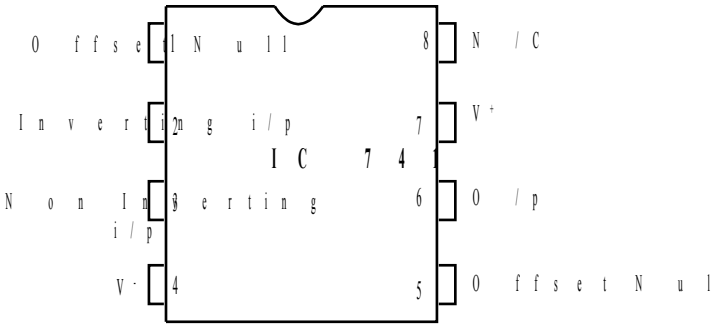
STUDY OF OP-AMP

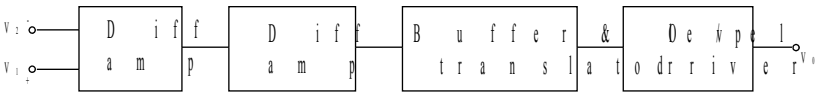
An operational amplifier or op-amp is a linear integrated circuit that has a very high voltage gain, high input impedance and low output impedance. Op-amp is basically a differential amplifier whose basic function is to amplify the difference between two input signals.

Op-amp has five basic terminals, that is, two input terminals, one o/p terminal and two power supply terminals. Pin2 is called the inverting input terminal and it gives opposite polarity at the output if a signal is applied to it. It produces a phase shift of 180^{o} between input and output. Pin3 is called the non-inverting terminal that amplifies the input signal without inversion, i.e., there is no phase shift or i/p is in phase with o/p. The op-amp usually amplifies the difference between the voltages applied to its two input terminals. Two further terminals pins 7 and 4 are provided for the connection of positive and negative power supply voltages respectively. Terminals 1 and 5 are used for dc offset. The pin 8 marked NC indicates 'No Connection'.

Study of op-amp



Block schematic of op-amp



The block diagram of op-amp shows 2 difference amplifiers, a buffer for less loading, a level translator for adjusting operating point to original level and o/p stage. An ideal op-amp should have the following characteristics:

1. Infinite bandwidth
2. infinite input resistance
3. infinite open loop gain
4. zero output resistance
5. zero offset.

Op-amps have two operating configurations; open loop and closed loop. In open loop configuration, it can operate as a switch but gain is uncontrolled. In closed loop configuration, gain can controlled by feed back resistance Rf and input resistance Rin.

EX.No:1

LINEAR OP-AMP CIRCUITS

Aim:

To design Voltage Follower, Inverting and Non inverting, Differentiator, Integrator, Subtractor, summing amplifier using op-amp and test its performance.

Apparatus required:

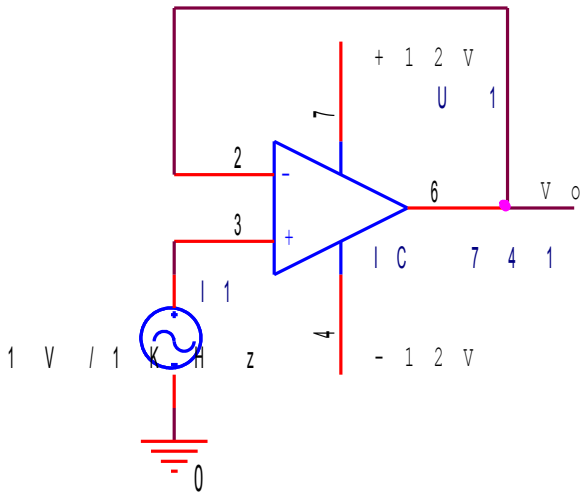
S.No	Components	Range	Quantity
1.	Op-amp	IC 741	1
2.	Dual trace supply	(0-30) V	1
3.	Function Generator	(0-1) MHz	1
4.	Resistors		
5.	Capacitors		
6	CRO	(0-30) MHz	1

1) Voltage Follower:

Design:

Vin = Vout [Unity Gain] & Rin = ∞ & Rf = 0

Circuit Diagram

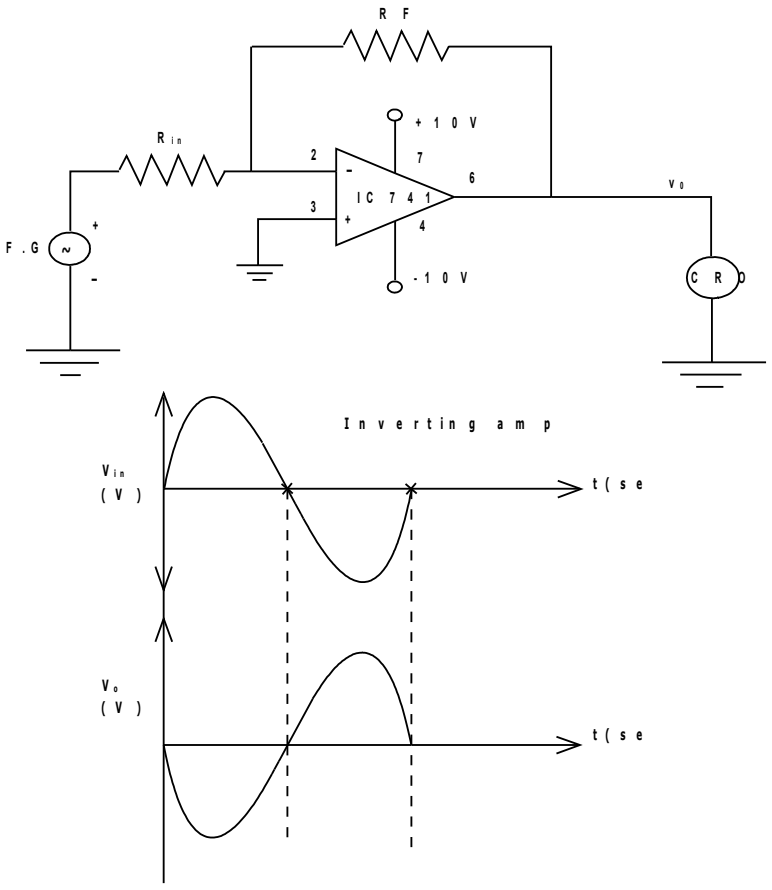


2) Inverting amplifier: [Closed Loop Configuration]

Design:

$A_{CL} = V_o/V_{in} = - R_f / R_{in}$; Assume R_{in} = _____; Gain = _____;

Circuit Diagram:



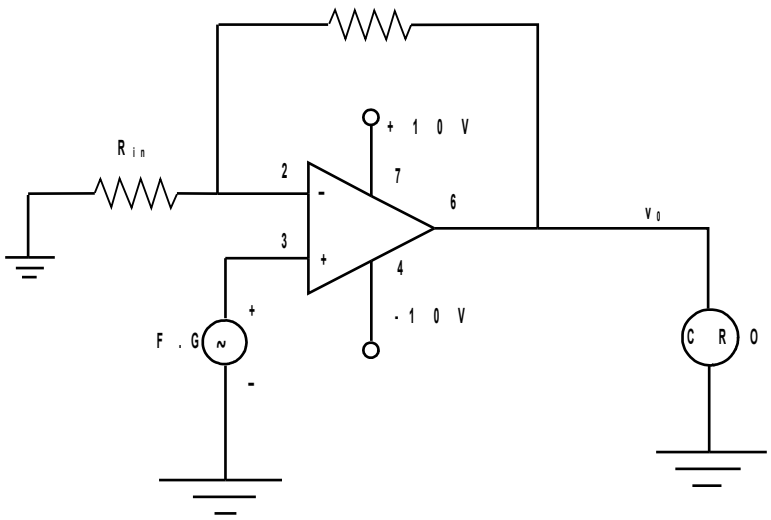
3) Non inverting amplifier: [Closed Loop Configuration]

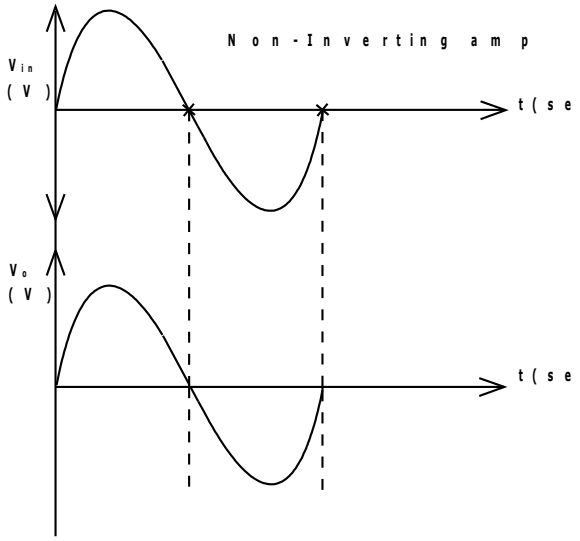
Design:

$A_{CL} = V_o / V_{in} = 1 + R_f / R_{in;}$ Assume R_{in} = _____; Gain = _____;

Circuit Diagram

-





4) Differentiator:

Design:

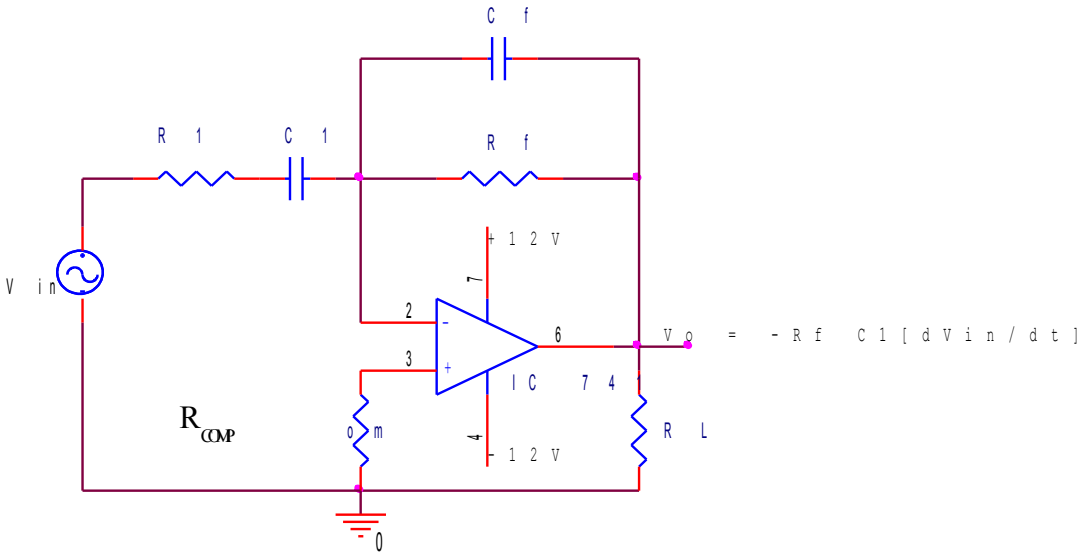
Step1: Select f_a equal to the highest frequency of the input signal to be differentiated. Then assuming a value of $C_1 < 1\mu$ F. Calculate the value of R_f.

Step2: Choose $f_b = 20\ f_a$ and calculate the values of R_1 and C_f so that $R_1C_1 = R_f C_f$.

f_a = _____KHz ; f_b = _____KHz ;$C_1 = 0.1\ \mu f$; $R_{COMP} = R_f$; $R_L = 10K\Omega$

$f_a = 1/ [2\pi R_f C_1]$; $R_f = 1/2\pi\ C_1\ f_a$; $f_b = 1/ [2\pi R_1 C_1]$;$R_1 = 1/2\pi\ C_1\ f_b$;$R_1C_1 = R_f C_f$; $C_f = R1C1/ R_f$

Circuit Diagram



Observation:

For sine wave input:

Peak to peak amplitude of the input = volts.

Frequency of the input = Hz

Peak to peak amplitude of the output = volts.

Frequency of the output = Hz

For square wave input:

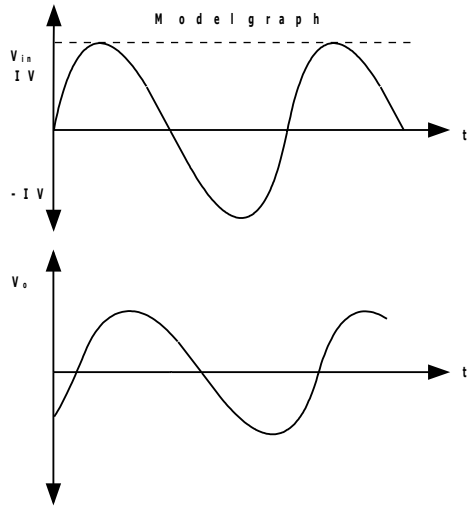
Peak to peak amplitude of the input = volts.

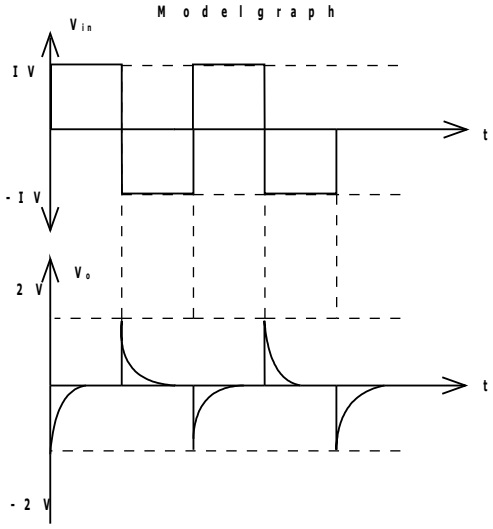
Frequency of the input = Hz

Peak to peak amplitude of the output = volts.

