LINEAR IC APPLICATIONS LAB

B.Tech, ECE, V-Semester, R20 Regulation

PREPARED BY
Dr. B.Y.V.N.R. Swamy
Mr. P.Venkateswara Rao

&

VERIFIED BY
Dr. Y.S.V. Raman

Department of Electronics and Communications Engineering,
Lakireddy Bali Reddy College of Engineering,
L.B. Reddy Nagar, Mylavaram – 521 230

B.Tech. (VSem.) R20 -Regulation	PC: LINEAR IC APPLICATIONS LAB	0	0	3	1.5

Pre-requisites: Fundamentals of Electronic Devices, Analog Circuits.

Course Educational Objective: This course provides the practical exposure on development of Integrated Circuit (IC) based arithmetic applications realization, Design of Active Filters and different waveform generators and design of multivibrators using 555 timer and its applications.

Course Outcomes: At the end of the course, the student will be able to

CO1: Demonstrate the characteristics and applications of Op-Amps.

CO2: Apply the 555 Timer circuit concepts for the realization of waveform generators.

CO3: Design Active filters, arithmetic circuits, waveform generators and data converters using Op-Amp.

CO4: Adapt effective Communication, presentation and report writing skills

LIST OF EXPERIMENTS

(Any of the 10 experiments are required to be conducted)

1. Verification of functionality of Inverting and Non-inverting amplifiers for the Sine and Square wave inputs.
2. Realization of Adder and Subtractor using Op-Amp.
3. Realization of Differentiator and Integrator using Op-Amp.
4. Design, Realization and plot the frequency response of First order Low pass and High pass filters using Op-Amp.
5. Design and plot the frequency response of Band pass filter using Op-Amp.
6. Design and Realization of Op-Amp based Astable multivibrator for Square wave generation.
7. Design and Realization of 555 timer based Monostable multivibrator for Pulse generation.
8. Construction of Schmitt Trigger using Op-Amp and calculate UTP and LTP values.
9. Design and Realization of 555 timer based Astable multivibrator for square wave generation.
10. Design and Realization of RC phase shift Oscillator for sinusoidal signal generation using Op-Amp.
11. Design and Realization of Function generator for square wave and triangular waves using Op-Amps.
12. Design and Realization of Pulse generators using Op-Amp.
13. Design and Realization of 3-bit Digital to Analog converter using Op-Amp.

Verification of Functionality of Inverting and Non-Inverting Amplifiers for the Sine and Square Wave Inputs	EXPT. NO: 1
	DATE:

1. AIM:

To verify the functionality of Inverting and Non-inverting amplifiers for the Sine and Square wave inputs.

2. EQUIPMENTS REQUIRED:

1. Plus – Minus DC Power supply- [(0 - ±15) V, 50 mA]
2. IC741.
3. Bread board.
4. Resistors
5. Function Generator
6. Dual Trace Oscilloscope
7. Digital Multimeter

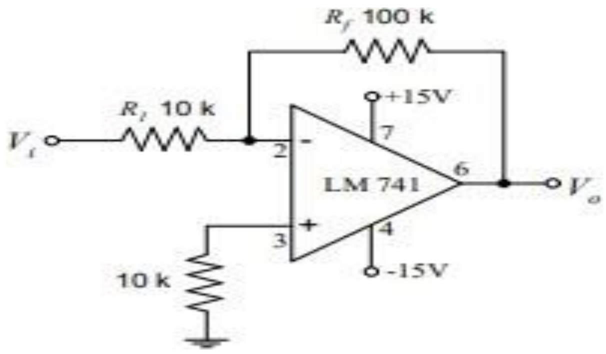
3. THEORY:

Inverting Amplifier: In inverting amplifier input signal V_i is applied to the inverting input terminal through R_1 and non-inverting input terminal is grounded. The output voltage V_o is fed back to the inverting input terminal through R_f- R_1 network where R_f is the feedback resistor.

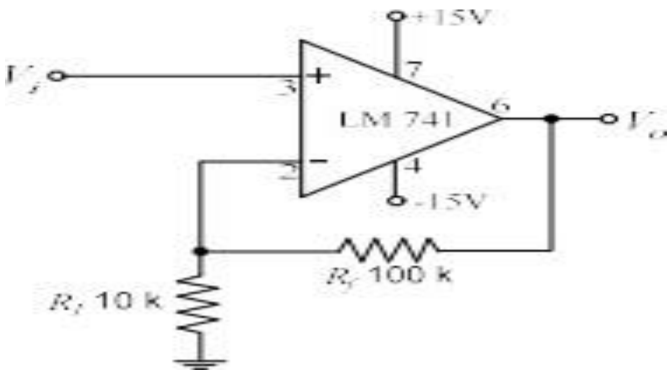
Non-Inverting Amplifier: In Non-inverting amplifier input signal V_i is applied to the non-inverting input terminal. The output voltage V_o is fed back to the inverting input terminal through R_f- R_1 network where R_f is the feedback resistor.

4. CIRCUIT DIAGRAM:

(a) Inverting Amplifier:



(b) Non-Inverting Amplifier:



5. PROCEDURE:

For Inverting Amplifier:

1. Connect the circuit as per the circuit diagram.
2. Apply proper biasing voltages at the corresponding pin numbers of the IC.
3. Select R_1, Rf and R_{comp} as 10K,100K and 10K respectively.
4. Apply the Sine and square input to the inverting terminal and observe the output.

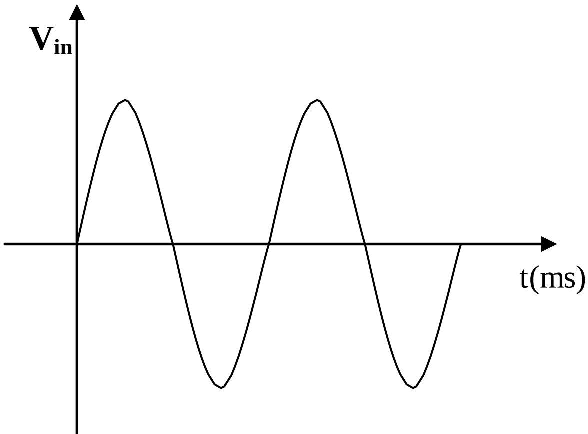
For Non-Inverting Amplifier:

1. Connect the circuit as per the circuit diagram.
2. Apply proper biasing voltages at the corresponding pin numbers of the IC.
3. Select R_1 and R_f as 10K and 100K respectively.
4. Apply the Sine and square input to the Non inverting terminal and observe the output.

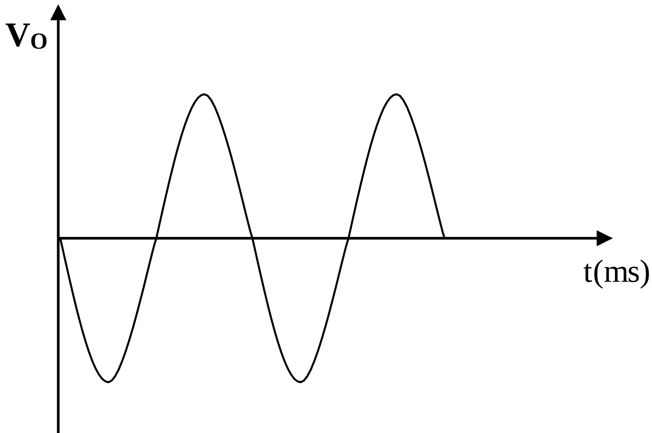
6. EXPECTED GRAPH:

Inverting Amplifier for Sine Wave input:

Input waveform:

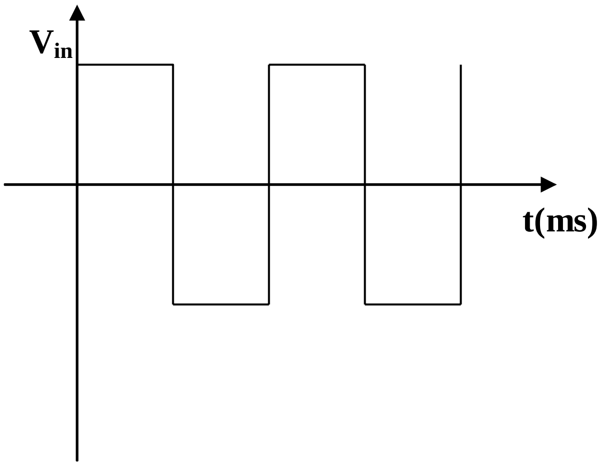


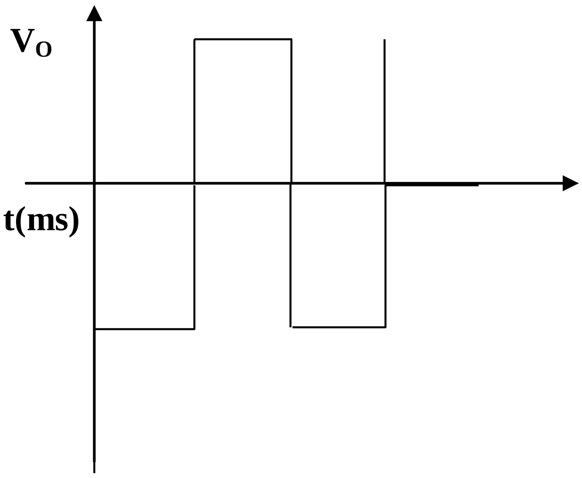
Output waveform:



Inverting Amplifier for Square Wave input:

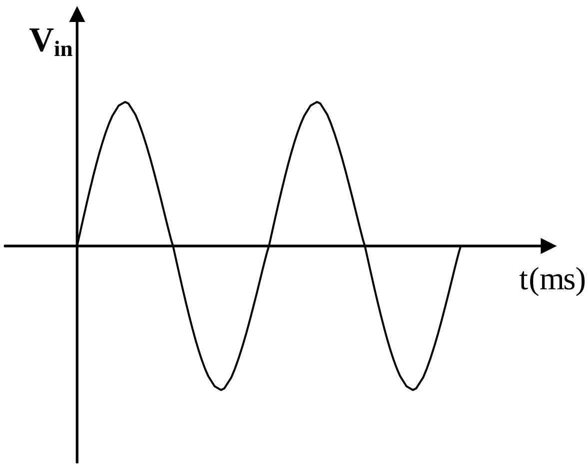
Input waveform: **Output waveform:**

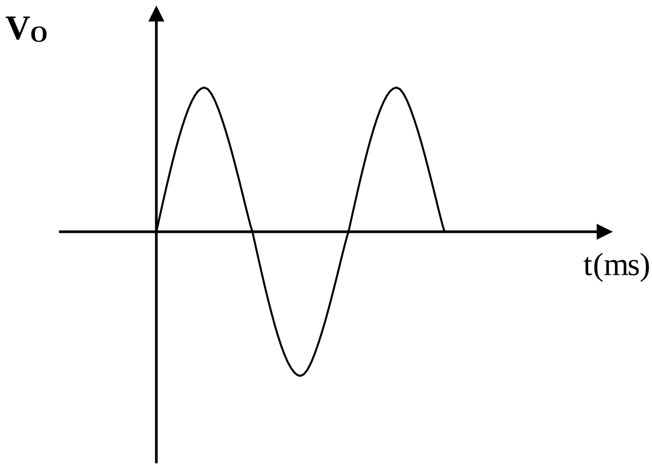




Non-Inverting Amplifier for Sine Wave input:

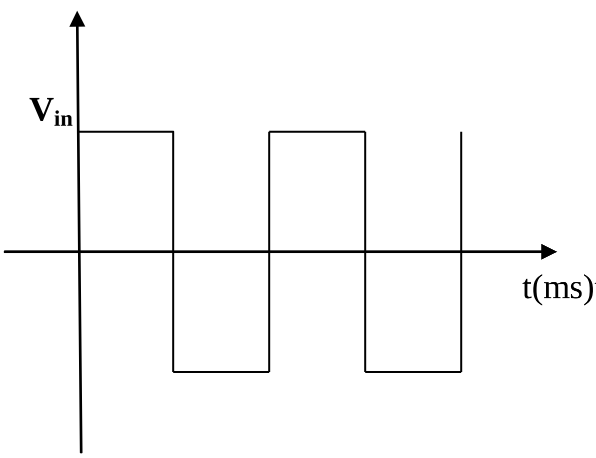
Input waveform: **Output waveform:**

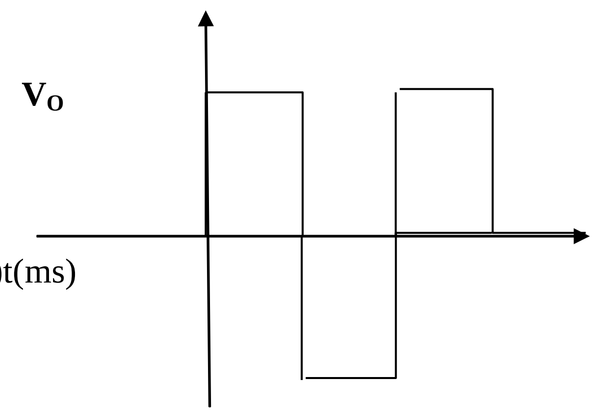




Non-Inverting Amplifier for Square Wave input:

Input waveform: **Output waveform:**





7. PRECAUTIONS:

1. Set power supplies to zero volts before connecting the circuit diagram.
2. Insert Op amp with correct pin numbers.
3. Avoid loose connections.

8. RESULT:

Hence the experiment on Inverting and Non-inverting amplifiers for the Sine and Square wave inputs are observed and verified.

