_{CE}$ = $V_{CC}/2$ = 5V, $V_E$ = $V_{CC}/10$,

$R_E = V_E / I_E = 470\Omega$

$R_C = \dfrac{V_{CC} - V_{CE} - V_E}{I_C} = 2.2K\Omega$

$V_2 = V_{BE} + V_E = 1.7V$

$V_{CC} = V_1 + V_2 =$

$10.1V R_1$

$= V_1/10{*}I_B = 56K\Omega$ $R_2 =$

$V_2/9{*}I_B=12K\Omega$

$R_B=R_1 \parallel R_2=20.78K\Omega$

$F_O$=10 KHz

$X_{CE} = R_E / 10 = 47$

$C_E = 10 / 2 * F_O * R_E = 47\mu F$

**Design of Tank circuit**:

**Assume f = 30 KHz and C = 0.01µF Frequency of oscillation f = 1/ (2π√LC)Then L = 6mH;**

$L = L1 + L_2$

Let $L_1 = L_2$ $L_1 = L / 2 = 3mH$ $L_1 = L_2 = 3mH$

**Barkhausen Criterion:**

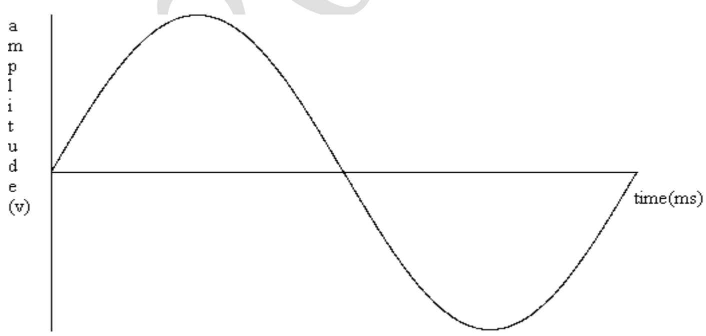
$h_{fe} = L1 / L_2$

$= 3\ mH / 3\ mH = 1$

**TABULATION:**

| S.No | Capacitor | Inductor ($L_1 = L_2$) | Theoretical frequency in KHz | Amplitude in Volts | Time period in ms | Observed frequency in KHz |
|---|---|---|---|---|---|---|
| 1 | 0.01uF | | 40K | | | |
| 2 | 0.01uF | | 10K | | | |
| 3 | 0.01uF | | 20K | | | |
| 4 | 0.01uF | | 30K | | | |

**MODEL GRAPH:**



**POST LAB QUESTIONS:**

1. How many capacitors are there in a tank of Hartley Oscillator?
2. Write the expressions for frequency of Hartley Oscillator?
3. How to tune a Hartley oscillator?
4. What are the disadvantages of Hartley Oscillators over Colpitts Oscillators?
5. What will be the oscillator frequency of Hartley oscillator if inductance L1, L2 are equal to 1mH and 2mH respectively and capacitor C is 10 nF.



**RESULT:**

Thus the Hartley oscillator is designed and analyzed as per the design.

Designedfrequency =

Obtained frequency =

| | |
|---|---|
| **Ex. No: 3(b)** | **DESIGN AND ANALYSIS OF COLPITTS OSCILLATOR** |
| **Date:** | |

## AIM:

To design and construct a Colpitts oscillator for a given frequency and test the result with the design.

## EQUIPMENTS REQUIRED:

| S.NO | EQUIPMENTS | RANGE | QUANTITY |
|---|---|---|---|
| 1 | Transistor | BC 107 | 1 |
| 2 | Resistor | 56KΩ,2.2KΩ,12KΩ,470Ω | Each one |
| 3 | Capacitor | 22 μF, 2.2μF, 47μF,0.01 μF | 2.2μF - 1,<br>22 μF-1,<br>0.01 μF -2<br>47μF – 1 |
| 4 | Inductor | mH | 1 |
| 5 | CRO | (0-30 )MHz | 1 |
| 6 | RPS | (0-30) V | 1 |
| 7 | Bread board | - | 1 |
| 8 | Connecting wires | - | As required |

## PRE LAB QUESTIONS:

1. Which type of feedback is used by Colpitts oscillator?
2. Which component of Colpitts oscillator is used in feedback system?
3. Which network is used to give feedback to transistor?
4. How many capacitors are there in a tank of Colpitts oscillator?
5. Which configuration of the transistor amplifier is used for Colpitts oscillator?

## THEORY:

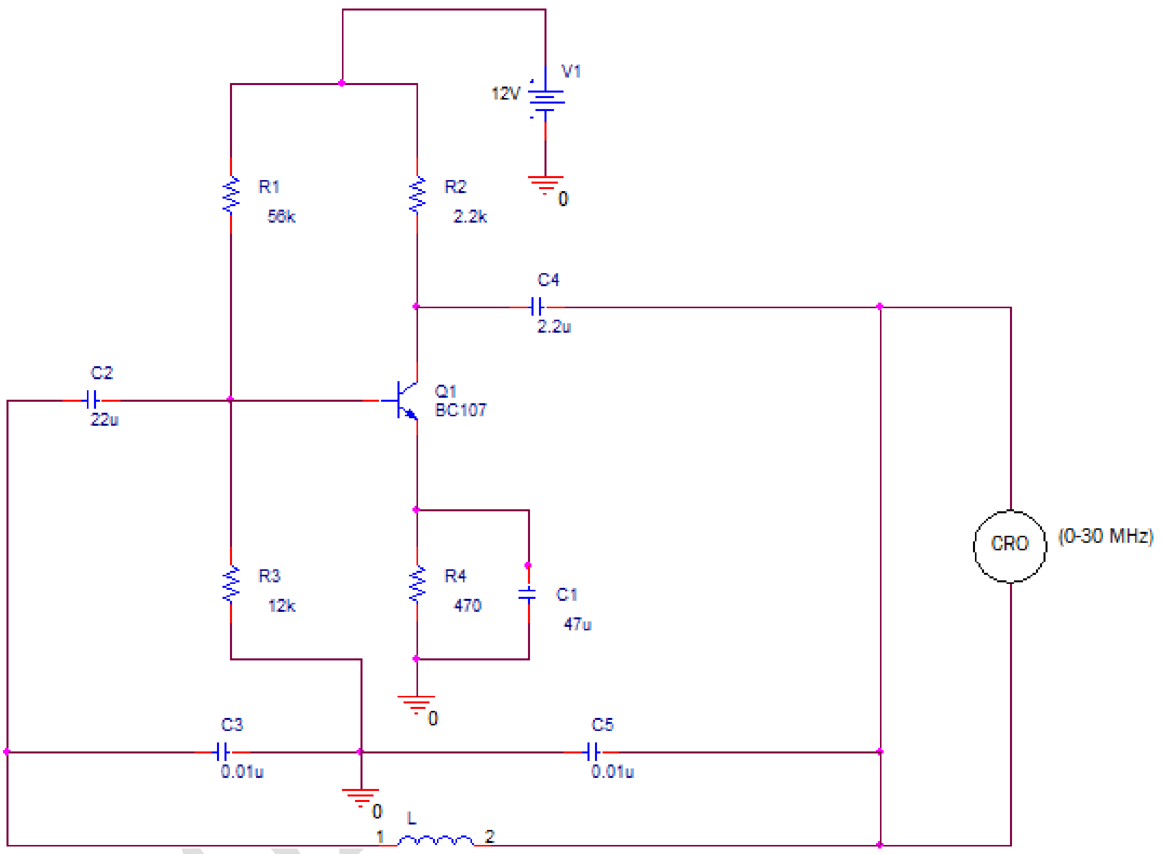
The feedback network consisting of inductors L and capacitor C1 and $C_2$ determines the frequency of the oscillator. When the supply voltage +Vcc is switched on, a transient current is produced in the tank circuit and consequently damped harmonic oscillations are set up in the circuit. The oscillatory current in the tank circuit produces ac voltages across C1 and C2. As terminal 3 is earthed it will be at zero potential. If terminal 1 is at positive potential with respect to 3 at any instant, terminal 2 will be a negative potential with respect to 3 at the same instant. Thus the phase difference between the terminals 1 and 2 is always 180. In the CE mode, the transistor provides the phase difference of 180 between the input and output. Therefore the total phase shift is 360. Thus at the frequency determinant for the tank circuit, the necessary condition for sustained oscillations is satisfied. If the feedback is adjusted so that the loop gain Aβ = 1 the circuit acts as an oscillator.

In radio frequency transmitter to generate high frequency carrier signals LC oscillators are used.

**PROCEDURE:**

1. Design the Colpitts oscillator as per the design procedure.
2. The connections are made as per the circuit diagram.
3. Measure the output sine wave form using CRO.
4. The value of frequency is calculated and compared with the design.

**CIRCUIT DIAGRAM:**




**DESIGN PROCEDURE:**

Assume f = 30 KHz and L = 5mH, C=5.6nF

Frequency of oscillation $f = 1 / (2\pi\sqrt{LC})$

And $C = C_1C_2 / (C_1 + C_2)$

Let $C_1 = 0.01\mu F$

⇨ $C2 = 0.01\mu F$

**Barkhausen Criterion:**

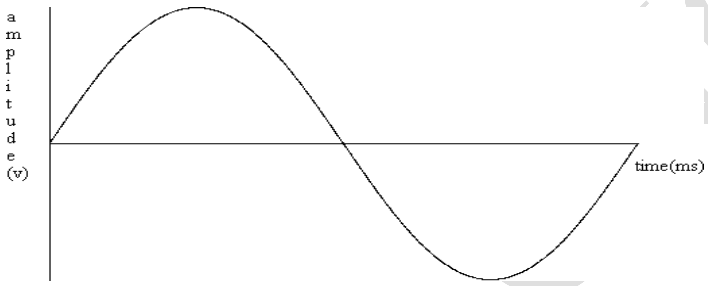
$h_{fe} = C_1 / C_2$

$= 0.01\mu F / 0.01\mu F = 1$

## TABULATION:

| S.No | Capacitor ($C_1 = C_2$) | Inductor L | Theoretical frequency in KHz | Amplitude in Volts | Time period in ms | Observed frequency in KHz |
|---|---|---|---|---|---|---|
| 1 | 0.01uF | | 40K | | | |
| 2 | 0.01uF | | 10K | | | |
| 3 | 0.01uF | | 20K | | | |
| 4 | 0.01uF | | 30K | | | |

## MODEL GRAPH:




## POST LAB QUESTIONS:

1. How many capacitors are there in a tank of colpitts oscillator?
2. In colpitts oscillator , What factors are improved by capacitor circuit configuration?
3. What is the condition for Colpitts oscillator circuit?
4. What are the adavantages of Colpitts Oscillators?
5. What type of circuit is called if the circuit with two capacitors and one inductors?

## RESULT:

Thus the Colpitts oscillator is designed and analyzed as per the design.

Designed frequency =

Obtained frequency =

| Ex. No: 4 | DESIGN AND ANALYSIS OF RC INTEGRATOR & DIFFERENTIATOR |
|---|---|
| Date: | |

**AIM:**

To design and analyze the RC integrator, differentiators circuits.

**EQUIPMENTS REQUIRED:**

| S.NO | EQUIPMENTS | RANGE | QUANTITY |
|---|---|---|---|
| 1 | Resistor | 1KΩ | 1 |
| 2 | Capacitor | 0.1uF | 1 |
| 3 | CRO | (0-30 )MHz | 1 |
| 4 | Function generator | (0-3) MHz | 1 |
| 5 | Bread board | - | 1 |
| 6 | Connecting wires | - | As required |

**PRE LAB QUESTIONS:**

1. What are integrators and differentiators?
2. What are the applications of integrators?
3. Op-amp is used mostly as an integrator than a differentiator.Why?
4. Mention some applications of differentiators.
5. Draw the frequency response curves for integrator and differentiator.

**THEORY:**

**INTEGRATOR:**

For a low pass RC circuit, if the time constant is very large as compared to the time required by the input signal to make an appreciable change, the circuit acts as an integrator. Under this case, the drop across C is negligible compared to drop across R. Thus the entire input $V_i$ (t) can be assumed to be appearing across R. Then the current i is given by, $V_r = V_i = iR$; $i = V_i / R$

**DIFFERENTIATOR:**

For a high pass RC circuit, if time constant is very small as compared to the time required by the input signal to make an appreciable change the circuit acts as a differentiator.

i = C ( dVc /
dt)i = C( dVi
/ dt)

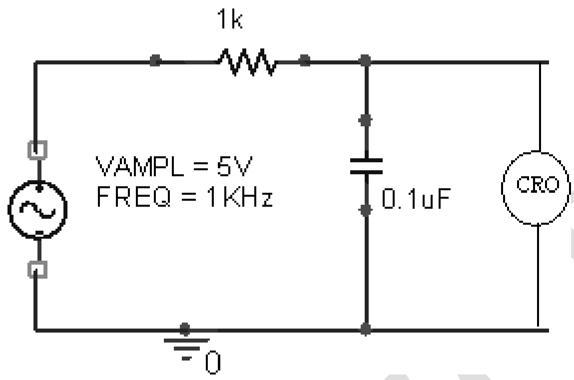
Vo = iR

Vo = RC (dVi / dt)

**PROCEDURE:**

1. The circuit connections are made as per the circuit diagram Integrator.
2. Apply the square wave as a input signal to the circuit.
3. The time period and amplitude of the output wave is noted from CRO and the wave form isdrawn in agraph.
4. Interchange the resistor and capacitor position to get Differentiator circuit, then Perform thesteps 1,2, 3.