Frequency of the output = Hz

Model Graph:





5) Integrator:

Design:

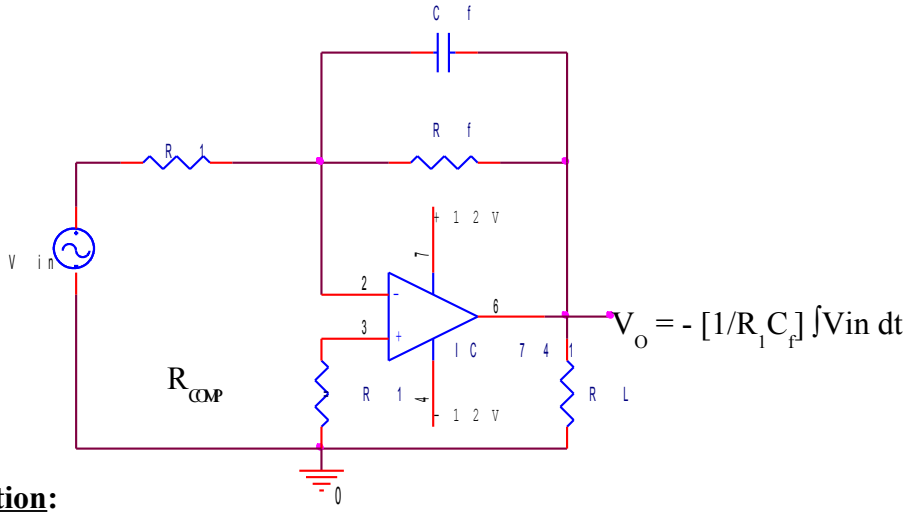
Generally the value of the f_a and in turn R_1C_f and $R_f C_f$ values should be selected such that $f_a < f_b$. From the frequency response we can observe that f_a is the frequency at which the gain is 0 db and f_b is the frequency at which the gain is limited. Maximum input signal frequency = 1 KHz.

Condition is time period of the input signal is larger than or equal to R_fC_f (i.e.) $T \geq R_1C_f$

f_b = _____KHz ; $f_a = f_b/10$; $R_f = 10R_1$; $R_{COMP} = R_{1;} R_{L\ \&}\ R_1 = 10K\Omega$

$f_a = 1/ [2\pi R_fC_f]$; RfC_f = 1msec &; Cf = 1msec/100K

Circuit Diagram:



Observation:

For sine wave input:

Peak to peak amplitude of the input = volts.

Frequency of the input = Hz

Peak to peak amplitude of the output = volts.

Frequency of the output = Hz

For square wave input:

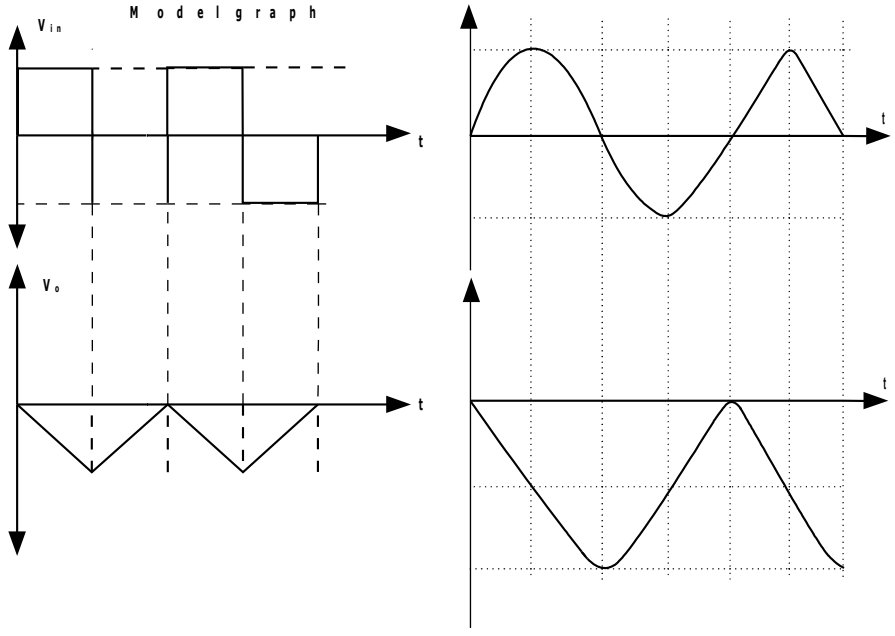
Peak to peak amplitude of the input = volts.

Frequency of the input = Hz

Peak to peak amplitude of the output = volts.

Frequency of the output = Hz

Model Graph:

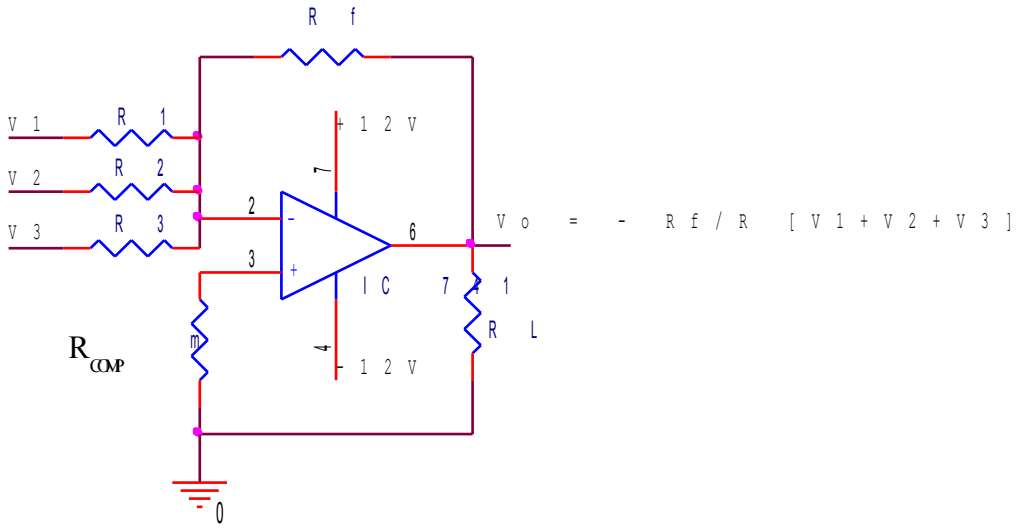


6) <u>Summing Amplifier</u>: [Inverting]

<u>Design</u>: $R_1=R_2=R_3=R$ & $R = R_f = \underline{4.7}K\Omega$; $R_L = 10\ K\Omega$; $R_{COMP} = R_1 \| R_2 \| R_3 \| R_f$

$V_O = - R_f / R\ [V_1+V_2+V_3]$

<u>Circuit Diagram</u>

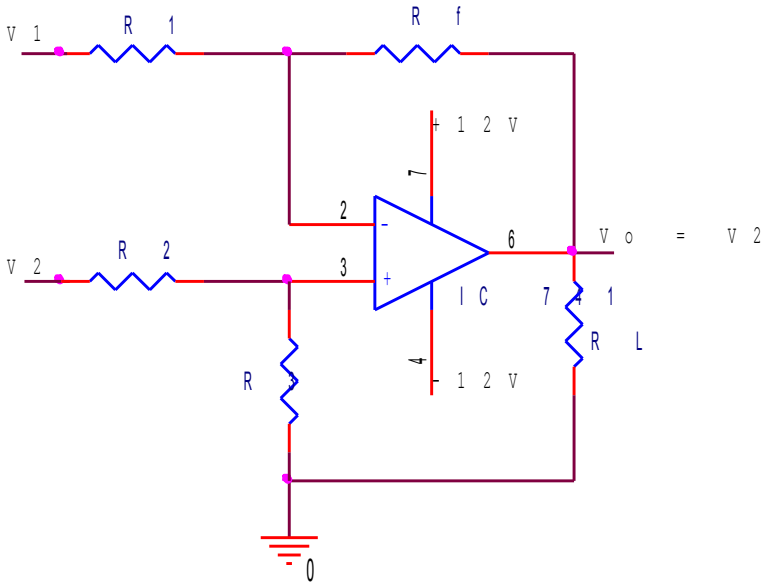


7) <u>Subtractor</u>: [Differential Configuration]

<u>Design:</u> $R_1=R_2=R_3=R_f = R =$ _____

$V_O = R_f / R\ [V_2 - V_1]$

<u>Circuit Diagram:</u>



Procedure:

1. Connect the components as per the circuit diagram.
2. Set the input voltage using {F.G [for 1 to 5]. and DC Supply [for 6 & 7]}observe the output waveform at Pin no.6
3. Connect CRO at Pin no.6 and measure 0/p voltage and note it down.
4. Plot the output waveforms

Viva Question:

1. What is an op-amp?
2. Give the characteristics of an ideal op-amp:
3. How a non-inverting amplifier can be courted into voltage follower?
4. What is the necessity of negative feedback?
5. What are 4 building blocks of an op-amp?
6. What is the purpose of shunting C_f across R_f and connecting R_1 in series with the input signal?
7. What are the applications of Differentiator?
8. What do you mean by unity gain bandwidth?
9. What did you observe at the output when the signal frequency is increased above f_a?
10. How would you eliminate the high frequency noise in integrator?
11. What are the main applications of the Integrator?
12. Is it possible to design an analog computer using integrator and differentiator?
13. What happens to the output of integrator when input signal frequency goes below f_a?

Result:

Thus Voltage Follower, Inverting and Non inverting, Differentiator, Integrator, Subtractor, summing amplifier using op-amp was designed and tested.

Ex.No.2

COMPARATOR CIRCUITS

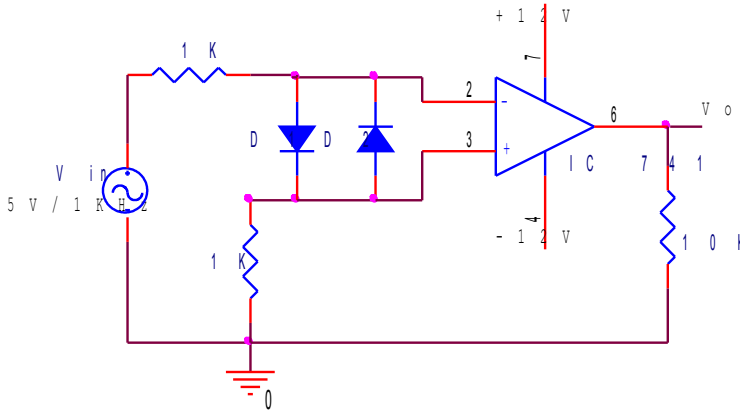
Aim:

To design and test the following circuits, Zero crossing detector, Window detector and Schmitt trigger using Op-amp

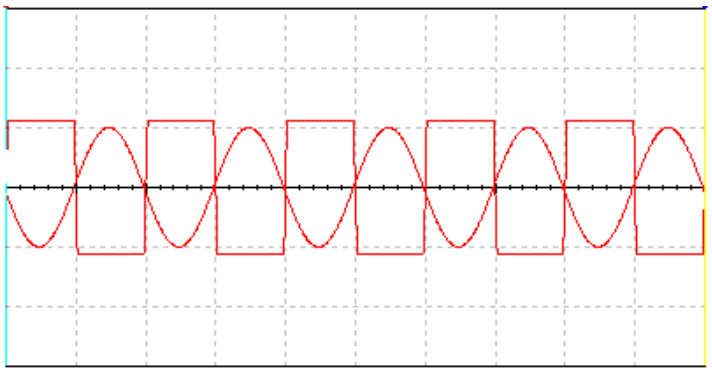
Apparatus Required.

S.No	Component	Range	Quantity
1.	Op amp	IC 741	1
2.	F.G	(0-1) MHz	1
3.	Resistors		
4.	CRO	(0-30) MHz	1
5.	DTS	(0-30) V	1
6.	Diode	1N4007	2

1) Zero Crossing Detector: [Sine wave to square wave converter]



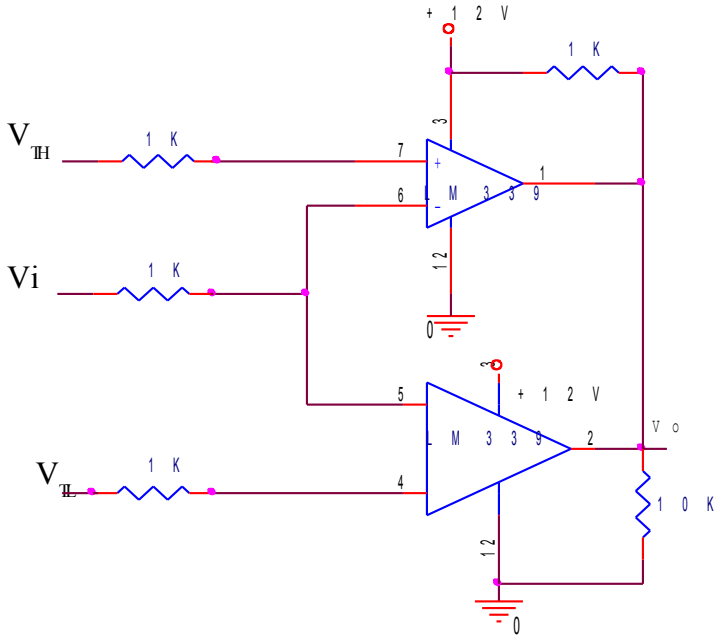
Model Graph:



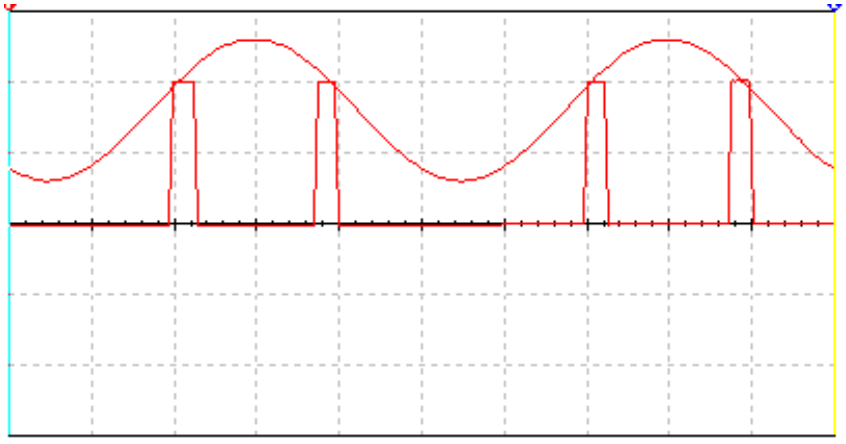
2) Window Detector:

Condition	Output
$V_{TL} < V_i > V_{TH}$	$V_O = V_{CC}$
$V_i > V_{TH}$ OR $V_i < V_{TL}$	$V_O = 0$

Circuit Diagram:



Model Graph:



Observation:

S.No	V_{TH}	V_{TL}	V_i	V_o
1	6V	3V	5V/1KHz	
2	3V	0V	5V/1KHz	
3	0V	6V	5V/1KHz	

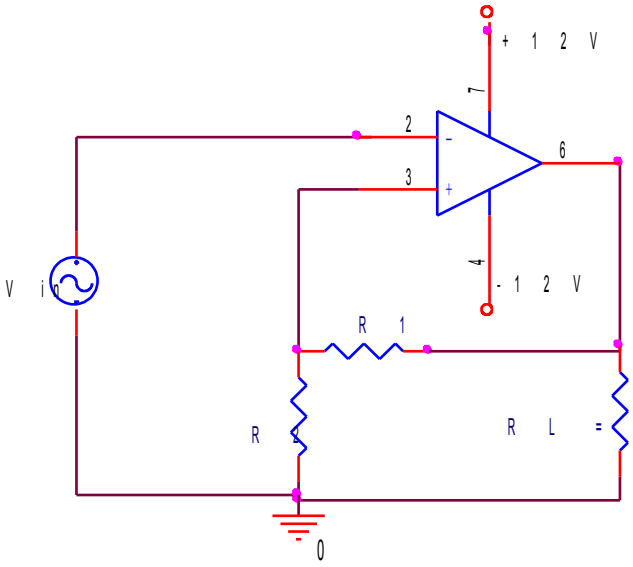
3) Schmitt Trigger:

Design

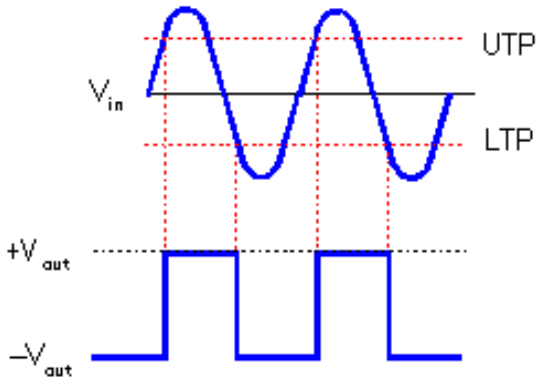
V_{CC} = 12 V; V_{SAT} = 0.9 V_{CC}; R1= 47KΩ; R2 = 120Ω

V_{UT} = + [V_{SAT} R_2] / [R_1+R_2] & V_{LT} = - [V_{SAT} R_2] / [R_1+R_2] & HYSTERSIS [H] = V_{UT} - V_{LT}

Circuit Diagram



Model Graph



Procedure

1. Connect the circuit as shown in the circuit
2. Set the input voltage as 5V (p-p) at 1KHz. (Input should be always less than V_{cc})
3. Note down the output voltage at CRO
4. To observe the phase difference between the input and the output, set the CRO in dual Mode and switch the trigger source in CRO to CHI.
5. Plot the input and output waveforms on the graph.

Observation:

Peak to peak amplitude of the output = Volts.

Frequency = Hz.

Upper threshold voltage = Volts.

Lower threshold voltage = Volts.

Viva Questions:

1. What is Hysteresis? What parameter determines Hysteresis?
2. How would you recognize that positive feedback is being used in the Op-amp circuit?
3. What do you mean by upper and lower threshold voltage in Schmitt Trigger?
4. What is the difference between a basic comparator and the Schmitt trigger?
5. What is a sample and hold circuit? Why is it needed?
6. What is a voltage limiting, and why is it needed?
7. What is the name of the circuit that is used to detect the peak value of the Nonsinusoidal input waveforms?
8. How will you produce, definite Hysteris in a Schmitt trigger using op-amp?

Result:

Thus comparator circuits using op-amp was designed & tested.

Ex. No 3

SAMPLE AND HOLD CIRCUITS

Aim

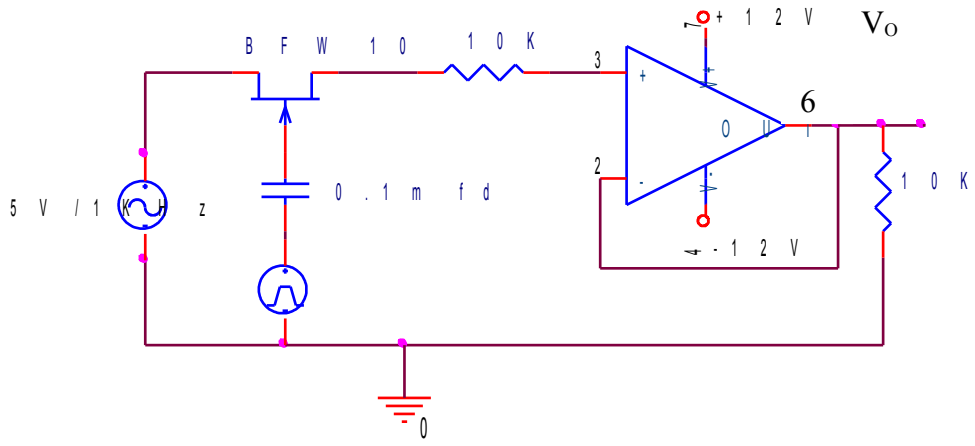
To construct Sample and Hold circuit using op-amp and plot its waveforms.

Apparatus Required:

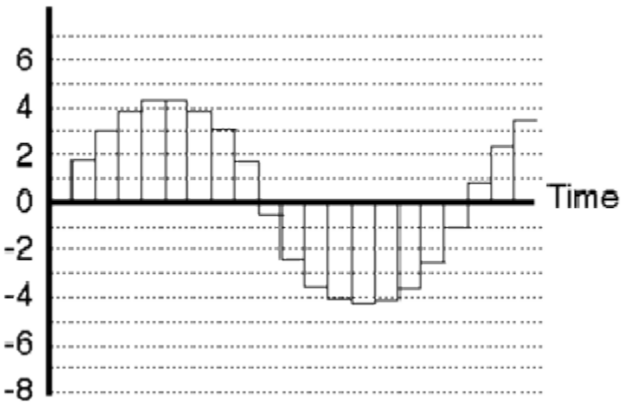
S.No	Component	Range	Quantity
1.	Op amp	IC 741	1
2.	DTS	(0-30) V	1
3.	CRO		1
4.	Resistor		1
5.	Capacitors	–	–
6.	JFET	BFW 10	1
7.	Function Generator	(0-1) MHz	2

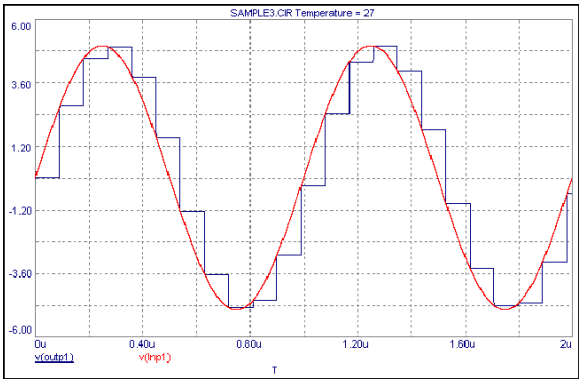
Circuit Diagram:

-



Model Graph





Procedure

1. Connect the circuit as shown in the circuit diagram.
2. Set the input Sine waveform as 5V/1 KHz at FG1
3. Set the square waveform as 10V/100KHz at FG 2
4. Note down the output voltage at CRO
5. Plot the input and output waveforms on the graph.

Result:

Thus the sample and hold circuit using Op-amp was constructed and tested.

Ex. No 4

Astable and Monostable Multivibrators using op-amp

Aim

To design Astable and monostable Multivibrators using op-amp and plot its waveforms.

Apparatus Required:

S.No	Component	Range	Quantity
1.	Op amp	IC 741	1
2.	DTS	(0-30) V	1
3.	CRO		1
4.	Resistor		1
5.	Capacitors	–	–
6.	Diode	IN4001	2
7.	Probes	–	1

Design:

1. Monostable Multivibrators:

$\beta = R_2/R_1+R_2$ [$\beta = 0.5$ & $R_1 = 10$ K]

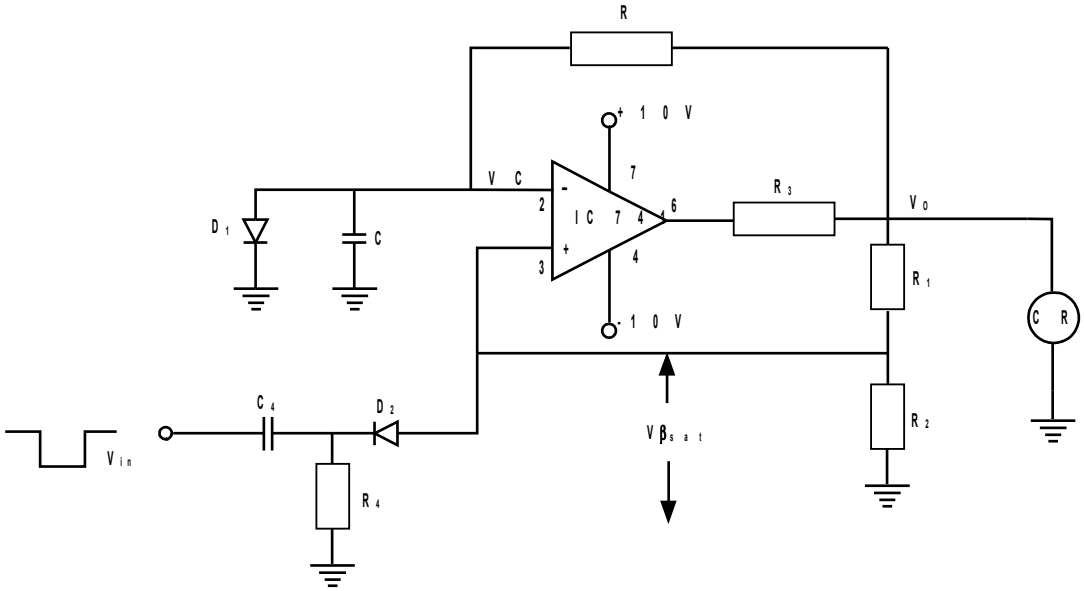
Find R_2 = ; R3 = 1K; R4 = 10K;

Let F =_____KHz ; C= 1mfd; C4 = 0.1mfd

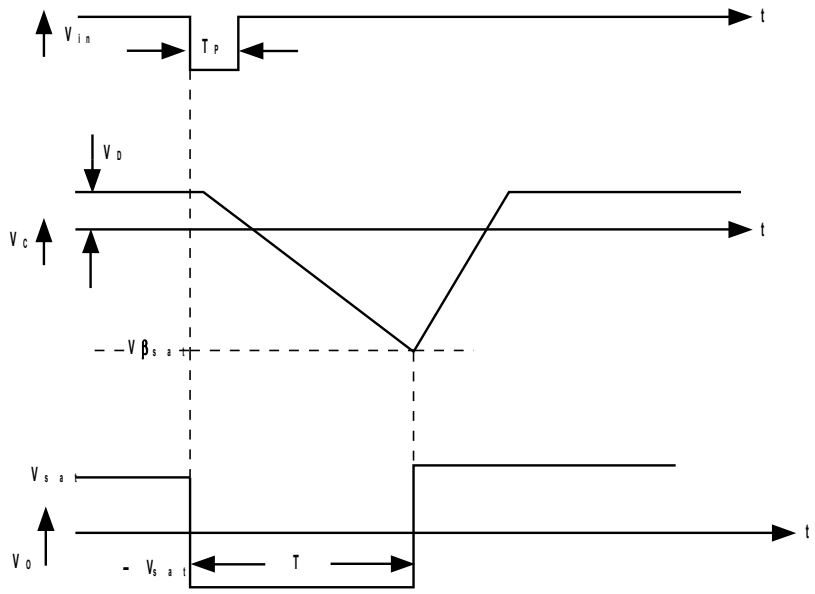
Pulse width, T = 0.69RC

Find R =

Circuit Diagram



Model graph:



Procedure:

1. Make the connections as shown in circuit diagram.
2. A trigger pulse is given through differentiator circuit through pin no.3
3. Observe the pulse waveform at pin no.6 using CRO and note down the time period.
4. Plot the waveform on the graph.

2. Astable Multivibrators:

Design:

$T = 2RC$

$R_1 = 1.16\ R_2$

Given $f_O =$ _____KHz

Frequency of Oscillation fo = 1 / 2 RC if $R_1 = 1.16R_2$

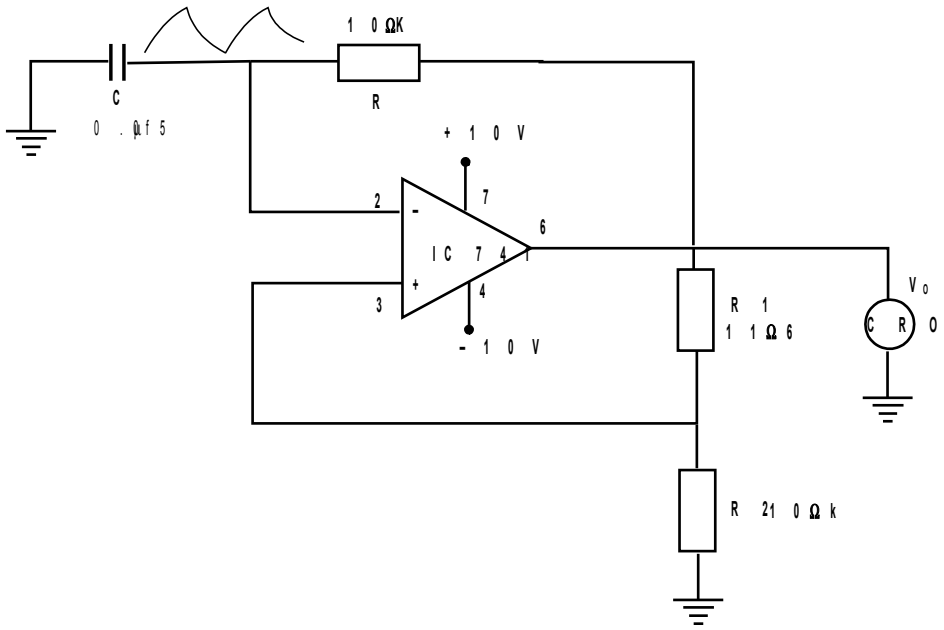
Let $R_2 = 10\ K\Omega$

$R_1 = 10 * 1.16 = 11.6 K\Omega$

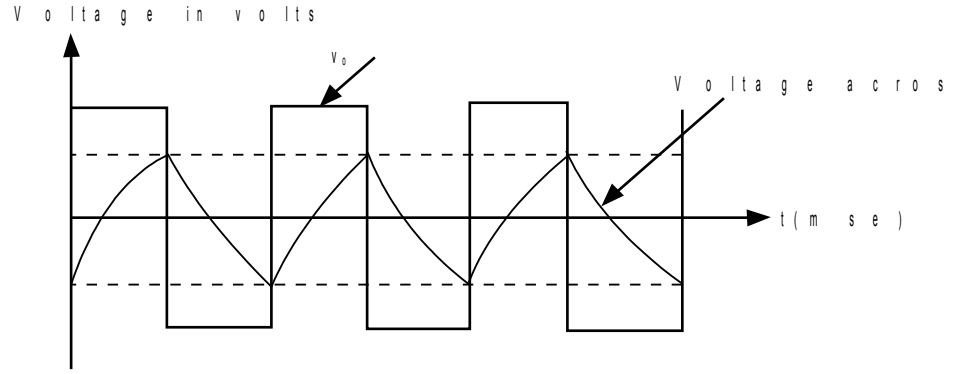
Let $C = 0.05\ \mu F$

$R = 1 / 2\ fC = 1/ (2 * 1 * 10^3 * 0.05 * 10^{-6}) =$

Circuit Diagram



Model graph



Procedure:

1. Make the connections as shown in the circuit diagram
2. Keep the CRO channel switch in ground and adjust the horizontal line on the x axis so that it coincides with the central line.
3. Select the suitable voltage sensitivity and time base on the CRO.
4. Check for the correct polarity of the supply voltage to op-amp and switch on power supply to the circuit.
5. Observe the waveform at the output and across the capacitor. Measure the frequency of oscillation and the amplitude. Compare with the designed value.
6. Plot the Waveform on the graph.