9. VIVA -VOCE QUESTIONS:

1. What is an Op Amp?
2. What are the different applications of an Op amp?
3. What is the main building block of an Op amp?
4. Which type of feedback is present in an inverting amplifier?
5. Which type of feedback is present in a non-inverting amplifier

Realization of Adder and Subtractor using Op-Amp	EXPT. NO: 2
	DATE:

1. AIM: To realize Adder and Subtractor circuits using Op-Amp.

2. EQUIPMENTS REQUIRED:

1. Plus – Minus DC Power supply- [(0 - ±15) V, 50 mA]
2. IC741.
3. Bread board.
4. Resistors
5. Function Generator
6. Dual Trace Oscilloscope
7. Digital Multimeter

3. THEORY:

Operational amplifier may be used to design a circuit whose output is the sum of several input signals. Such a circuit is called a summing amplifier or a summer circuit. A typical summing amplifier has two input voltages and two input resistors R_1, R_2 and a feedback resistance R_f and also a compensating resistor R_{comp}. The expression for two input adder circuit can be written as

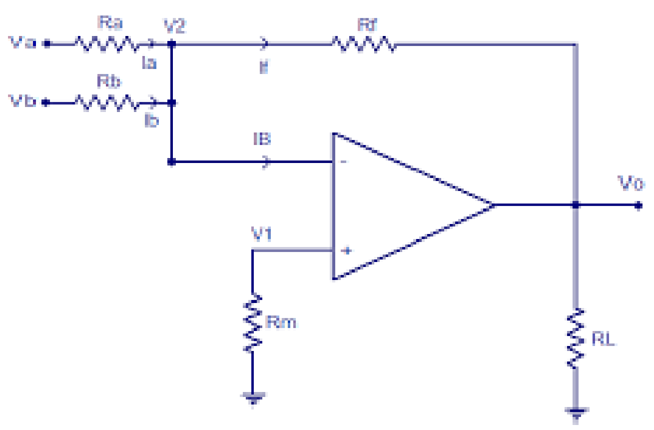
$$V_o = -(V_1+V_2) \text{ if } R_1=R_2=R_f.$$

A basic differential amplifier can be used as a subtractor circuit if all the resistors are equal in value. The expression for a two input subtractor can be written as

$$V_o = (V_1-V_2) \text{ if } R_1=R_2=R_3= R_f.$$

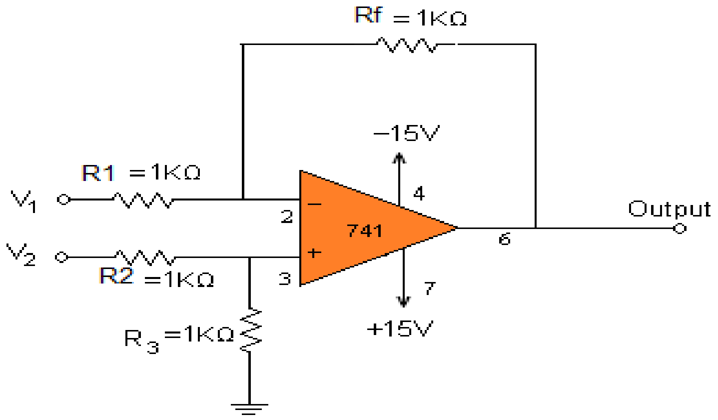
4. CIRCUIT DIAGRAM:

(a) Adder circuit:



Summing amplifier

(b) Subtractor circuit:



5. PROCEDURE:

For Adder circuit:

1. Connect the circuit as per the circuit diagram.
2. Apply proper biasing voltages at the corresponding pin numbers of the IC.
3. The value as considered R_1=R_2=R_f=10k and R_{comp}= R_1//R_2//R_f= 3.3k firstly and for various values of V_1& V_2 we get the output as addition of V_1& V_2.

For Subtractor circuit:

1. Connect the circuit as per the circuit diagram.
2. Apply proper biasing voltages at the corresponding pin numbers of the IC.
3. Select R_1=R_2=R_3=R_f=1k and for various values of V_1& V_2 we get the output as subtraction of V_1& V_2.

6. TABULAR COLUMN:

Adder:

Expected Results:

S.No:	V_1(v)	V_2(v)	V_o (v)
1	1	0.5	
2	2	1	
3	2.5	1.5	

Observed Results:

S.No:	V_1(v)	V_2(v)	V_o (v)
1	1	0.5	
2	2	1	
3	2.5	1.5	

Subtractor:

Expected Results:

S.No:	V_1(v)	V_2(v)	V_o (v)
1	1	0.5	
2	3	1	
3	2.5	2.5	

Observed Results:

S.No:	V_1(v)	V_2(v)	V_o (v)
1	1	0.5	
2	3	1	
3	2.5	2.5	

7. PRECAUTIONS:

1. Set power supplies to zero volts before connecting the circuit diagram.
2. Insert Op amp with correct pin numbers.
3. Avoid loose connections.

8. RESULT:

Hence the experiment on Adder and Subtractor circuits using Op-Amp is performed and their outputs have been verified with theoretical values.

9. VIVA -VOCE QUESTIONS:

1. What is an Op Amp?
2. What are the different applications of an Op amp?
3. What is the main building block of an Op amp?
4. What is differential amplifier?
5. Classify differential amplifier.

Realization of Differentiator and Integrator using Op-Amp	EXPT. NO: 3
	DATE:

1. AIM:

To realize Differentiator and Integrator circuits using Op-Amp for various input waveforms.

2. EQUIPMENTS REQUIRED:

1. Plus – Minus DC Power supply- [(0 - ±15) V, 50 mA]
2. IC741.
3. Bread board.
4. Resistors
5. Capacitors
6. Dual Trace Oscilloscope

3. THEORY:

DIFFERENTIATOR:

One of the simplest of the Op amp circuits that contain capacitor is the differentiating amplifier or differentiator. As the name suggests the circuit performs the mathematical operation of differentiation i.e., the output waveform is the derivative of the input waveform. The differentiator may be constructed from a basic inverting amplifier if an input resistor $R_1$is replaced by a capacitor C_1.

The output voltage V_O is equal to R_FC_1 times the negative instantaneous rate of change of the input voltage V_{in} with time. Since the differentiator performs the reverse of the integrator's function, a cosine wave input will produce a sine wave output or a triangular input will produce a square wave output.

INTEGRATOR:

If we interchange the resistor and capacitor of the Differentiator circuit, we obtain the circuit of an Integrator. As the name suggests the circuit performs the mathematical operation of Integration i.e., the output waveform is the integration of the input waveform.

The output voltage V_O is directly proportional to the negative integral of the input voltage and inversely proportional to the time constant R_1C_f**.** For example, if the input is a sine wave, then the output is cosine wave; or if the input is a square wave, the output will be a triangular wave. The resistor R_f limits the low frequency gain and hence minimizes the variation in the output voltage.

4. DESIGN PROCEDURE:

For Differentiator:

(i) Choose f_a equal to the highest frequency of the input signal. Assume a practical value of C_1 (< 1μF) and then calculate R_f. ($f_a = 1/\ 2\pi R_f C_1$)

(ii) Choose $f_b = 10\ f_a$(say). Now calculate the values of R_1 and C_f so that $R_1C_1 = R_fC_f$ ($f_b = 1/\ 2\pi R_1C_1$)

(iii) A resistor $R_{comp} = (R_1 // R_f)$ is normally connected to non-inverting input terminal to compensate for the input bias current.

For Integrator:

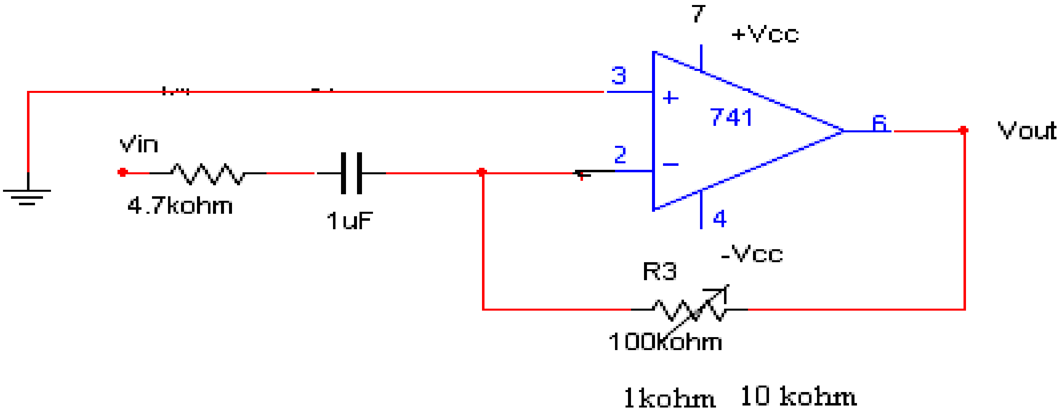
(i) Choose R_fC_f = period of the signal to be integrated = $1/\ f_a$. ($f_a = 1/\ 2\pi R_fC_f$)

(ii) Choose a value of C_1 (<1μF).

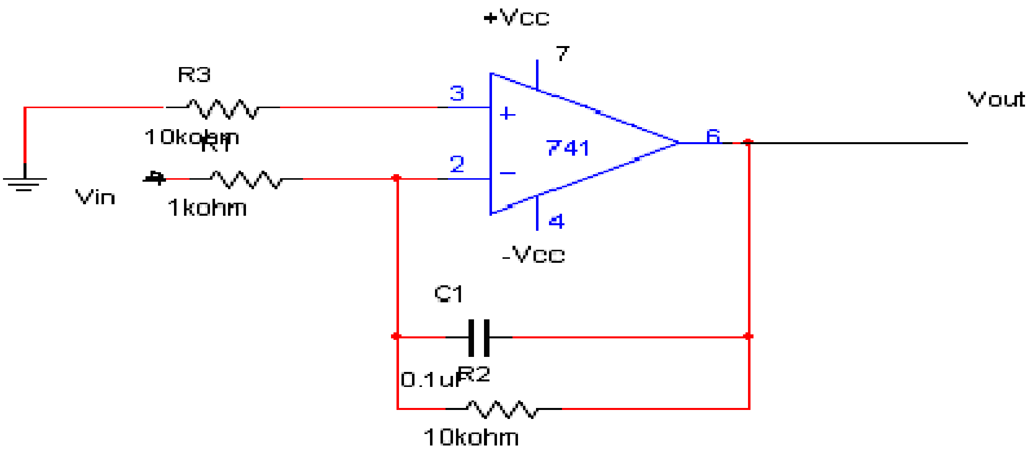
(iii) Choose $R_f = 10\ R_1$.

5. CIRCUIT DIAGRAM:

Differentiator:



Integrator:



6. PROCEDURE:

For Differentiator:

1. Connections are given as per the circuit diagram. Adjust the Function generator to produce a 5-volt peak sine wave at 100 Hz.
2. Observe input voltage V_s and output voltage V_O simultaneously on the oscilloscope. Measure and record the peak value of V_O and the phase angle of V_O with respect to V_s.
3. Repeat step (2) while increasing the frequency of the input signal. Find the maximum frequency at which the circuit performs differentiation. Compare it with the calculated value of f_a.

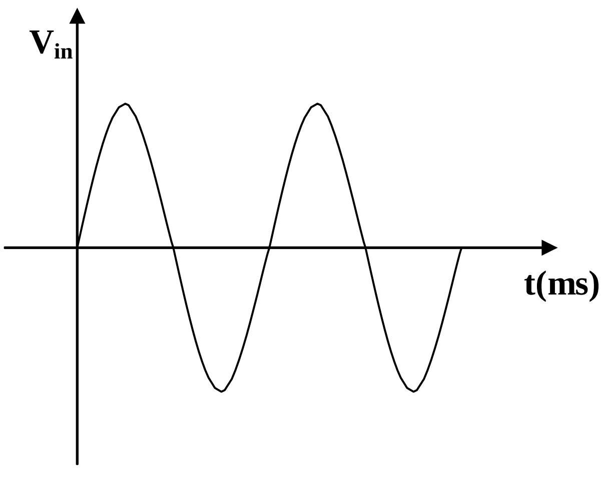
For Integrator:

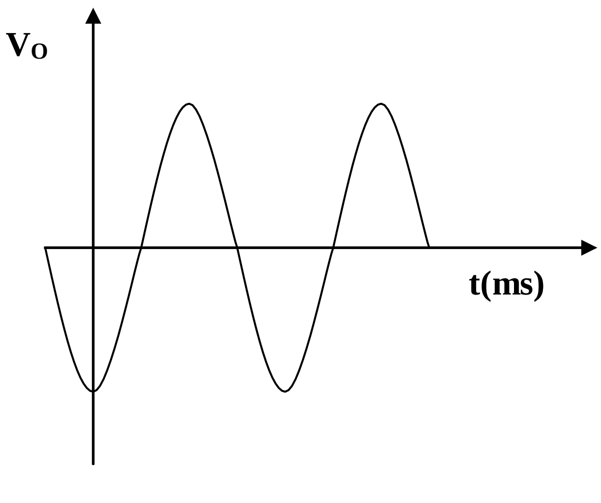
1. Connections are given as per the circuit diagram. Adjust the Function generator to produce a square wave of 1 V peak-to-peak amplitude at 500 Hz. View simultaneously V_O and V_{in}.
2. Slowly adjust the input frequency until the output is a good triangular waveform. Measure the amplitude and frequency of the input and output waveforms.
3. Verify the following relationship between R_fC_f and input frequency.
4. For perfect integration $f > f_a = 1/2\pi R_f C_f$.
5. Now set the function generator to a sine wave of 1 volt peak to peak and frequency of 500 Hz.Adjust the frequency of the input until the output is a negative going cosine wave. Measure the frequency and amplitude of the input and output waveforms.