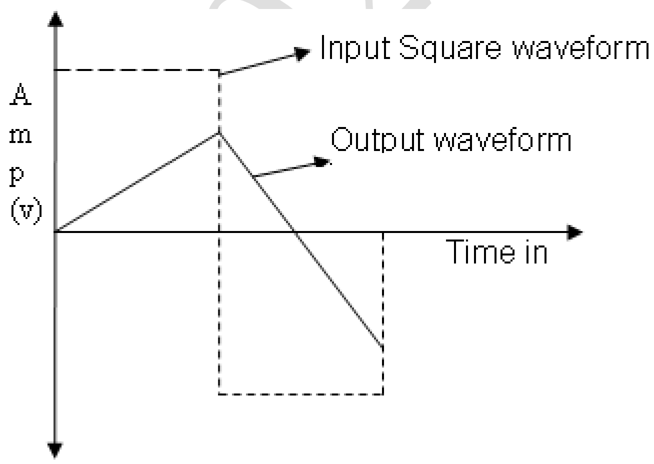
**INTEGRATOR:**



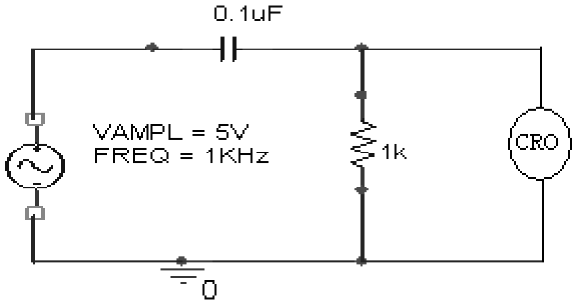

**TABULATION:**

| S.No | Amplitude in V | Time period in ms |
|---|---|---|
| | | |

**MODEL GRAPH:**



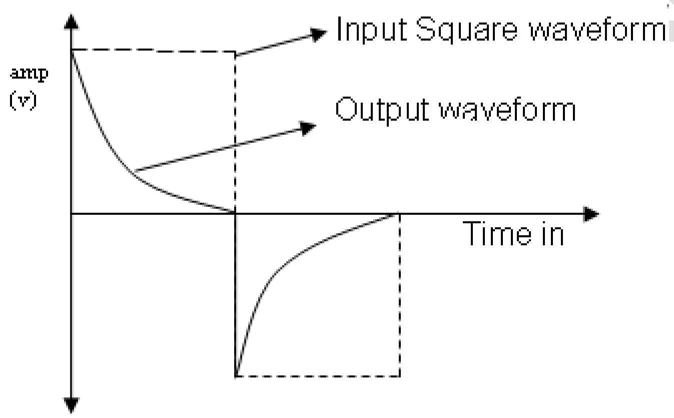
**DIFFRENTIATOR:**




**TABULATION:**

| S.No | Amplitude in V | Time period in ms |
|---|---|---|
| | | |

**MODEL GRAPH:**




**POST LAB QUESTIONS:**

1. What is the requirement of a high pass filter to act as differentiator?
2. What is time constant of an RC circuit?
3. What is the requirement of a low pass filter to act as integrator?
4. What are the modes in which op-amp is operated with finite gain and infinite gain?
5. Differences between active and passive differentiators

**RESULT:**

Thus the output waveforms of integrator and differentiator were obtained and the graphs were drawn.

| Ex. No: 5 | DESIGN AND ANALYSIS OF CLIPPERS & CLAMPERS |
|---|---|
| Date: | |

**AIM:**

To design and analyze the clippers, clampers circuits.

**EQUIPMENTS REQUIRED:**

| S.NO | EQUIPMENTS | RANGE | QUANTITY |
|---|---|---|---|
| 1 | Diode | IN4007 | 1 |
| 2 | Resistor | 1K | 1 |
| 3 | Capacitor | 0.1uF | 1 |
| 4 | CRO | (0-30 )MHz | 1 |
| 5 | RPS | (0-30) V | 1 |
| 6 | Bread board | - | 1 |
| 7 | Connecting wires | - | As required |

**PRE LAB QUESTIONS:**

1. What is clipper circuit?
2. Describe the different types of clipper circuit.
3. What is operating principle of the clamper circuit?
4. Describe the function of the Clamper circuit.
5. Explain the Positive clipper, Negative clipper, Biased clipper and Combination clipper

**THEORY:**

**CLIPPER:**

Clipper is a circuit which is used to clip off unwanted portion of the waveform, without distorting the remaining part of the waveform. When the diode is connected in series with the load, such a circuit is called a series clipper. The clipper level is determined by the reference voltage $V_{ref}$ and could be obtained by the supply voltage. When the supply voltage is positive, the circuit is said tobe positive reference clipper.

**CLAMPER:**

The clamper which is used to add a dc level as per the requirements to the ac output signal is called clampers. The capacitor, diode and resistors are the 3 basic elements of a clamper circuit. They are also called as dc inserter circuits or dc resonators. They are positive and negative clampers depending on whether positive dc or negative dc shift is introduced. A positive clamper adds a positivelevel to the ac output. During the positive half cycle of Vi, the diode is reverse biased

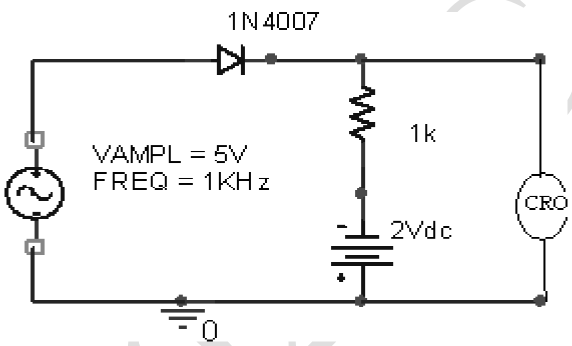
and the capacitor starts discharge. During the negative half cycle, the diode gets forward biased and the capacitor chargesto maximum level Vm. A negative clamper adds a negative level to the ac output.

**PROCEDURE:**

1. The circuit connections are made as per the circuit diagram.
2. Apply the sine wave as an input signal to the circuit.
3. The time period and amplitude of the output wave is noted from CRO and the waveform is drawn in a graph.
4. Repeat the above procedures for all the circuit diagram.
   drawn in a graph.
5. Repeat the above procedures for all the circuit diagram.

**SERIES CLIPPERS:**

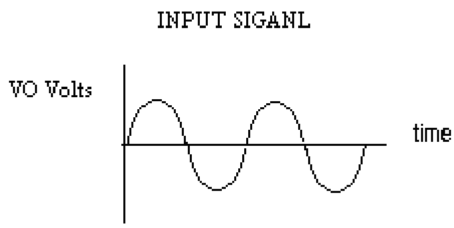
**Negative clipper with negative reference:**

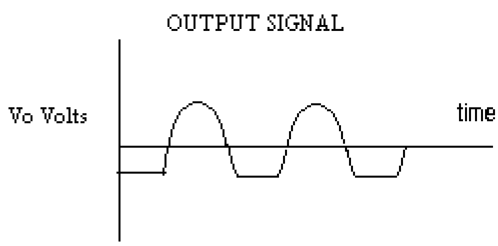



**TABULATION:**

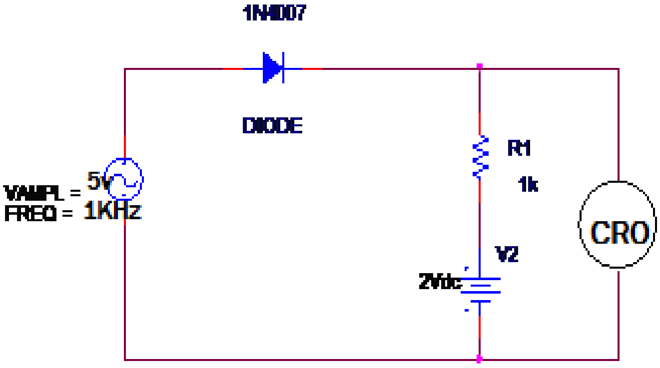
| S.No | Amplitude in V | Time period in ms |
|---|---|---|
| | | |

**MODEL GRAPH:**



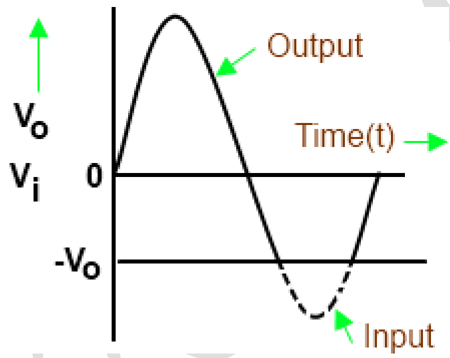
**Negative clipper with positive reference**




**TABULATION:**

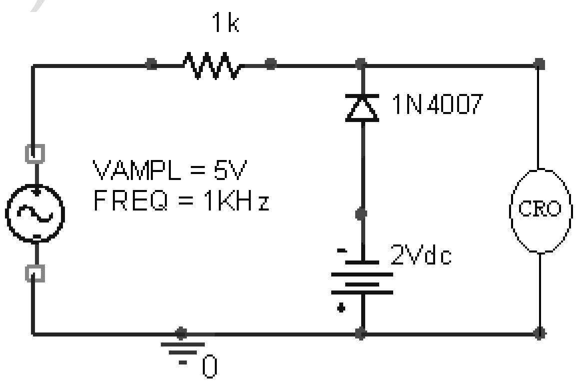
| S.No | Amplitude in V | Time period in ms |
|---|---|---|
| | | |

**MODEL GRAPH:**




**PARALLEL CLIPPERS:**

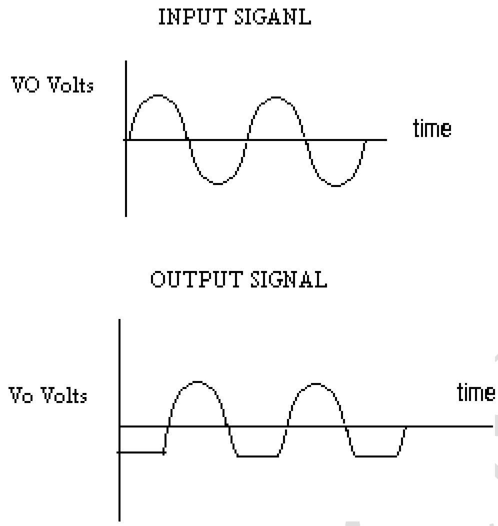
**Negative clipper with negative reference:**



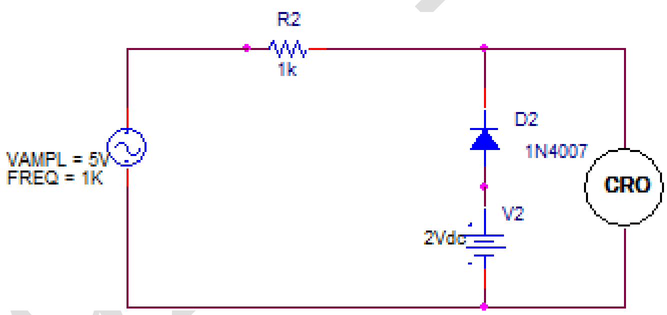
## TABULATION:

| S.No | Amplitude in V | Time period in ms |
|---|---|---|
| | | |

## MODEL GRAPH:



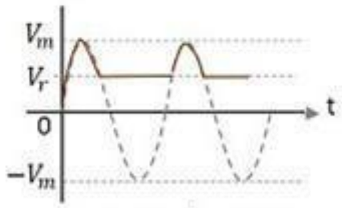

## Negative clipper with positive reference:




## TABULATION:

| S.No | Amplitude in V | Time period in ms |
|---|---|---|
| | | |

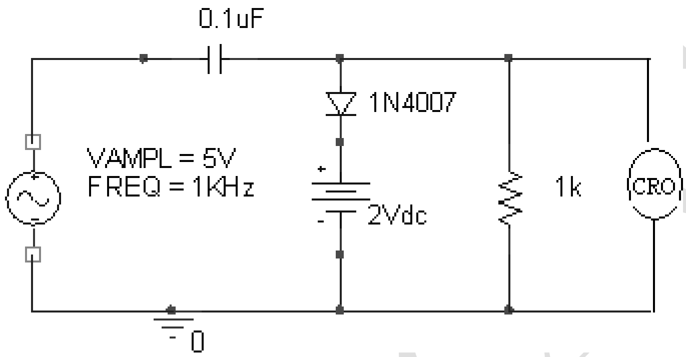
**MODEL GRAPH**




Practical Output Waveform

**CLAMPER:**

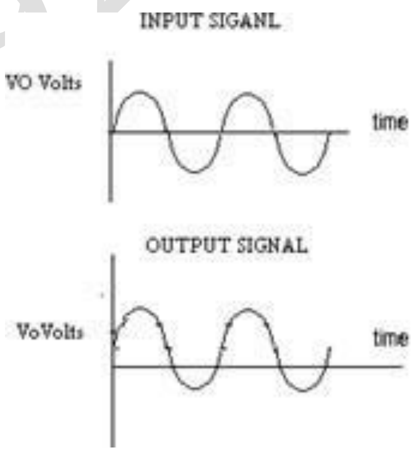
**Positive clamper:**



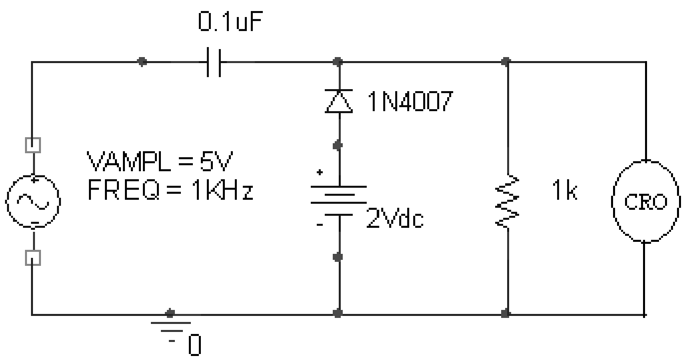

**TABULATION:**

| S.No | Amplitude in V | Time period in ms |
|---|---|---|
| | | |

**MODEL GRAPH:**



**Negative Clamper:**

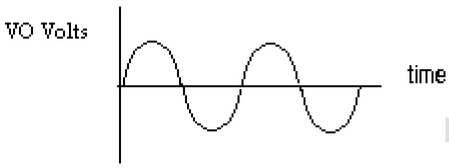



**TABULATION:**

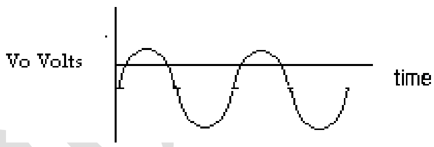
| S.No | Amplitude in V | Time period in ms |
|---|---|---|
| | | |

**MODEL GRAPH:**

INPUT SIGANL




OUTPUT SIGNAL




**POST LAB QUESTIONS:**

1. Which circuit parameter the clipper circuit consists of?
2. Whether the original signal change at the output of the clamper circuit?
3. What is meaning of the Positive clamper and Negative clamper?
4. What is the application of clipper circuit?
5. What is the another name for clipper and clamper?

