WEST GODAVRI INSTITUTE OF SCIENCE & ENGINEERING

DEPARTMENT OF ELECTRONICS & COMMUNICATION ENGINEERING

ANALOG ICs AND APPLICATIONS LABORATORY MANUAL

(R-20)

Prepared by
Sk.M.Unnisha Begum

for

III-B.Tech : I-Sem

(ELECTRONICS &COMMUNICATION ENGINEERING)

WEST GODAVARI INSTITUTE OF SCIENCE AND ENGINEERING
(Approved by AICTE –New Delhi and Affiliated to JNTU – Kakinada)
An ISO 9001 – 2008 Certified College

AVAPADU, PRAKASARAOPALEM – 534112, Nallajerla Mandal, W.G.Dist. (A.P)

JAWAHARLAL NEHRU TECHNOLOGICAL UNIVERSITY: KAKINADA

KAKINADA - 533 003, Andhra Pradesh, India

ELECTRONICS AND COMMUNICATION ENGINEERING

(Applicable for batches admitted from 2022-2023)

III Year – I Semester

L	T	P	C
0	0	3	1.5

ANALOG ICs AND APPLICATIONS LAB

List of Experiments

Minimum Twelve Experiments to be conducted:

1. Study of OP AMPs – IC 741, IC 555, IC 565, IC 566, IC 1496 – functioning, parameters and Specifications.
2. OP AMP Applications – Adder, Subtractor, Comparator Circuits.
3. Integrator and Differentiator Circuits using IC 741.
4. Active Filter Applications – LPF, HPF (first order)
5. Active Filter Applications – BPF, Band Reject (Wideband) and Notch Filters.
6. IC 741 Oscillator Circuits – Phase Shift and Wien Bridge Oscillators.
7. Function Generator using OP AMPs.
8. IC 555 Timer – Monostable Operation Circuit.
9. IC 555 Timer – Astable Operation Circuit.
10. Schmitt Trigger Circuits – using IC 741 and IC 555.
11. IC 565 – PLL Applications.
12. IC 566 – VCO Applications.
13. 4 bit DAC using OP AMP.

EXPERIMENT NO: 1 **Date:**

STUDY OF OP-AMPS

AIM: To study the pin configurations, specifications & functioning of different integrated circuits used in the practical applications.

APPARATUS REQUIRED:

a) IC µA 741 OP-Am
b) NE ISE 555/SE 555C
c) VCO IC 566
d) Phase Locked Loop NE/SE 565
e) IC 723 Voltage Regulator
f) Three Terminal Voltage Regulators

a) µA 741 OP-AMP

Pin configuration

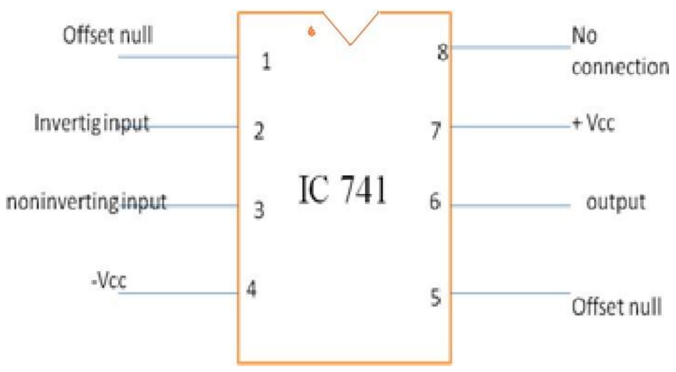


Fig 1.1 Pin diagram for IC 741

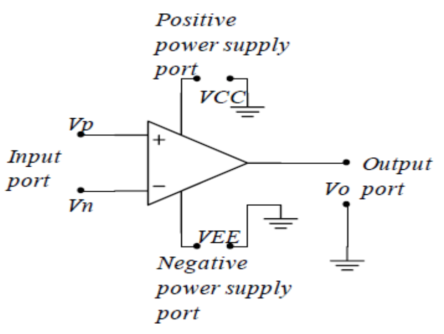


Fig 1.2 Symbol for IC 741

The operational amplifier (op-amp) is a voltage controlled voltage source with very high gain. It is a five terminal four port active element. The symbol of the op-amp with the associated terminals and ports is shown in Figures.

The power supply voltages *VCC* and *VEE* power the operational amplifier and in general define the output voltage range of the amplifier. The terminals labeled with the "+" and the "-" signs are called non-inverting and inverting respectively. The input voltage Vp and Vn and the output voltage Vo are referenced to ground.

Specifications

1. Supply voltage:

µA 741A, µA741, µA741E	±22v
µA 741C	±18v

2. Internal power dissipation

Dip package	310mw
Differential input voltage	±30v

3.Operating temperature range

Military(µA 741A, µA741)	-55^0 to $+125^0$ C.
4.Commercial (µA 741E, µA 741C)	0^0 C to $+70^0$ C.
5.Input offset voltage	1.0 mV.
6.Input Bias current	80 nA.
7.PSRR	30µV/V.
8.Input resistance	2MΩ.
9.CMRR	90dB.
10. Output resistance	75Ω.
11. Bandwidth	1.0 MHz.
12. Slew rate	0.5 V/µ sec.

Applications of IC 741: Adder, substractor, comparator, filters, oscillators

b) NE / SE 555 TIMER

Pin configuration

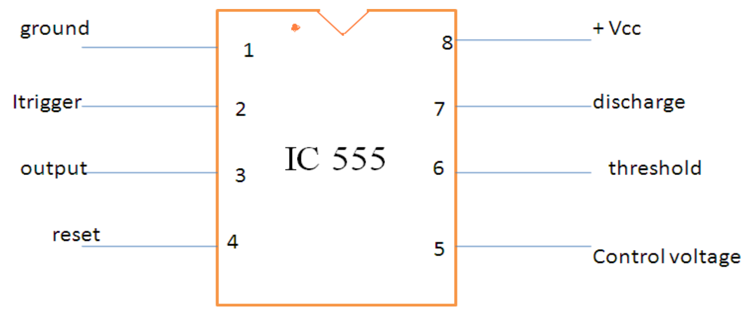


Fig 1.3 Pin diagram for IC 555

One of the most versatile linear ICs is the 555 timer which was first introduced in early 1970 by Signetic Corporation giving the name as SE/NE 555 timer. This IC is a monolithic timing circuit that can produce accurate and highly stable time delays or oscillation. Like other commonly used op-amps, this IC is also very much reliable, easy to use and cheaper in cost. It has a variety of applications including monostable and astable multivibrators, dc-dc converters, digital logic probes, waveform generators, analog frequency meters and tachometers, temperature measurement and control devices, voltage regulators etc. The timer basically operates in one of the two modes either as a monostable (one-shot) multivibrator or as an astable (free-running) multivibrator.The SE 555 is designed for the operating temperature range from – 55°C to 125° while the NE 555 operates over a temperature range of 0° to 70°C.

Specifications:

1. Supply voltage	4.5V to 18V
2. Supply current	3mA
3. Output voltage (low)	0.1V
4. Output voltage (high)	12.5V & 3.3V
5. Maximum operating frequency	500 kHz
6. Timing	µsec to hours

Applications of IC 555: Multi-vibrators, Oscillators, generation of PWM

c) IC 566 pin configuration

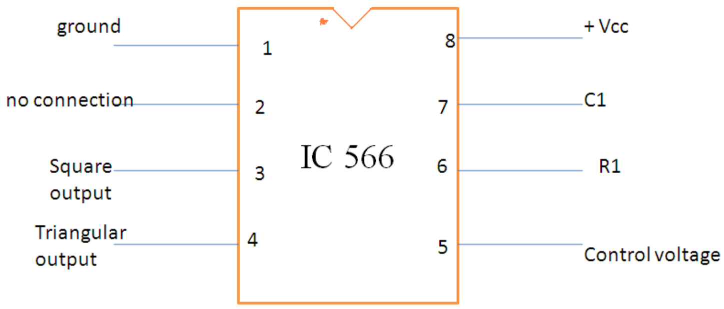


Fig 1.4 Pin diagram for IC 566

The NE/SE566 Function Generator is a voltage-controlled oscillator of exceptional linearity with buffered square wave and triangle wave outputs. The frequency of oscillation is determined by an external resistor and capacitor and the voltage applied to the control terminal. The oscillator can be programmed over a ten-to-one frequency range by proper selection of an external resistance and modulated over a ten-to-one range by the control voltage, with exceptional linearity

Specifications:

1. Operating supply voltage	12V to 24V
2. Operating supply current	12.5mA
3. Input voltage	$3V_{p-p}$
4. Operating temperature	0 to 70°C
5. Power dissipation	30mw

Applications of VCO: Frequency modulation, Voltage to frequency converter

d) NE / SE 565 PHASE LOCKED LOOP
Pin configuration

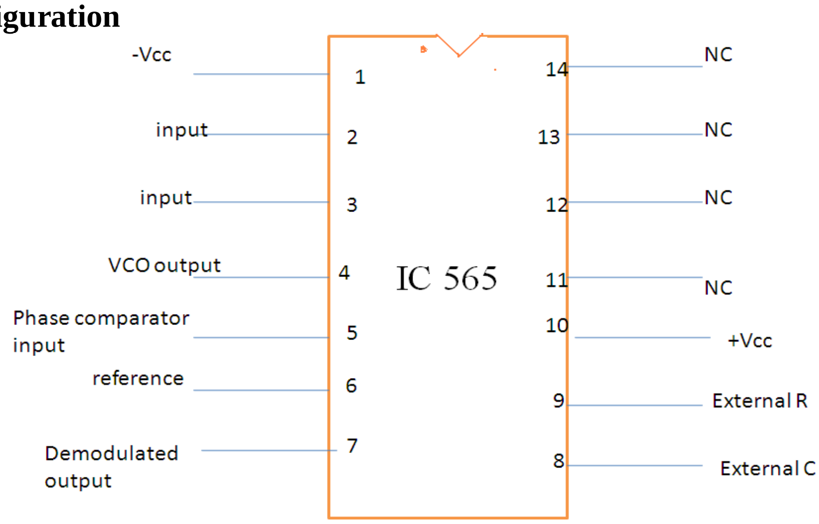


Fig 1.5 Pin diagram for IC 565

Specifications:

1. Maximum supply voltage	26v
2. Input voltage	3v (p-p)
3. Power dissipation	300mw
4. Operating temperature range	NE565-0^0 to 700C, SE 565-55to 1250C
5. Supply voltage	12v
6. Supply current	8mA
7. Output current sink	1mA
8. Output current source	10mA

Applications of IC 565: FM demodulation

e) IC 723 VOLTAGE REGULATOR

Pin configuration

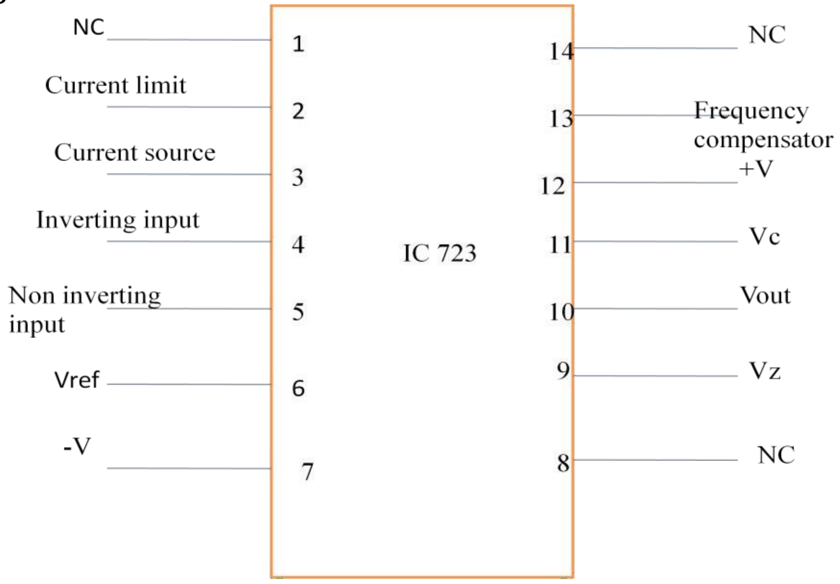


Fig 1.6 Pin diagram for IC 723

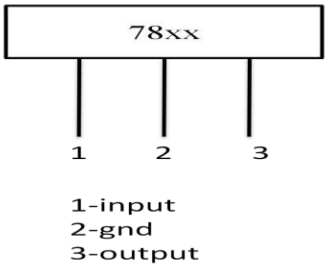
The 723 voltage regulator is commonly used for series voltage regulator applications. It can be used as both positive and negative voltage regulator. It has an ability to provide up to 150 mA of current to the load, but this can be increased more than 10A by using power transistors. It also comes with comparatively low standby current drain, and provision is made for either linear or fold-back current limiting. LM723 IC can also be used as a temperature controller, current regulator or shunt regulator and it is available in both Dual-In-Line and Metal Can packages. The input voltage ranges from 9.5 to 40V and it can regulate voltage from 2V to 37V.

Specifications:

1. Input voltage	40v max
2. Output voltage	2v to 37v
3. Output current	150mA
4. Input regulation	0.02%
5. Load regulation	0.03%
6. Operating temperature	55^0C to125^0C

f) THREE TERMINAL VOLTAGE REGULATORS

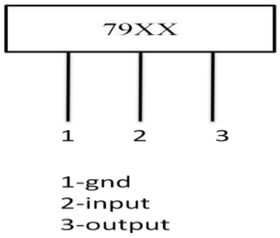
i) IC 78XX (Positive Voltage Regulators)
Pin configuration



Specifications

1. Input voltage
   For 5V to 18V regulated output — 35V.
   Up to 24V regulated output — 40V.
2. Internal power dissipation — internally limited.
3. Storage temperature range — -65^0 C to 150^0 C.
4. Operating junction Temperature range
   µA7800 — -55^0 C to 150^0 C.
   µA7800C — 0^0 C to 125^0 C.

ii) IC 79XX (Negative Voltage Regulators)
Pin configuration



Specifications:

1. Input voltage
   For -5v to -18v regulated output — -35V
   For -24v regulated output — -40V
2. Internal power dissipation — internally limited
3. Storage temperature range — -65^o C to 150^o C
4. Operating junction temperature range
   µA7800 — -55^o C to 150^o C
   µA7800C — 0^o C to 125^o C

RESULT:

The pin configurations, specifications & functioning of different integrated circuits used in the practical applications have been studied

EXPERIMENT NO: 2 **Date:**

APPLICATIONS OF OPERATIONAL AMPLIFIER (IC 741)

AIM: To design and study the operation of IC 741 Operational amplifier as

a) Adder
b) Subtractor
c) Comparator

APPARATUS REQUIRED:

1. Bread Board.
2. Function Generator
3. Cathode Ray Oscilloscope.
4. Digital Multimeter.
5. Regulated Power Supply (Dual Channel).
6. Connecting Wires.

COMPONENTS REQUIRED:

1. IC 741	:1No
2. Resistor………10kΩ	:5No

CIRCUIT DIAGRAMS:

a) ADDER

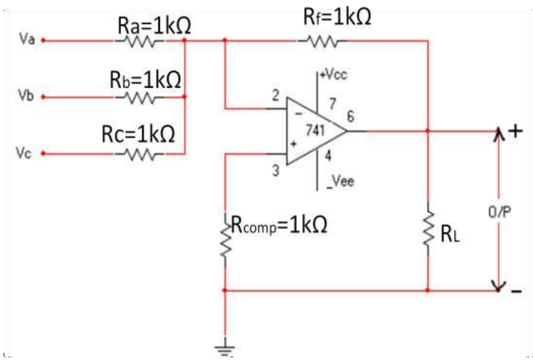


Fig 2.1 adder

b) SUBTRACTOR

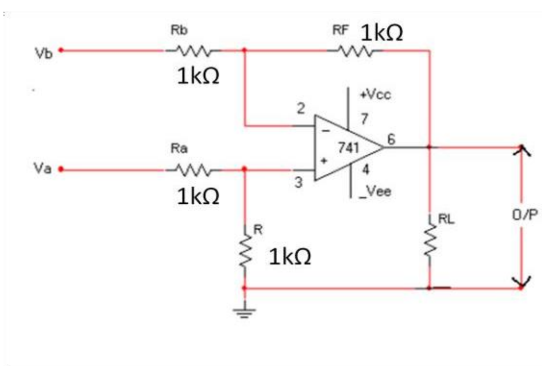


Fig 2.2 subatractor

c) COMPARATOR

i. Non-Inverting Comparator

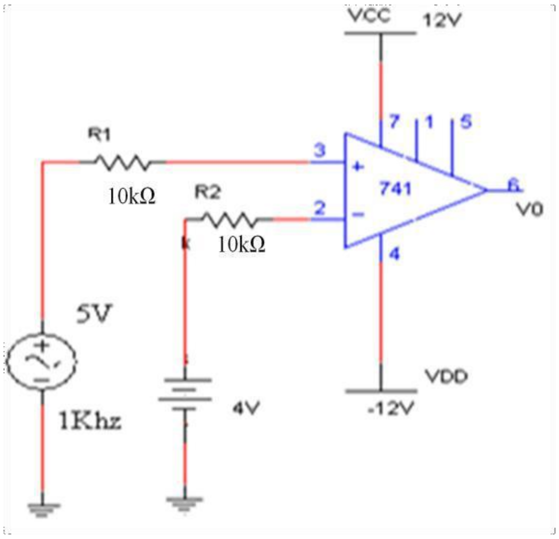


Fig 2.3 comparator

THEORY:

Adder:

A typical summing amplifier (Inverting Adder) with three inputs Va ,Vb & Vc applied at the inverting terminal of IC741 is shown in fig(1). The following analysis is carried out assuming that the Op-Amp is an ideal one, that is AOL = ∞, Ri = ∞ & R0=0; since the input bias current is assumed to be zero, there is no voltage drop across the resistor Rcomp and hence the non inverting input terminal is at ground potential.