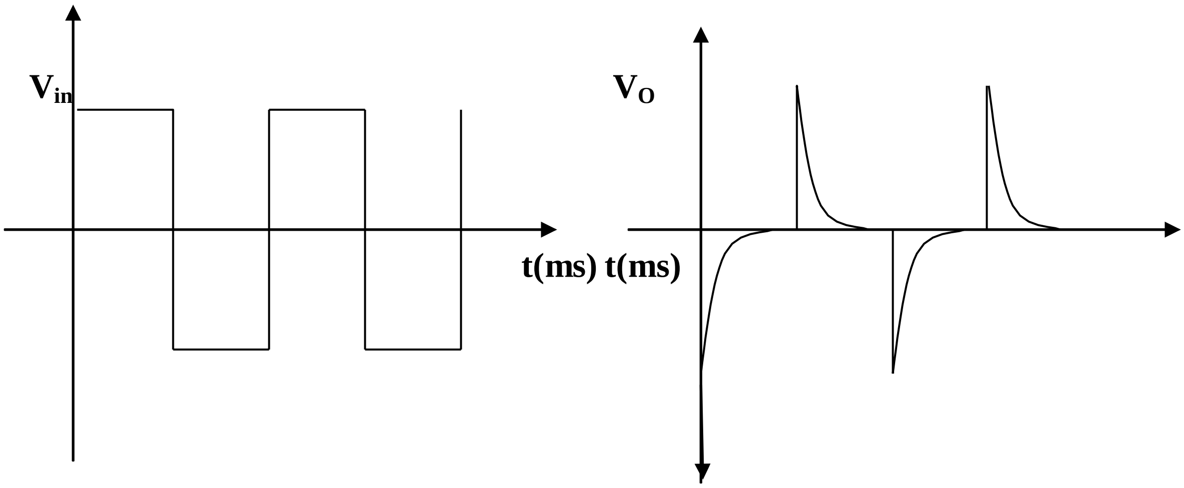
7. EXPECTED WAVEFORMS:

For Differentiator:

INPUT (V_{in}) **OUTPUT(V_O)**

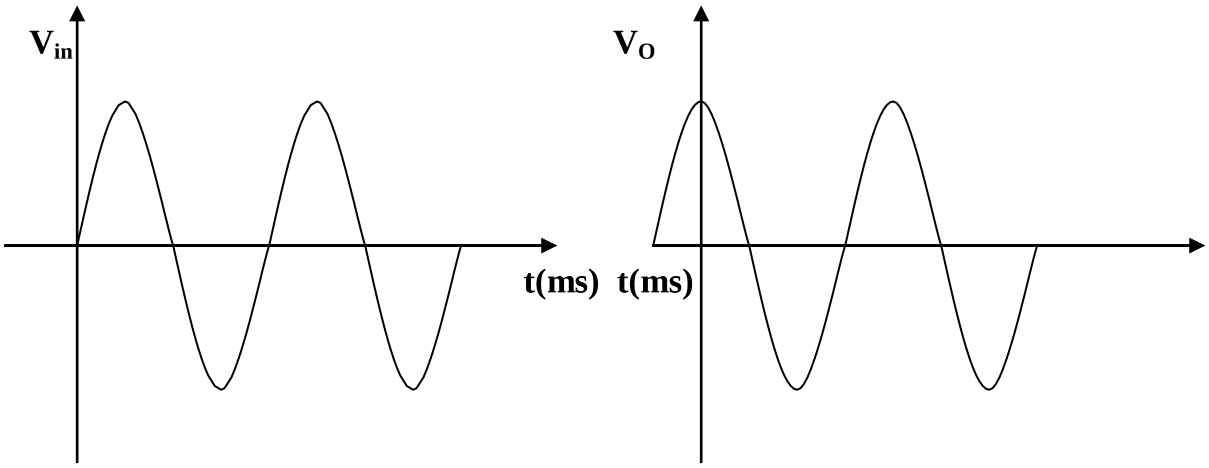


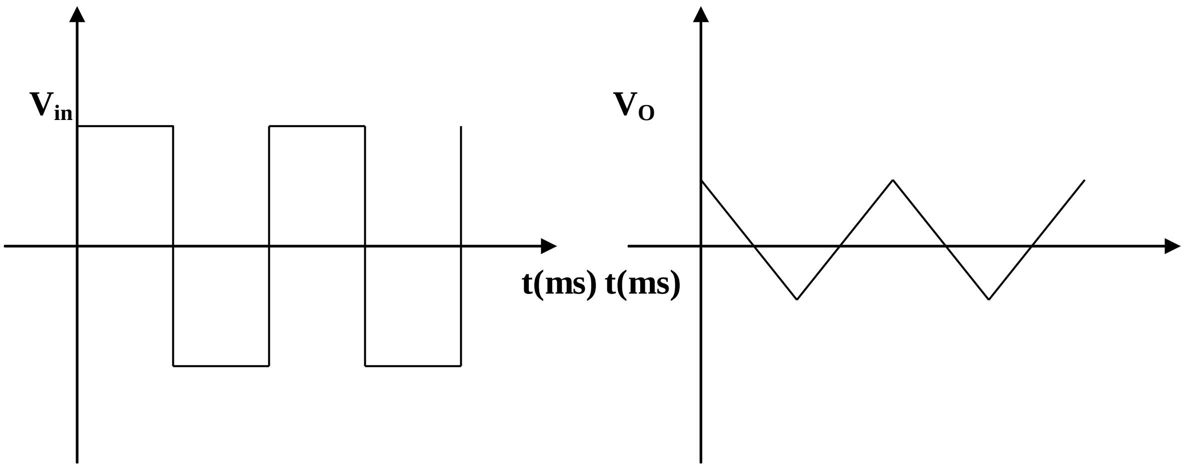




For Integrator:

INPUT (V_{in}) **OUTPUT(V_O)**





8. PRECAUTIONS:

1. Set power supplies to zero volts before connecting the circuit diagram.
2. Insert Op amp with correct pin numbers.
3. Avoid loose connections.

9. RESULT:

Hence the experiment on the design of Integrator circuit and Differentiator circuit using IC 741 has been done and results have been verified and drawn for various input waveforms.

10. VIVA -VOCE QUESTIONS:

1. What do you mean by a Integrator circuit?
2. What do you mean by a Differentiator circuit?
3. Write the output voltage expression in a Differentiator circuit?
4. Which circuits are preferred for analog computers?
5. What output we would get if we give input as a square wave to a Integrator circuit?
6. What output we would get if we give input as a square wave to a Differentiator circuit?
7. What happens to the Integrator output if the input is at low frequencies or at f=0Hz?

Design, Realization and plot the frequency response of First order Low pass and High pass filters using Op-Amp	EXPT. NO: 4
	DATE:

1. AIM:

To design Low pass filter and a High pass filter of first order at a given cut off frequency with a given pass band gain A_V=5 and to plot the frequency response curve.

2. EQUIPMENTS REQUIRED:

2.1 Plus – Minus DC Power supply- [(0 - ±15) V, 50 mA]

2.2 IC741.

2.3 Bread board.

2.4 Resistors

2.5 Capacitors

2.6 Dual Trace Oscilloscope

3. THEORY:

The frequency selective circuit that passes electrical signal of desired frequencies and attenuates the signals of frequency outside the band is called an electric filter. Filter can be classified into active filter and passive filter. Active filter use active elements like Op amps, transistors along with resistors, inductors and capacitors. Due to flexibility in gain and frequency adjustments, active filters can be easily controlled. Based on the shape of the freq response, the filter can be classified into Butter worth, Chebyshev and Bessel.

A filter that attenuates high frequencies and passes low frequencies is called low pass filter. A LPF can be constructed by connecting an RC network to the non-inverting input terminal of the op amp. Here RC network acts for filtering. Resistors R_1 and R_f determine the gain for the filters. Here the cut off frequency is denoted by f_H.

Simply interchanging frequency determining resistors and capacitors in low pass filters gives High pass filters. A HPF attenuates low frequencies and passes high frequencies. Here the cut off frequency is denoted by f_L. Lower cutoff frequency (f_L) is the frequency at which the magnitude of the gain is 0.707 times its pass band value. All frequencies higher than f_L are pass band frequencies, with the highest frequency determined by the closed loop bandwidth of the op-amp.

In the stop band (from 100Hz to 1 KHz) the gain increases at the rate of 20db/decade. In the pass band the gain remains constant at 6.db. The upper frequency limit of the pass band is set by the closed loop bandwidth of the op-amp.

4. DESIGN PROCEDURE:

For Low pass filter:

4.1 Know the cut off frequency $\mathbf{f_H}$ **.**

4.2) Choose **C= 0.1μF**

4.3) Calculate R by using $\mathbf{f_H = 1/\ 2\pi RC}$

4.4) Choose the value of gain A_V and R_1= 1K and from these values

Calculate $\mathbf{R_f}$.

For High pass filter:

4.1 Know the cut off frequency $\mathbf{f_L}$ **.**

4.2) Choose **C= 0.1μF**

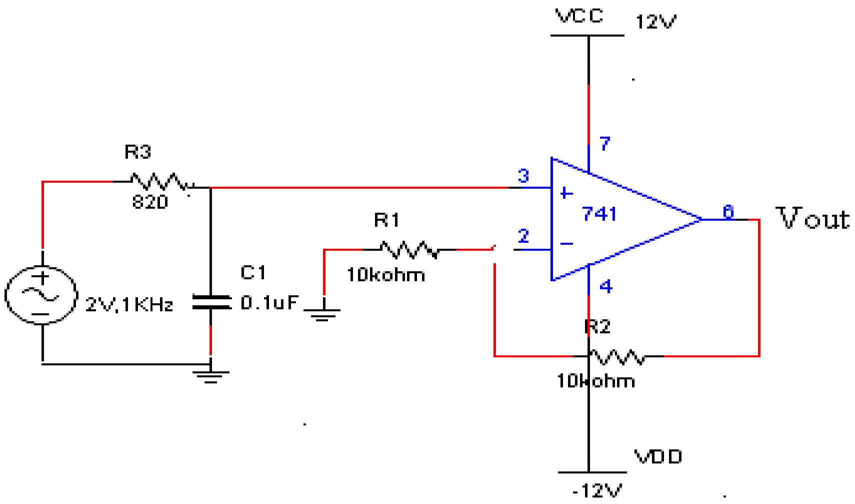
4.3) Calculate R by using $\mathbf{f_L = 1/\ 2\pi RC}$

4.4) Choose the value of gain A_V and R_1= 1K and from these values

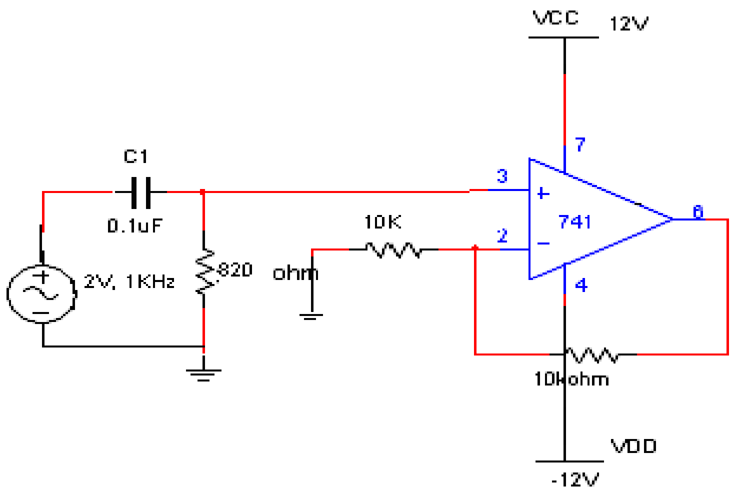
Calculate $\mathbf{R_f}$.

5. CIRCUIT DIAGRAM:

Low pass filter:



High pass filter:



6. PROCEDURE:

For Low pass filter:

6.1 Design the circuit as per the given specifications and connect it as shownin the Circuit diagram.

6.2 Keep input voltage at 1V using the function generator.

6.3 Keeping input voltage constant; vary the frequency from 0Hz to 200KHz.

6.4 Find higher cut-off frequency practically.

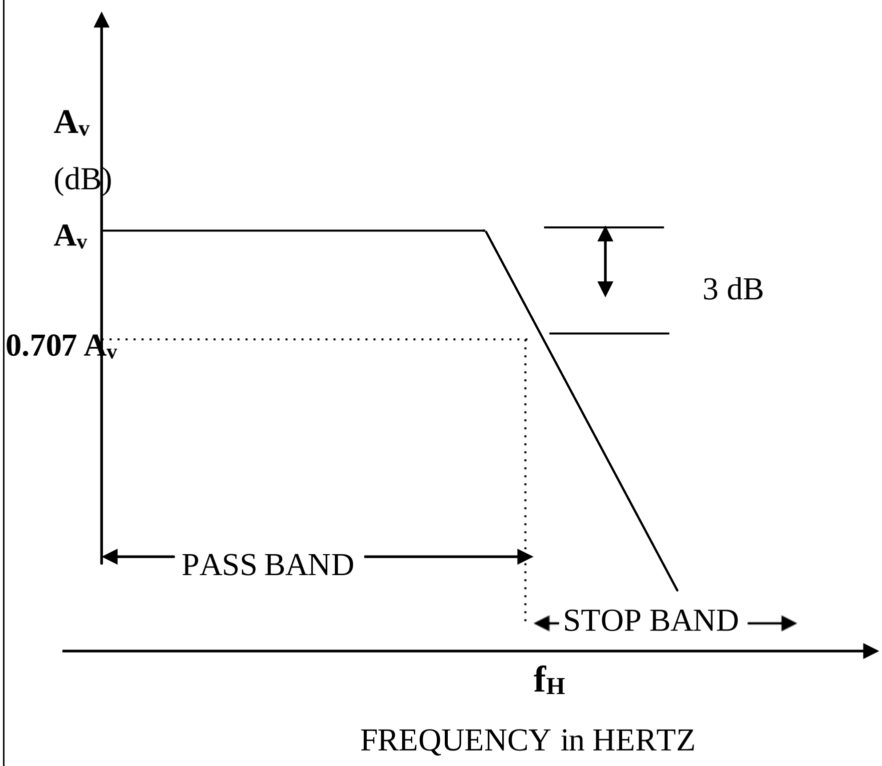
6.5 Plot the graph between voltage gain in decibels & frequency.