**RESULT:**

Thus the output waveforms of clipper and clamper were obtained and the graphs were drawn.

| Ex. No: 6<br>Date: | **INSTRUMENTATION AMPLIFIER** |
|---|---|

**AIM:**

To design an instrumentation amplifier and obtain the output for various gain.

**APPARATUS REQUIRED:**

| S.NO. | COMPONENTS NAME | SPECIFICATION & RANGE | QUANTITY |
|---|---|---|---|
| 1. | Bread board | - | 1 |
| 2. | Op amp | IC741 | 3 |
| 3. | Function generator | 3 MHz | 2 |
| 4. | CRO | 30 MHz | 1 |
| 5. | Dual power supply | +12/-12V | 1 |
| 6. | Resistors | | Each 1 |
| 7. | Connecting wires | Single strand | As required |

**PRE LAB QUESTIONS:**

1. What is an Instrumentation amplifier?
2. what are the adavantages of instrumentation amplifier?
3. What are the Applications of Instrumentation Amplifier?
4. Difference between Operational Amplifier and Instrumentation Amplifier
5. What are the features of instrumentation amplifier?

**THEORY:**

Instrumentation amplifier is a kind of differential amplifier with additional input buffer stages. The addition of input buffer stages makes it easy to match (impedance matching) the amplifier with the preceding stage. Instrumentation are commonly used in industrial test and measurement application.The instrumentation amplifier also has some useful features like low offset voltage, high CMRR (Common mode rejection ratio), high input resistance, high gain etc.

The two non-inverting amplifiers form a differential input stage acting as buffer amplifiers with a gain of 1 + 2R2/R1 for differential input signals and unity gain for common mode input signals.

Since amplifiers A1 and A2 are closed loop negative feedback amplifiers, we can expect the voltage at Va tobe equal to the input voltage V1. Likewise, the voltage at Vb to be equal to the value at V2.

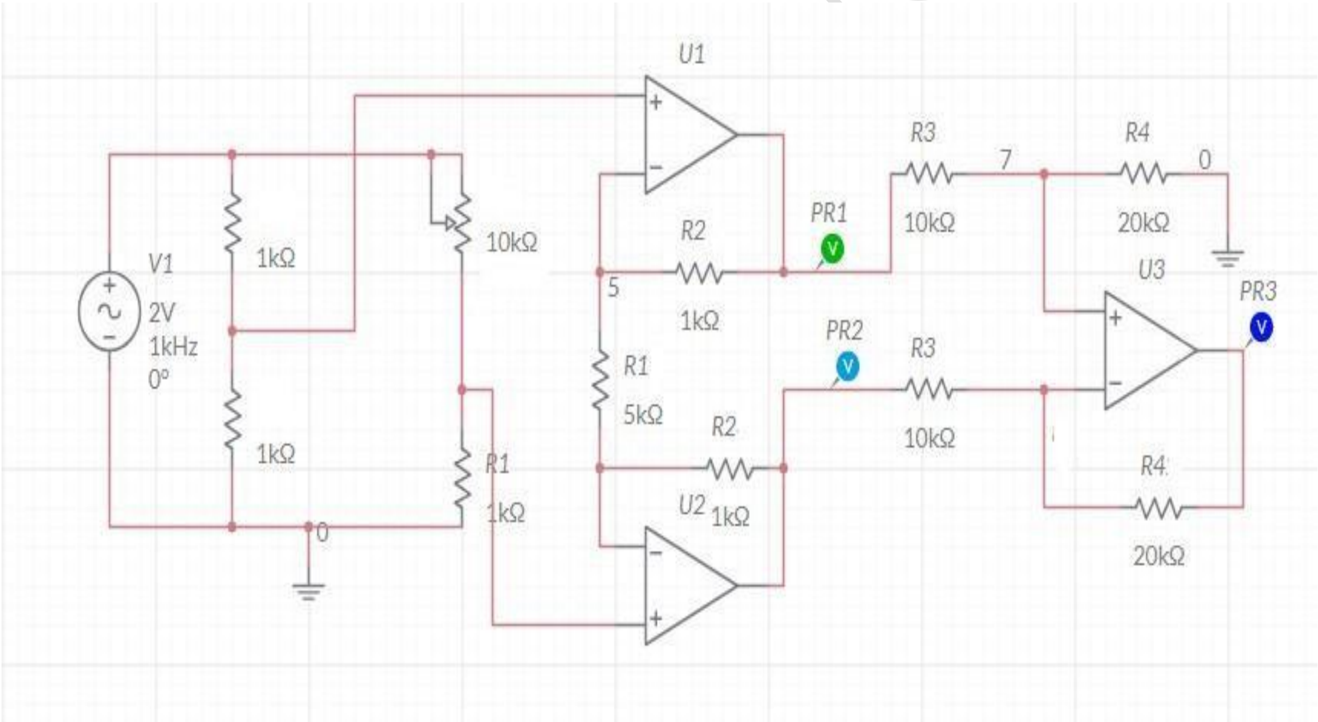
As the op-amps take no current at their input terminals (virtual earth), the same current must flow through the three resistor network of R2, R1 and R2 connected across the op-amp outputs. This means then that the voltage on the upper end of R1 will be equal to V1and the voltage at the lowerend of R1 to be equal to V2.

The voltage output from the differential op-amp A3 acting as a subtractor, is simply the difference between its two inputs ( V2 – V1 ) and which is amplified by the gain of A3which may be one, unity,(assuming that R3 = R4). Then we have a general expression for overall voltage gain of the instrumentation amplifier circuit as:

$$V_{out} = (V_2 - V_1)\left[1 + \frac{2R_2}{R_1}\right]\frac{R_4}{R_3}$$



**CIRCUIT DIAGRAM:**



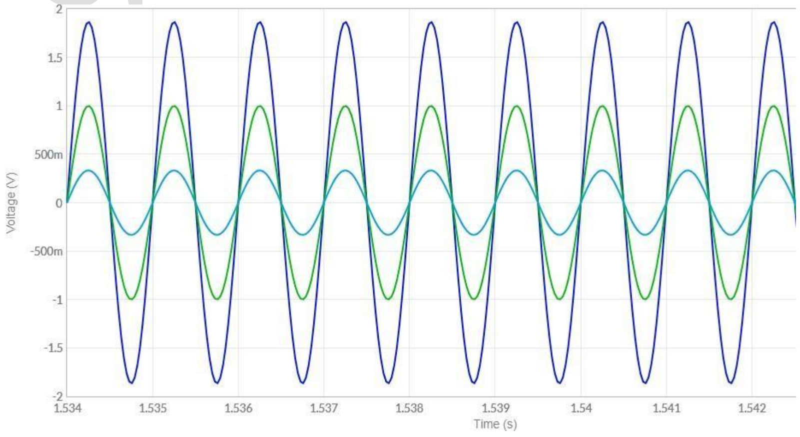
## PROCEDURE:

1. Connections are given as per the circuit diagram.
2. Input signal is connected to the circuit from the signal generator.
3. The input and output signals of the circuit observed from the dual channels 1 and 2 of the CRO.
4. Suitable voltage sensitivity and time-base on CRO is selected.
5. Change the gain setting resistor value and observe the output.

## Tabulation:

| Input /Output | Theoretical | | | Practical | | |
|---|---|---|---|---|---|---|
| | Amplitude | Time | | | | |
| Input | | | | | | |
| Output | V1 | V2 | Vo | V1 | V2 | Vo |
| | | | | | | |
| | | | | | | |
| | | | | | | |
| | | | | | | |

## Output:



**RESULT:**

Thus the Instrumentation amplifier was designed and verified.

## Exp.No: 7 Active Low Pass, High Pass and Band Pass Filters

Date:

**Aim:**

To design Low pass, High pass and Band pass active filters using Op-amp and obtain frequency response.

**Apparatus Required:**

| S.NO. | COMPONENTS NAME | SPECIFICATION & RANGE | QUANTITY |
|---|---|---|---|
| 1. | Bread board | - | 1 |
| 2. | Op-amp | IC741 | 1 |
| 3. | CRO | 30 MHz | 1 |
| 4. | Dual Power supply | +12/-12V | 1 |
| 5. | Capacitor | | Each 1 |
| 6. | Resistors | | Each 1 |
| 7. | Function generator | 3 MHz | 1 |
| 8. | Connecting wires | Single strand | As required |



## Theory:

A filter is often used in electronic circuits to block (or allow) a select frequency to thecircuit. An op-amp is used to design a filters, so it is called Active filters. There are Four types active filters likeLow pass, High pass, band pass and band stop . A low pass filter is used in circuits that only allow low frequencies to pass through (below the Cutoff frequency). It is often used to block high frequencies and AC current in a circuit.A high pass filter is used in circuits that only require high frequencies to operate (above the cut off frequency). It blocks most low frequencies & DC component.A band pass filter is a combination of a high pass anda low pass filter. It allows only a select range of frequencies to pass through. It is designedsuch a way that the cut off frequency of the low pass filter is higher than the cut off frequency of the high pass filter, hence allowing only a select range of the frequenciesto pass through.

## Procedure:

1. Connections are given as per the circuit diagram.
2. Input signal is connected to the circuit from the signal generator.
3. The input and output signals of the filter channels 1 and 2 of the CRO are connected.
4. Suitable voltage sensitivity and time-base on CRO is selected.
5. The correct polarity is checked.
6. The above steps are repeated for second order filter.

**Design Low Pass Filter:**

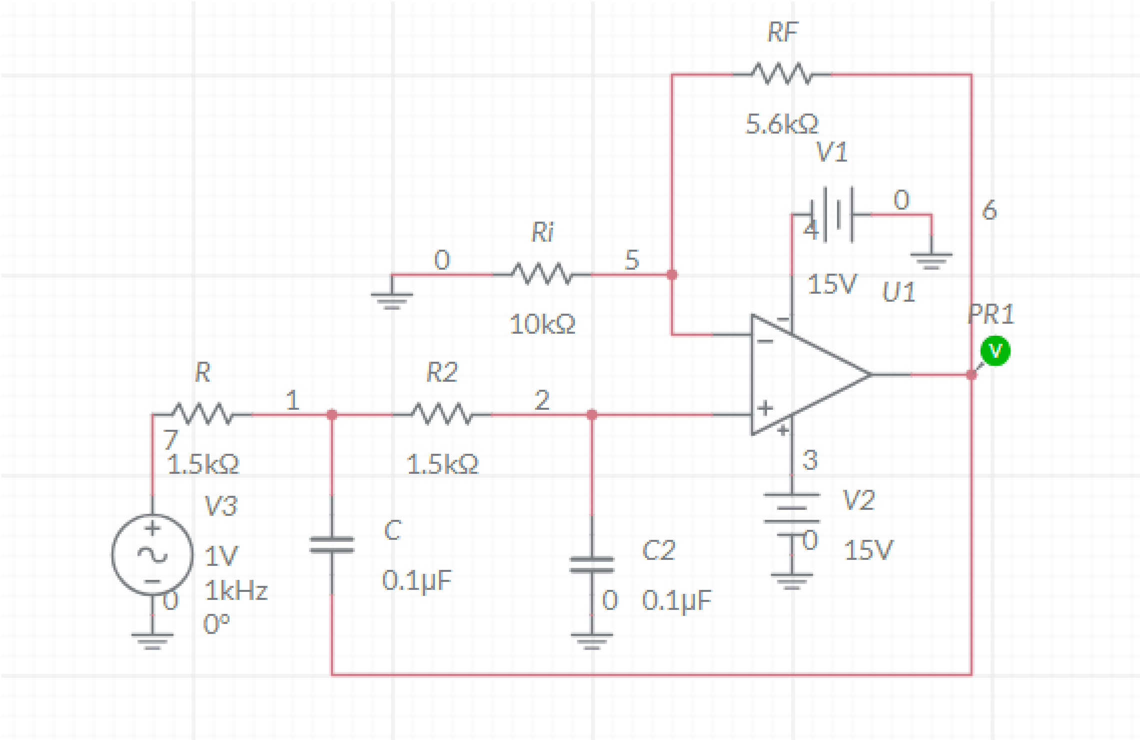
**Design Second order Butterworth Low Pass filter having upper cut off frequency 1KHz anddetermineits frequency response.**

The following steps are used for the design of active LPF.