The voltage at node „A" is zero as the non- inverting input terminal is grounded. The nodal equation by KCL at node „a" is given as

$$\frac{v_a}{R_a} + \frac{v_b}{R_b} + \frac{v_c}{R_c} + \frac{v_0}{R_f} = 0$$

$$v_0 = -((\frac{R_f}{R_a})\, v_a + (\frac{R_f}{R_b})\, v_b + (\frac{R_f}{R_c})\, v_c)$$

Case (1):- $R_a=R_b=R_c=R_f$

$$V_0 = -(V_a + V_b + V_c)$$

Case (2):- $R_a=R_b=R_c=3R_f$

$$V_0 = -(V_a + V_b + V_c)/3$$

Subtractor

A typical subtractor with two inputs Va & Vb applied at the non-inverting terminal & Inverting terminal of IC741 respectively is shown in fig(2). The following analysis is carried out assuming that the Op-Amp is an ideal one, that is AOL = ∞, Ri = ∞ & R0=0;

Let Ra = Rb= Rf = R,
V_O= Va – Vb

COMPARATOR:

A comparator is a circuit which compares a signal voltage applied at one input of an op-amp with a known reference voltage at the other input. It is basically an open loop op-amp with output ±Vsat as in the ideal transfer characteristics. It is clear that the change in the output state takes place with an increment in input Vi of only 2mv.This is the uncertainty region where output cannot be directly defined There are basically 2 types of comparators.
1. Non inverting comparator and.
2. Inverting comparator.

PROCEDURE:

<u>Part-I</u>

Adder

1. Connect the Adder circuit as shown in fig.1 with Ra = Rb = Rc = Rf = 10KΩ, RL =100KΩ and R = 250Ω on the CDS board
2. Switch „ON‟ the power supply and apply + 15V to pin no.7 and -15V to pin no.4 of the IC741.
3. Apply the input voltages from the regulated supplies to the corresponding inputs at the inverting Input terminal of IC741 (pin no.2).
3. Connect the Digital Multimeter at the Output terminals (pin no.6), and note down the Output voltage and verify with theoretical values.
4. Repeat the above steps for different input voltages.

Subtractor

1. Connect the subtractor circuit as shown in fig.2 with Ra = R_b = R_f = R = 10KΩ and R_L =100KΩ on the CDS board
2. Switch „ON‟ the power supply and apply + 15V to pin no.7 and -15V to pin no.4 of the
3. Apply the input voltages from the regulated supplies to the corresponding inputs at the inverting & non-inverting input terminals of IC741 (pin no.2 & 3 respectively).
4. Connect the Digital Multimeter at the Output terminals (pin no.6), and note down the output voltage and verify with theoretical values.
5. Repeat the above steps for different input voltages

<u>Part-II</u>

Comparator

1. Connect the comparator circuit as shown in fig.3.
2. Connect the 1MHz function generator to the input terminals. Apply 1V signal at non-inverting terminals of the op-amp IC741.
3. Connect the 20MHz C.R.O at the output terminals.

4. Keep 1V reference voltage at the Inverting terminal of the Op-amp. When Vin is less than the Vref, then output voltage is at –Vsat because of the higher input voltage at negative terminal. Therefore the output voltage is at logic low level
5. Now, Keep –1V reference voltage. When Vref is less than the Vin, then the output voltage is at +Vsat because of the higher input voltage at positive terminal. Hence, the output voltage is at logic high level.
6. Observe and record the output voltage and waveforms.

OBSERVATIONS
ADDER:

V1(volts)	V2(volts)	Theoretical V0=-(V1+V2)	Practical V0 =-(V1+V2)

SUBTRACTOR:

V1(volts)	V2(volts)	Theoretical V0=(V1-V2)	Practical V0 =(V1-V2)

<u>Observations for comparator:</u>

Input signal

Amplitude = Time period =

Output signal

Amplitude = Time period =

EXPECTED WAVEFORMS FOR COMPARATOR:

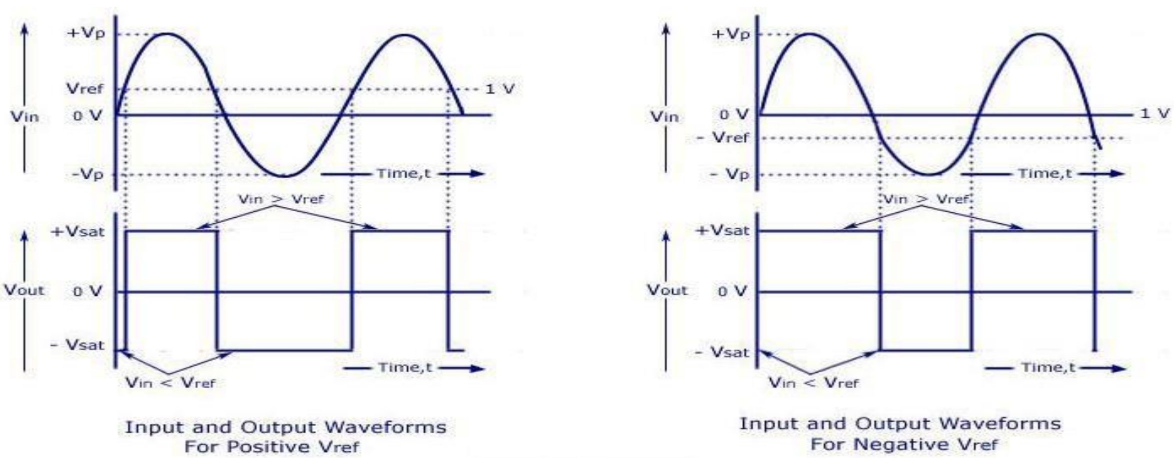


Fig 2.4 output wave form of a non inverting comparator for +v_{ref} and -Vref

PRECAUTIONS:
1. Make null adjustment before applying the input signal.
2. Maintain proper Vcc levels.

Applications of adder and subtractor:
1. Digital signal processing
2. Communication

Applications of comparator:
1. Zero crossing detector
2. Level detector
3. Time marker generator
4. Window detector

RESULT:

Adder and Subtractor are designed using 741 Op – Amp and the experimental results were compared with the theoretical values.

Applied input signal is compared with reference voltages in a comparator using 741 Op – Amp and the corresponding waveforms were noted.

EXPERIMENT NO: 3 **Date:**

INTEGRATOR AND DIFFERENTIATOR

AIM: To design, construct and verify the response of

a) Integrator using Op-amp IC741 for sine and square wave inputs at 1 KHz frequency.
b) Differentiator using Op-amp IC741 for sine and square wave inputs at 1 KHz frequency.

APPARATUS REQUIRED:

1. Bread Board / CDS Board.
2. Function Generator (1MHz).
3. Cathode Ray Oscilloscope (20MHz/30 MHz)
4. Regulated Power Supply (Dual Channel).
5. Connecting Wires.