For High pass filter:

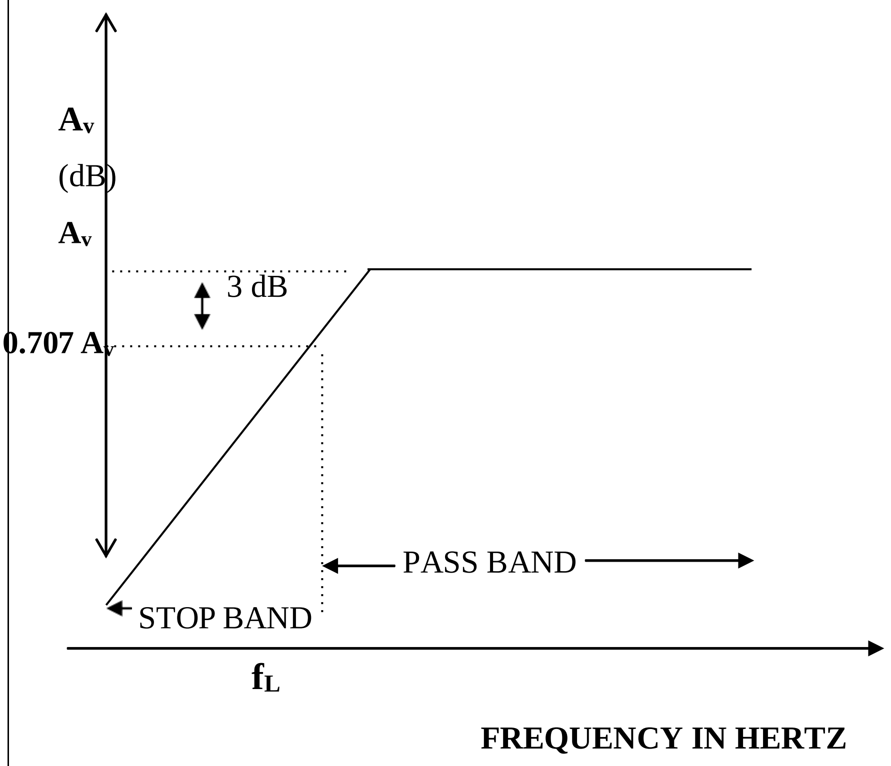
- Design the circuit as per the given specifications and connect it as shown in the Circuit diagram.
- Keep input voltage at 1V using the function generator.
- Keeping input voltage constant; vary the frequency from 0Hz to 500KHz.
- Find lower cut-off frequency practically.
- Plot the graph between voltage gain in decibels & frequency.

7. EXPECTED FREQUENCY RESPONSE:

Low pass filter:



High pass filter:



8. PRECAUTIONS:

1) Keep current knob of power supply in maximum position.
2) Check the op amp before connections
3) Avoid loose contacts

9. RESULT:

Hence the design of Low pass filter and High pass filter of first order has been done and the frequency response has been drawn for given design specifications and the lower cut off frequency and higher cut off frequency has been found practically.

10. VIVA -VOCE QUESTIONS:

1. What is a filter circuit?
2. Distinguish between low pass filter and high pass filter?
3. Tell the applications of low pass filter and high pass filter?
4. What happens to the filter response when the order of a filter increases?
5. What is the difference between amplifier and a filter?
6. What is the difference between first order and second order filters?
7. What do you mean by active devices and passive devices?

Design and plot the frequency response of Band pass filter using Op-Amp	EXPT. NO: 5
	DATE:

1. **AIM:** To design a Band pass filter of first order at a given cut off frequency and hence to plot the frequency response curve and also to find the Q-factor.

2. EQUIPMENTS REQUIRED:

1 Plus – Minus DC Power supply- [(0 - ±15) V, 50 mA]

2 IC741

3 Bread board

4 Resistors

5 Capacitors

6 Dual Trace Oscilloscope

3. THEORY:

The frequency selective circuit that passes electrical signal of desired frequencies and attenuates the signals of frequency outside the band is called an electric filter.

A Band pass filter has a pass band between two cut off frequencies f_H and f_L such that $f_H > f_L$. Any input frequency outside this pass band is attenuated. There are two types of band pass filters (i) Wide band pass filter and (ii) Narrow band pass filter. For a wide band pass filter Q-factor is less than 10. Similarly for a Narrow band pass filter Q-factor is greater than 10. A Wide band pass filter is can be formed by cascading high pass and low pass filter sections. To obtain a 20-dB/decade-band pass, first order high pass and low pass filter sections are cascaded. A narrow band pass filter is realized by using multiple feedbacks.

4. DESIGN PROCEDURE:

(a) Wide band pass filter:

For Low pass filter section:

4.1) Know the cut off frequency $\mathbf{f_H}$ **.**

4.2) Choose **C= 0.1μF**

4.3) Calculate R by using $\mathbf{f_H = 1/ 2\pi RC}$

4.4) Choose the value of gain A_V and R_1= 1K and from these valuesCalculate $\mathbf{R_f}$.

For High pass filter:

4.1) Know the cut off frequency $\mathbf{f_L}$**.**

4.2) Choose **C'= 0.1μF**

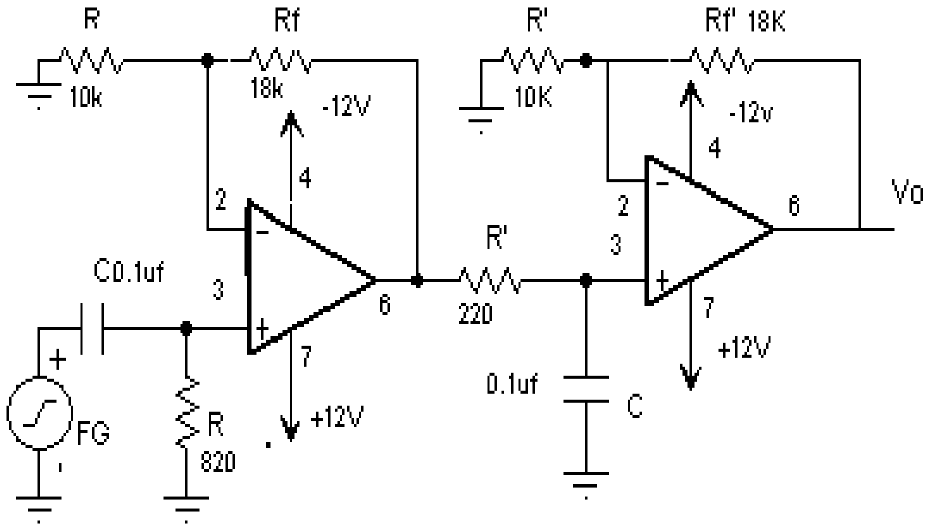
4.3) Calculate R' by using $\mathbf{f_L = 1/ 2\pi R'C'}$

4.4) Choose the value of gain A_V and R_1'= 1K and from these values

Calculate $\mathbf{R_f}$'.

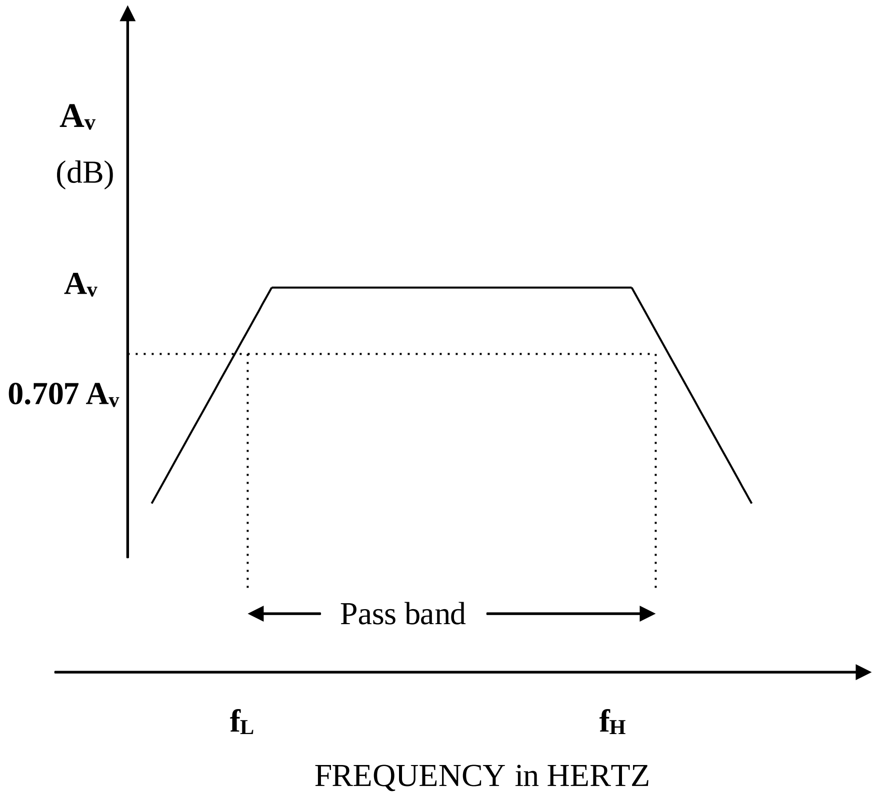
5. CIRCUIT DIAGRAM:

5.1: Wide Bandpass filter:



6. EXPECTED FREQUENCY RESPONSE:

6.1 Wide bandpass filter:



7. PROCEDURE:

Wide band pass filter:

1) Design the circuit as per the given specifications and connect it as shown in the Circuit diagram.
2) Keeps input voltage at 1V using the function generator.
3) Keeping input voltage constant; vary the frequency from 1Hz to 1000 KHz.
4) Note down the corresponding output voltages.
5) Plot the graph between voltage gain in decibels & frequency and find out the higher cut off frequency and lower cut off frequency and compare With the theoretical frequencies.
6) From the graph find the bandwidth of the filter and hence calculate the Q – factor of the filter by using **$Q = (f_o / BW)$** and also calculate center frequency **$f_O = \sqrt{f_H f_L}$**

8. RESULT:

Hence the design and study of Band pass filter has been done for a given cut off frequency and the frequency response has been drawn and the Q-factor has been found out.

9. PRECAUTIONS:

1) Keep current knob of power supply in maximum position.
2) Check the op amp before connections.
3) Avoid loose contacts.
4) Avoid parallax error while observing output in CRO.

10. VIVA -VOCE QUESTIONS:

1. What is a filter circuit?
2. Distinguish between Band pass filter and Band reject filter?
3. Tell the applications of Narrow band reject filter?
4. What is the other name of Narrow band reject filter?
5. What happens to the filter response when the order of a filter increases?
6. Tell the applications of band pass filters
7. What is the difference between first order and second order filters?
8. How can you realize a Wide band reject filter?
9. What is the difference between amplifier and a filter?
10. How can you realize a Wide band pass filter?

Design and Realization of Op-Amp based Astable Multivibrator For Square Wave generation	EXPT. NO: 6
	DATE:

1. AIM:

To Design and generate Op-Amp based Astable Multivibrator for Square wave generation and also to compare the theoretical frequency with the practical frequency.

2. EQUIPMENTS REQUIRED:

1. Plus – Minus DC Power supply- [(0 - ±15) V, 50 mA]
2. IC741.
3. Bread board.
4. Resistors
5. Capacitors
6. Dual Trace Oscilloscope

3. THEORY:

Operational amplifier may be used to design a circuit capable of generating a variety of output waveforms. A simple Op amp square wave oscillator is also called as a free running oscillator. The principle of generation of square wave output is to force an Op amp to operate in the saturation region. A fraction $\beta = R_2 / (R_1+R_2)$ of the output is fed back to the non-inverting input terminal. The output is also fed back to the inverting terminal after integrating by means of a low pass RC combination. Whenever input at the inverting input terminal just exceeds V_{ref}, switching takes place resulting in a square wave output. In Astable multivibrator, both the states are quasi stable.

4. DESIGN PROCEDURE:

For square wave generator:

1) know the Theoretical frequency F.