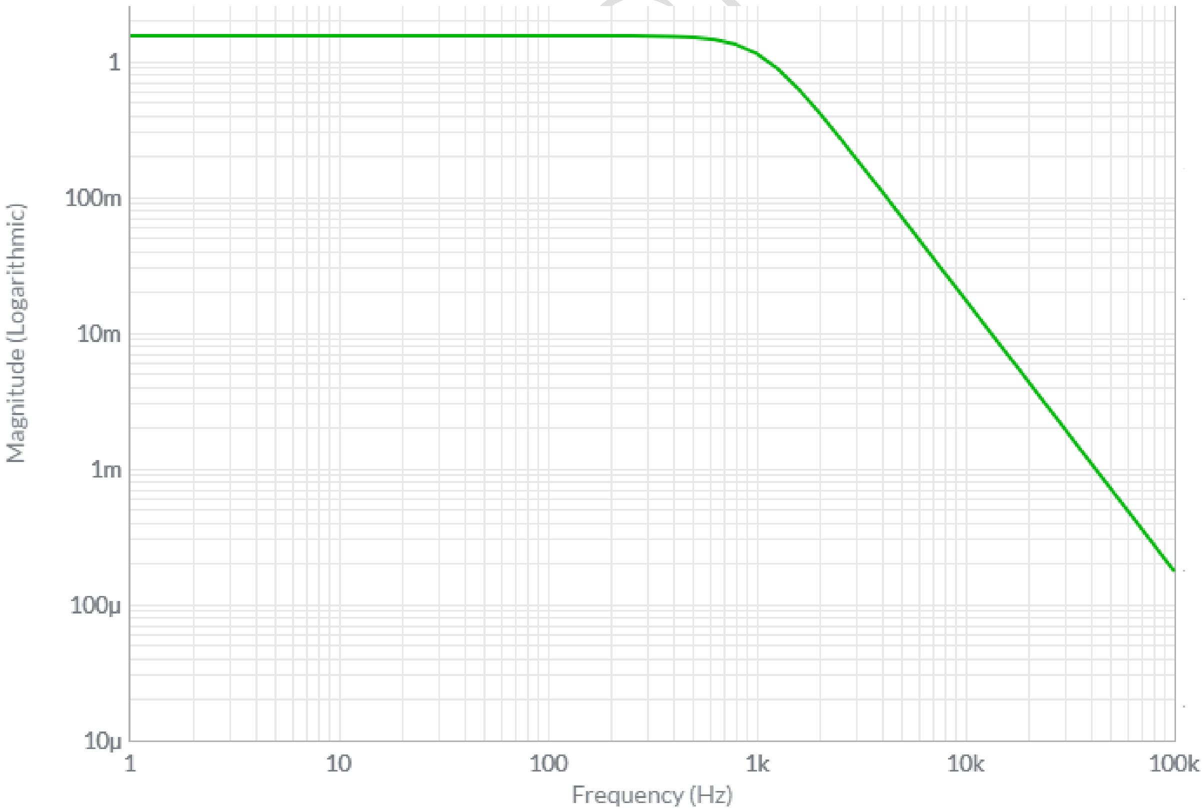
1. The value of high cut off frequency $f_H$ is chosen.
2. The value of capacitor C is selected such that its value is $\leq 1\mu F$.
3. By knowing the values of $f_H$and C, the value of R can be calculated using $f_H = \frac{1}{2\pi RC}$
4. Finally the values of $R_1$ and $R_f$ are selected depending on the designed pass band gain by using $A = 1 + \left(\frac{R_f}{R_1}\right)$

**Circuit Diagram:-**

**Low Pass filter Circuit:**




**Output:**



**Low Pass Filter Tabulation:**

| S.No | Frequency (Hz) | O/p voltage(v) | Gain=Vo/Vin | Gain=20log(Vo/Vin) |
|---|---|---|---|---|
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |

**Second order High Pass Filter:**

The high pass filter is the complement of the low pass filter. Thus the high pass filter can be obtained by interchanging R and C in the circuit of low pass configuration. A high pass filter allows only frequencies above a certain bread point to pass through and at terminates the low frequency components. The range of frequencies beyond its lower cut off frequency $f_L$ is called stop band.

**Design:-**

$$f_L = 1\mathbf{K}HZ, C = 0.1\mu F, Gain,\ Av = 2$$

$$f_L = \frac{1}{2\pi\sqrt{R_2R_3C_2C_3}}$$

$$Let\ R_2 = R_3 = R$$
$$C_2 = C_3 = C$$

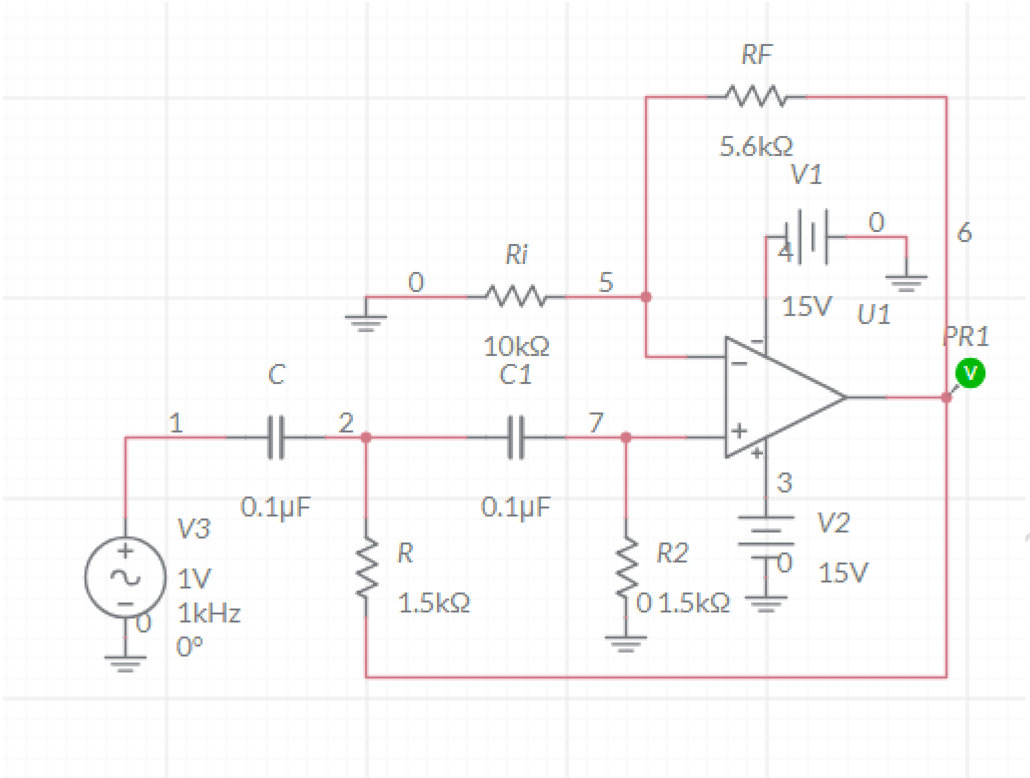
$$R_2 = R_3 = \frac{1}{2\pi f LC}$$

$$R_2 = R_3 = 1.5k\Omega$$

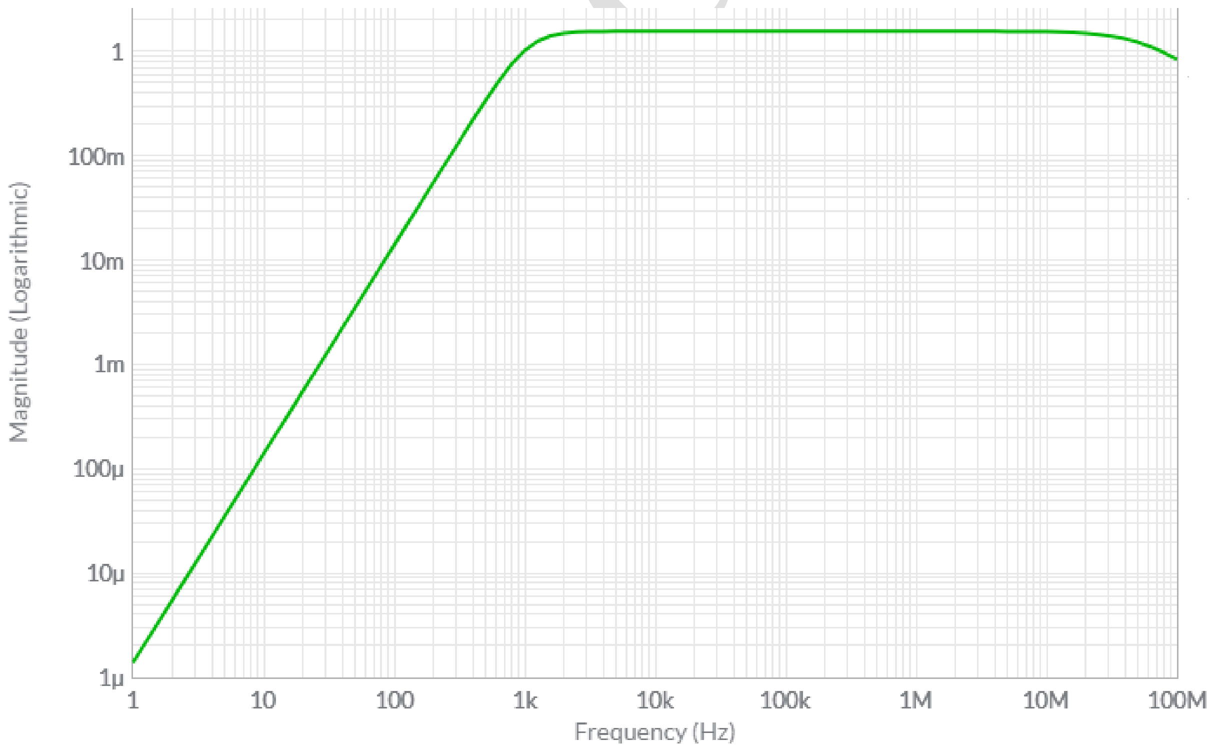
$$A = 1 + \frac{R_f}{R_1} = 2$$

$$\Rightarrow R_f = R_1 = 10k\Omega (given)$$

**Second order High Pass Filter Circuit:**




**Output:**



**High Pass Filter Tabulation:**

| S.No | Frequency (Hz) | O/p voltage(v) | Gain=Vo/Vin | Gain=20log(Vo/Vin) |
|---|---|---|---|---|
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |

**BPF:-**

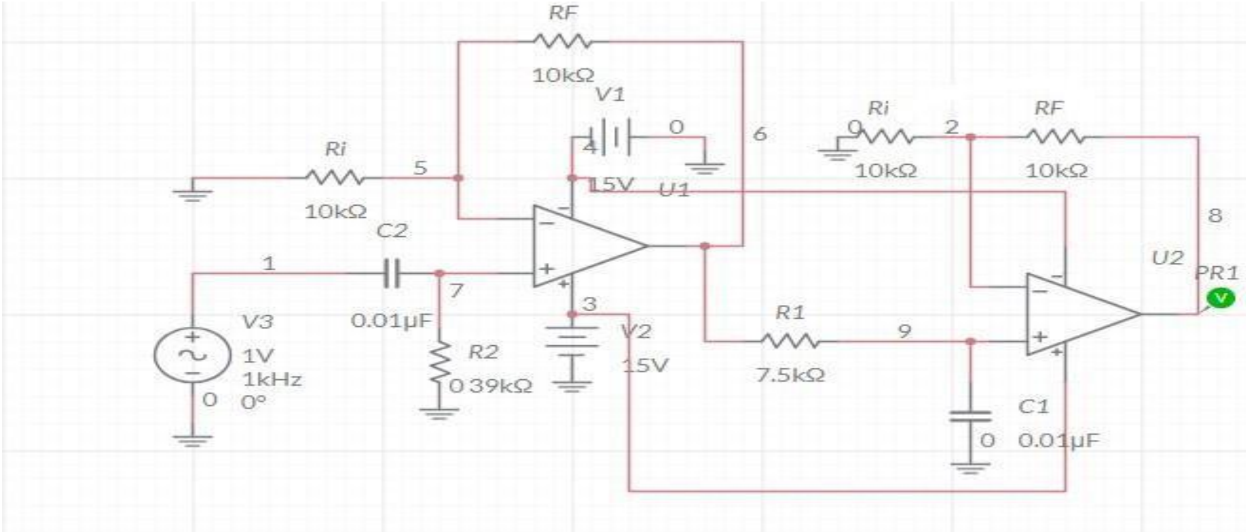
The BPF is the combination of high and low pass filters and this allows a specified range of frequencies to pass through. It has two stop bands in range of frequencies between 0 to $f_L$ and beyond $f_H$. The band b/w $f_L$ and $f_H$is called pass band. Hence its bandwidth is $(f_L-f_H)$. This filter has a maximum gain at the resonant frequency ($f_r$) which is defined as

$$f_r = \sqrt{f_H f_L}$$

The figure of merit (or) quality factor Q is given by

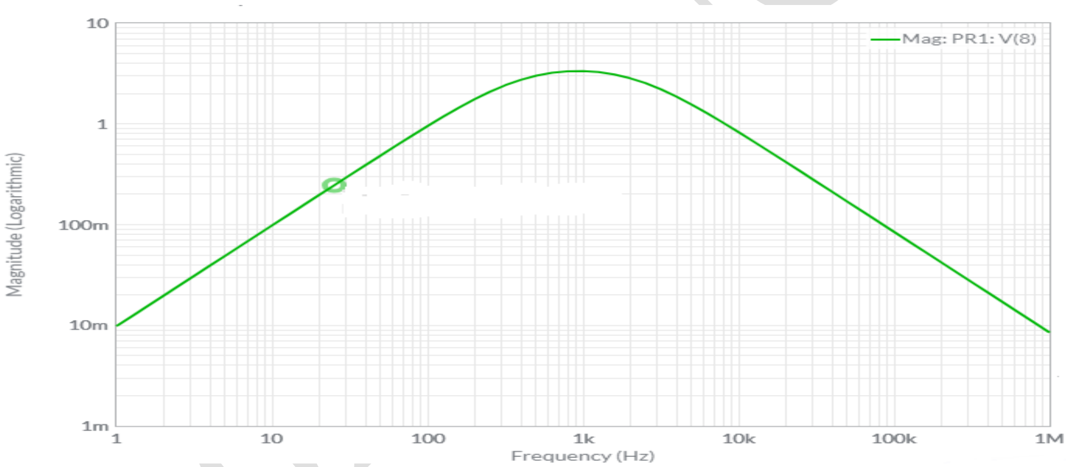
$$Q = \frac{f_r}{f_H - f_L} = \frac{f_r}{BW}$$

**Band Pass Filter Circuit:**




**HPF** **LPF**

**Output:**




**Result:**

The circuit was designed and their frequency responses were plotted.