Questions:

1. What is other name for Astable Multivibrators?
2. How an Op-amp is used to generate square wave?
3. What are the changes to be done in a symmetric square wave generator to generate asymmetric square wave?

Result:

Thus Astable & Monostable Multivibrators were designed using op-amp and the waveforms were plotted.

EXP.NO: 5

MULTIVIBRATORS USING IC 555

Aim:

To design and test an Astable and Monostable Multivibrators using 555 timer with duty cycles ratio.

Apparatus Required:

S.No	Component	Range	Quantity
1.	555 TIMER		1
2.	Resistors	3.3K, 6.8k	1
3.	Capacitors	0.1 μ F, 0.01μ F	2
4.	Diode	In4001	1
5.	CRO		1
6.	Power supply	± 15 V	1
7.	Probe		2
8.	Bread Board		1

Astable Multivibrators using 555

Fig shows the 555 timer connected as an Astable Multivibrators. Initially, when the output is high. Capacitor C starts charging towards V_{cc} through R_A and R_B. As soon as capacitor voltage equals 2/3 V_{cc} upper comparator (UC) triggers the flip flop and the output switches low. Now capacitor C starts discharging through R_B and transistor Q_1.

When the voltage across C equals 1/3 V_{cc} lower comparator (LC), output triggers the flip-flop and the output goes high. Then the cycle repeats.

The capacitor is periodically charged and discharged between 2/3 V_{cc} and 1/3 V_{cc} respectively. The time during which the capacitor charges form 1/3 V_{cc} to 2/3 V_{cc} is equal to the time the output is high and is given by

$$T_c = 0.69(R_A+R_B)C \qquad (1)$$

Where R_A and R_B are in Ohms and C is in farads. Similarly the time during which the capacitor discharges from 2/3 V_{cc} to 1/3 V_{cc} is equal to the time the output is low and is given by

$$T_d = 0.69\ R_B\ C \qquad (2)$$

The total period of the output waveform is

$$\mathbf{T = T_c + T_d = 0.69 (R_A + 2R_B) C} \quad (3)$$

The frequency of oscillation

$$f_o = 1 / T = 1.45 / (R_A + 2R_B)C \quad (4)$$

Eqn (4) shows that fo is independent of supply voltage Vcc

The duty cycle is the ratio of the time t_d during which the output is low to the total time period T. This definition is applicable to 555 Astable Multivibrators only; conventionally the duty cycle ratio is defined as the ratio as the time during which the output is high to the total time period.

$$\therefore \quad \text{Duty cycle} = t_d \, T \times 100$$

$$R_B + R_A + 2R_B \times 100 \quad (5)$$

To obtain 50% duty cycle a diode should be connected across R_B and R_A must be a combination of a fixed resistor and a potentiometer. So that the potentiometer can be adjusted for the exact square waves

DESIGN:

Design an Astable Multivibrators for a frequency of _____KHz with a duty cycle ratio of D = 50 %

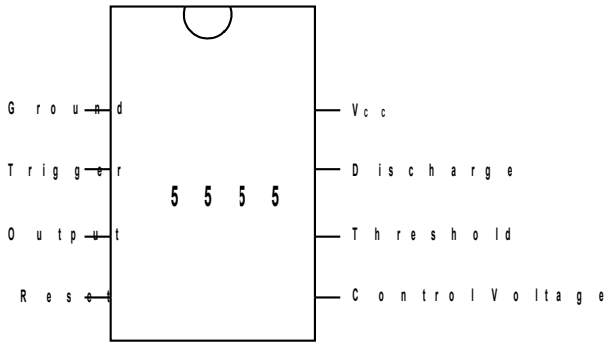
$$\mathbf{fo = 1/T = 1.45 / (R_A + 2R_B)C}$$

Choosing C = 1 μ F; $R_A = 560$

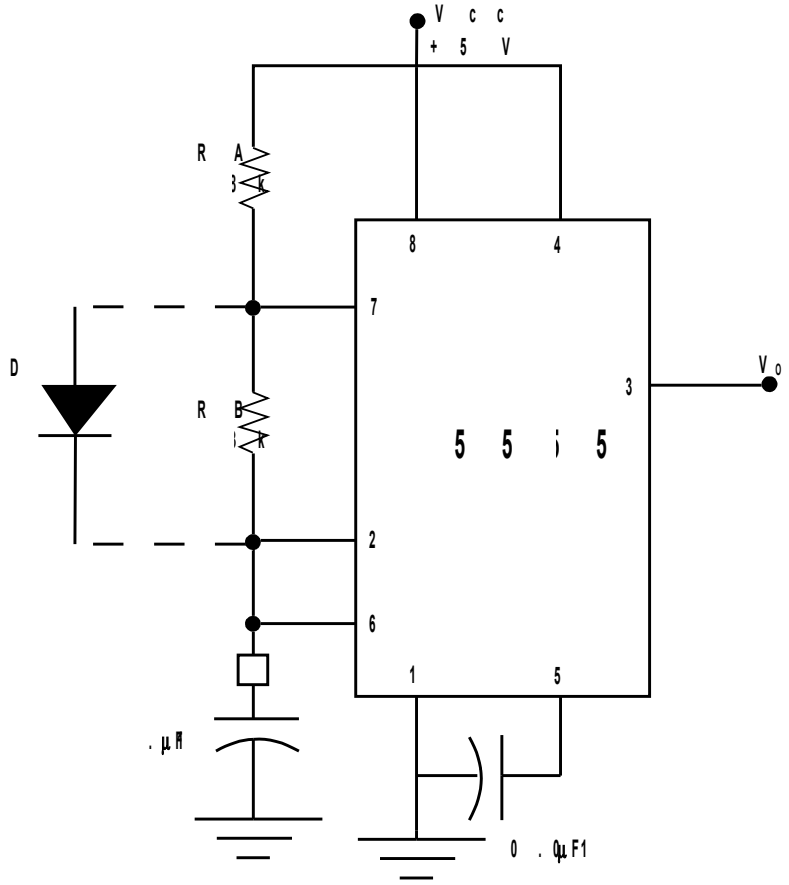
$D = R_B / R_A + 2R_B = 0.5$ [50%]

$R_B =$ _____

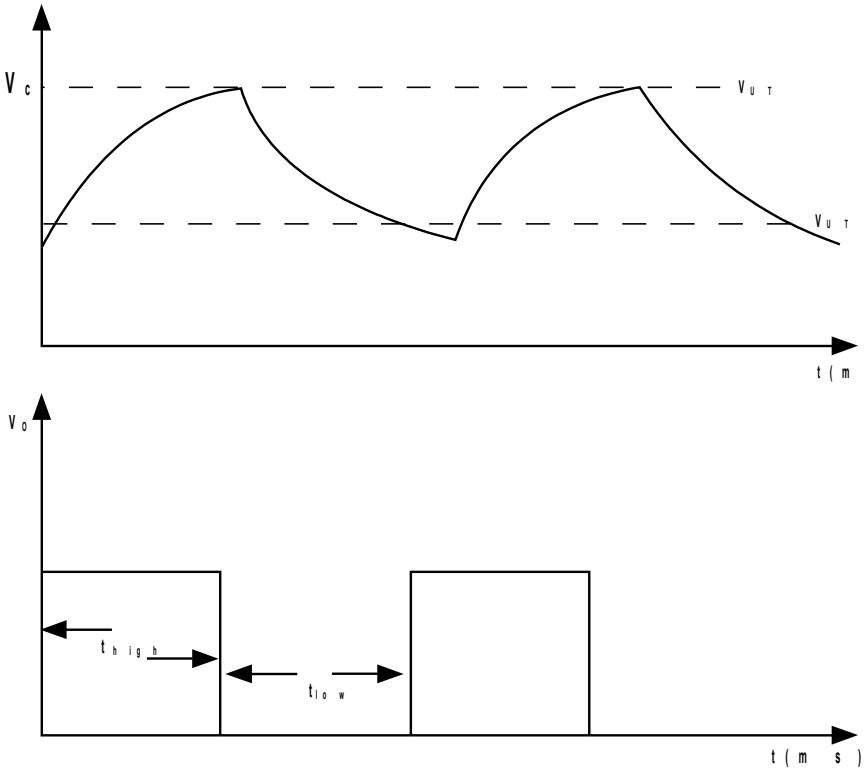
Pin diagram:



Circuit Diagram



Model Graph



Procedure:

1. Rig-up the circuit of 555 Astable Multivibrators as shown in fig with the designed value of components.
2. Connect the CRO probes to pin 3 and 2 to display the output signal and the voltage across the timing capacitor. Set suitable voltage sensitively and time-base on the CRO.
3. Switch on the power supply to CRO and the circuit.
4. Observe the waveforms on the CRO and draw to scale on a graph sheet. Measure the voltage levels at which the capacitor starts charging and discharging, output high and low timings and frequency.
5. Switch off the power supply. Connect a diode across R_B as shown in dashed lines in fig to make the Astable with 50 % duty cycle ratio. Switch on the power supply. Observe the output waveform. Draw to scale on a graph sheet.

Monostable Multivibrators using 555

Monostable Multivibrators has one stable state and other is a quasi stable state. The circuit is useful for generating single output pulse at adjustable time duration in response to a triggering signal. The width of the output pulse depends only on external components, resistor and a capacitor.

The stable state is the output low and quasi stable state is the output high. In the stable state transistor Q1 is 'on' and capacitor C is shorted out to ground. However upon application of a negative trigger pulse to pin2, Q1 is turned 'off' which releases the short circuit across the external capacitor C and drives the output high. The capacitor C now starts charging up towards V_{cc} through R_A. However when the voltage across C equal 2/3 V_{cc} the upper comparator output switches form low to high which in turn drives the output to its low state via the output of the flip flop. At the same time the output of the flip flop turns Q1 'on' and hence C rapidly discharges through the transistor. The output remains low until a trigger is again applied. Then the cycle repeats.

The pulse width of the trigger input must be smaller than the expected pulse width of the output. The trigger pulse must be of negative going signal with amplitude larger than 1/3 Vcc. The width of the output pulse is given by,

$$T = 1.1\ R_A C$$

Design:

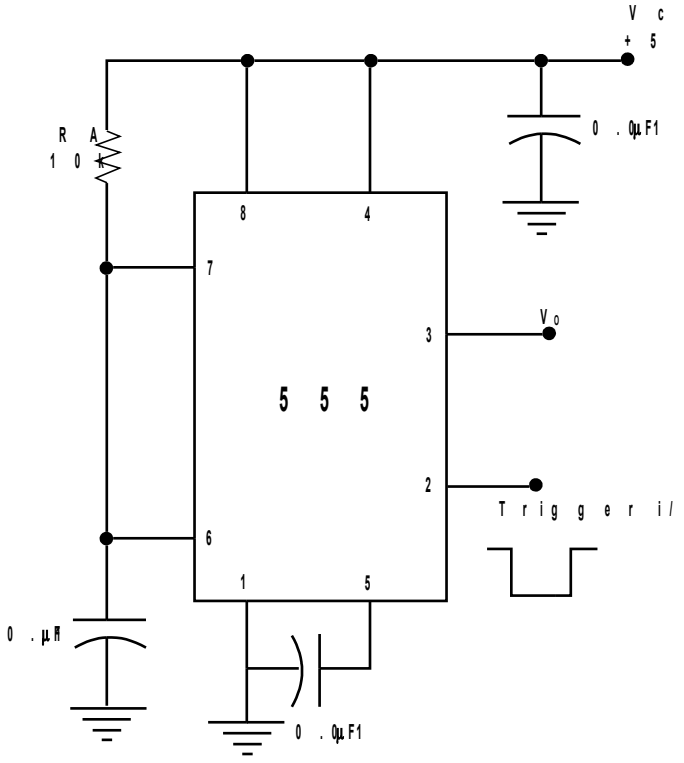
Given a pulse width of duration of 100 μ s

Let C = 0.01 mfd; F = _____KHz

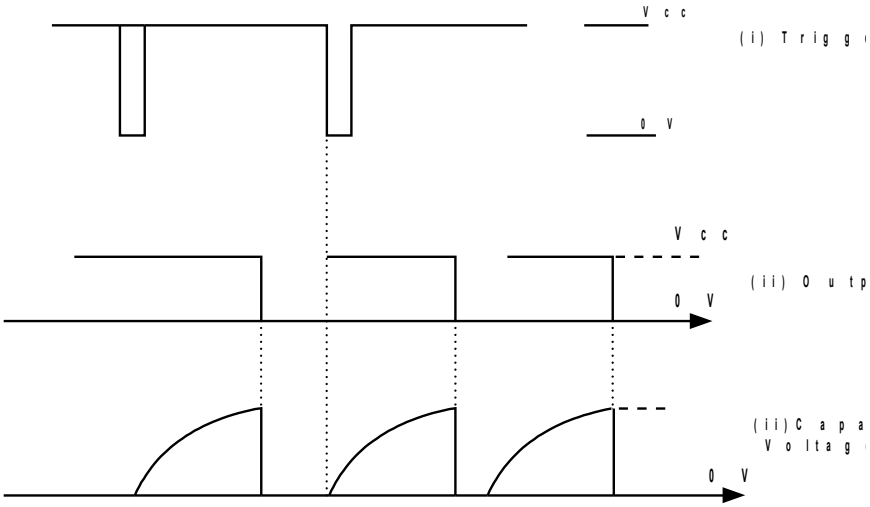
Here, T= 1.1 R_AC

So, R_A =

Circuit Diagram:



Model Diagram:



Procedure:

1. Rig-up the circuit of 555 monostable Multivibrators as shown in fig with the designed value of components.
2. Connect the trigger input to pin 2 of 555 timer form the function generator.
3. Connect the CRO probes to pin 3 and 2 to display the output signal and the voltage across the timing capacitor. Set suitable voltage sensitively and time-base on the CRO.
4. Switch on the power supply to CRO and the circuit.
5. Observe the waveforms on the CRO and draw to scale on a graph sheet. Measure the voltage levels at which the capacitor starts charging and discharging, output high and low timings along with trigger pulse.