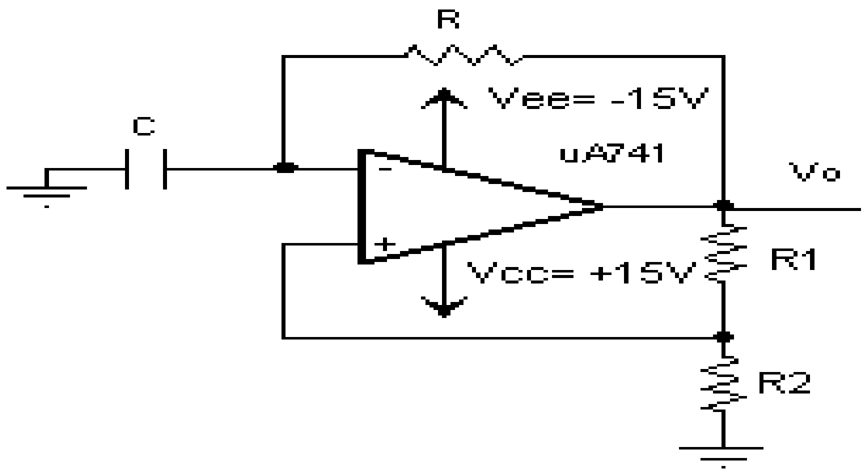
2) Choose C= 0.1μF

3) Calculate R by using F=1/2RC.

4) Choose $R_2 = 1.16R_1$.

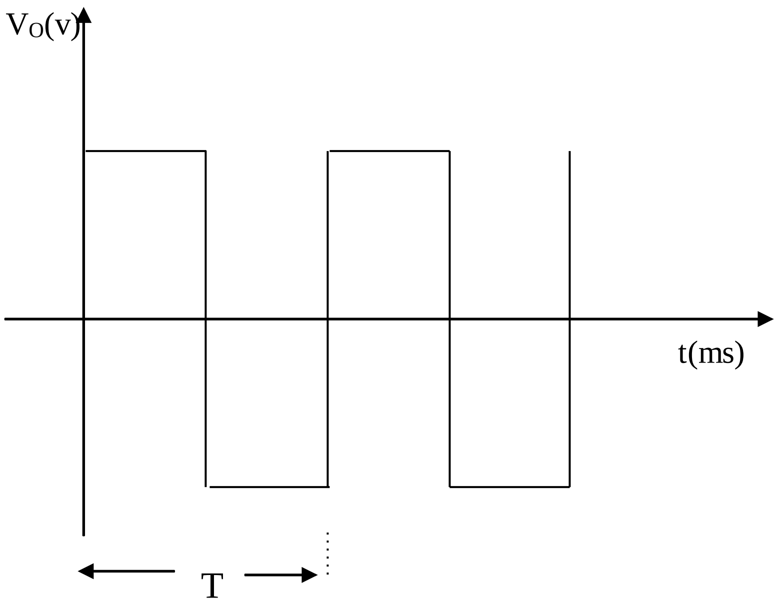
5. CIRCUIT DIAGRAM:

For square wave generator:



6. EXPECTED WAVEFORMS:

6.1: For square wave generator:



7. PROCEDURE:

7.1: For Square wave generator:

1. Connect the circuit as per the circuit diagram.
2. Apply proper biasing voltages at the corresponding pin numbers of the IC.
3. Observe the output waveforms at the CRO and Compare the theoretical frequency with the Practical frequency
4. Draw the graph for the observed output waveform.

8. PRECAUTIONS:

1) Keep current knob of power supply in maximum position.
2) Check the op amp before connections.
3) Avoid loose contacts.
4) Avoid parallax error while observing output in CRO.

9. RESULT:

Hence the design of square wave generator and Triangular wave generator has been done and the observed practical frequencies have been compared with theoretical frequencies.

10. VIVA -VOCE QUESTIONS:

1. What is the other name for a square wave generator?
2. What is an Op Amp?
3. What happens to the output of an Integrator circuit when we give input as a square wave?
4. Write the output time period expression for an Astable multivibrator using 741 Op amp?
5. Tell the function of resistors $R_1$and R_2 in the square wave generator circuit?
6. What is the function of a comparator circuit when operated in open loop mode?

Design and Realization of 555 timer based Monostable multivibrator for Pulse generation	EXPT. NO: 7
	DATE:

1. AIM:

To Design and Realization of 555 timer based Monostable multivibrator for Pulse generation and to compare the theoretical frequency with the practical frequency

2. EQUIPMENTS REQUIRED:

1. Plus – Minus DC Power supply- [(0 - ±15) V, 50 mA]
2. IC555.
3. Bread board.
4. Resistors
5. Capacitors
6. Diode
7. Dual Trace Oscilloscope

3. THEORY:

Monostable Multi vibrator:

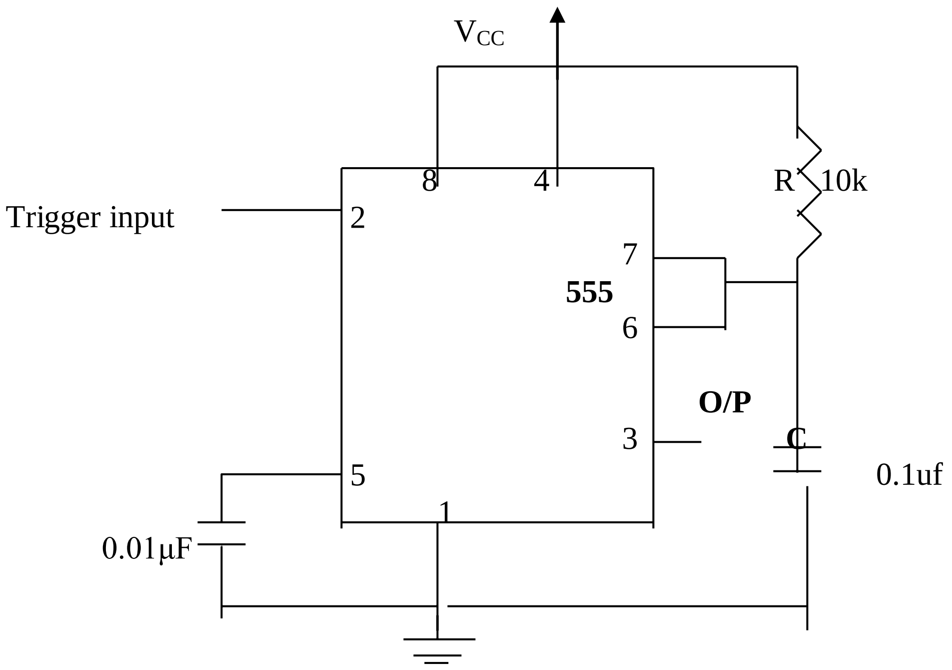
The 555 timer can be used with supply voltage in the range of +5 v to +18 v and can Drive upto 200 mAmps. It is compatible with both TTL and CMOS logic circuits because of the widerange of supply voltage the 555 timer is versatile and easy to use in monostable multivibrator we will provide external triggering in order to make the timer to switch over to high state (unstable). This is also called as one-short multivibrator.

4. DESIGN PROCEDURE:

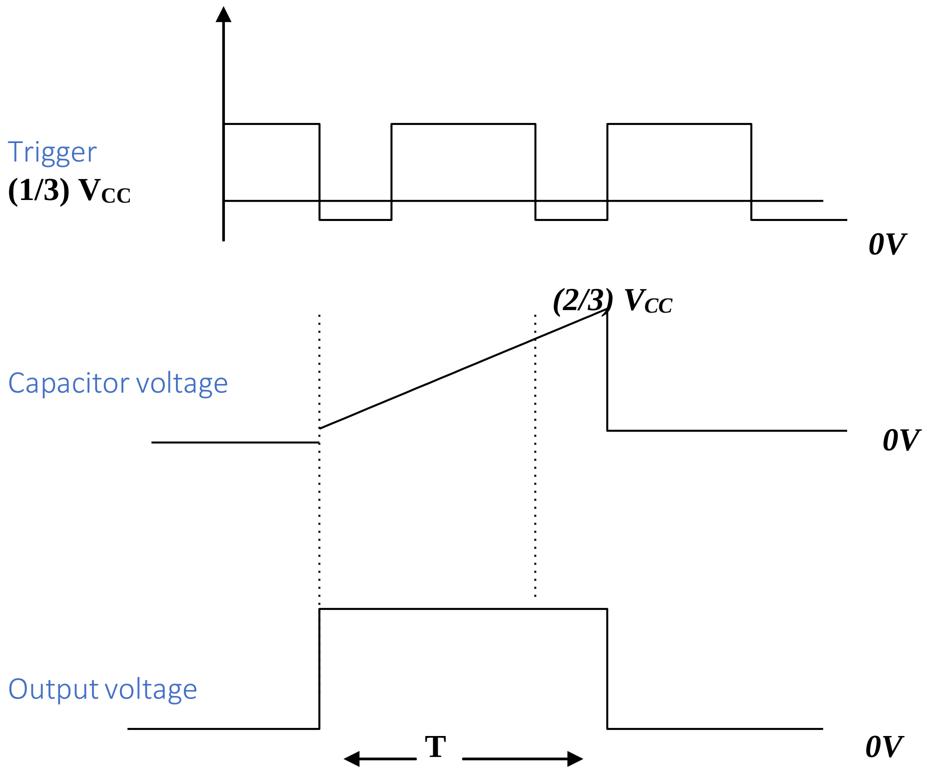
(i) Choose a value of **C=0.1 µF**and theoretical pulse width (**T_P**) will be given.

(ii) Calculate the value of R by using **T_P = 1.1RC.**

5. CIRCUIT DIAGRAM:



6. EXPECTED WAVEFORMS:Additional pulse has no effect on output



7. PROCEDURE:

1. Connect the circuit using the component values as obtained in part 4.
2. Apply trigger input with amplitude of 5 volt and at a frequency of 1 KHz and observe the output waveform.
3. Observe and sketch the capacitor voltage waveform (pin–6) and output waveform (pin–3).
4. Measure the pulse width of the output waveform and compare with theoretical pulse width.
5. Draw graphs for output waveforms.

8. PRECAUTIONS:

1) Keep current knob of power supply in maximum position.

2) Check the op amp before connections.

3) Avoid loose contacts.

4) Avoid parallax error while observing output in CRO.

9. RESULT:

Monostable Multivibrator circuit is designed for a given pulse width and is compared with the obtained pulse width.

10. VIVA -VOCE QUESTIONS:

1. What is a Multivibrator circuit?
2. How many stable states are present in a Monostable multivibrator circuit?
3. Write the expression for pulse with in a Monostable multivibrator circuit?
4. Tell any three applications of a Monostable multivibrator circuit?
5. Can a Monostable multivibrator circuit be used to produce Sinusoidal waveforms? Explain?
6. What is the difference between Monostable multivibrator circuit and Astable multivibrator circuit?

Construction of Schmitt Trigger using Op-Amp and calculate UTP and LTP values	EXPT. NO: 08
	DATE:

1. AIM:

To Design a Schmitt trigger circuit using IC 741 and verify the output wave forms.

2. EQUIPMENTS REQUIRED:

1. Plus – Minus DC Power supply- [(0 - ±15) V, 50 mA]
2. IC741.
3. Bread board.
4. Resistors
5. Capacitors
6. Dual Trace Oscilloscope

3. DESIGN:

1. To design a Schmitt trigger for a given value of $+V_{sat}$, $-V_{sat}$, UTP and LTP.
2. $+V_{sat} = V_{cc} - 3V$ (approx)
3. $-V_{sat} = V_{ee} + 3V$ (approx)
4. Select V_{cc} and V_{ee} as per the design requirements.
5. UTP and LTP depends on β i.e. $R_2 / (R_1 + R_2)$. Select the resistors as per the UTP and LTP requirements.
6. Design a Schmitt Trigger for $+V_{sat} = 9V$ and $-V_{sat} = -6V$ and UTP and LTP as 3V and -2V. For this requirement, $V_{cc} = +12V$ and $V_{ee} = -9V$ and the $\beta = 1/3$ (R_1 can be selected as 1K it gives R_2 as 2K)

4. CIRCUIT DIAGRAMS:

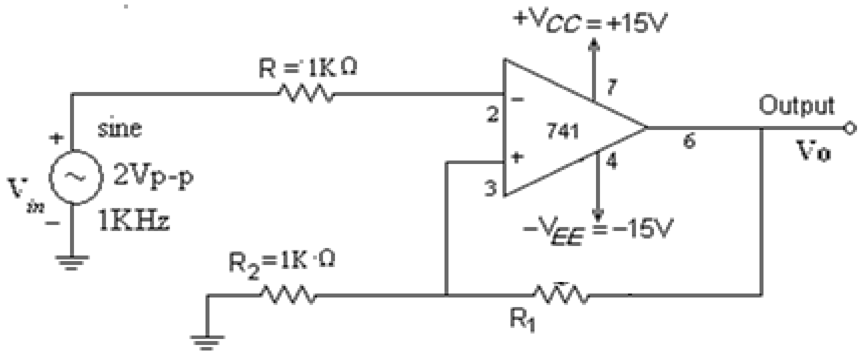
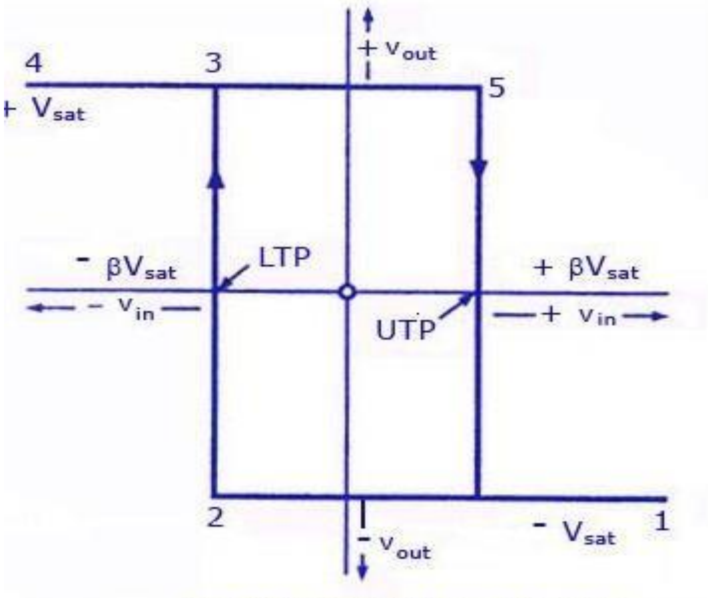


Fig 1. Schmitt trigger using IC 741



Input Output Characteristics

5. PROCEDURE:

Using IC 741:

Connect the circuit as shown in fig 1(a) as Schmitt trigger using IC 741.