COMPONENTS REQUIRED:

IC 741	1No
Resistor-------1kΩ	1No
10kΩ	2No
100kΩ	1No
Capacitor-----0,1µf	1No

CIRCUIT DIAGRAMS:

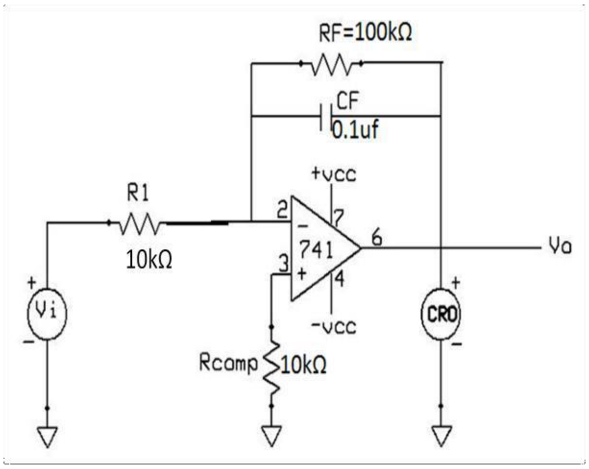


Fig 3.1 Integrator

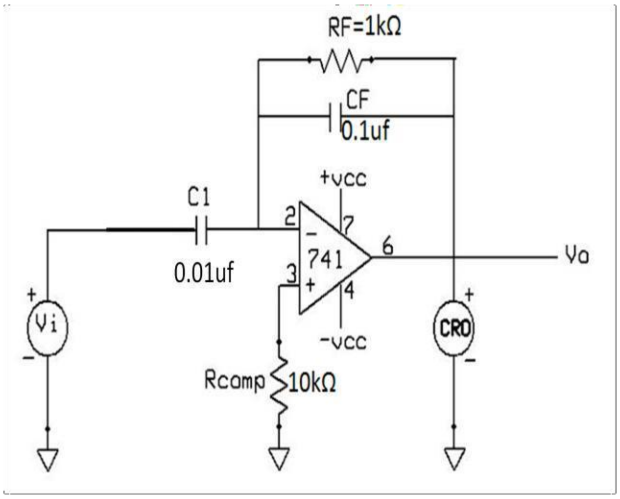


Fig 3.2 Differentiator

THEORY:

The integrator

A circuit in which the output voltage waveform is the integration of the input is called integrator. $V_0=-1/R_4C_1$

The above equation indicates that the output voltage is directly proportional to the negative

integral of the input voltage and inversely proportional to the time constant R1CF. For Example if the input is a sine wave, the output will be a cosine wave or if the input is a square wave, the output will be a triangular wave.

1. When the input signal frequency is ZERO, the integrator works as an open – loop amplifier. This is because of the capacitor C_F acts as an open circuit (X_{CF} =1/ωCF = infinite for f=0).
2. Therefore the ideal integrator becomes unstable & suffers with low frequency noise. To overcome this problem R_F is connected across the feedback capacitor C_F. Thus RF limits the low-frequency gain and hence minimizes the variations in the output voltage.
3. Frequency f_b at which the gain of the integrator is 0 dB, is given by

$$f_b = 1/2\pi R1CF$$

4. Both the stability and the low – frequency roll-off problems can be corrected by the addition of a resistors R_F in the feedback path.

NOTE: The input signal will be integrated properly if the time period T of the input signal is greater than or equal to R_FC_F.

The Differentiator

The differentiator circuit performs the mathematical operation of differentiation. That is the output waveform is the derivative of the input waveform. Therefore

$$V_O = R_f C_1 \, dVin / dt$$

1. The above equation indicates that the output voltage is directly proportional to the derivative of the input voltage and also proportional to the time constant R_FC_1.
2. For Example if the input is a sine wave, the output will be a cosine wave or if the input is a square wave, the output will be spikes.
3. The reactance of the circuit increases with increase in frequency at a rate of 20dB/ decade. This makes the circuit unstable. In other words the gain of an ideal differentiator circuit is direct dependent on input signal frequency. Therefore at high frequencies (f=∞), the gain of the circuit becomes infinite making the system unstable.
4. The input impedance XC1 decreases with increase in frequency, which makes the circuit very susceptible to high frequency noise.
5. The frequency response of the basic differentiator is shown in fig 3.4 In this fig fa is the frequency at which the gain is 0 dB.

$$fa = 1/2\pi R_FC_1$$

6. Both the stability and the high – frequency noise problem can be corrected by the addition of two components R_1 and C_F as shown in fig 3.2.The frequency response of which is

shown in fig 3.4. From f to fa the gain decreases at 40dB/decade. This 40 dB/decade change in gain is caused by the R_1C_1 and R_FC_F combinations

NOTE: The input signal will be differentiated properly if the time period T of the input signal is greater than or equal to R_F C_1.

PROCEDURE:

Integrator

1. Connect the circuit as shown in fig 3.1 on the breadboard.
2. Switch „ON" the power supply and apply + 15V to pin no.7 and -15V to pin no.4 of the IC741.
3. Apply a sine wave input signal of 2V peak-to-peak amplitude at 1 KHz frequency from the function generator (at pin no.2 of the IC741).
4. Connect the C.R.O at (pin no.6) the output terminals.
5. Observe and plot the input & output voltage waveforms.
6. Measure the output voltage (Vo) from the experimental results.

Differentiator

1. Connect the circuit as shown in fig 3.2 on the breadboard.
2. Switch „ON" the power supply and apply + 15V to pin no.7 and -15V to pin no.4 of the IC741.
3. Apply a sine wave input signal of 2V peak-to-peak amplitude at 1 KHz frequency from the function generator (at pin no.2 of the IC741).
4. Connect the C.R.O at (pin no.6) the output terminals.
5. Observe and plot the input & output voltage waveforms.
6. Measure the output voltage (Vo) from the experimental results.

EXPECTED WAVEFORMS:

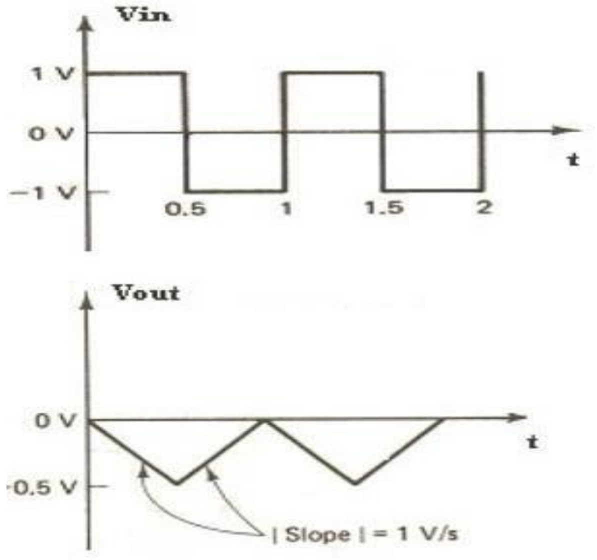


Fig 3.3 Output waveform of Integrator

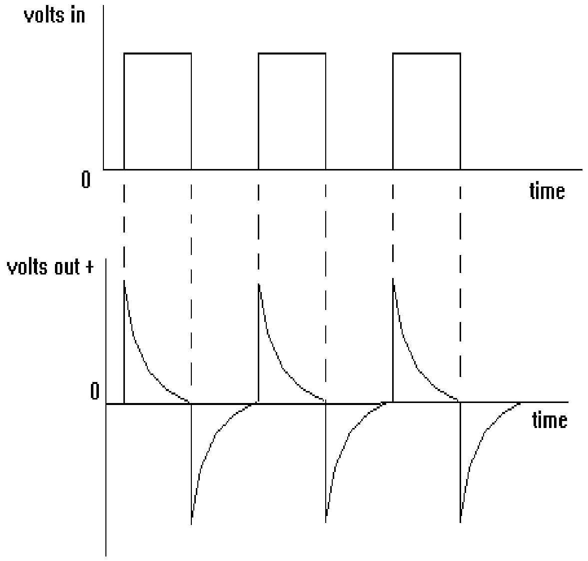


Fig 3.4 Output waveform of Differentiator

<u>Observations for integrator:</u>

Input signal

Amplitude = Time period =

Output signal

Amplitude = Time period =

<u>Observations for differentiator:</u>

Input signal

Amplitude = Time period =

Output signal

Amplitude = Time period =

Applications of integrator:
1. In the analog computers
2. In solving differential equations
3. In analog to digital converters
4. In various signal wave shaping circuits
5. In ramp generators

Applications of differentiator:
1. In wave shaping circuits
2. as a rate of change detector in FM demodulators

RESULT: The Integrator & Differentiator circuits were constructed using IC 741 and verified their response for sine & square wave inputs.

EXPERIMENT NO: 4 **Date:**

FREQUENCY RESPONSE OF LOW PASS AND HIGH PASS ACTIVE FILTERS

AIM: To design, construct and plot the frequency response of

a) First order low pass filter with cut-off frequency of 5 KHz
b) First order high pass filter with a cut-off frequency of 1 KHz.