Questions:

1. What are the features of 555 timer?
2. What are the applications of 555 timer?
3. Define duty cycle ratio.
4. What are the applications of monostable Multivibrators?
5. What is meant by quasi stable state?
6. What should be the amplitude of trigger pulse?

Result:

Thus the Astable Multivibrators and Monostable Multivibrators using 555 timer is designed and tested.

EX.No:6

DAC CONVERTORS

Aim:-

To design R-2R ladder type DAC using op-amp.

Components Required:-

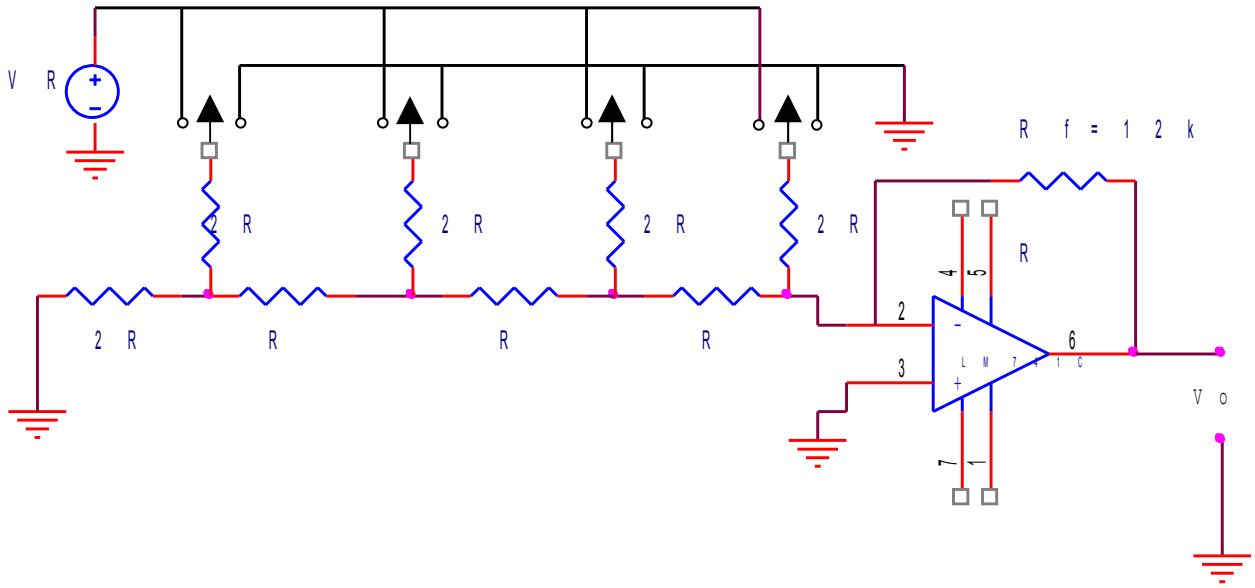
S.No	Components	Range	Quantity
1.	Op-amp	IC 741	1
2.	Resistors	10KΩ ,20KΩ	1
3.	DPDT(switch)		1
4.	Dual Tracking Supply	(0-30)V	1
5.	Voltage Source	(0-30)V	1

Theory:-

In R-2R ladder type D to A converter, only two values of resistor is used (i.e. R and 2R). Hence it is suitable for integrated circuit fabrication. The typical values of R are from 2.5KΩ to 10KΩ . In this output voltage is a weighted sum of digital inputs. Since the resistive ladder is a linear network, the principle of super position can be used to find the total analog output voltage for a particular digital input by adding the output voltages caused by the individual digital inputs.

Circuit Diagram:-

4-Bit R/2R Ladder DAC:



Design:-

Output voltage, $V_o = -V_R \frac{R_f}{R}\left[\frac{b_1}{2^1}+\frac{b_2}{2^2}+\frac{b_3}{2^3}+\frac{b_4}{2^4}\right]$

Binary value=1000(given)
Output voltage=6v (given)
Reference resistor =10KΩ (given)
Reference Voltage, V_R=10V (given)

$\therefore$ R_f =12kΩ

Resolution,

$$V = \frac{1}{2^n} \times \frac{V_R}{R} \times R_f$$

$$V = \frac{1}{2^4} \times \frac{10V}{10k\Omega} \times 12k\Omega$$

$$V = 0.75$$

Questions:

1. Mention any two specifications of a DAC.
2. Name any two types of ADC.
3. In a binary ladder network of a DAC, the value of the smaller resistance is 10 kΩ .What is the resistance value of the other set?
4. What output voltage would be produced by a DAC whose output range is 0 to 10V and whose input binary number is 10 (for a 2 – bit DAC)?
5. What is the range value for resistor (R) in DAC?

Procedure:-

1. Connections are given as per the circuit diagram.
2. The power supply is switched on.
3. Reference voltage is set as 10V.
4. Binary values are applied according to the binary input values.
5. The output voltage is noted down.
6. The output voltage obtained is compared with the given output voltage.

Result:-

Thus the R-2R ladder type DAC was designed using Op-amp.

EX.No:7

FREQUENCY RESPONSE OF 2nd ORDER LPF & HPF

Aim:-

To design and test the frequency response of a second order LPF and HPF.

Components Required:-

S.No	Components	Range	Quantity
1.	Op-amp	IC 741	1
2.	Resistors		
3.	Capacitor	O.01μ f	2
4.	CRO		1
5.	Power Supply	± 15V	1
6.	Probe		2
7.	Bread Board		1

Theory:-

LPF:-

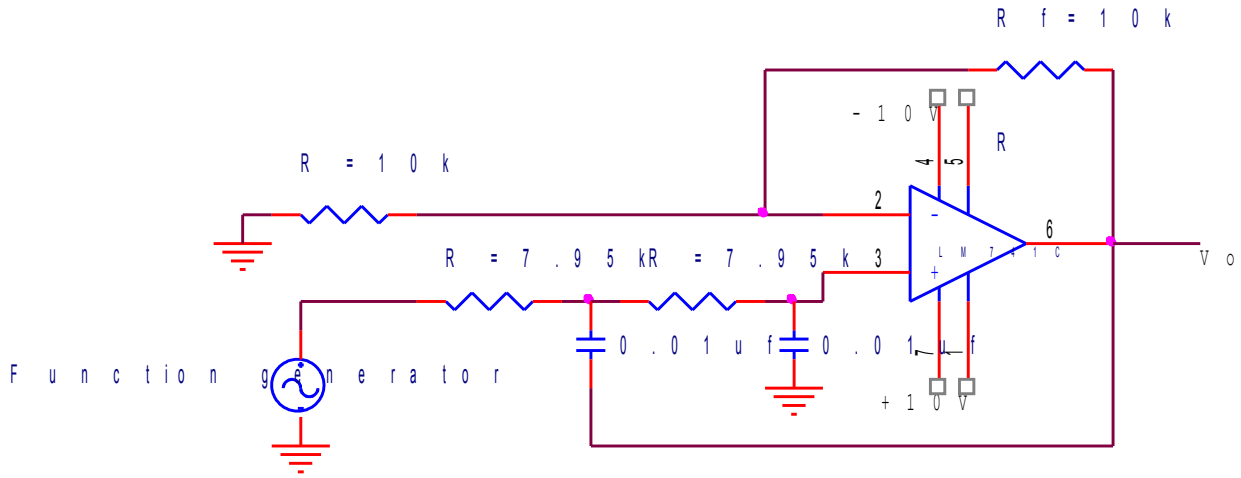
A LPF allows only low frequency signals up to a certain break-point f_H to pass through, while suppressing high frequency components. The range of frequency from 0 to higher cut off frequency f_H is called pass band and the range of frequencies beyond f_H is called stop band.

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 ≤1μ 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:-

Second Order LPF:



Design:-

Second order:-

Given frequency, f_H = 2 KHz and gain =2

Let C=0.01μ f

The frequency, $f_H = \frac{1}{2\pi\sqrt{\left(2\times10^{3}\right)\left(0.01\times10^{-6}\right)}}$

Set, $R_2 = R_3 = R$

$C_2 = C_3 = C$

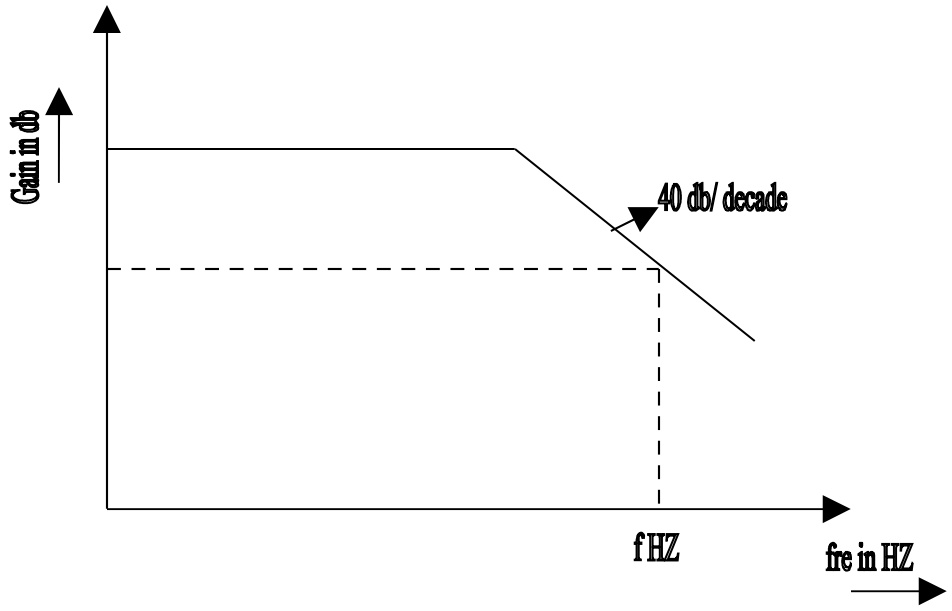
$$\therefore f_H = \frac{1}{2\pi RC}$$

Tabulation

Second order LPF Vin=1V

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

Model graph:-

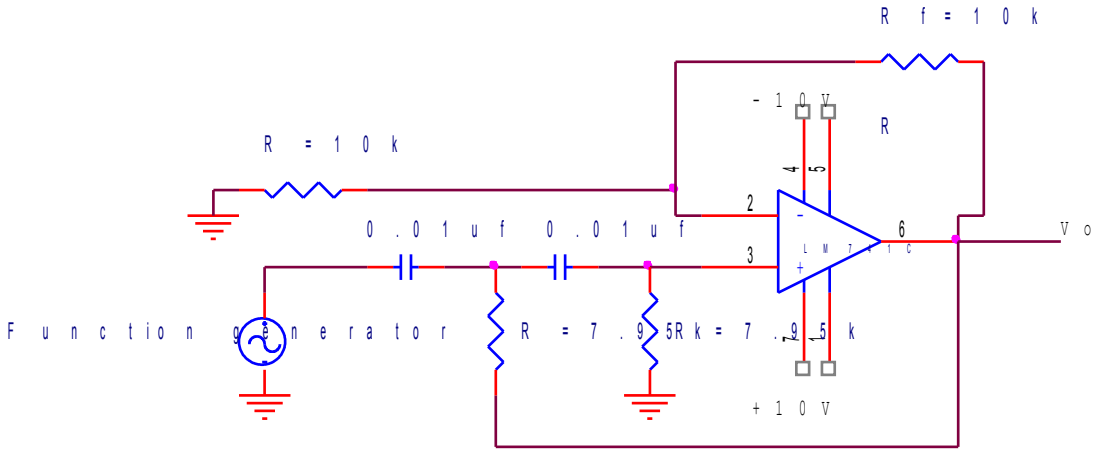


Second order HPF:

Theory:-

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.

Circuit Diagram:-
Second Order HPF:



Design:-

$$f_L = 2\mathbf{K}HZ$$

$$C = 0.01\mu F$$

$$Gain\ ,\ Av = 2$$

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

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

$$R_2 = R_3 = \frac{1}{2\pi fLC}$$

$$R_2 = R_3 = 7.95\,k\Omega$$

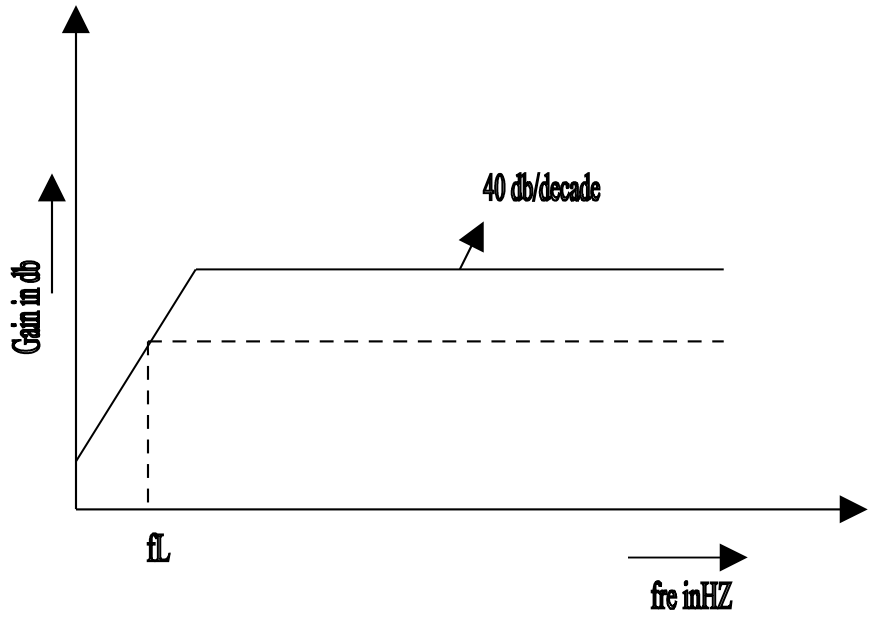
$$A = 1 + \frac{R_f}{R_1} = 2$$

$$\therefore R_f = R_1 = 10\,k\Omega(given\ )$$

Vin=1V

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

Model graph:-



Procedure:-

LPF:-

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.