<u>Low pass filter</u>:

The upper cut off frequency (Designed) = KHz.

The upper cut off frequency (Obtained) = KHz.

<u>High pass filter</u>:

The lower cut off frequency (Designed) = KHz.

The lower cut off frequency (Obtained) = KHz.

<u>Band pass filter:</u>

Centre Frequency = KHz
.

Quality Factor =

**Ex.No: 8 PLL characteristics and its use as Frequency Multiplier, Clock synchronization.Date:**

**Aim:**

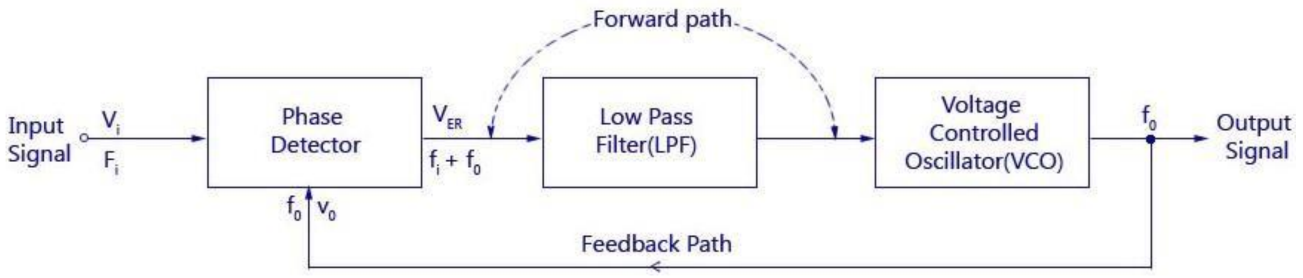
1. **To study the PLL characteristics.**
2. **To use PLL as frequency multiplier and clock synchronization.**

**Apparatus Required:**

| S.NO. | EQUIPMENTS& COMPONENTS | SPECIFICATION | QUANTITY |
|---|---|---|---|
| 1. | Bread board | - | 1 |
| 2. | IC | IC565, IC7490 | 1 |
| 3. | Resistors | 20K,2K,10K,4.7K | Each 1 |
| | Capacitor | 0.001µF,10µF,0.01µF | Each 1 |
| 4. | Function generator | 3 MHz | 1 |
| 5. | CRO | 30 MHz | 1 |
| 6. | Dual power supply | +6/-6V | 1 |
| 7. | Connecting wires | Single strand | 10 wires |

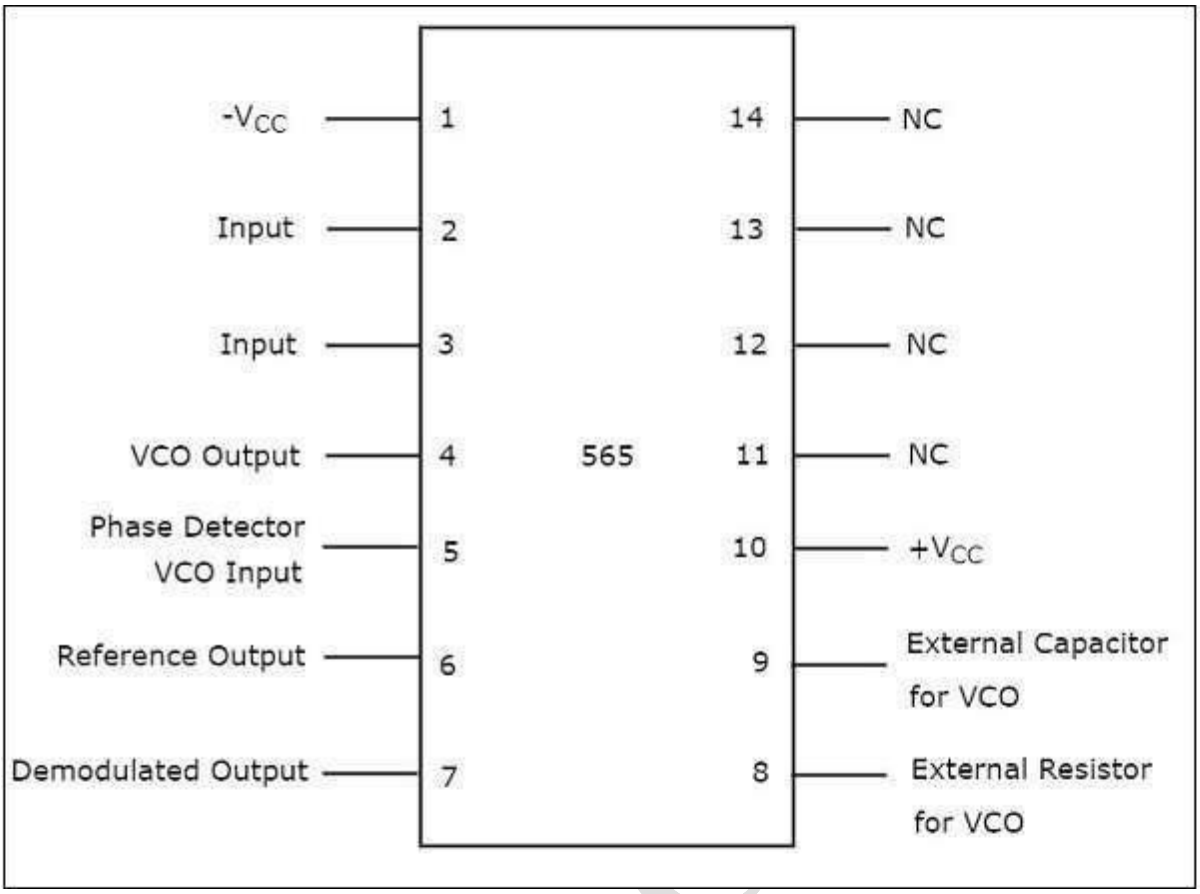
**Theory:**

The PLL IC 565 is usable over the frequency range 0.1 Hz to 500 kHz. It has highly stable centre frequency and is able to achieve a very linear FM detection. The output of VCO is capable of producing TTL compatible square wave. The dual supply is in the range of ±6V to ±12V. The IC can also be operated from single supply in the range 12V to 24V.




The phase locked loop consists of a phase detector, a voltage control oscillator and, in between them,a low pass filter is fixed. The input signal „Vi‟ with an input frequency „Fi‟ is conceded by a phase detector. Basically the phase detector is a comparator which compares the input frequency fi through the feedback frequency fo. The output of the phase detector is (fi+fo) which is a DC voltage. The outof the phase detector, i.e., DC voltage is input to the low pass filter (LPF); it removes the high frequency noise and produces a steady DC level, i.e., Fi-Fo. The Vf is also a dynamic characteristic of the PLL.

The following figure shows the pin-out and the internal block schematic of PLL IC LM 565.




**Design:**

Center frequency or VCO frequency

$$f_{out} = \frac{1.2}{4R_1C_1} \text{ Hz}$$

$$Lock\ in\ range = f_L = \pm 8\frac{f_{out}}{V} \quad ; \quad V = V^+ - (V^-)$$

$$Ca = f \quad = \pm\left[\frac{f_L}{2\pi \times 3.6 \times 10^3 \times}\right]^{\frac{1}{2}}$$

**PLL Characteristics Circuit:**

**PLL characteristics:**

**PLL used as Frequency Multiplier:**

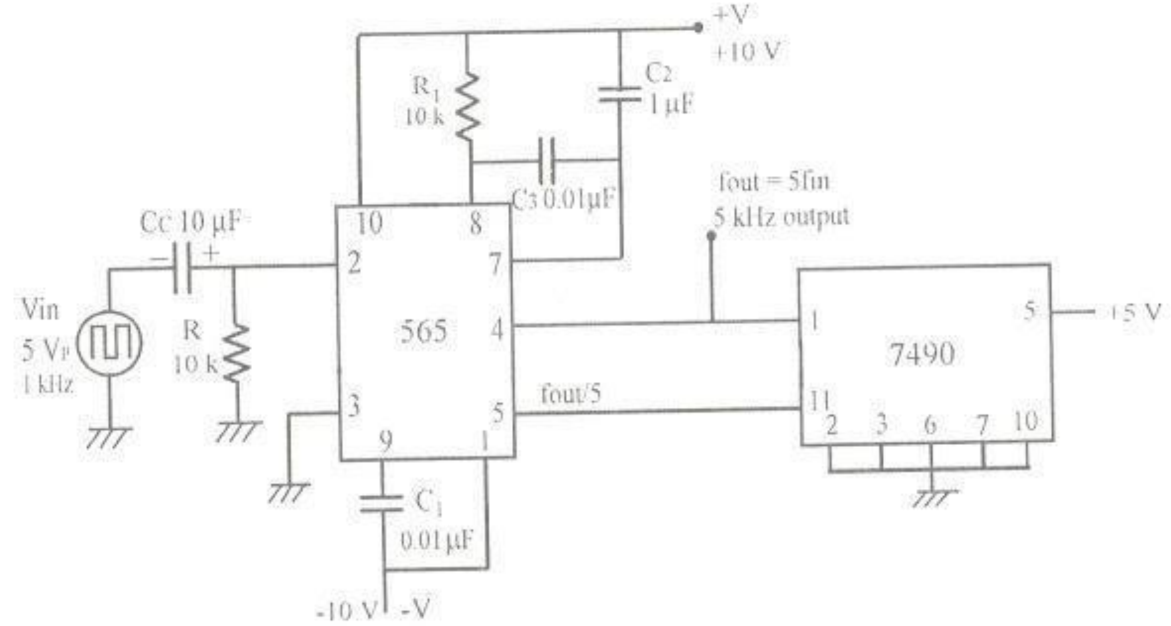
## Design:

Let V+ =10 V and V- = -10

Let the input frequency be 1 Khz, and the output frequency 5 Khz,VCO should run at 5 Khz frequency , fo=(1.2/4*R1*C1)=5Khz

Take C1=0.01μF Then R1=6K
Take C2=10μF and C3=0.001μF

use Cc=10μF and R=10k for ac coupling of input signal

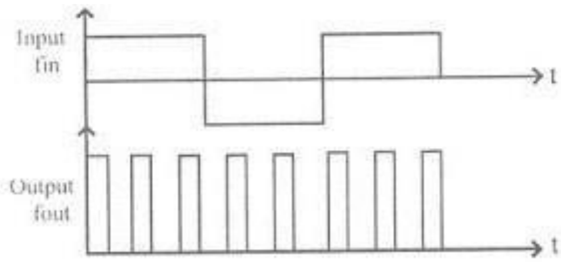


+V
+10 V
R1
10 k
C2
1 µF
C3 0.01µF
fout = 5fin
5 kHz output
Cc 10 µF
Vin
5 Vp
1 kHz
R
10 k
565
7490
fout/5
+5 V
C1
0.01µF
-10 V
-V

**Procedure:**

- Check the component using multi meter
- setup the circuit stage by stage on the breadboard
- verify the working of circuit separately.
- complete the circuit and apply 5V p-p ,1Khz square wave
- observe the multiplied frequency output on the CRO
- plot the output wave form on the graph sheet

**Output waveform:**






**Result:**

Thus, the PLL circuit was assembled successfully.

**Ex.No: 9** **R-2R Ladder Type D- A Converter using Op-amp.**

Date:

**Aim:**

**To design R-2R Ladder Type D- A Converter using Op-amp and observe the output.**

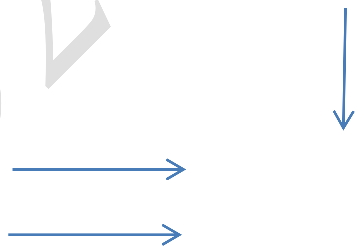
**Apparatus required:**

| S.NO. | EQUIPMENTS& COMPONENTS | SPECIFICATION | QUANTITY |
|---|---|---|---|
| 1. | Bread board | - | 1 |
| 2. | Op-amp | IC741 | 1 |
| 3. | Resistors | 2.5K | 5 |
| 4. | Function generator | 3 MHz | 1 |
| 5. | CRO | 30 MHz | 1 |
| 6. | Dual power supply | +6/-6V | 1 |
| 7. | Connecting wires | Single strand | 10 wires |

**Theory:**

A digital-to-analog converter (DAC, D/A, D2A or D-to-A) is a circuit that converts digital data(usually binary) into an analog signal (current or voltage). One important specification of a DAC is its resolution. It can be defined by the numbers of bits or its step size. Wide range of resistors used Weighted Resistor type DAC. This can be avoided by using R-2R ladder type DAC where only two values of resistors are required.