APPARATUS REQUIRED:

1. Bread Board / CDC Board.
2. Function Generator (1MHz).
3. Cathode Ray Oscilloscope (20MHz/30 MHz)
4. Regulated Power Supply (Dual Channel).
5. Connecting Wires.

COMPONENTS REQUIRED:

IC 741	1No
Resistor-------1kΩ	1No
10kΩ	2No
4.7kΩ	1No
Capacitor-----0.1µf	1No

CIRCUIT DIAGRAMS:

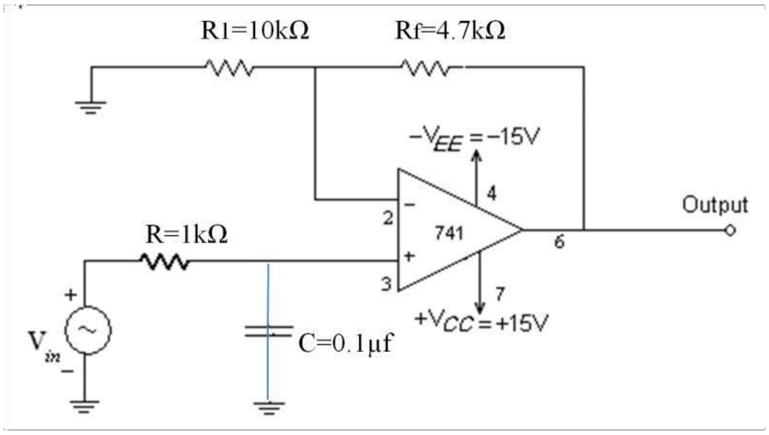


Fig 4.1 Low pass filter

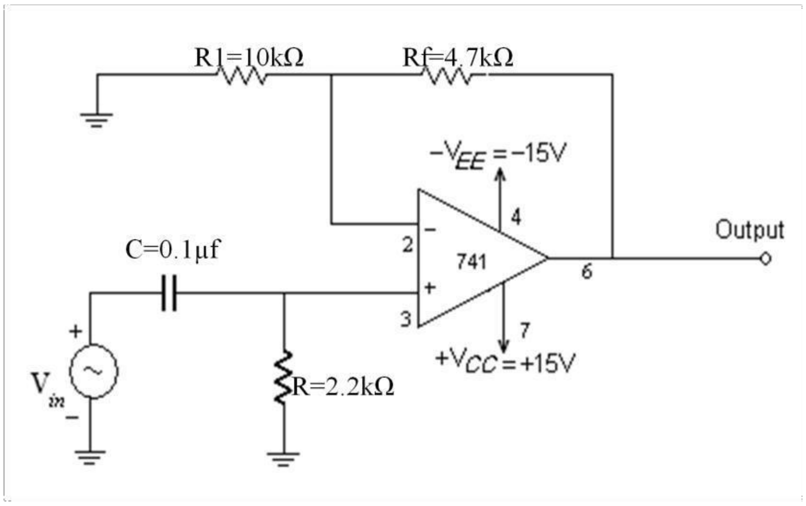


Fig 4.2 High pass filter

THEORY:

A first order filter consists of a single RC network connected to the non-inverting input terminal of the op-Amp as shown in the figure. Resistors R_1 & R_f determine the gain of the filter in the pass band. Components R & C determine the cutoff frequency of the filter.

Low-Pass filter: The circuit of 1st order low-pas filter is shown in fig.1 & its frequency response is as shown in the fig3. The dashed curve in the fig 4.3 indicates the ideal response & solid curve indicates practical filter response. It is not possible to achieve ideal characteristics. However with special design techniques (Higher order filters) it is possible to closely approximate the ideal response. Active filters are typically specified by the voltage transfer function,

H(s) = V0 (s)/ Vi(s)……………… (1) (under steady state conditions)

High Pass Filter: The circuit of 1st order high pass filter is shown in fig.2 & its frequency response is as shown in the fig4.4 the dashed curve in the fig 4.4 indicates the ideal response & solid curve indicates practical filter response. When an input signal is applied to High pass filter, the signals at high frequencies are passed through circuit and signals at low frequencies are rejected. That is the signal which are having frequencies less than the lower cutoff frequency f_L are rejected and the signal with frequency greater the lower cut off frequency f_L are passed through the circuit. That is

1. For $f > f_L$, Vo(s) /Vi(s) = Maximum and is called as pass band.
2. For $f < f_L$, Vo(s) /vi(s) = 0 and is called as the stop band

PROCEDURE:

1. Connections are made as per the circuit diagram.
2. Apply sine wave of amplitude 4Vp-p to the non inverting input terminal.
3. Values the input signal frequency.
4. Note down the corresponding output voltage.
5. Calculate gain in db.
6. Tabulate the values.
7. Plot a graph between frequency and gain.
8. Identify stop band and pass band from the graph

OBSERVATIONS:

Low Pass Filter

Input signal amplitude:

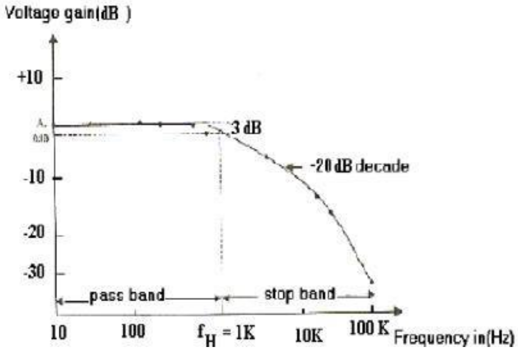
Frequency(Hz)	V0(V)	Gain =(V0/Vi)	Gain in db= 20log(V0/Vi)

High Pass Filter

Input signal amplitude:

Frequency(Hz)	V0(V)	Gain =(V0/Vi)	Gain in db= 20log(V0/Vi)

EXPECTED WAVEFORMS:



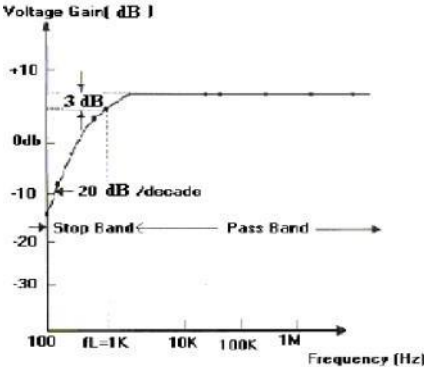


fig 4.3 Frequency response of 1st Order LPF   fig 4.4 Frequency response of 1st Order HPF

Applications of filters:

1. In communications systems, use filters to suppress noise, to isolate a single communication from many channels, to prevent spillover of adjacent bands, and to recover the original message signal from modulated signals.

2. In instrumentation systems, engineers use filters to select desired frequency components or eliminate undesired ones. In addition, we can use these filters to limit the bandwidth of analog signals before converting them to digital signals. You also need these filters to convert the digital signals back to analog representations.

3. In audio systems, engineers use filters in crossover networks to send different frequencies to different speakers. In the music industry, record and playback applications require fine control of frequency components.

4. In biomedical systems, filters are used to interface physiological sensors with data logging and diagnostic equipment.

RESULT: The first order LPF & HPF are designed for a chosen cutoff frequency and the frequency response curves were plotted between voltage gain (dB) and frequency (Hz).

EXPERIMENT NO: 5 **Date:**

FREQUENCY RESPONSE OF BAND PASS AND BAND REJECT ACTIVE FILTERS

AIM: To design, construct and study the frequency response of

a) Band pass filter
b) Band reject filter

APPARATUS REQUIRED:

1. Bread Board / CDS Board.
2. Function Generator
3. Cathode Ray Oscilloscope
4. Regulated Power Supply (Dual Channel).
5. Connecting Wires.