HPF

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.

Result:-

Thus the second order Low pass filter and High pass filter were designed using Op-amp and its cut off frequency was determined.

EX.No:8

FREQENCY RESPONSE OF 2nd ORDER BSF & BPF

Aim:-

To design and test the frequency response of a second order LPF and HPF.

Components Required:-

S.No	Components	Range	Quantity
1.	Op-amp	IC 741	3
2.	Resistors		
3.	Capacitor	O.01μ f, O.05μ f	2
4.	CRO		1
5.	Power Supply	± 15V	1
6.	Probe		2
7.	Bread Board		1

Theory:-

BSF:-

BSF is the logical inverse of band pass filter which does not allows a specified range of frequencies to pass through. It has two pass bands in the range of frequencies between 0 to f_L and beyond f_H. The band between f_L and f_H is called stop band. BSF is also called Band Reject Filter (BRF) or Band Elimination Filter (BEF).

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

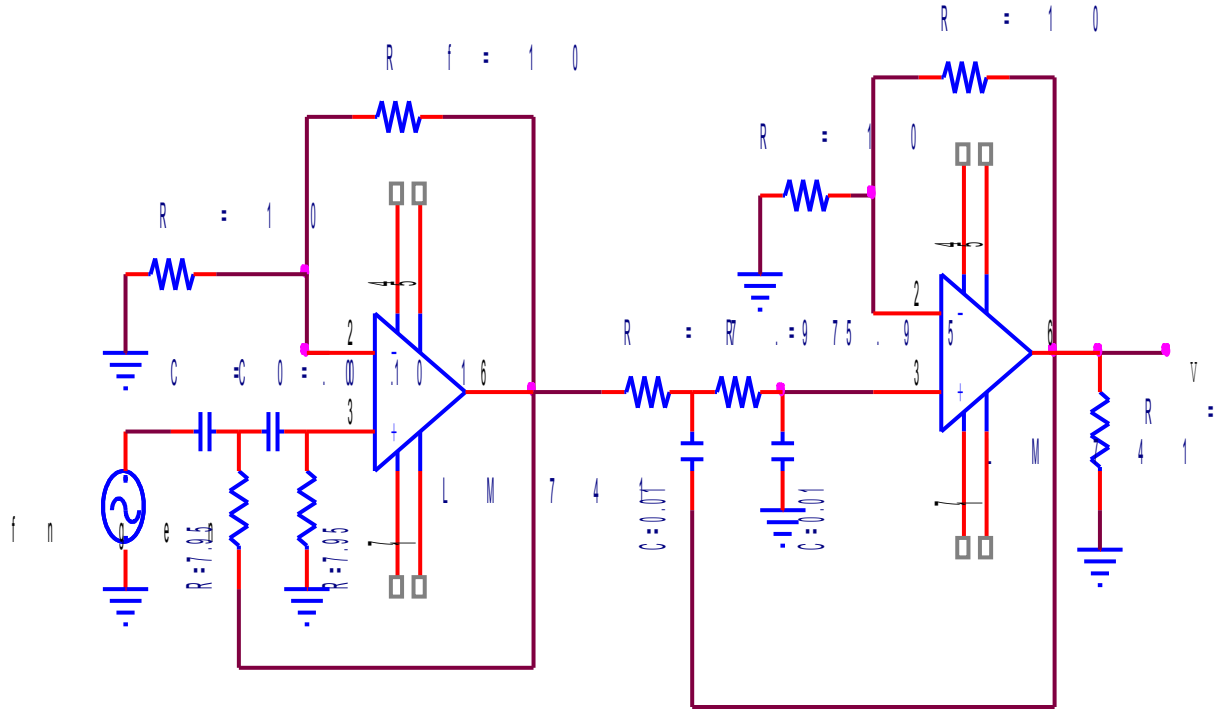
$$f_r = \sqrt{f_H f_L}$$

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

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

Circuit Diagram:-

BPF



Design:-
BSF:-

f_H=200Hz
f_L=1kHz

Low pass section:-

f_H=200Hz
Let C^1=0.05μ f
Then,

$$R^1 = \frac{1}{2\pi f_H c^1}$$

$$R^1 = \frac{1}{2\pi(200)(0.05 \times 10^{-6})}$$

$$R^1 = 15.9K\Omega$$

$$C^1 = 0.05\mu f$$

High Pass Section:-

$f_L = 1\text{KHZ}$

$C = 0.01\mu f$

$$R = \frac{1}{2\pi f_L C}$$

$$= \frac{1}{2\pi(1\times10^{3})(0.01\times10^{-6})}$$

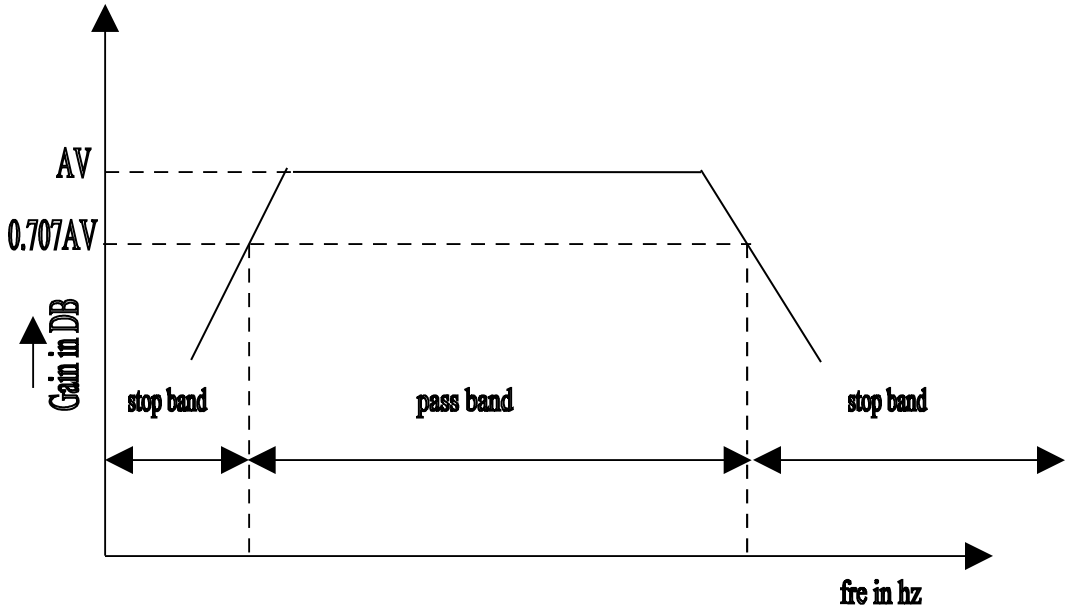
$R = 15.9\text{K}\Omega$

Gain, Av=2 for each section

$\therefore R_1 = R_f = R_1^1 = R_f^1 = 10\,\text{K}\Omega$

Model graph:-

BPF:-



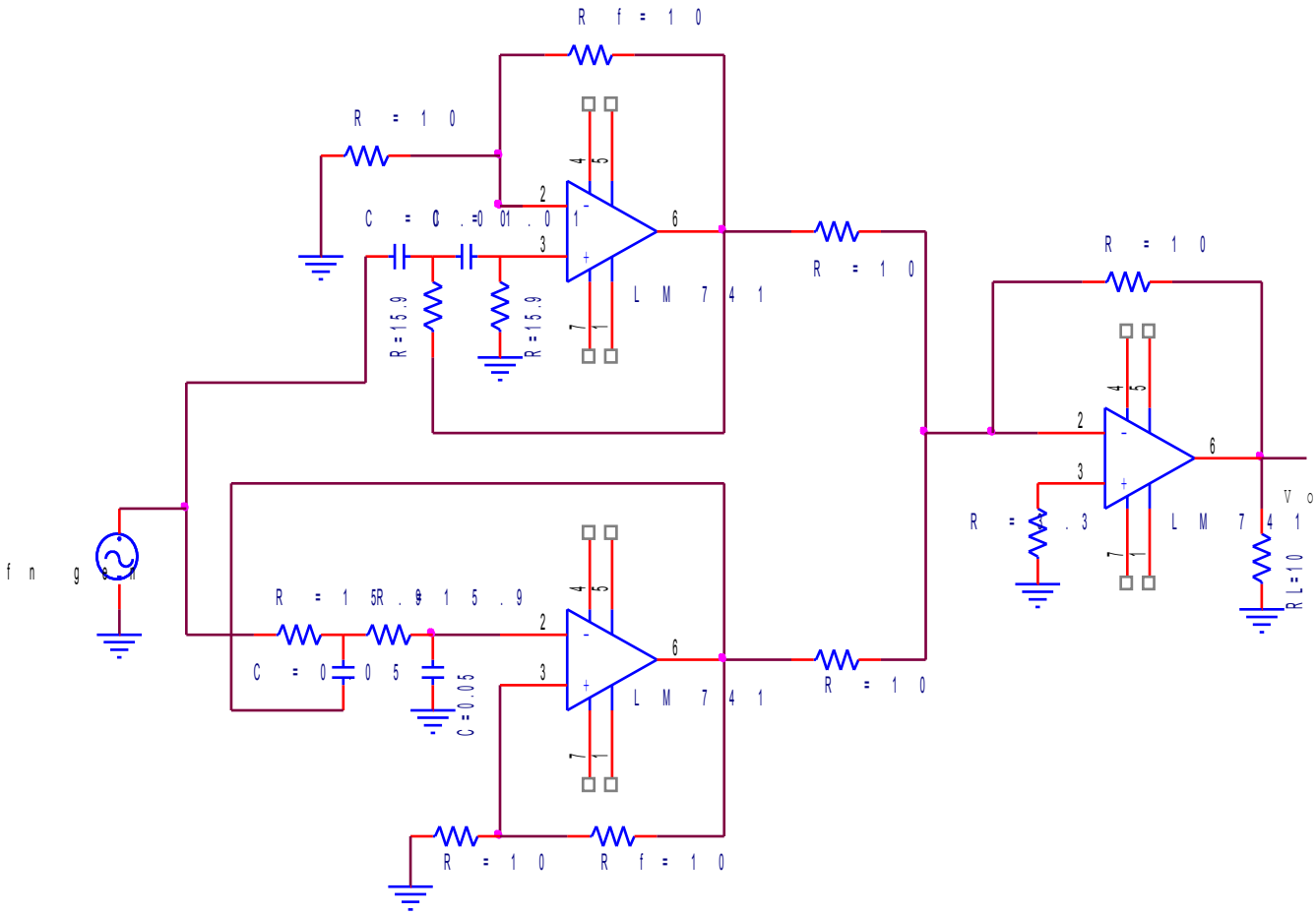
Tabulation:-

BPF Vin=50mv

S.No	Frequency (Hz)	Vo(volts)	Gain=20log(Vo/Vin)

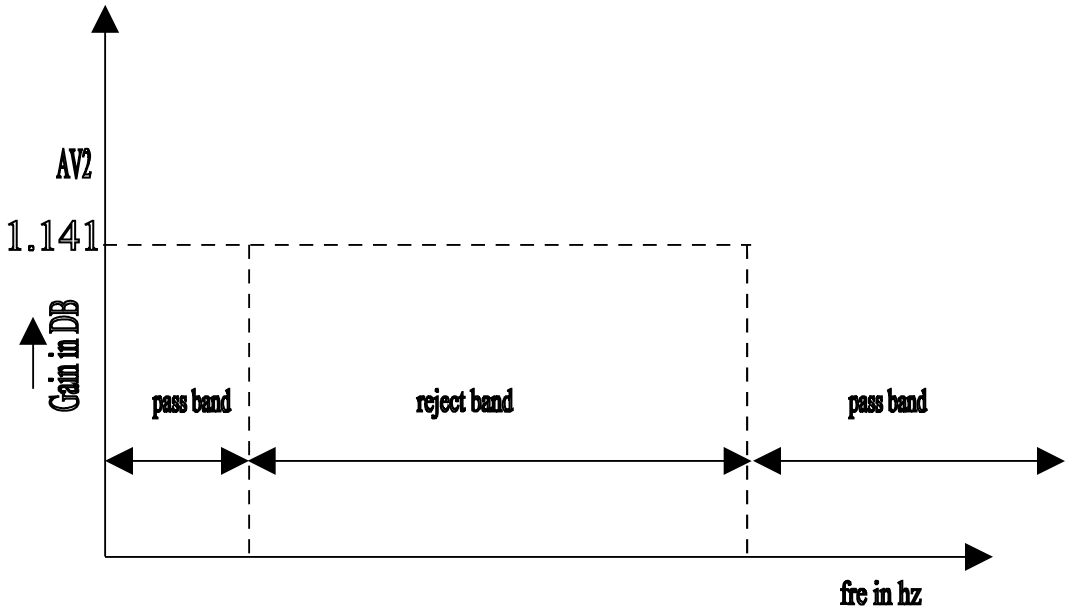
Circuit Diagram:-

BSF



Model graph:-

BSF:-



Tabulation:-

BSF

Vin=50mv

S.No	Frequency (Hz)	Vo(volts)	Gain=20log(Vo/Vin)

Procedure:

BSF,BPF:-

1. The input signal is connected to the circuit from the signal generator.
2. The input and output signals are connected to the filter.
3. The suitable voltage is selected.
4. The correct polarity is checked.
5. The steps are repeated.

Result:-

Thus the frequency response of second order BPF and BSF filter was designed and tested.

EX.No:9

APPLICATION OF MULTIPLIER

Aim:-

To test the applications such as Analog divider, Squarer and square rooter using the multiplier IC.

Components Required:-

S.No	Component	Range	Quantity
1.	Multiplier IC	AD 633	1
2.	Op-amp	IC 741	1
3.	Resistor		
4.	Powersupply	± 15V	1
5.	Bread Board		1
6.	CRO		1

Theory:-

Voltage Squarer:-

The inputs can be positive or negative represented by any corresponding voltage level between o and 10V. The input voltage V_i is to be squared is connected to both the input terminals (ie $V_x=V_y=V_i$) and the output is $V_o=KV_i^2$.