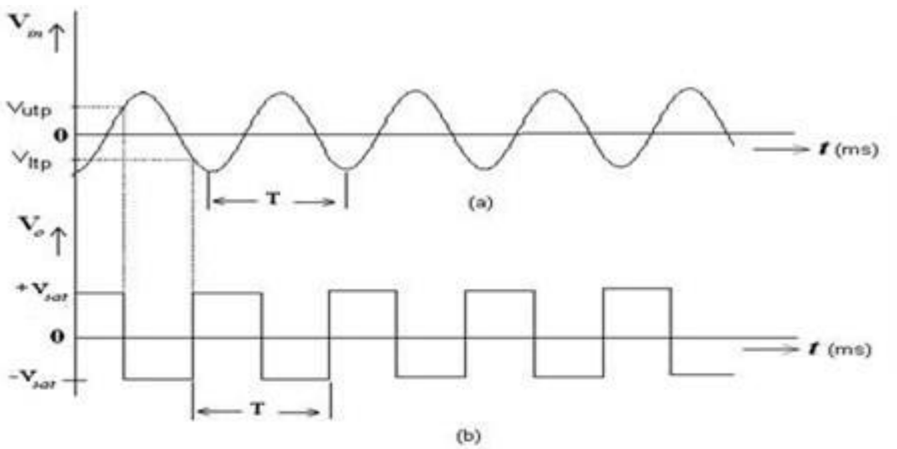
Give a 5 V_{p-p} sine wave of 1 kHz as input.

Observe the wave form on CRO and measure UTP and LTP, V_{sat} and - V_{sat}.

Use X-Y mode in CRO and observe hysteresis curve.

Repeat the above experiment for R_1 = 5.1Kohms and 15 Kohms and observe the effect.

6. EXPECTED WAVEFORMS:



(a) Input wave form (b) output wave form

7. OBSERVATIONS:

Parameter	Input	Output
Voltage(V_{p-p}), V		
Time period - Positive Half cycle(ms)		
Time period - Negative Half cycle(ms)		
	Theoretical	**Observed**
UTP		
LTP		

8. RESULT: Designed and verified Schmitt trigger circuit using IC 741.

9. VIVA -VOCE QUESTIONS:

1. What is a Schmitt Trigger?
2. Define Hysteresis loop ?
3. Write the applications of Schmitt Trigger

Design and Realization of 555 timer based Astable multivibrator for square wave generation	EXPT.NO: 09
	DATE :

1. AIM:

To design an Astable Multivibrator circuit for a given frequency and determine its Duty cycle by using IC 555 timer.

2. EQUIPMENTS REQUIRED:

1. DC Power supply- [(0-5) V, 50 mA]
2. Dual Trace Oscilloscope (30 MHz)
3. IC 555 timer.
4. Breadboard.
5. Resistors and Capacitors

3. THEORY:

Astable multivibrator, multivibrator often called a free running oscillatorand Square wave generator. This circuit does not require an external triggering to change the state of the output. The time during which the output is either high or low is determined by two resistors and a capacitor, which are externally connected to 555 timer.

Initially when the output is high, capacitor C starts charging towards V_{CC} through R_A& R_B. However as soon as voltage across the capacitor equals (2/3)V_{CC} , comparator 1 triggers the flipflop and the output switches low. Now C starts discharging through R_B and transistor Q_1. When the voltage across C equals (1/3) V_{CC}, comparator 2 output triggers the flip-flop and the output goes high then the cycle repeats.

4. DESIGN PROCEDURE:

In Astable mode **free running frequency(f) = 1.45 / (R_A+2R_B)C**

%Duty cycle (D) =[(R_A+R_B)/(R_A+2R_B)] * 100

Step (i) Choose the value of capacitor C= 0.1μF.

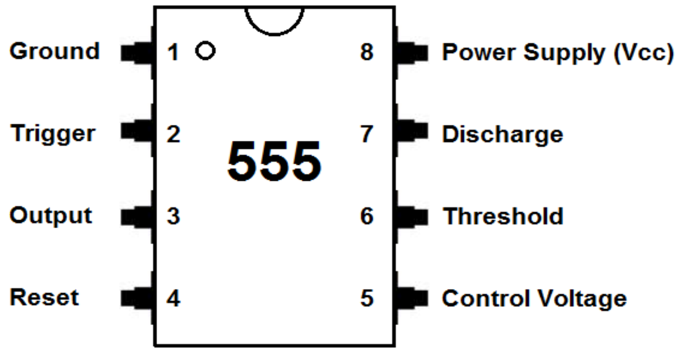
Step (ii) From the above two expressions calculate the values of resistors R_A and R_B.

Output voltage (V_O) = V_{CC}

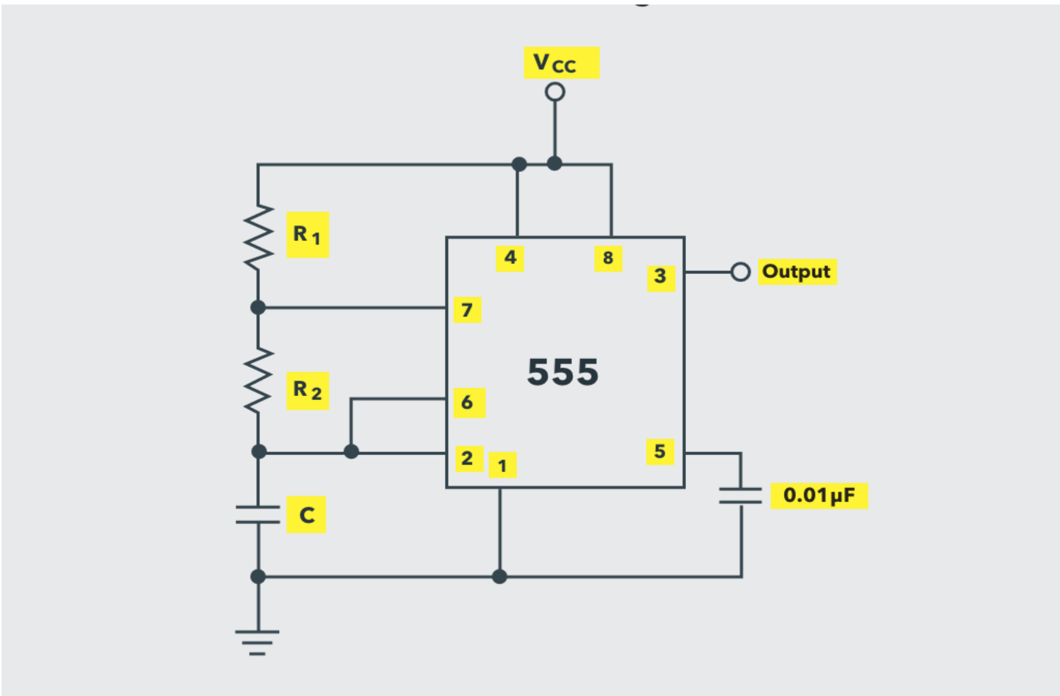
Voltage across capacitor (V_C) = [*(2/3) V_{CC} - (1/3) V_{CC}]*

% Duty Cycle (D) = [T_{high} / (T_{high} + T_{low})] *100

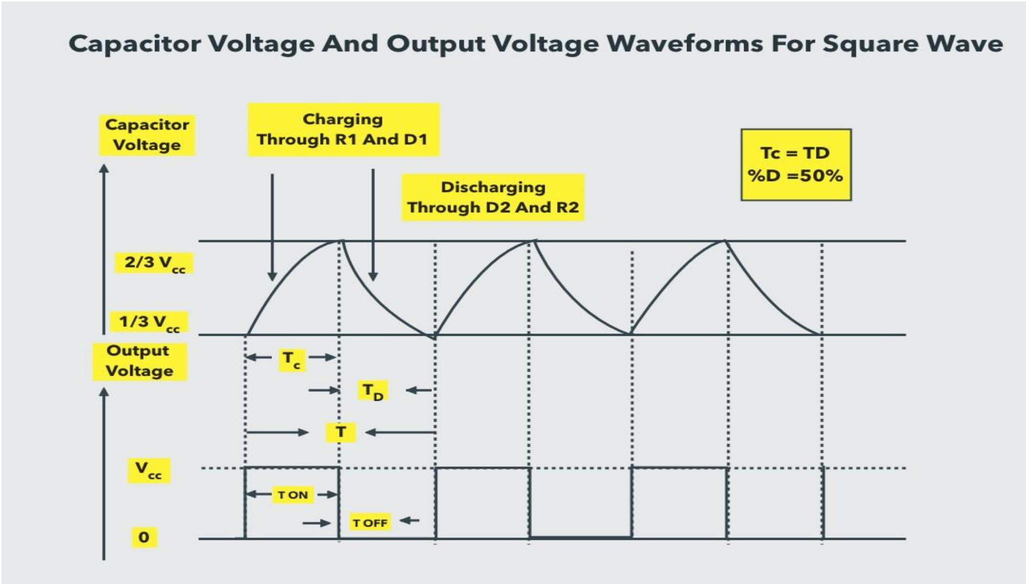
5. PIN DIAGRAM:



6. CIRCUIT DIAGRAM:



6. EXPECTED WAVEFORMS:



7. PROCEDURE:

1. Connect the circuit using the component values as obtained in part 4.
2. Observe and sketch the capacitor voltage waveform (pin– 6) and output waveform (pin– 3).
3. Measure the values of T_{high} and T_{low} of output waveform and calculate % Duty cycle (D) and frequency.
4. Draw graphs for output waveforms.

8. PRECAUTIONS:

1) Keep the current knob of the power supply in maximum position.
2) Check the op-amp before connections.
3) Avoid loose contacts.
4) Avoid parallax error while observing output in CRO.

9. RESULT:

An Astable Multivibrator circuit is designed for a given frequency and its Duty cycle is determined.

10. VIVA -VOCE QUESTIONS:

1. What is a Multivibrator circuit?
2. How many stable states are present in an Astable multivibrator circuit?
3. Write the expression for frequency and % Duty cycle in an Astable
4. multivibrator circuit?
5. Tell any three applications of an Astable multivibrator circuit?
6. Is the Astable multivibrator circuit be used to produce Sinusoidal waveforms? Explain?f)
7. What is the difference between the Monostable multivibrator circuit and the Astable multivibrator circuit?

Design and Realization of RC phase shift Oscillator for sinusoidal signal generation using Op-Amp	EXPT. NO: 10
	DATE:

1. AIM:

To Design a Phase shift and Wien bridge oscillators for a particular frequency and compare them with practical frequencies.

2. EQUIPMENTS REQUIRED:

1. Plus – Minus DC Power supply- [(0 - ±15) V, 50 mA]
2. IC741.
3. Bread board.
4. Resistors –
5. Capacitors
6. Dual Trace Oscilloscope
7. Digital Multimeter

3. THEORY:

Operational amplifier may be used to design an oscillator circuit capable of generating a variety of output waveforms. Basically the function of an oscillator is to generate alternating current or voltage waveforms. More precisely, an oscillator circuit generates a repetitive waveform of fixed amplitude and frequency without any external input signal.

An oscillator is a type of feedback amplifier in which part of the output is feedback to the input via a feedback circuit .If the signal feedback is of proper magnitude and phase; the circuit produces alternating current and voltages. The two most commonly used audio frequency oscillators are (i) RC Phase shift oscillator (ii) Wien bridge oscillator.

In RC phase shift oscillator an op amp is used as the amplifying stage and three RC cascaded networks as the feedback network. The Op amp is used in the inverting mode; therefore any signal that appears at the inverting terminal is shifted by 180 degrees at output. The cascaded RC network provides an additional 180-degree phase shift required for oscillation. The frequency of the oscillator depends upon RC components connected in the feedback network.

4. DESIGN PROCEDURE:

For RC Phase shift oscillator

1) Know the Theoretical frequency F.

2) Choose **C= 0.1μF**

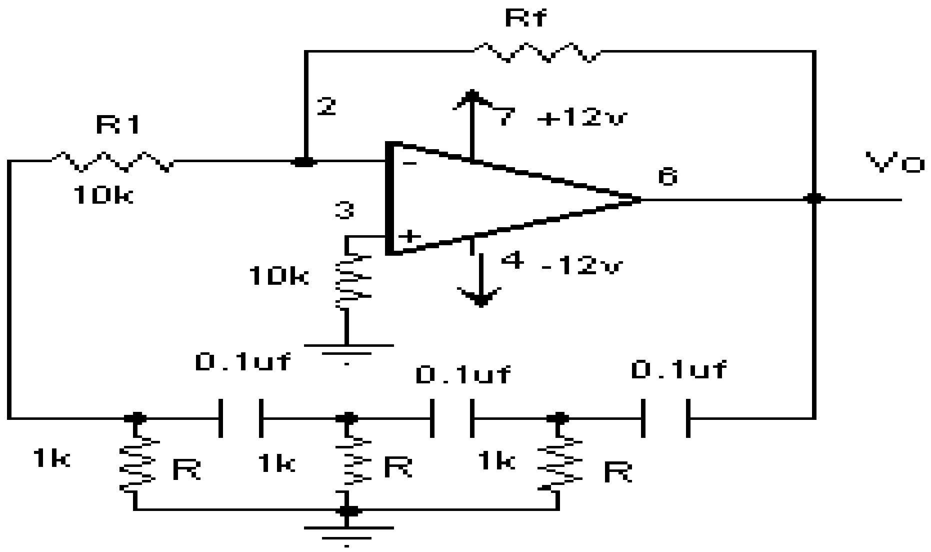
3) Calculate **R** by using $F=1/2\pi RC\sqrt{6}$.