COMPONENTS REQUIRED:

IC 741	1No
Resistor-------1kΩ	1No
10kΩ	2No
4.7kΩ	1No
Capacitor-----0.1µf	1No

CIRCUIT DIAGRAMS: BAND PASS FILTER

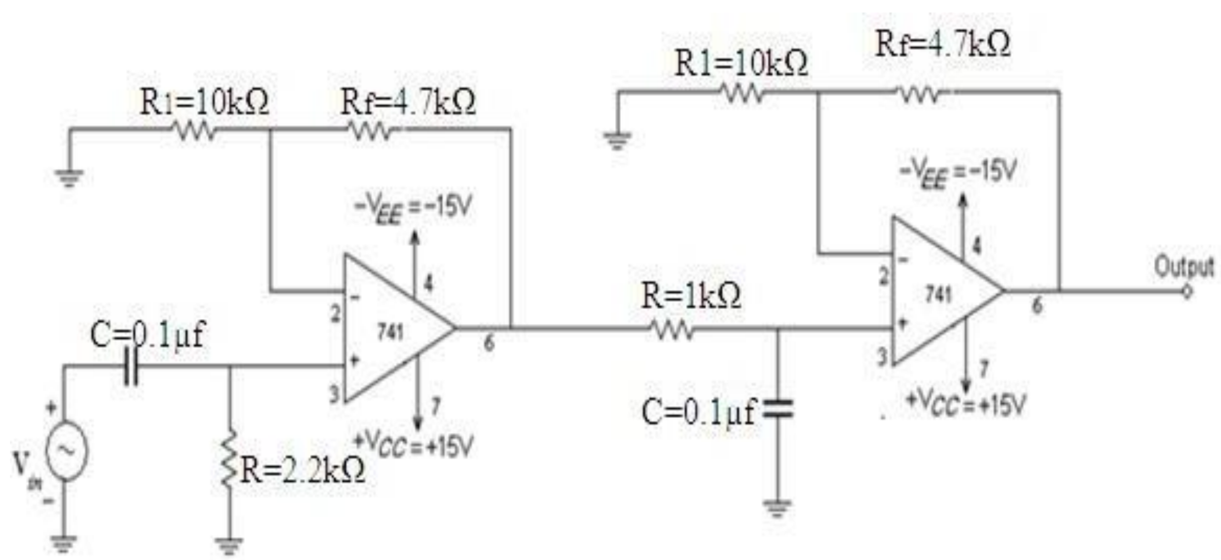


Fig 5.1 Band pass filter

BAND REJECT FILTER

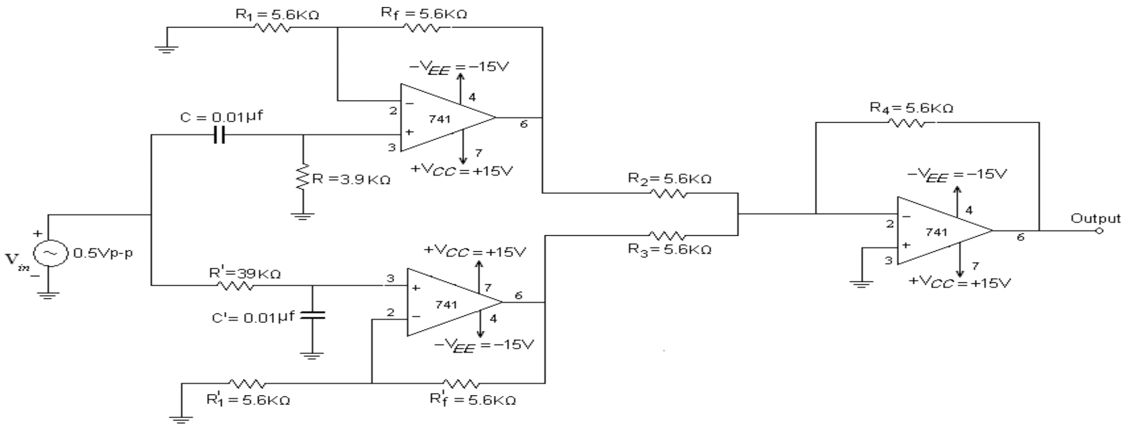


Fig 5.2 Band reject filter

THEORY:
BAND PASS FILTER:

A Band Pass Filter (BPF) has a pass band between the lower cut-off frequency, f_L & the higher cut-off frequency f_H, such that $f_H > f_L$. When the input frequency is zero, the gain of the filter will be zero. As the input signal frequency increases from zero to f_L, the gain will increase at a rate 20dB/decade up to 3dB less than its maximum value. If the input signal frequency increases beyond f_L, the gain will reach its maximum value and remains constant up to high frequencies as shown in the Fig 5.3. When the input signal frequency reaches the higher cut-off frequency, f_H, the gain will fall 3dB less from its maximum value. If the input signal frequency increases beyond f_H, the gain will decreases to zero at rate of 20dB/decade. After reaching the total pass band region, the gain of the filter is constant up to its designed f_H (high cut off frequency).

There is a phase shift between input and output voltages of BPF as a function of frequency in its Pass Band region. This filter passes all frequencies equally well i.e. the output and input voltages are equal in amplitude for all frequencies. This highest frequency up to which the input and output amplitudes remain equal is dependent of the unity gain bandwidth of Op – Amp. At this frequency, the phase shift between input and output becomes maximum.

BAND REJECT FILTER:

A Band Reject Filter (BRF) has a stop band between the cutoff frequencies f_H & f_L such that $f_H < f_L$. When the input signal frequency is zero, the gain of the BPF will be maximum and will remains constant as the input signal frequency increases. At the higher cut off frequency f_H, the gain becomes 3dB less than its maximum value. As the input signal frequency increases beyond f_H, the gain of the filter decreases & becomes zero at the central (f_C) or operating frequency (f_O). After this center frequency f_C, the gain increases to 3dB less than its maximum value at the lower cut-off frequency, f_L. As the input signal frequency increases beyond f_L the gain increases to the maximum value and becomes constant.

There is a phase shift between input and output voltages of BPF in its "Pass band region". This filter passes all the frequencies equally well i.e. output and input voltages are equal in (magnitude) amplitude for all frequencies. This highest frequency up to which the input and output amplitude remains equal is dependent on the unity gain bandwidth of the Op- Amp. However at this frequency, the phase shift between the input and output is maximum.

PROCEDURE:

1. Make the circuit connection as shown in figure.
2. Connect the signal generator to input terminals. And connect the C.R.O at output terminals of the trainer & switch on the trainer.
3. Apply the input signal frequency from 100Hz to 10 KHz.
4. Record the input frequency, Input voltage and Output voltage. Find the gain of the B.P.F using the formula. The gain magnitude in dB is equal to 20 Log (Vo/Vi).

OBSERVATION TABLES: Band Pass Filter:

Input signal amplitude:

Frequency(Hz)	V0(V)	Gain =(V0/Vi)	Gain in db= 20log(V0/Vi)

Band Reject Filter:

Input signal amplitude:

Frequency(Hz)	V0(V)	Gain =(V0/Vi)	Gain in db= 20log(V0/Vi)

EXPECTED WAVEFORMS:

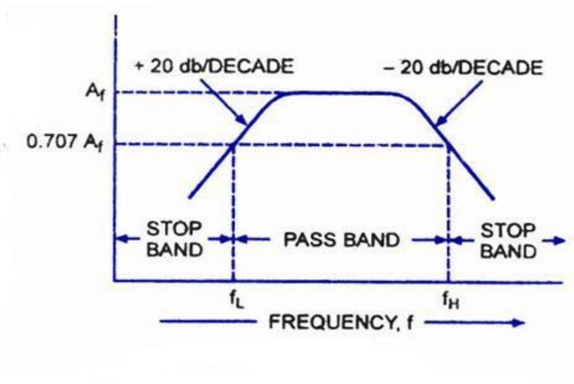


Fig 5.3 frequency response for band pass filter

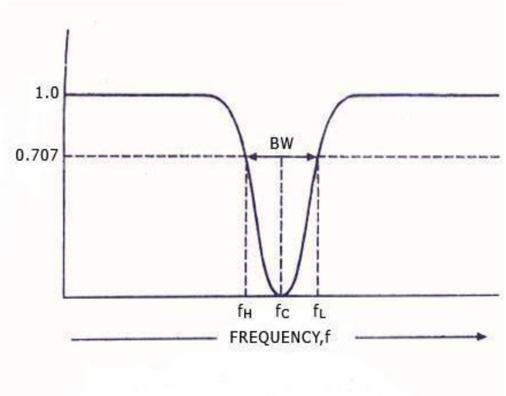


Fig 5.4 frequency response for band reject filters

RESULT: The band pass & band reject filters have been designed for chosen f_L, f_H and frequency responses were plotted between voltage gain (in dB) and input frequency.

EXPERIMENT NO: 6 **Date:**

IC 741 Oscillator Circuits –RC Phase Shift and Wein Bridge Oscillators.

AIM: To study the Operation of Wein – Bridge Oscillator and RC phase shift oscillator using IC 741 Op-Amp and to determine the frequency of Oscillations.