Voltage Divider:-

The voltage divider circuit is constructed by using a multiplier and an Op-amp. This circuit produces the ratio of two input signals. The division is achieved by connecting the multiplier in the feedback loop of an Op-amp. The voltages V_{den} and V_{num} represent the two input voltages. V_{den} form one input of the multiplier and output of Op-amp V_{OA} forms the second input. The output V_{om} of the multiplier is connected back to the inverting input terminal of Op-amp in the feedback loop.

Square Rooter:-

The square root of a signal is determined by connecting both inputs of the multiplier to the output of the Op-amp. Then, the output voltage of the multiplier V_{om} is equal in magnitude but opposite in polarity to V_i. The input voltage V_i must be negative otherwise the Op-amp saturates.

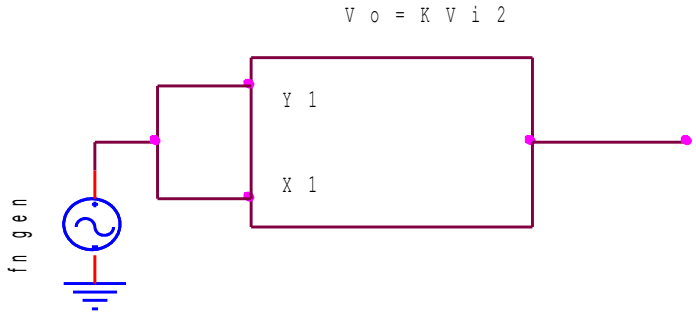
The range of V_i is b/w-1v and -10v

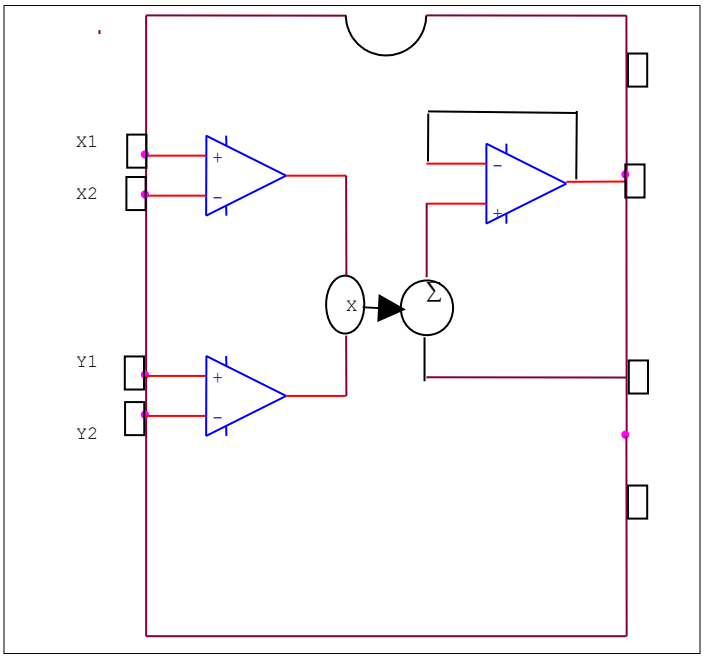
$$V_o = \sqrt{10\,|V_i|}$$

Thus, the V_O is equal to the square root of 10times the absolute magnitude of V_i.

Circuit Diagram:-

Voltage squarer:-



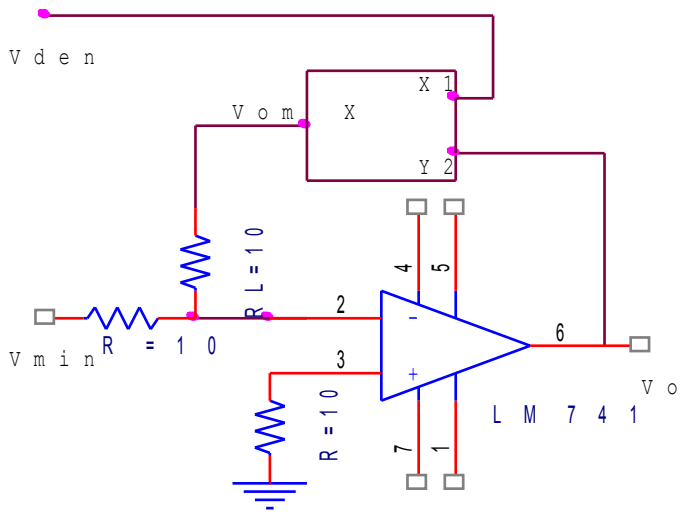


Observation:-

Input voltage amp time=

Output voltage amp time=

Voltage divider:-



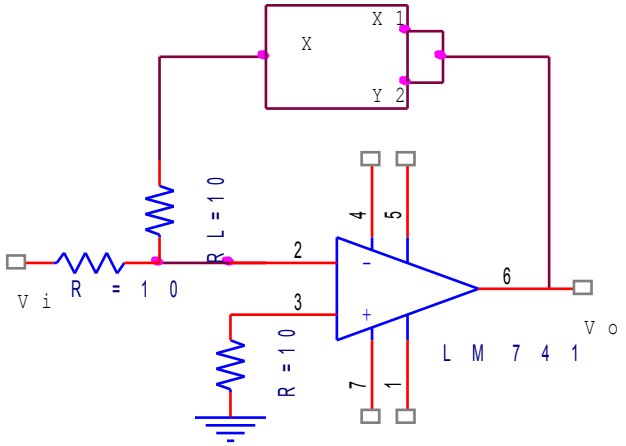
Output VoA=-Vmin/Vden

Observation:-

Vmin voltage amp time=

Vden voltage amp time=

Square rooter:-



$$v_o = \sqrt{10\,|Vi|}$$

Observation:-

Input voltage amp time=

Output voltage amp time=

Procedure:

Voltage Squarer:-

1. Connections are given as per the circuit diagram.
2. Input is connected to the input terminals
3. Supply is connected to the corresponding terminals.
4. Output is noted down.

Voltage Divider:-

1. Connections are given as per the circuit diagram.

2. Input is applied to the input terminals.
3. Power supply is connected to the corresponding terminals.
4. Output is noted down.

Square Rooter:-

1. Connections are given as per the circuit diagram.
2. Input is applied to the input terminals.
3. Power supply is connected to the corresponding terminals.
4. Output is noted down.

Result:-

Thus the voltage Squarer, Voltage Divider and Square Rooter were tested using multiplier IC.

EXP.NO.10:-

OSCILLATORS USING OPERATIONAL AMPLIFIER

Aim:

To design the following sine wave oscillators

a) Wein Bridge Oscillator with the frequency of 1 KHz.
b) RC Phase shift oscillator with the frequency of 200 Hz.

Components Required:

S.No	Components	Range	Quantity
1.	Op-amp	IC 741	1
2.	Dual trace supply	(0-30) V	1
3.	Function Generator	(0-2) MHz	1
4.	Resistors		
5.	Capacitors		
6	CRO	(0-30) MHz	1
7	Probes	--	--

Equations Related to the Experiments:

a) Wein Bridge Oscillator

Closed loop gain $A_v = (1+R_f/R_1) = 3$

Frequency of Oscillation $f_a = 1/(2\pi\ RC)$

b) RC Phase shift Oscillator:

Gain $A_v = [R_f/R_1] = 29$

Frequency of oscillation $f_a = 1\sqrt{6} * 2 * \pi * RC$

1) Wein Bridge Oscillator:

Design:

Gain required for sustained oscillation is $A_v = 1/\beta = 3$

(PASS BAND GAIN) (i.e.) $1+R_f/R_1 = 3$

$\therefore\ R_f = 2R_1$

Frequency of Oscillation $f_o = 1/2\pi\ R\ C$

Given $f_o = 1$ KHz

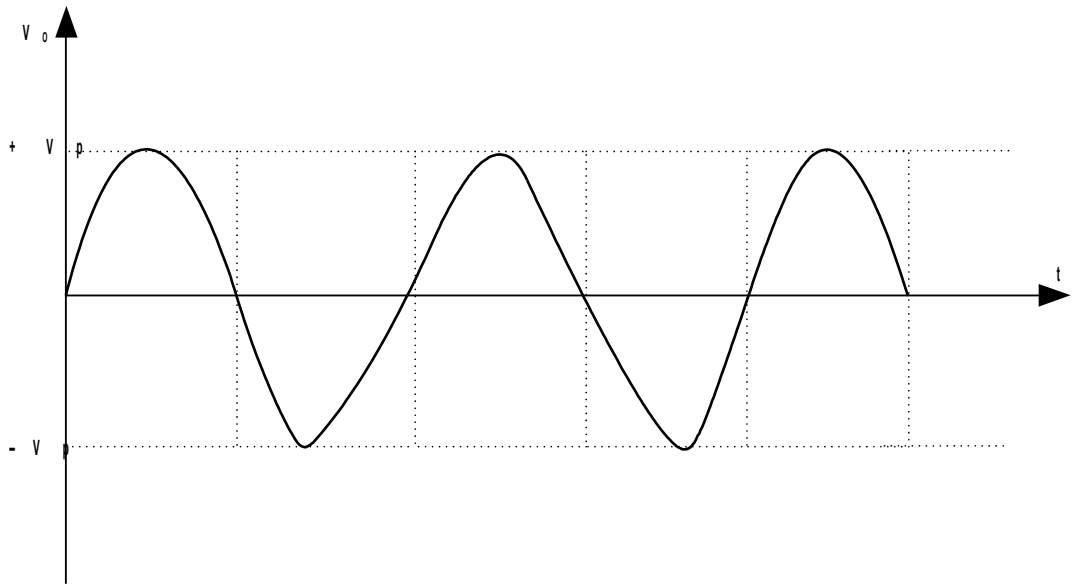
Let C = 0.05 μ F

$\therefore\ R = 1/2\ \pi\ f_oC$

R = 3.2 KΩ

Let R1 = 10 KΩ $\quad\quad\quad\quad\quad\quad\quad\quad$ $\therefore$ Rf = 2 * 10 KΩ

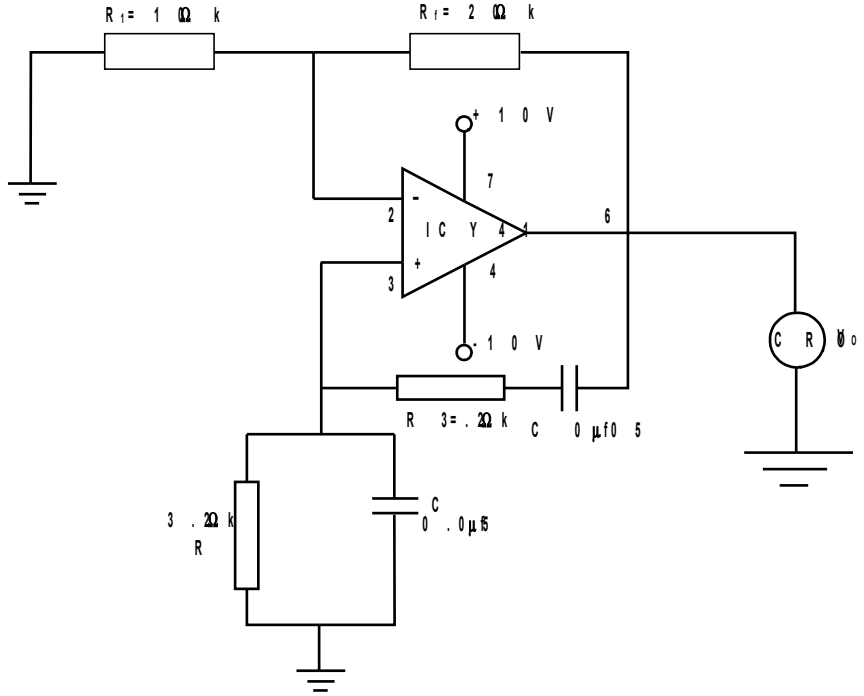
Model Graph:



Procedure:

1. Connect the components as shown in the circuit 5.1

Circuit 5.1:



2. Switch on the power supply and CRO.
3. Note down the output voltage at CRO.
4. Plot the output waveform on the graph.
5. Redesign the circuit to generate the sine wave of frequency 2KHz.
6. Compare the output with the theoretical value of oscillation.

Observation:

Peak to peak amplitude of the output = Volts.
Frequency of oscillation = Hz.

Questions:

1. State the two conditions for oscillations.

2. Classify the Oscillators?
3. Define an oscillator?
4. What is the frequency range generated by Wein Bridge Oscillator?
5. What is frequency stability?

2) RC Phase Shift Oscillators:

Design:

Frequency of oscillation fo = 1/(√6*2*Π *RC)

Av = [Rf/R1] = 29

R_1 = 10 R

R_f = 29 R_1

Given fo = 200 Hz.

Let C = 0.1μ F

$$R = 1/\left(\sqrt{6} * 2\pi * fo * C\right)$$

$$= 1/\left(\sqrt{6} * 2 * \pi * 200 * 0.1 * 10^{-6}\right)$$

= KΩ

To prevent the loading of amplifier by RC network, R1 ≥ 10R

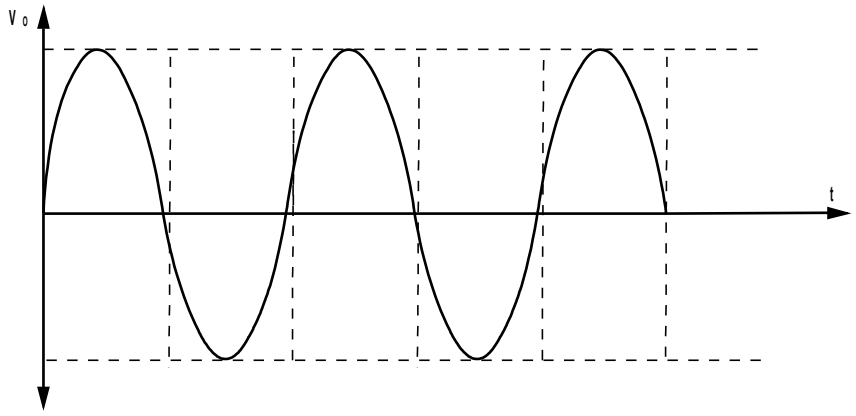
$$\therefore R1 = 10 * \text{------} = K\Omega$$

Since Rf = 29R1

$$Rf = 29 * \text{------}$$

= MΩ

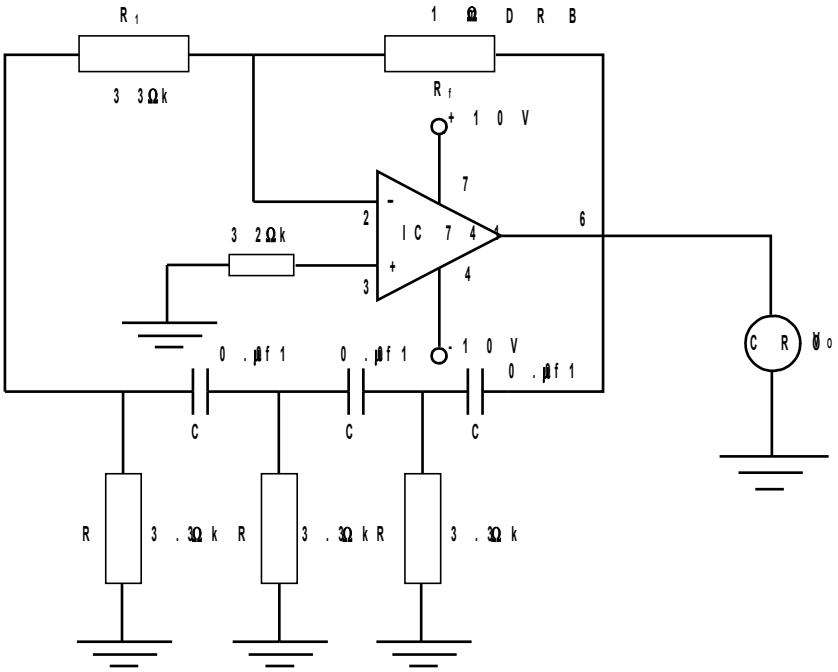
Model Graph:



Procedure:

1. Connect the circuits as shown in the circuit 5.2
2. Switch on the power supply.

Circuit 5.2:



3. Note down the output voltage on the CRO.
4. Plot the output waveforms on the graph.
5. Redesign the circuit to generate the sine wave of 1 KHz.
6. Plot the output waveform on the graph.
7. Compare the practical value of the frequency with the theoretical value.