4) Choose $R_1 \geq 10R$.

5) Calculate R_f by using the expression $R_f=29R_1$.

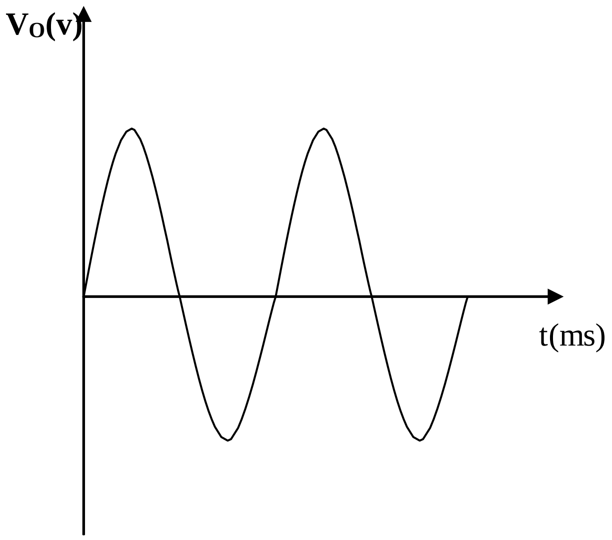
5. CIRCUIT DIAGRAM:

RC Phase Shift Oscillator:



6. EXPECTED WAVEFORMS:

RC Phase shift oscillator:



7. PROCEDURE:

For RC Phase shift oscillator:

1. Connect the circuit as per the circuit diagram.
2. Apply proper biasing voltages at the corresponding pin numbers of the IC.
3. Observe the output waveforms at the CRO and Compare the theoretical frequency with the Practical frequency
4. Draw the graph for the observed output waveform.

8. PRECAUTIONS:

(1) Keep current knob of power supply in maximum position.

(2) Check the op amp before connections and Avoid loose contacts.

9. RESULT:

Hence the design of RC Phase shift oscillator has been done for a given frequency and the theoretical frequency has been compared with Practical frequency.

10. VIVA -VOCE QUESTIONS:

1. What is a Op Amp?
2. What is an Oscillator circuit?
3. What are the conditions required to maintain oscillations?
4. Give any two examples for Audio oscillators?
5. Write the expression for frequency of oscillations in a RC Phase shift oscillator?
6. Write the expression for frequency of oscillations in a Wien bridge oscillator?

7.Which is the minimum gain required to maintain oscillations in an RC Phase shiftoscillator?

8. Which is the minimum gain required to maintain oscillations in a Wien bridge oscillator?

Design and Realization of Function Generator for Square wave and triangular waves using Op-Amp	EXPT. NO: 11
	DATE:

1.AIM:

To Design and generate various time varying waveforms using Op amps and also to compare the theoretical frequency with the practical frequency.

2.EQUIPMENTS REQUIRED:

1. Plus – Minus DC Power supply- [(0 - ±15) V, 50 mA]
2. IC741.
3. Bread board.
4. Resistors
5. Capacitors
6. Dual Trace Oscilloscope

3.THEORY:

Operational amplifier may be used to design a circuit capable of generating a variety of output waveforms. A simple Op amp square wave oscillator is also called as a free running oscillator. The principle of generation of square wave output is to force an Op amp to operate in the saturation region. A fraction $\beta = R_2 / (R_1+R_2)$ of the output is fed back to the non-inverting input terminal. The output is also fed back to the inverting terminal after integrating by means of a low pass RC combination. Whenever input at the inverting input terminal just exceeds V_{ref}, switching takes place resulting in a square wave output. In Astable multivibrator, both the states are quasi stable.

A triangular wave can be simply obtained by integrating a square wave. The Frequency of the square wave and triangular wave is same. The triangular wave that is a designed here use less number of components. It basically consists of a two level comparator followed by an integrator. The output of the comparator A_1 is a square wave of amplitude $\pm V_{sat}$ and is applied to the inverting input terminal of the integrator A2 producing a triangular wave. This triangular wave is fed back as input to the comparator A_1 through a voltage divider R_2R_3.

3. DESIGN PROCEDURE:

For square wave generator:

1) Know the Theoretical frequency F.
2) Choose C= 0.01μF
3) Calculate R by using F=1/2RC.
4) Choose $R_2 = 1.16R_1$.

For Triangular wave generator:

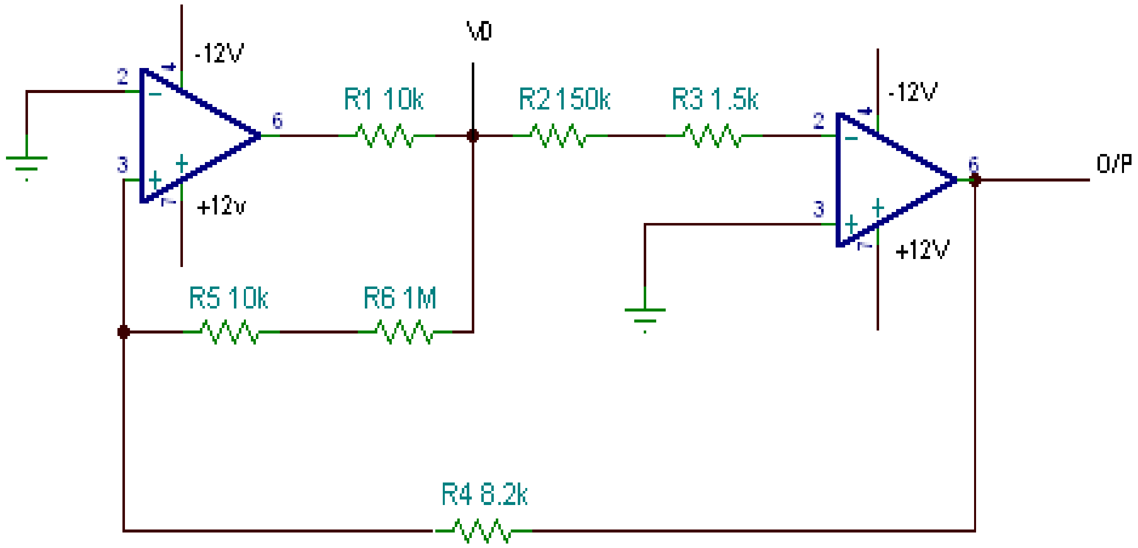
1) Know the Theoretical frequency F and output peak to peak voltage.

2) Choose C_1= 0.05µF

3) Calculate R_1 by using $F = R_3 / 4R_1C_1R_2$ and $V_o = (2R_2/R_3)V_{sat}$.

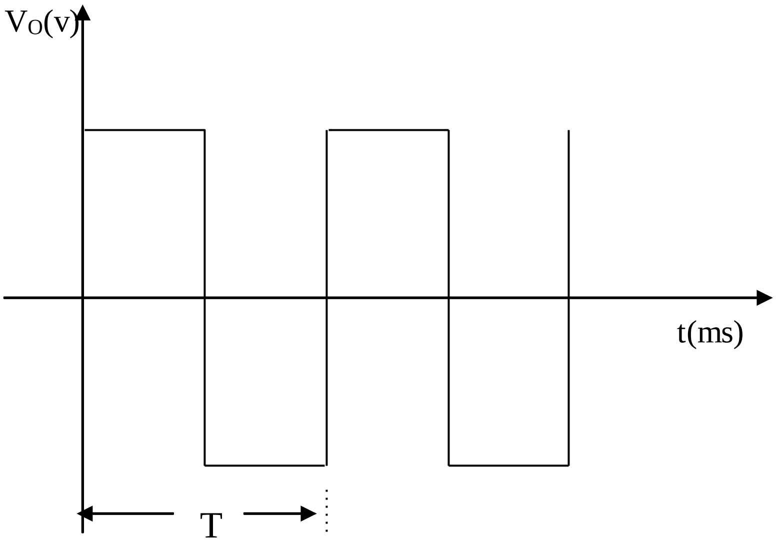
5. CIRCUIT DIAGRAM:

For square wave Triangular wave generator:

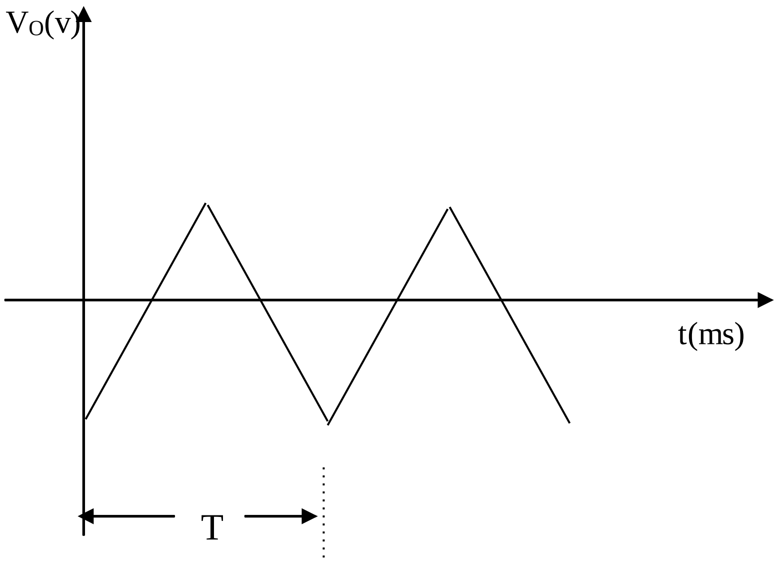


6. EXPECTED WAVEFORMS:

6.1 : For square wave generator:



6.2: For Triangular wave generator



7. PROCEDURE:

7.1: For Square wave generator:

1. Connect the circuit as per the circuit diagram.
2. Apply proper biasing voltages at the corresponding pin numbers of the IC.
3. Observe the output waveforms at the CRO and Compare the theoretical frequency with the Practical frequency
4. Draw the graph for the observed output waveform.

7.2: For Triangular wave generator:

5. Connect the circuit as per the circuit diagram.
6. Apply proper biasing voltages at the corresponding pin numbers of the IC.
7. Observe the output waveforms at the CRO and Compare the theoretical frequency with the Practical frequency
8. Draw the graph for the observed output waveform.

8. PRECAUTIONS:

1) Keep current knob of power supply in maximum position.
2) Check the op amp before connections.
3) Avoid loose contacts.
4) Avoid parallax error while observing output in CRO.

9. RESULT:

Hence the design of Square wave generator and Triangular wave generator has been done and the observed practical frequencies have been compared with theoretical frequencies.

10. VIVA -VOCE QUESTIONS:

1. What is the other name for a square wave generator?
2. What is an Op Amp?
3. What happens to the output of an Integrator circuit when we give input as a square wave?
4. Write the output time period expression for an Astable multivibrator using 741 Op amp?
5. Tell the function of resistors R_1 and R_2 in the square wave generator circuit?
6. What is the function of a comparator circuit when operated in open loop mode?

Design and Realization of Pulse generators using Op-Amp	EXPT. NO: 12
	DATE:

1. AIM:

To Design and Realization of Pulse generators using Op-Amp and also to compare the theoretical frequency with the practical frequency

2. EQUIPMENTS REQUIRED:

1. Plus – Minus DC Power supply- [(0 - ±15) V, 50 mA]
2. IC741.
3. Bread board.
4. Resistors
5. Capacitors
6. Dual Trace Oscilloscope
7. Diode

3. THEORY:

Monostable multivibrator is a square wave shaping circuit having one stable state and another quasi stable state. It can be made to switch to other state by the application of triggering pulse, but it then returns to its stable state after a time interval determined the RC value. It is often referred as single shot multivibrator. It is also used as a gating circuit and delay circuit. The time period is

$T = RC \ln ((1+V_D / Vsat)/ (1-\beta))$ Where $\beta = R_2 / (R_1+R_2)$, V_D is diode cut-in Voltage and V_{sat}

= 15VIfV$_{sat}$>> V$_D$andR$_1$=R$_2$sothatβ= 0.5, thenT=0.69RC.