APPARATUS REQUIRED:

1. CDS Board/ Bread Board
2. Regulated DC power Supply
3. C.R.O
4. Connecting patch chords

COMPONENTS REQUIRED:

IC 741	1No
Resistor ------1kΩ	4No
1.5kΩ	2No
15kΩ	1No
100kΩ	1No
Capacitor-----0.1µf	3No

CIRCUIT DIAGRAM:

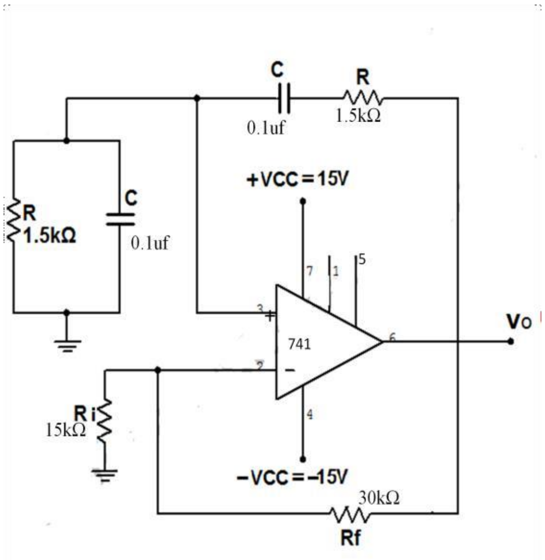


Fig 6.1 Wein bridge oscillators

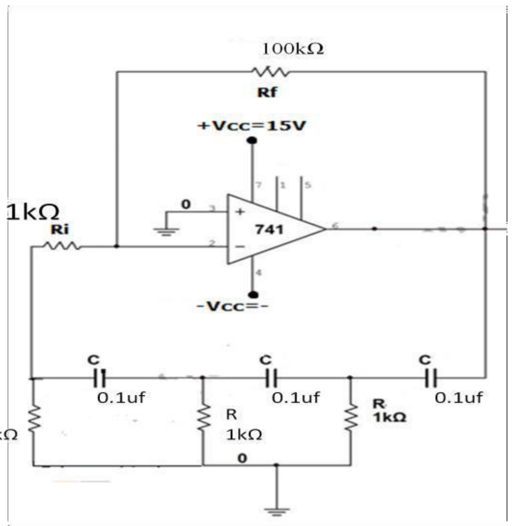


Fig 6.2 RC phase shift oscillator

THEORY:
Wein bridge oscillator:

The most commonly used audio frequency oscillator is weinbridge oscillator. From the figure shown above it may be noted that the feedback signal in the circuit is connected to the positive terminal so that the Op-Amp is working as a non-inverting amplifier. Therefore the feedback network needs not to provide any phase shift. The circuit can be viewed as a weinbridge

with a series RC network in one arm and a parallel RC network in the adjoining arm. The addition of zero phases around the circuit is achieved by balancing the bridge.
The frequency of the oscillations in Wein Bridge is given by

$$F_0 = 1/\ 2\pi RC$$

At F_O, The feedback factor β is equal to 1/3. Therefore for sustained oscillation, the amplifier must have a gain of precisely 3. However from practical point of view, A_V may be slightly less or greater than 3. For $A_V < 3$ the oscillations will either die down or fail to start when power is first applied and for $A_V > 3$, the oscillations will be growing.

RC phase shift oscillator:

Phase shift oscillator which consists of an Op-Amp as the amplifying stage & three RC cascaded networks as the feedback circuit that provides feedback voltage from the output back to the input of the amplifier. The output is used in inverting mode. Therefore any signal that appears at the inverting terminal is shifted by 180^0 phase shift required for oscillation is provided by the cascaded RC networks. Thus the total phase shift of the cascaded RC networks is exactly 360^0 or 00. At some specific frequency when the phase shift of the cascaded RC network is exactly 180^0 and the gain of the amplifier is sufficiently large and circuit oscillates at that frequency.

$$\frac{R_f}{R_1} = 29 \ \Rightarrow \ R_f = 29R_1$$

Thus the circuit will produce a sinusoidal waveform of frequency f_o if the gain is 29 and the total phase shift around the circuit is exactly 360^0.

PROCEDURE:

Wein bridge oscillator:

1. Connect the circuit as shown in the figure1.
2. Connect the C.R.O at the output terminals and observe the output waveform.
3. Record the output waveform and measure the practical frequency from the waveform.
4. For different values of R, calculate theoretical frequency using the formula $F_O = 1/ 2\pi RC$ and also measure the frequency of output signal from the waveform.
5. Compare the theoretical and practical frequencies of the output signal.

RC phase shift oscillator:

1. Connect the circuit as shown in figure1.
2. Connect Oscilloscope at output terminals V_0 observe the output sine wave.
3. Record the output waveform and measure the practical frequency of the output waveform.
4. From the given values of R & C calculate theoretical frequency using the formula $f=1/2\pi RC\sqrt{6}$.
5. Compare the theoretical and practical frequencies.

Observations for Wein bridge oscillator:

Output signal

Amplitude =   Time period =

S.NO	**R (in Ohms)**	**C (in µF)**	**f theoretical = 1/2∏RC (in Hz)**	**f practical (in Hz)**

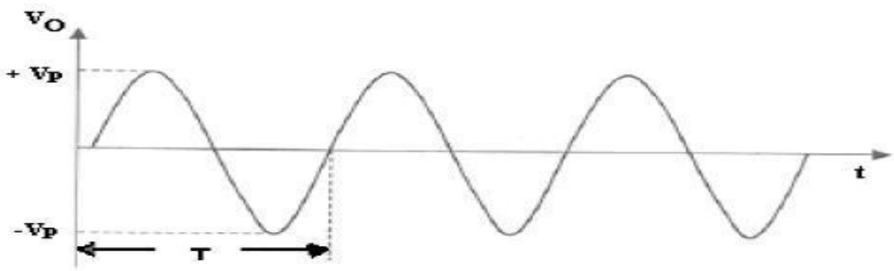
RC phase shift oscillator:

Output signal

Amplitude = Time period =

S.NO	R (in Ohms)	C (in μF)	f theoretical = $1/2\pi RC\sqrt{6}$ (in Hz)	f practical (in Hz)

EXPECTED WAVEFORMS:



RESULT: Operation of Wein –Bridge and RC phase shift oscillators using IC 741 Op-Amp is studied and frequency of the oscillations is determined.

EXPERIMENT NO: 7 **Date:**

FUNCTION GENERATOR USING IC 741 OP – AMP

AIM: Study of op-Amp as function generator that produces various specific waveforms for test purpose over a wide range of frequencies.

APPARATUS:

1. Function Generator Trainer kit.
2. C. R.O.
3. Digital Multimeter.
4. Connecting Wires.

CIRCUIT DIAGRAM:

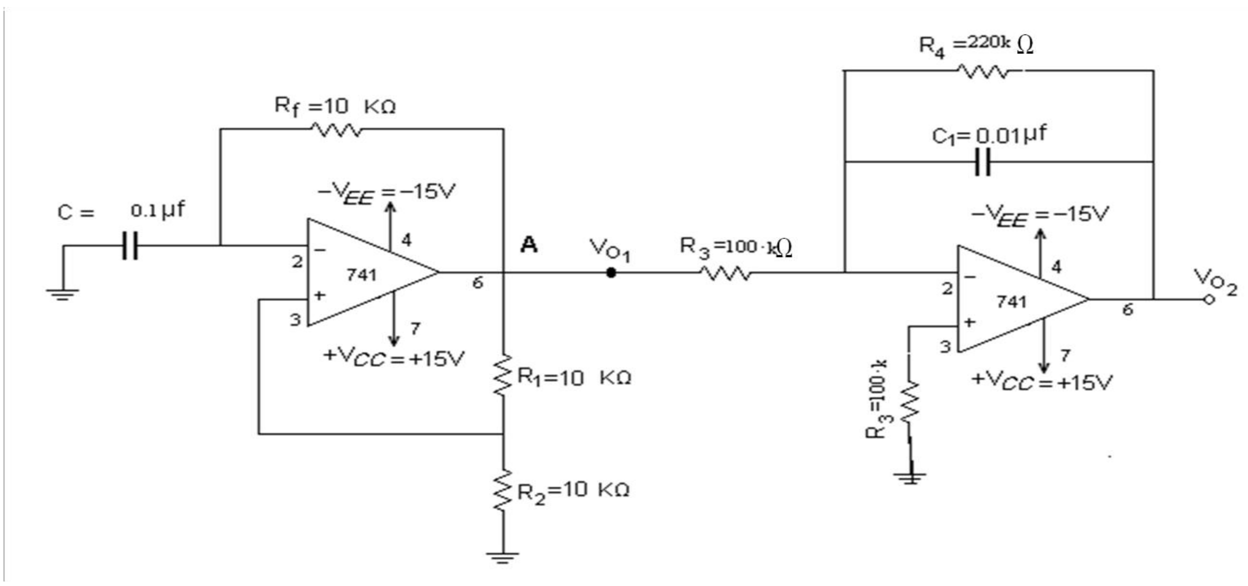


Fig 7.1 Function generator using ic 741 op – amp

THEORY:

Function generator is a signal generator that produces various specific waveforms for test purposes over a wide range of frequencies. In laboratory type function generator generally one of the functions (sine, square & triangle) is generated using dedicated chips or standard circuits and converts it in to required signal.