Observation:

Peak to peak amplitude of the sine wave = Volts

Frequency of Oscillation (obtained) = Hz.

Questions:

1. What is the frequency range generated by RC phase shift Oscillator?
2. In RC phase shift oscillator how the total phase shift of 180° around the loop is achieved?

Result:

Thus wien bridge oscillator and RC Phase shift oscillator was designed using op-amp and tested.

EXP.NO.11:- VOLTAGE REGULATION USING IC LM723

AIM :

To design a high current, low voltage and high voltage linear variable dc regulated power supply and test its line and load regulation.

COMPONENTS REQUIRED :

S.NO	COMPONENTS	SPECIFICATION	QUANTITY
1.	Transistors	TIP122,2N3055	1 each
2.	Integrated Circuit	LM723	1
3.	Digital Ammeter	(0 – 10) A	1
4.	Digital Voltmeter	(0 – 20) V	1
5.	Variable Power Supply	(0 – 30) V-2A	1
6.	Resistors	300Ω ,430Ω ,1KΩ ,678KΩ ,67 8Ω 1Ω	1 each 2
7.	Capacitors	0.1µ F,100pF	1 each
9.	Rheostat	(0 – 350) Ω	1

CIRCUIT DIAGRAM : Low Voltage Regulator

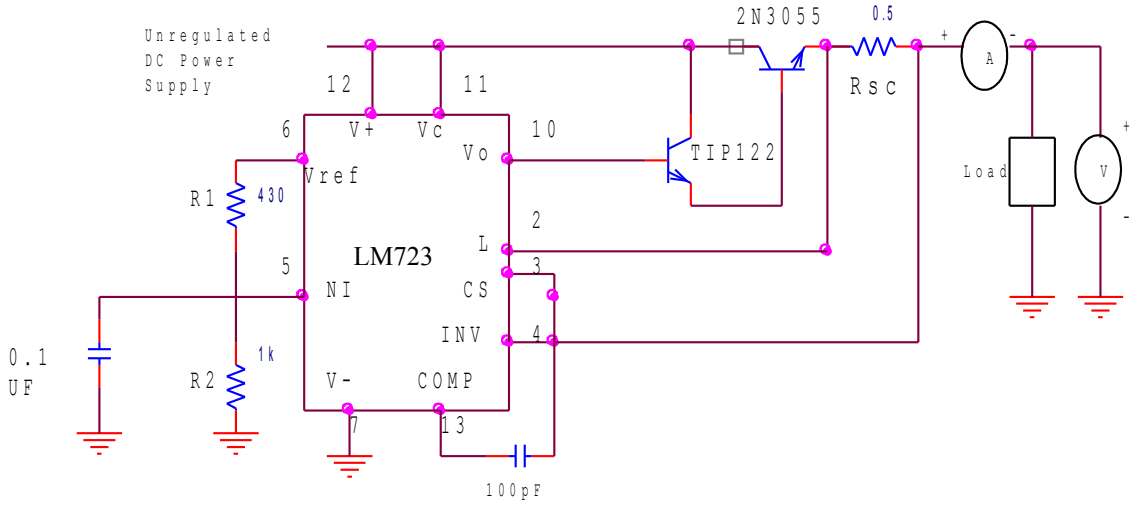


Fig. 1.1

DESIGN:

Output voltage → V_O

Reference voltage→ Vref

Rprotect → Minimum Resistance to protect the output from short circuit.

Low Voltage Regulator :

Given : Vo=5V, Vref = 7.15 V

To calculate R1, R2 ,R3 and Rsc.

Vo = Vref (R2 / (R1 + R2))

5 / 7.15 = (R2 / (R1 + R2))

(R1 + R2) 0.699= R2

0.699R1 = 0.301 R2 , R1 = 0.4306 R2

Select **R2 = 1 KΩ**

R1 = 1 KΩ * 0.4306 = 430Ω

R1 = 430Ω

R3 = R1 * R2 / (R1 + R2) , R3 = 430.6 *1000 /(430.6+1000)

R3 = 300Ω

Rsc = V_{sense} / I_{limit} = 0.5 /1A = 0.5Ω , **Rsc = 0.5Ω**

CIRCUIT DIAGRAM : High Voltage Regulator :

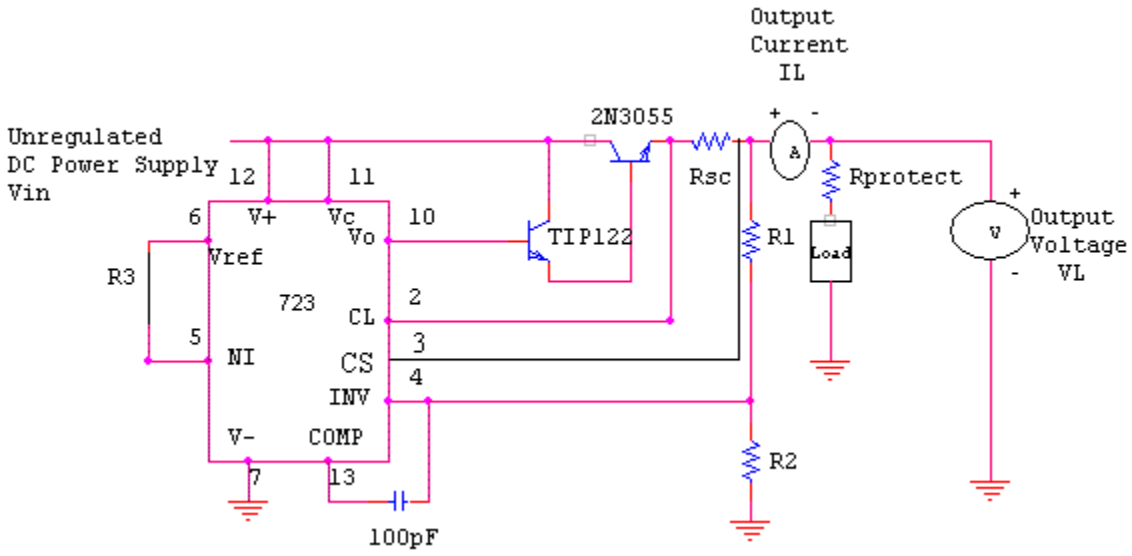


Fig. 1.2

High Voltage Regulator :

Given : Vo=12V, Vref = 7.15 V

To calculate R1, R2 ,R3 and Rsc.

Vo = Vref (1 + (R1 / R2))

12 / 7.15 = 1+ (R1 / R2)

(12 / 7.15) - 1 = (R1 / R2)

(R1 / R2) = 0.678

Select **R2 = 1 KΩ**

R1 = 1 KΩ * 0.678 = 678Ω

R1= 678Ω

$Rsc = V_{sense} / I_{limit} = 0.5 /1A = 0.5\Omega$

Rsc = 0.5Ω

Tabulation of the Measurements :

LOW VOLTAGE REGULATOR :

Line Regulation :

S.No.	Load Resistance R_{L1} =		Load Resistance R_{L2} =		Load Resistance R_{L3} =	
	Input Voltage Vin(Volts)	Output Voltage V_L(Volts)	Input Voltage Vin(Volts)	Output Voltage V_L (Volts)	Input Voltage Vin(Volts)	Output Voltage V_L (Volts)

Load Regulation :

S.No.	Input Voltage V_{in1} =		Input Voltage V_{in2} =		Input Voltage V_{in3} =	
	Output Current I_L (A)	Output Voltage V_L (Volts)	Output Current I_L (A)	Output Voltage V_L (Volts)	Output Current I_L (A)	Output Voltage V_L (Volts)

HIGH VOLTAGE REGULATOR :

Line Regulation :

S.No.	Load Resistance R_{L1} =		Load Resistance R_{L2} =		Load Resistance R_{L3} =	
	Input Voltage Vin(Volts)	Output Voltage V_L(Volts)	Input Voltage Vin(Volts)	Output Voltage V_L (Volts)	Input Voltage Vin(Volts)	Output Voltage V_L(Volts)

Load Regulation :

S.No.	Input Voltage V_{in1} =		Input Voltage V_{in2} =		Input Voltage V_{in3} =	
	Output Current I_L (A)	Output Voltage V_L(Volts)	Output Current I_L (A)	Output Voltage V_L(Volts)	Output Current I_L (A)	Output Voltage V_L(Volts)

Calculation of % Voltage Regulation :

% Voltage Regulation = (V_{dc} (NL) - V_{dc} (FL)) / V_{dc} (FL)

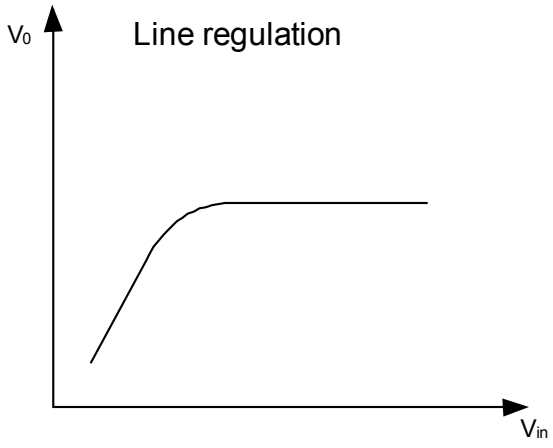
V_{dc} (NL) = D.C. output voltage on no load

V_{dc} (FL) = D.C. output voltage on full load

Model Graph :

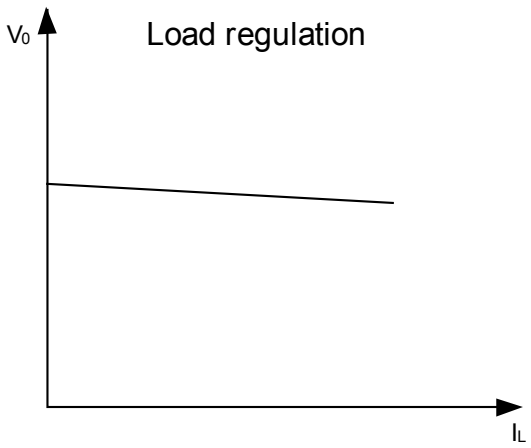
Line Regulation :

Input Voltage Vs Output Voltage :



Load Regulation :

Output Current Vs Output Voltage



PROCEDURE :

LOW VOLTAGE REGULATOR :

Line Regulation :

1. Give the circuit connection as per the circuit diagram shown in Fig 1.1.
2. Set the load Resistance to give load current of 0.25A.
3. Vary the input voltage from 7V to 18V and note down the corresponding output voltages.
4. Similarly set the load current (I_L) to 0.5A & 0.9A and make two more sets of measurements.

Load Regulation :

1. Set the input voltage to 10V.

2. Vary the load resistance in equal steps from 350Ω to 5Ω and note down the corresponding output voltage and load current.

3.Similarly set the input voltage (Vin) to 14V & 18V and make two more sets of measurements.

Lab Report :

1.Plot the line regulation by taking Input Voltage (Vin) along X-axis and Output Voltage (V_L) along Y-axis for various load currents.

2.Plot the load regulation by taking load current (I_L) along X-axis and Output Voltage (V_L) along Y-axis for various input voltages.

3.Calculate its % Voltage Regulation using the formula.

HIGH VOLTAGE REGULATOR :

Line Regulation :

1.Give the circuit connection as per the circuit diagram shown in Fig 1.2.

2.Set the load Resistance to give load current I_L of 0.25A.

3.Vary the input voltage from 7V to 18V and note down the corresponding output voltages.

4.Similarly set the load current (I_L) to 0.5A & 0.9A and make two more sets of measurements.

Load Regulation :

1. Set the input voltage to 10V.

2. Vary the load resistance in equal steps from 350Ω to 15Ω and note down the corresponding output voltage and load current.

3.Similarly set the input voltage (Vin) to 14V & 18V and make two more sets of measurements.

Lab Report :

1.Plot the line regulation by taking Input Voltage (Vin) along X-axis and Output Voltage (V_L) along Y-axis for various load currents.

2.Plot the load regulation by taking load current (I_L) along X-axis and Output Voltage (V_L) along Y-axis for various input voltages.

3.Calculate its % Voltage Regulation using the formula.

Result :

Thus the line and load regulation of a high current, low voltage and high voltage linear variable dc regulated power supply was designed and tested.

S.No	Low Voltage Regulator	High Voltage Regulator
% Voltage Regulation		

Questions :

i) Why minimum protect resistance in load is required? What will happen if it is not there?

ii) Did you short circuit the output and check whether the short circuit protection is working?

iii) What will you do if you are asked to design both high and low voltage regulators in one circuit?

iv) Give 10 example applications of the above circuits?

v) Why do you use a 100pF capacitor between 13 & 3,4?