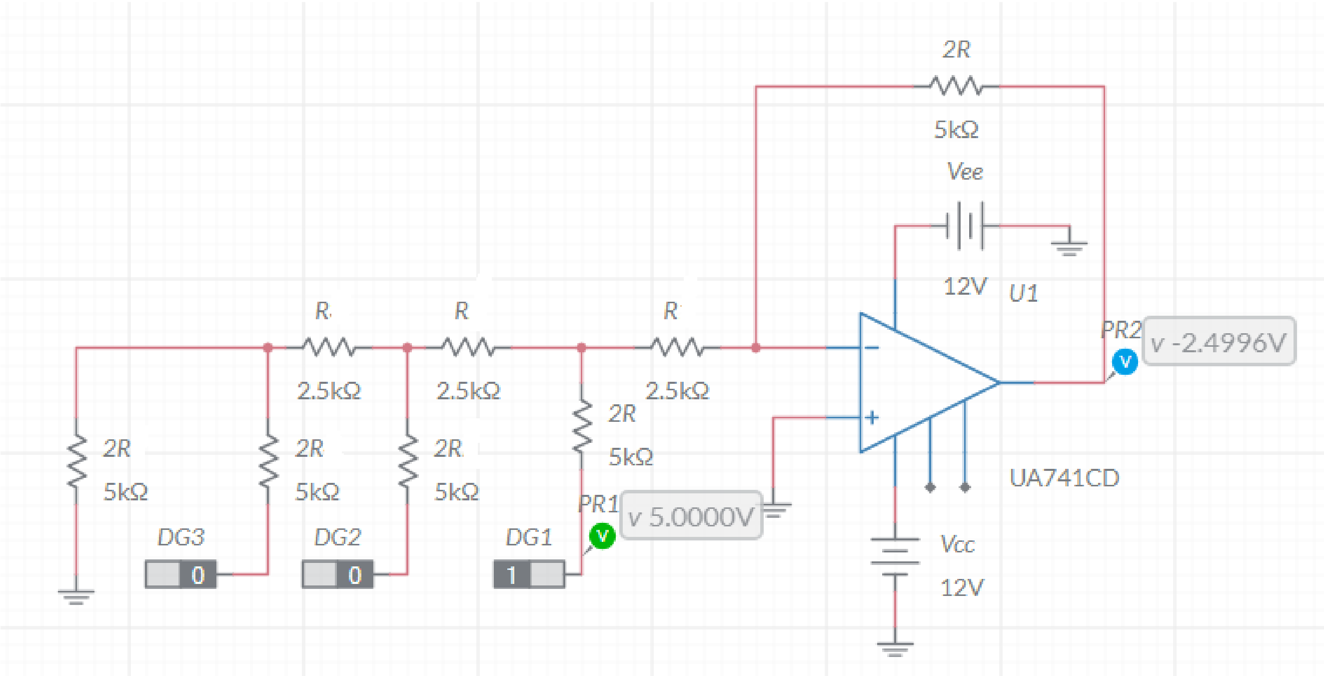
Basic Block diagram of DAC.



The output voltage of DAC;

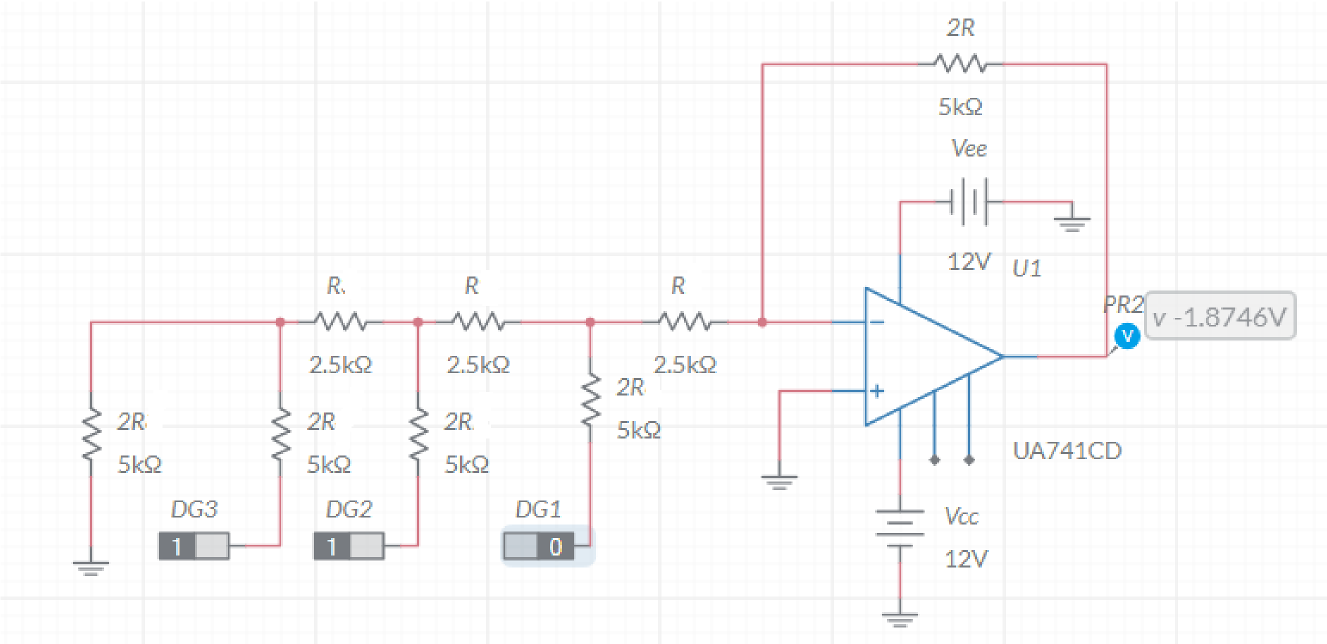
$$V_0 = (d_1 2^{-1} + d_2 2^{-2} + \cdots + d_n 2^{-n})$$

**To design 3 bit R-2R Digital to Analog converter to convert analog voltage ofbinary bit 100.**




**Theoretical Calculation:**

**If binary bit 011 :**




**Tabulation:**

**Vref = -**
**5V**

| d1 | d2 | d3 | Theoretical V0 | Practical V0 |
|---|---|---|---|---|
| 0 | 0 | 0 | | |
| 0 | 0 | 1 | | |
| 0 | 1 | 1 | | |
| 0 | 1 | 1 | | |
| 1 | 0 | 0 | | |
| 1 | 0 | 1 | | |
| 1 | 1 | 0 | | |
| 1 | 1 | 1 | | |

## Procedure:

- Check the component using multi meter.
- Setup the circuit stage by stage on the breadboard
- Verify the working of circuit separately.
- Complete the circuit and apply -5V ref if bit=1.
- Observe the output using multimeter.
- Plot the output wave form on the graph sheet.



## Result:

Thus the R-2R Ladder type using op-amp were designed and its output waveform were plotted

| Ex. No: 1 | TUNED COLLECTOR OSCILLATOR |
| --- | --- |
| Date: | |

**AIM**

To construct and simulate the Tuned collector oscillator by using pspice.

**SOFTWARE USED**

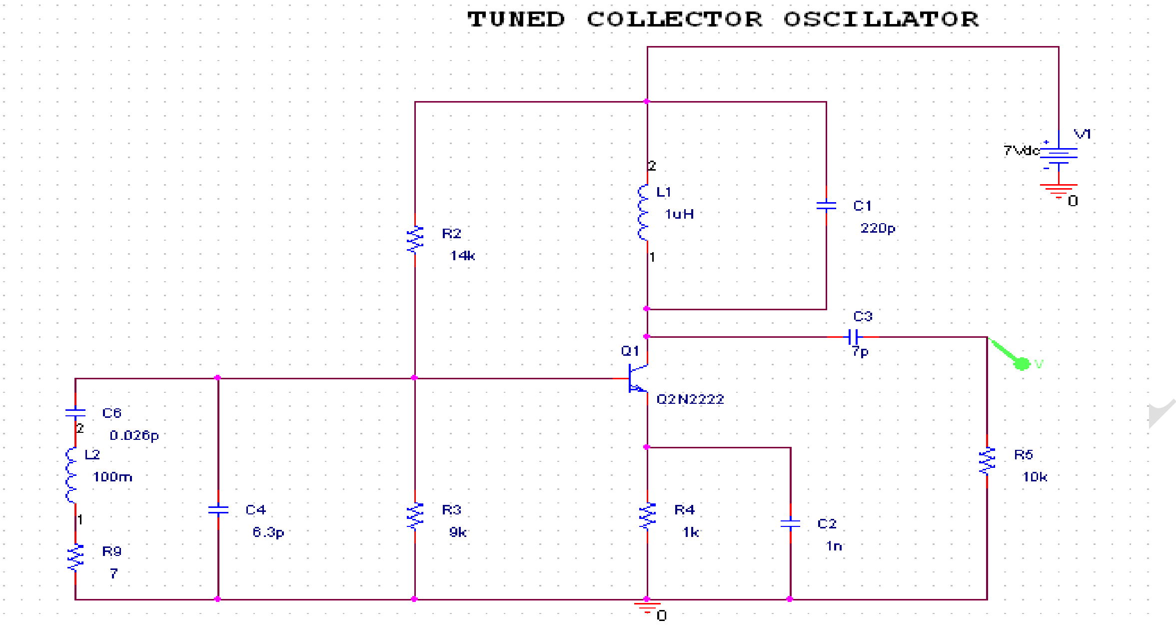
ORCAD 16.0

**PROCEDURE**

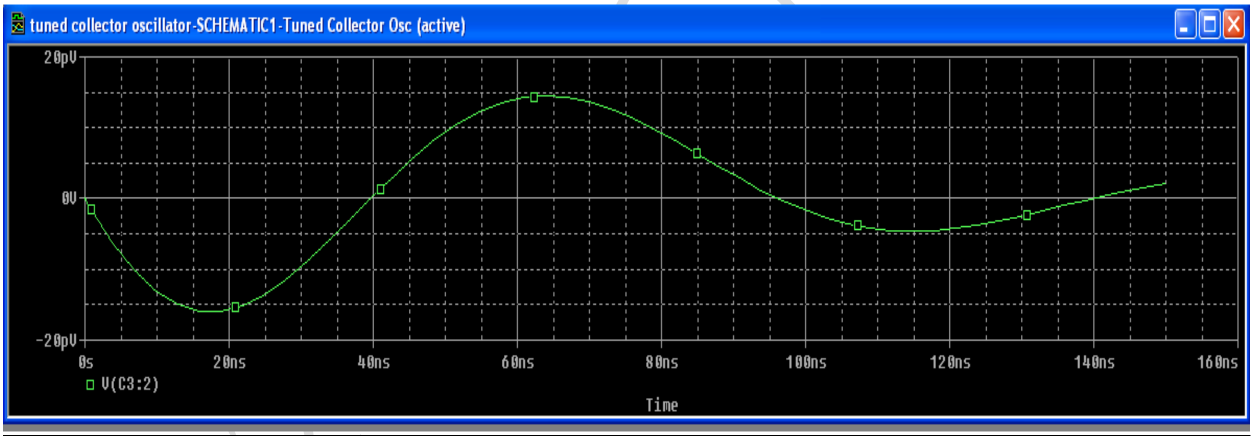
1. ORCAD - capture
2. Maximize the session log.
3. In file, open a new project and give the name to the project and choose analog or mixed A/B and specify the location say D:/ECE II
4. Create a blank project.
5. Click inside the window once and the tool bar appears.
6. Choose place part - add library - select a component in the library and select all the components - click open to add all the components.
7. Choose the components from the library and place in the worksheet.
8. Give connection using wire and properly ground the circuit.
9. Save the project and simulate it.
10. Place the marker at the points wherever the waveforms are to be viewed.
11. Click the option RUN and the output will be displayed.
12. If there are errors, correct and simulate it.
13. To view the waveform separately in plot – add plot to window – cut and paste the required waveforms.
14. The required input and output waveforms are taken printout.

## CIRCUIT DAIGRAM:

## TUNED COLLECTOR OSCILLATOR




## OUTPUT WAVEFORM:




## RESULT:

| Ex. No: 2 | TWIN - T OSCILLATOR/ WEIN BRIDGE OSCILLATOR |
|---|---|
| Date: | |

**AIM**

To construct and stimulate a Twin - T oscillator and Wein bridge oscillator using pspice.

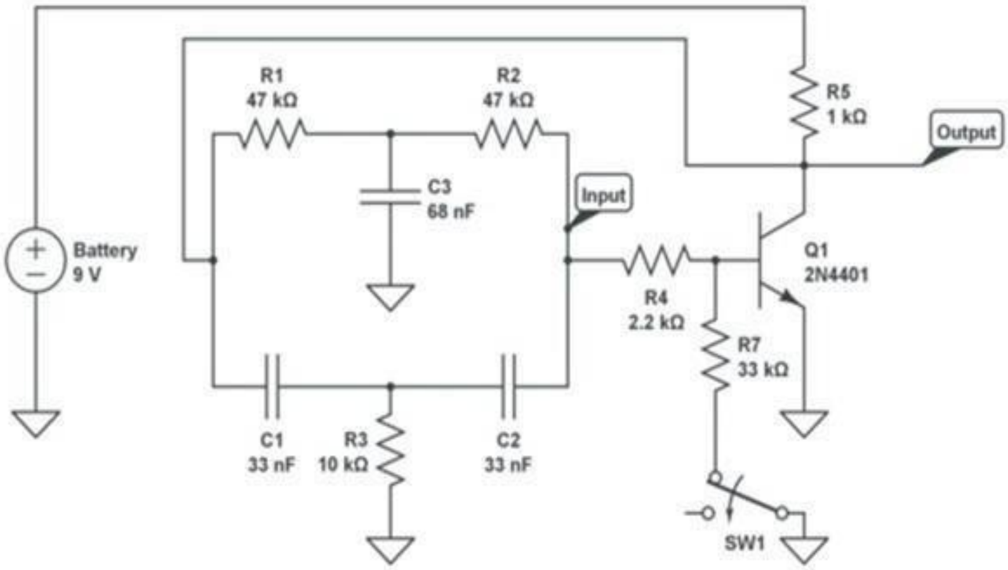
**SOFTWARE USED**

ORCAD 16.0

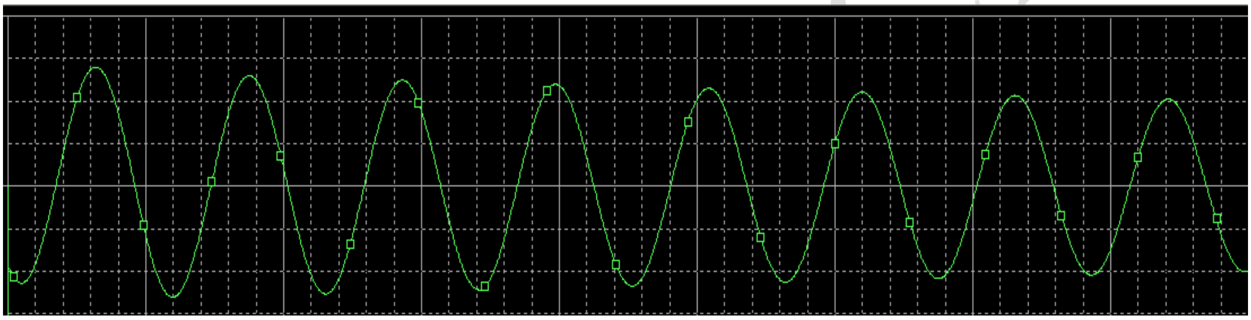
**PROCEDURE**

1. ORCAD - capture
2. Maximize the session log.
3. In file, open a new project and give the name to the project and choose analog or mixed A/B and specify the location say D:/ECE II
4. Create a blank project.
5. Click inside the window once and the tool bar appears.
6. Choose place part - add library - select a component in the library and select all the components - click open to add all the components.
7. Choose the components from the library and place in the worksheet.
8. Give connection using wire and properly ground the circuit.
9. Save the project and simulate it.
10. Place the marker at the points wherever the waveforms are to be viewed.
11. Click the option RUN and the output will be displayed.
12. If there are errors, correct and simulate it.
13. To view the waveform separately in plot – add plot to window – cut and paste the required waveforms.
14. The required input and output waveforms are taken printout.