4. DESIGN PROCEDURE:

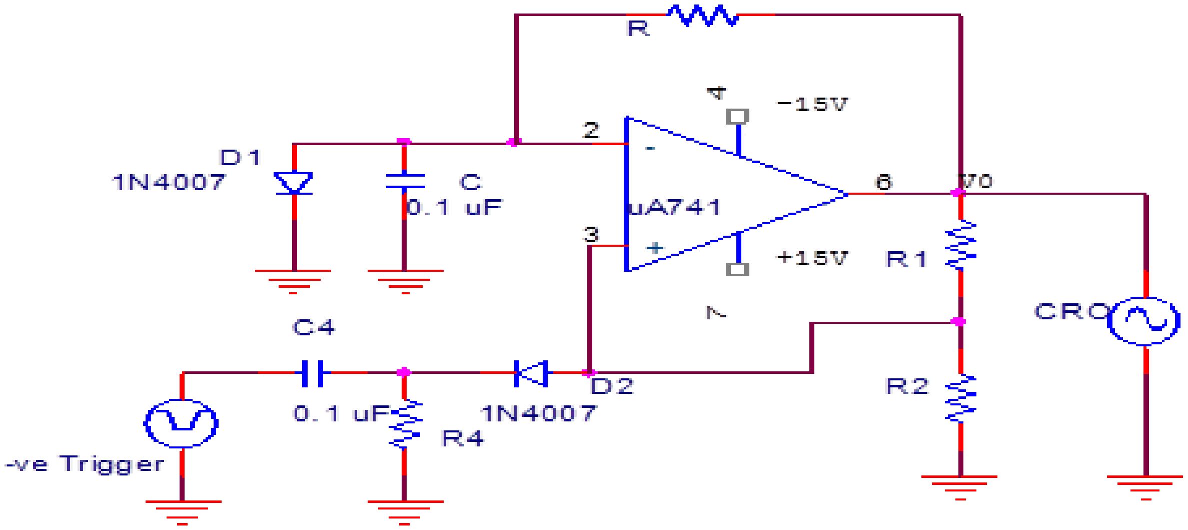
T=600µs,AssumeC=0.1µF;

T =0.69RC

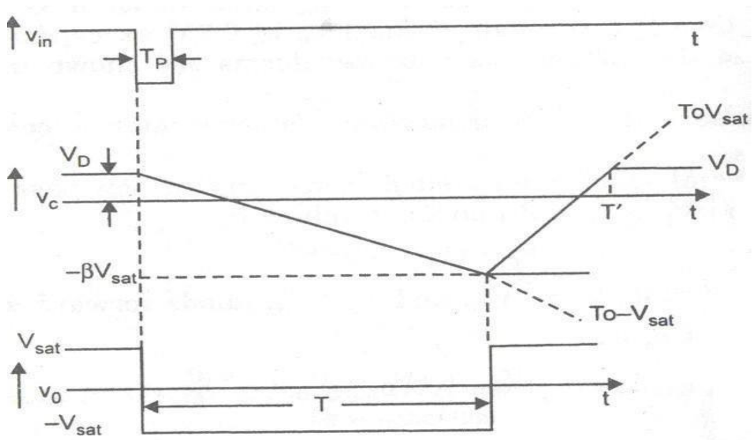
LetR$_1$=R$_2$=1kΩ;C=C$_4$=0.1µF;

R=600X10^6/(0.69X0.1X10^{-6})=

5. CIRCUIT DIAGRAM:



6. EXPECTED WAVEFORMS:



7. PROCEDURE:

1. The connection is made as per the circuit diagram.
2. Apply a negative going pulse as the input signal.
3. Observe the output voltage across the capacitor V_c and the output waveform V_0 and trace it.

TABULATION:

ResistanceR(KHz)	CapacitorC(µF)	TimePeriodT(ms)	FrequencyKHZ

8. PRECAUTIONS:

1) Keep current knob of power supply in maximum position.
2) Check the op amp before connections.
3) Avoid loose contacts.
4) Avoid parallax error while observing output in CRO.

9. RESULT:

A Monostable multivibrator is designed and constructed and the square wave output is obtained.

Time period of the output waveform(theoretical) =

Time period of the output waveform(practical) =

10. VIVA -VOCE QUESTIONS:

1. What is a Multivibrator circuit?
2. How many stable states are present in a Monostable multivibrator circuit?
3. Write the expression for pulse with in a Monostable multivibrator circuit?
4. Tell any three applications of a Monostable multivibrator circuit?
5. Can a Monostable multivibrator circuit be used to produce
6. Sinusoidal waveforms? Explain?
7. What is the difference between Monostable multivibrator circuit and
8. Astable multivibrator circuit?

Design and Realization of 3-bit Digital to Analog converter Using Op-Amp	EXPT. NO: 13
	DATE:

1. AIM:

(a) To construct a 3-bit DAC using R-2R ladder type technique and also by using Binary weighted resistor method.

(b) Also to plot the transfer characteristics and hence to calculate the resolution of the converter.

2. EQUIPMENTS REQUIRED:

2.1 Plus – Minus DC Power supply- [(0 - ±15) V, 50 mA]

2.2 IC741.

2.4 Bread board.

2.5 Resistors.

2.6 Fixed DC power supply.

2.7 Digital Multimeter.

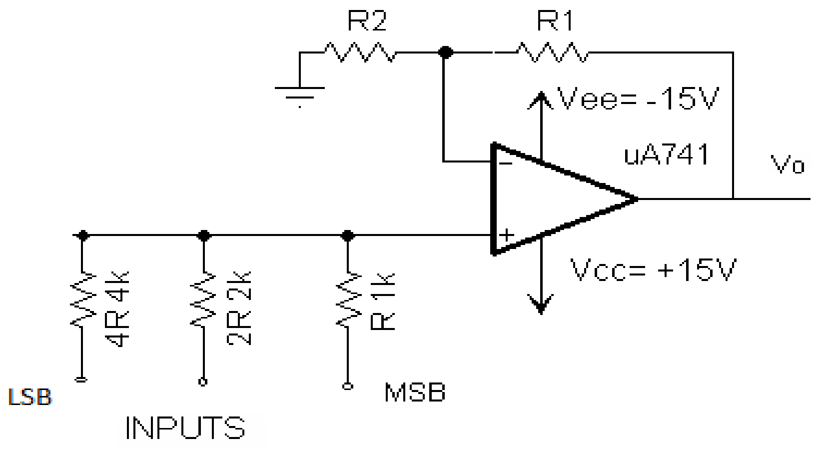
3. THEORY:

In most of the real world physical quantities such as voltage, current, temperature, pressures are available in analog form. It is difficult to process, store and transmit the analog signal without considerable error because of error and superimposition of noise. It is often convenient to express these variables in digital form. The advantage is that it gives better accuracy and reduces noise.

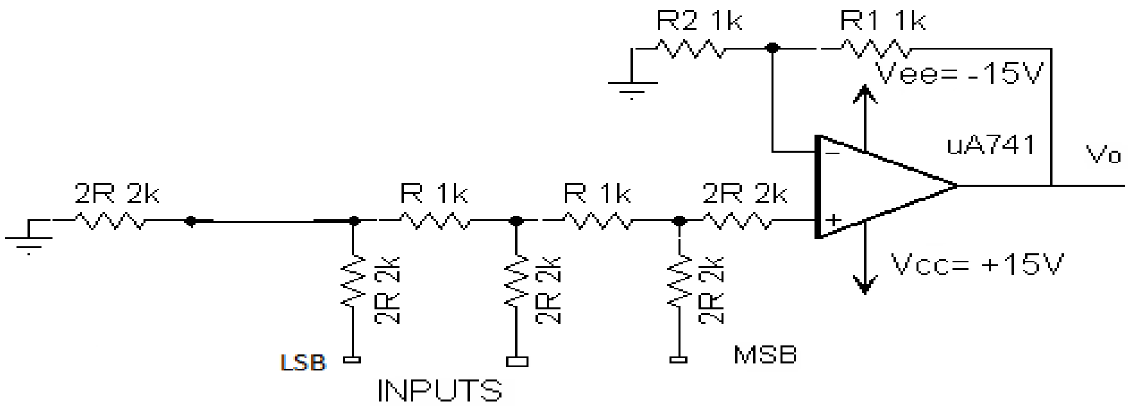
DAC's are available in IC form. There are various ways to implement the DAC circuits: (1) Binary weighted resistor method. (2) R-2R ladder type DAC. (3) Inverted R-2R ladder type DAC. In binary weighted resistor method the Op amp can either be connected in inverting mode or non-inverting mode. For example if the number of binary inputs is four then it is a 3 bit DAC. The problem with the DAC is that it requires binary weighted resistors, which may not be readily available, especially if the number of inputs is more than four. Also the accuracy and stability of a DAC depend upon the accuracy of the resistors and the tracking of each other with temperature.In R-2R ladder type DAC the circuit utilizes twice the number of resistors for the same number of bits (N) but of values R and 2R. The ladder used in this circuit is a current splitting device, and thus the ratio of the resistors is more critical than their absolute value. From the circuit we can clearly observe at any of the ladder nodes the resistance is 2R looking to the left or the right of toward the switch.

4. CIRCUIT DIAGRAM:

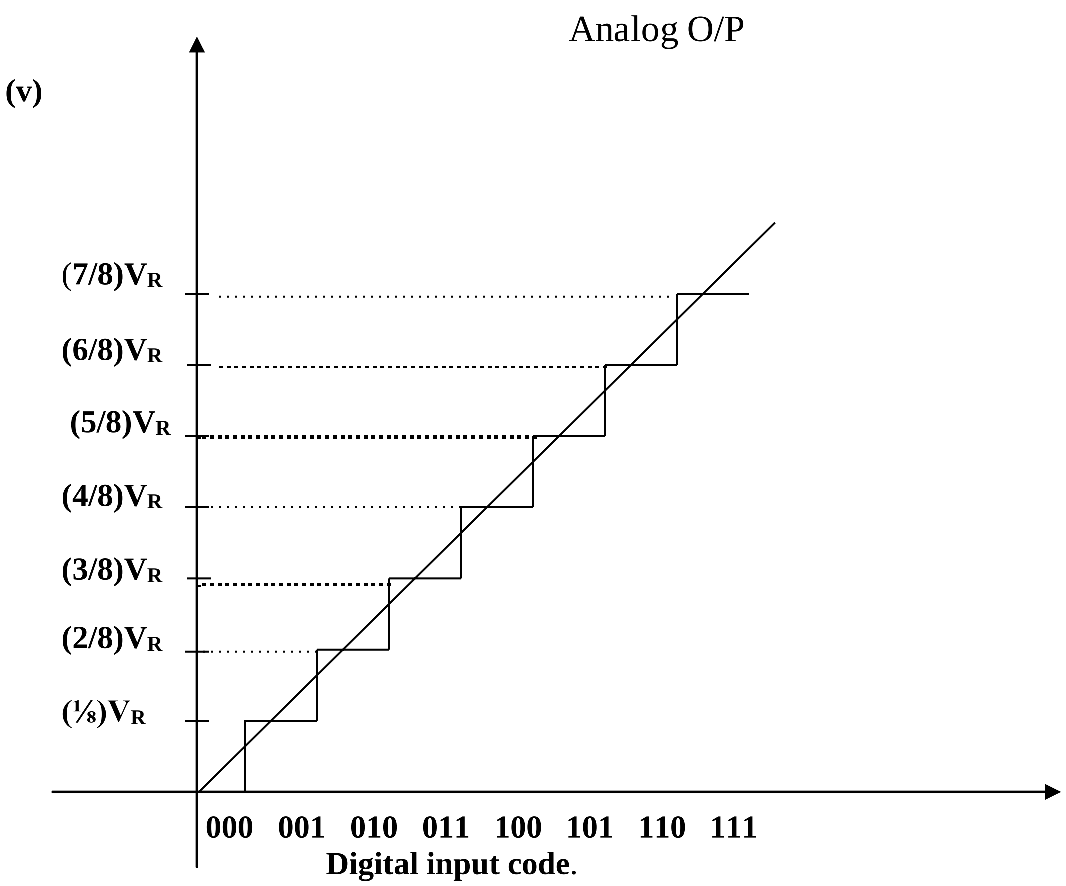
4.1 BINARY WEIGHED RESISTOR TYPE DAC:



4.2R- 2R LADDER TYPE DAC:



5. EXPECTED GRAPHS: (Transfer characteristics):



6. PROCEDURE:

1. Connect the circuit as per the circuit diagram shown.
2. Apply proper biasing voltages to the pin no:4 and pin no: 7.
3. Measure the output voltage for all the binary input combinations using a DMM.
4. Plot the graph of binary input voltage verses analog output voltage.
5. Measure the size of each step and hence calculate Resolution which is given by [V_{FS} /2^n-1].

7. TABULAR COLUMN:

7.1: USING BINARY WEIGHTED METHOD

S.No	DIGITAL INPUT			Analog output (V)
	B2	**B1**	**B0**	
1	**0**	**0**	**0**	
2	**0**	**0**	**0**	
3	**0**	**0**	**1**	
4	**0**	**0**	**1**	
5	**0**	**1**	**0**	
6	**0**	**1**	**0**	
7	**0**	**1**	**1**	
8	**0**	**1**	**1**	
9	**1**	**0**	**0**	
10	**1**	**0**	**0**	
11	**1**	**0**	**1**	
12	**1**	**0**	**1**	
13	**1**	**1**	**0**	
14	**1**	**1**	**0**	
15	**1**	**1**	**1**	
16	**1**	**1**	**1**	

7.2: USING R-2R LADDER METHOD

S.No	DIGITAL INPUT			Analog output (V)
	B2	**B1**	**B0**	
1	**0**	**0**	**0**	
2	**0**	**0**	**0**	
3	**0**	**0**	**1**	
4	**0**	**0**	**1**	
5	**0**	**1**	**0**	
6	**0**	**1**	**0**	
7	**0**	**1**	**1**	
8	**0**	**1**	**1**	
9	**1**	**0**	**0**	
10	**1**	**0**	**0**	
11	**1**	**0**	**1**	
12	**1**	**0**	**1**	
13	**1**	**1**	**0**	
14	**1**	**1**	**0**	
15	**1**	**1**	**1**	
16	**1**	**1**	**1**	

8. RESULT:

Hence the experiment on 3-bit DAC has been done and the transfer characteristics have been drawn and resolution has been found out.

9. VIVA -VOCE QUESTIONS:

1. What are ADC's and DAC's?
2. Define Resolution?
3. What are the different types of ADC's?
4. What are the different types of DAC's?
5. Which is the fastest ADC?
6. Tell about any two parameters of ADC's?