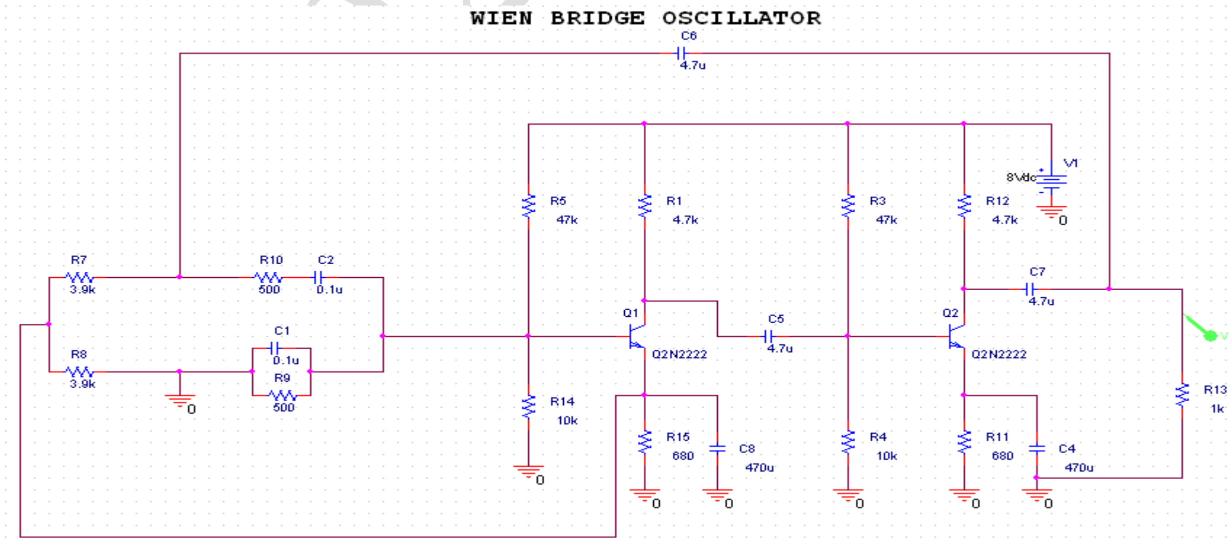
## CIRCUIT DIAGRAM:



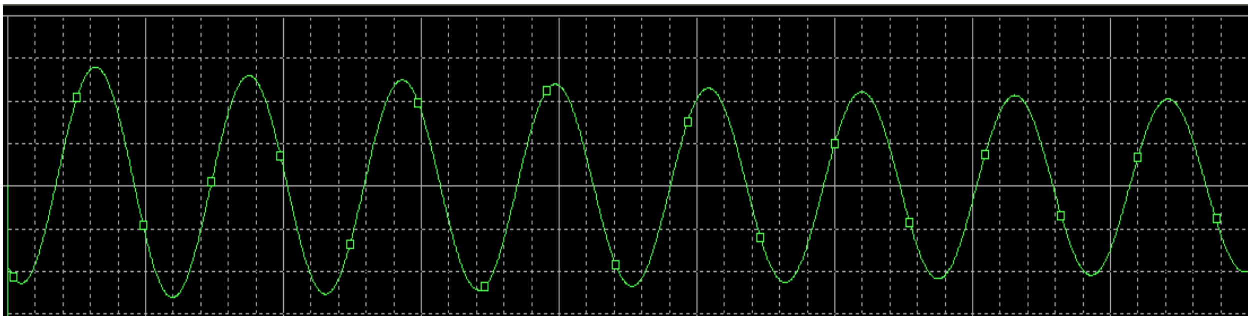

## MODEL GRAPH:



## CIRCUIT DIAGRAM:



**MODEL GRAPH:**



**RESULT:**

| Ex. No: 3 | DOUBLE AND STAGGER TUNED AMPLIFIERS |
|---|---|
| Date: | |

**AIM**

To construct and stimulate a Double and Stagger tuned amplifiers using pspice.

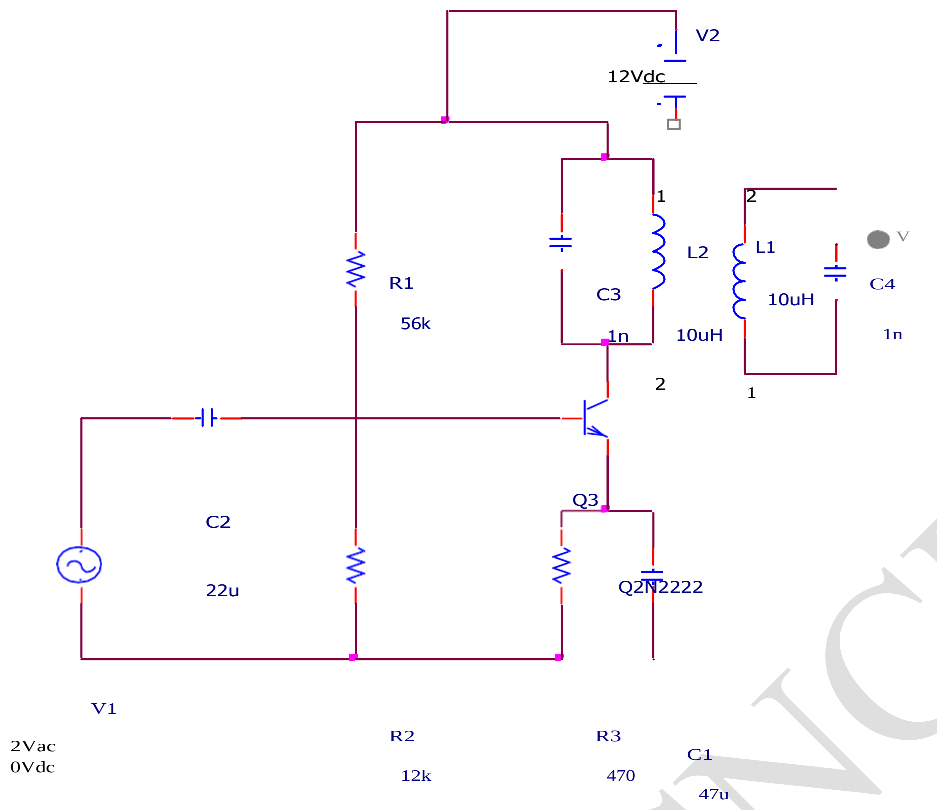
**SOFTWARE USED**

ORCAD 16.0

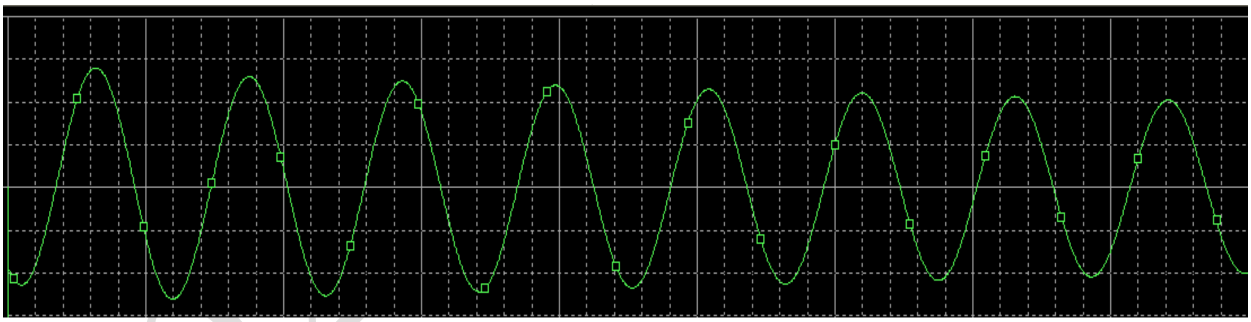
**PROCEDURE:**

1. ORCAD - capture
2. Maximize the session log.
3. In file, open a new project and give the name to the project and choose analog or mixed A/B and specify the location say D:/ECE II
4. Create a blank project.
5. Click inside the window once and the tool bar appears.
6. Choose place part - add library - select a component in the library and select all the components - click open to add all the components.
7. Choose the components from the library and place in the worksheet.
8. Give connection using wire and properly ground the circuit.
9. Save the project and simulate it.
10. Place the marker at the points wherever the waveforms are to be viewed.
11. Click the option RUN and the output will be displayed.
12. If there are errors, correct and simulate it.
13. To view the waveform separately in plot – add plot to window – cut and paste the required waveforms.
14. The required input and output waveforms are taken printout.

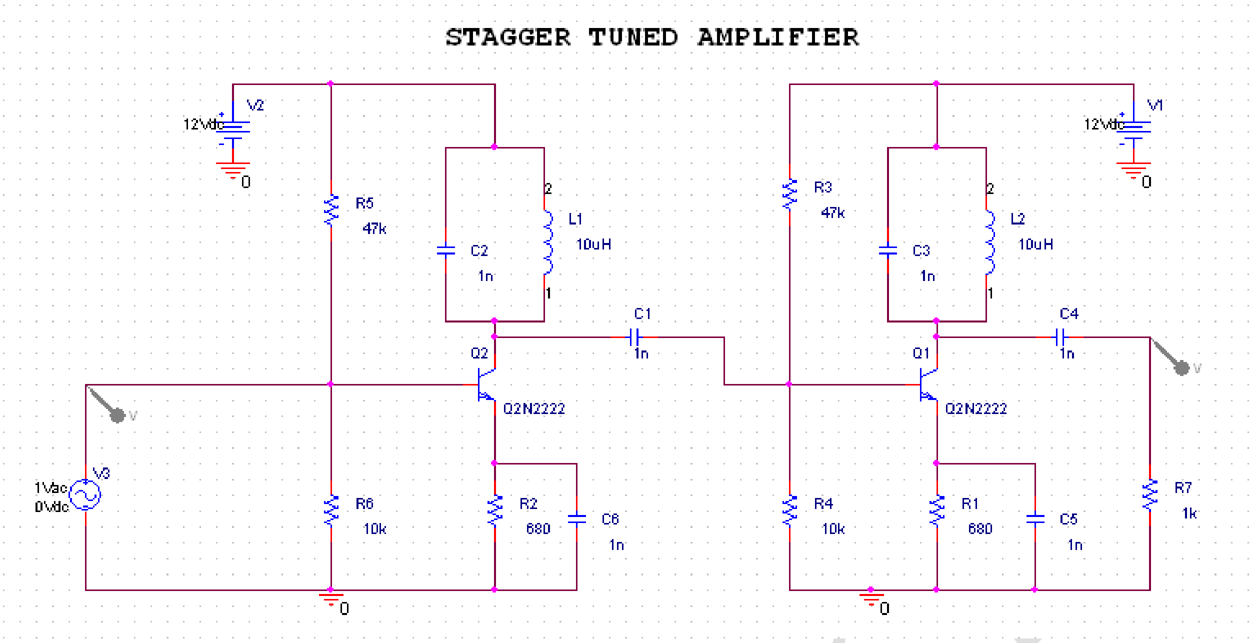
## DOUBLE TUNED AMPLIFIER



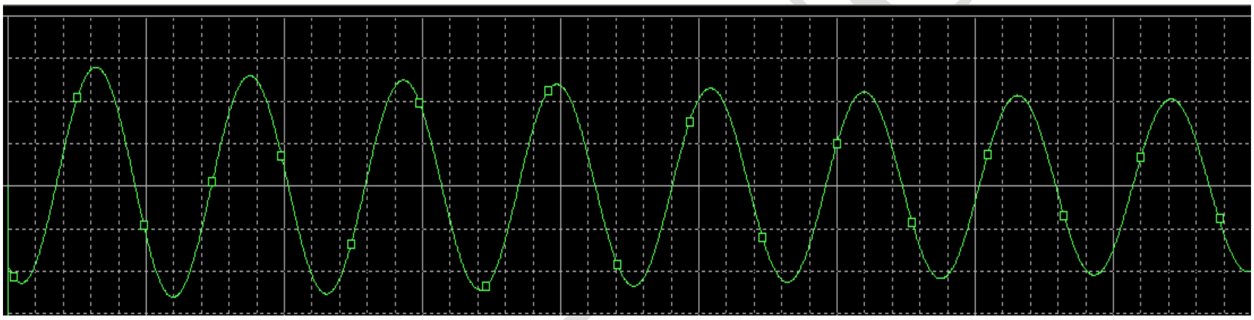

## OUTPUTWAVEFORM



## CIRCUIT DIAGRAM:




## OUTPUT WAVEFORM:



## RESULT:

| Ex. No: 4 | BISTABLE MULTIVIBRATOR |
|---|---|
| Date: | |

**AIM**

To construct and simulate the Bistable multivibrator by using pspice.

**SOFTWARE USED**

Multisim 13.0

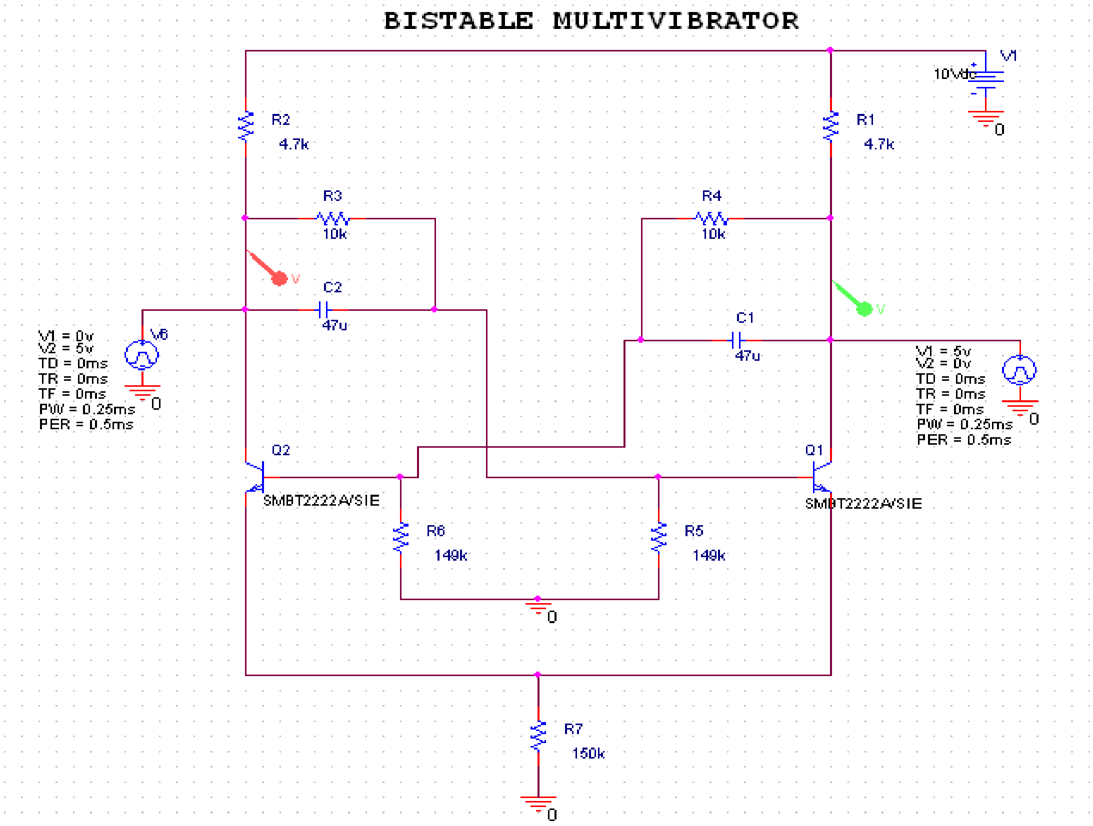
**PRELAB QUESTIONS:**

1. What is Multivibrator?
2. What are the other names of Bistable Multivibrator?
3. Mention the names of different kinds of triggering used in the circuit shown?
4. What is a stable state?
5. Describe the principle of fixed-bias Binary?

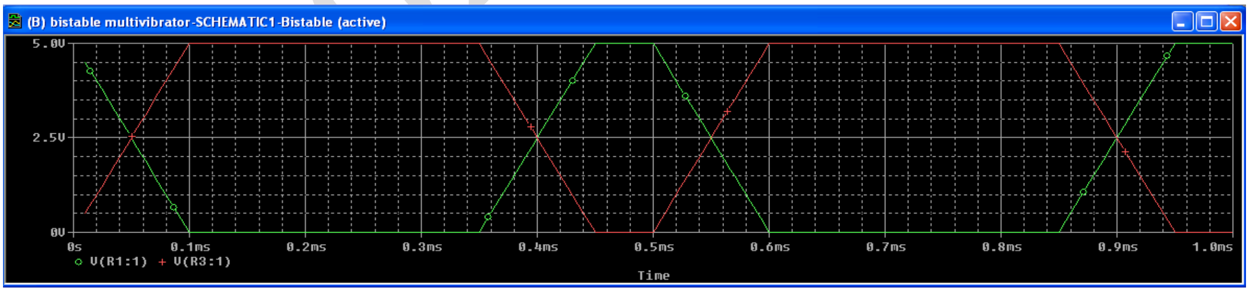
**PROCEDURE**

1. Open Multisim and create a “design”.
2. Draw a schematic diagram of the circuit (components and interconnections).
3. Connect signal sources to the circuit inputs , then stimulate the circuit to produce the output .
4. Connect the circuit outputs to one or more indicators to display the response of the circuit
5. Run the simulation and examine the results, copying and pasting Multisim windows into lab reports and other documents as needed.
6. Save the design.

## CIRCUIT DIAGRAM:




## OUTPUT WAVEFORM:




## RESULT:

| | |
|---|---|
| **Ex. No: 5** | **SCHMITT TRIGGER CIRCUIT WITH PREDICTABLE HYSTERESIS** |
| **Date:** | |

## AIM

To construct and simulate Schmitt trigger circuit by using pspice.

## SOFTWARE USED

Multisim 13.0

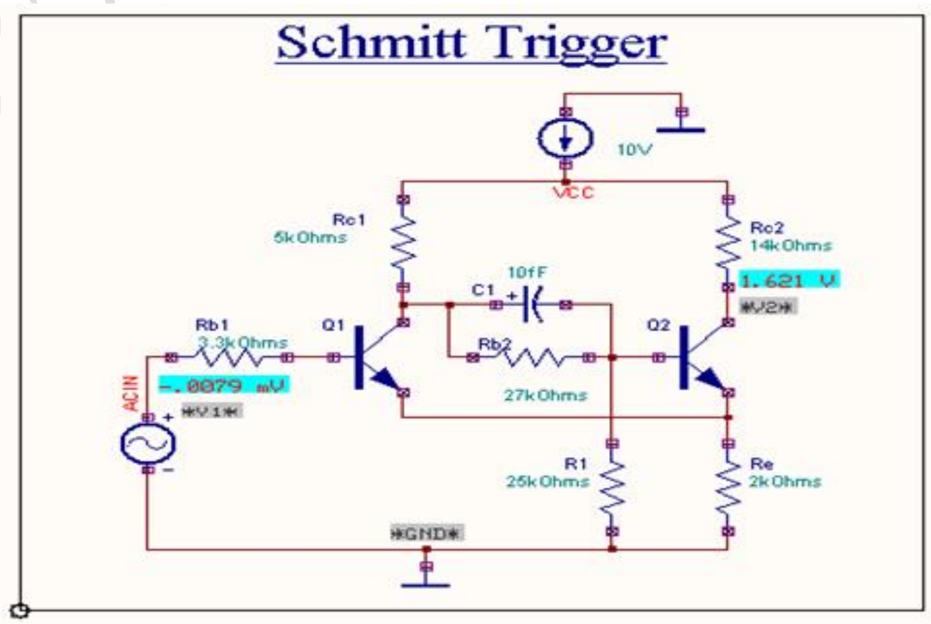
## PRELAB QUESTIONS:

1. What is Schmitt trigger?
2. What is the other name of a Schmitt Trigger ckt?
3. How many types of Schmitt trigger are used?
4. What is the difference between a Binary and Schmitt Trigger
5. Explain the working of Schmitt Trigger ckt?

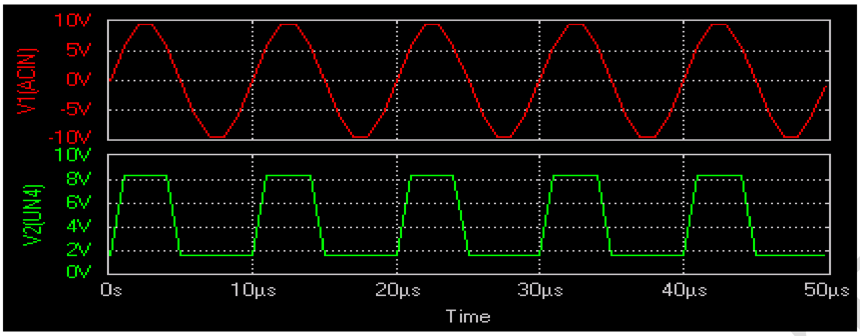
## PROCEDURE

7. Open Multisim and create a “design”.
8. Draw a schematic diagram of the circuit (components and interconnections).
9. Connect signal sources to the circuit inputs , then stimulate the circuit to produce the output .
10. Connect the circuit outputs to one or more indicators to display the response of the circuit
11. Run the simulation and examine the results, copying and pasting Multisim windows into lab reports and other documents as needed.
12. Save the design.

## CIRCUIT DIAGRAM



## OUTPUT WAVEFORM




## POST LAB QUESTIONS

1. Define UTP & LTP?
2. Explain how a Schmitt Trigger converts a sine wave to a square wave?
3. Explain Hysterisis with Schmitt Trigger?
4. What are the applications of Schmitt Trigger?
5. How noise can be eliminated on a given signal using Schmitt Trigger?

## RESULT:

Thus the Schmitt trigger circuit was designed and simulated using pspice simulator

| Ex. No: 6 | ANALYSIS OF POWER AMPLIFIERS |
|---|---|
| Date: | |

## AIM

To construct and simulate power amplifier circuit by using pspice.

## SOFTWARE USED

Multisim 13.0

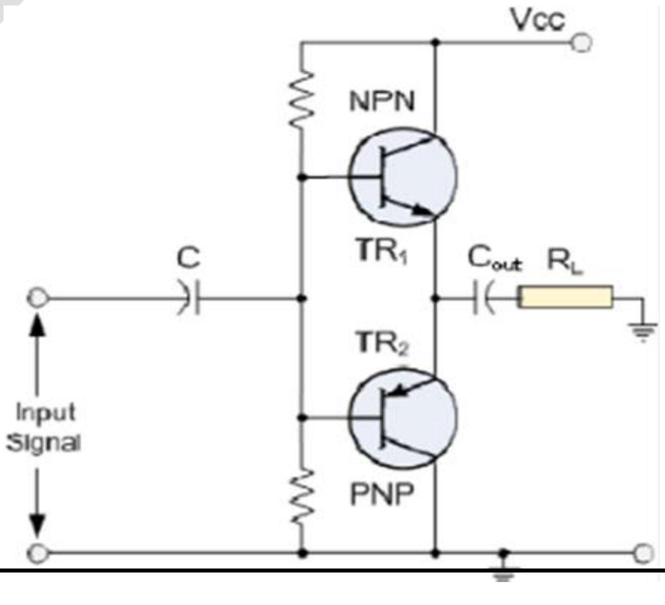
## PRELAB QUESTIONS:

1. List out some merits of PSPICE simulation tools.
2. What is the main difference between general amplifier and power amplifier ?
3. List out the classifications of power amplifier?
4. Define Power dissipation in power amplifier?
5. Mention any two applications of power amplifier?

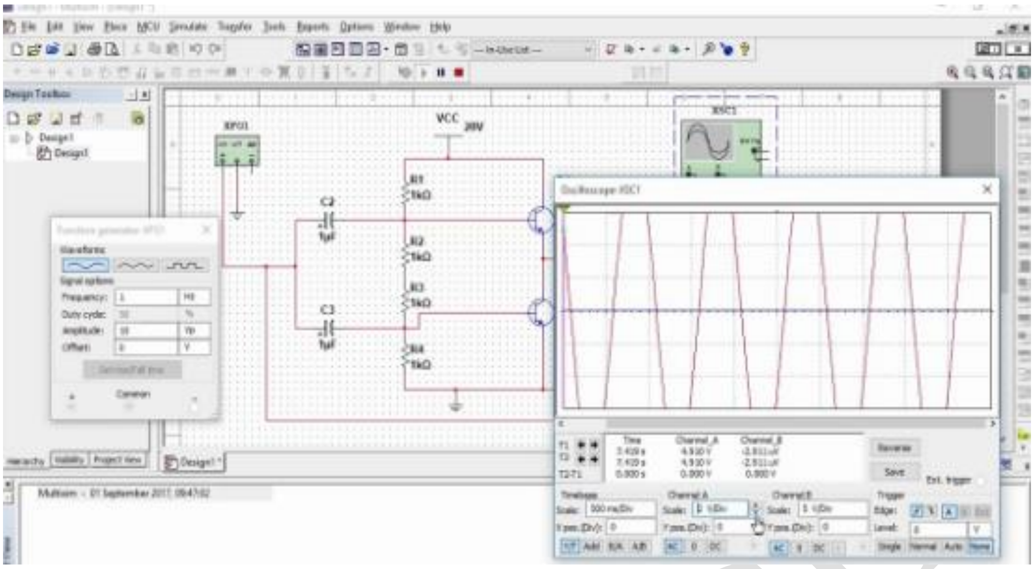
## PROCEDURE

1. Open Multisim and create a “design”.
2. Draw a schematic diagram of the circuit (components and interconnections).
3. Connect signal sources to the circuit inputs , then stimulate the circuit to produce the output .
4. Connect the circuit outputs to one or more indicators to display the response of the circuit
5. Run the simulation and examine the results, copying and pasting Multisim windows into lab reports and other documents as needed.
6. Save the design.

## CIRCUIT DIAGRAM



## OUTPUT WAVEFORM




## POST LAB QUESTIONS

1. What is called cross over distortion?
2. Define Total harmonic Distortion?
3. Examine the percentage of active region in various power amplifiers?
4. Which power amplifier has more efficient?
5. Why we can't get more current and Voltage gain using general amplifier?

## RESULT:

Thus power amplifier circuit was designed and simulated by using pspice simulator.