This consists of

1. Sine wave generator Using IC 741.
2. Square wave generator (Astable Multivibrator using IC 741)
3. Active integrator using IC 741

SQUARE WAVE GENERATOR (ASTABLE MULTIVIBRATOR)

In comparison to sine wave oscillations, square wave signals are generated when the Op-Amp is forced to operate in saturated region. That is the output of the Op-Amp is forced to swing between +Vsat & -Vsat, resulting in square wave output. The circuit arrangement of a square wave generator using IC 741 is shown in fig.

TRIANGULAR WAVE GENERATOR:

The circuit arrangement of a triangular wave generator is shown in Fig.2. A square wave from the square wave generator is fed to the integrator. The RC time constant of the integrator has been chosen in such a way that it is very small value compared to the time period of the incoming square wave. For the basic operation of integrator, it is known that the output of the integrator for a given square wave input is a triangle wave.

PROCEDURE:

1. Connect trainer kit to the 230V AC mains and switch on the supply.
2. Observe the output of the sine wave generator. If signal is not coming or distorted in shape adjust the gain trim pot provided on the kit until a good signal is obtained. Measure the signal frequency using Oscilloscope.
3. Observe the output of the square wave generator and measure the output signal frequency.
4. Observe the output of the Integrator (triangular wave generator) by varying the input signal frequency (square wave is internally connected to the circuit).
5. Measure the frequency of the triangular wave using CRO.

WAVEFORMS:

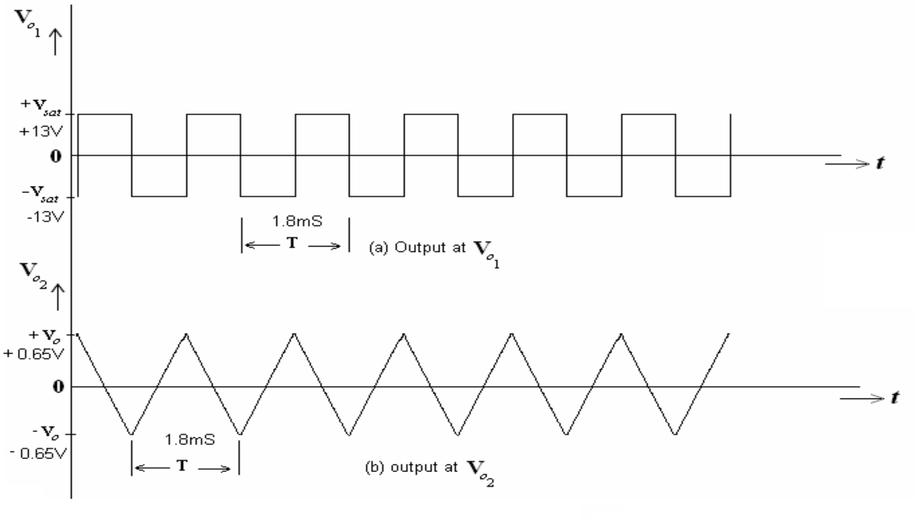


Fig 7.2:expected output waveforms of function generator

Observations:

Square wave Output signal

Amplitude = Time period =

Triangle wave Output signal

Amplitude = Time period =

Applications:

1. square wave generators
2. triangular wave generators
3. sine wave generators

RESULT: Hence studied op-Amp as function generator that produces Sine, square and triangular waveforms for test purpose over a wide range of frequencies.

EXPERIMENT NO: 8 **Date:**

MONOSTABLE MULTIVIBRATOR USING 555 TIMER

AIM: To design a Monostable Multivibrator using 555 timer to get 10msec. pulse output.

APPARATUS REQUIRED:

1. C.R.O
2. Regulated DC power Supply
3. Function generator
4. CDS Board/ Bread Board.
5. Connecting patch chords.

COMPONENTS REQUIRED:

IC 555	1No
Resistor-------10kΩ	1No
Capacitor-----0.1µf	1No
0.01µf	1No

CIRCUIT DIAGRAM:

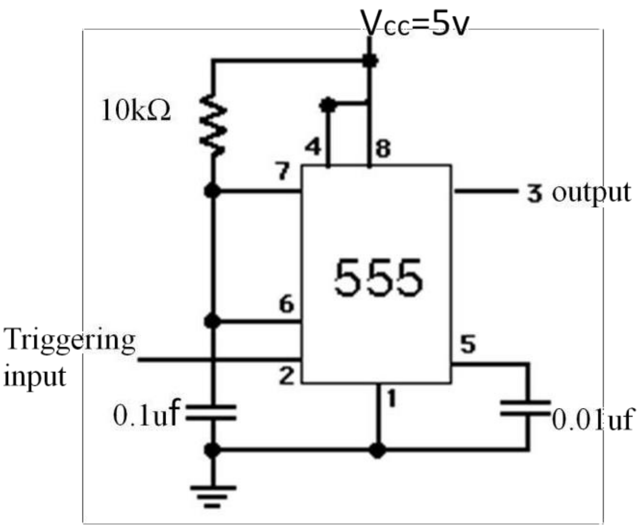


Fig 8.1: circuit diagram of monostable multivibrator

THEORY:

The 555 Timer is used in number of applications; it can be used as monostable, astable multivibrators, DC to DC converters, digital logic probes, analog frequency meters, voltage regulators and time delay circuits.

The IC 555 timer is 8-pin IC and it can operate in free- running (Astable) mode or in one-shot (Monostable) mode. It can produce accurate and highly stable time delays or oscillations. Monostable can also be called as One-shot Multivibarator. Fig 1.2 shows the Pin configuration of

Monostable Multivibrator. When the output is low, the circuit is in stable state, Transistor Q1 is ON and capacitor C is shorted out to ground. However, upon application of a negative trigger pulse to pin-2, transistor Q1 is turned OFF, which releases short circuit across the external capacitor and drives the output High. The capacitor C now starts charging up toward Vcc through R. However, when the voltage across the external capacitor equals 2/3 Vcc, the output of comparator1 switches from low to high, which inturn drives the output to its low state. The output, Q of the flip flop turns transistor Q1 ON, and hence, capacitor C rapidly discharges through the transistor. The output of the Monostable remains low until a trigger pulse is again applied. Then the cycle repeats. Fig (2) shows the trigger circuit & Fig.3 shows trigger input, output voltage and capacitor voltage waveforms.

Pulse width of the trigger input must be smaller than the expected pulse width of the output waveforms. Trigger pulse must be a negative going input signal with amplitude larger than 1/3 Vcc. The time during which the output remains high is given by

tp =1.1RC

Once triggered, the circuit‟s output will remain in the high state until the set time tp elapses. The output will not change its state even if an input trigger is applied again during this time interval tp.

DESIGN:

Step 1: Choose C=1μF.

Step 2: Since in monostable multivibrator, tp=1.1RC. Therefore R= tp / 1.1C

Step 3: Using equations, design the value of R.

PROCEDURE:

1. Connect the 555 timer in Monostable mode as shown in fig.8.1
2. Connect the C.R.O at the output terminals & observe the output.
3. Apply external trigger at the trigger input terminal (PIN 2) and observe the output of Monostable Multivibrator.
4. Record the trigger input, voltage across the capacitor & output waveforms and measure
5. The output pulse width. Verify results with the sample output waveforms as shown in fig
6. Calculate the time period of pulse (tp =1.1RC) theoretically & compare it with practical values.

OBSERVATION TABLE:

S.No	Theoritical value of o/p pulse width (in m.sec) $t_p = 1.1\,RC$	Practical value of output pulse width(in m.sec

EXPECTED WAVEFORMS:

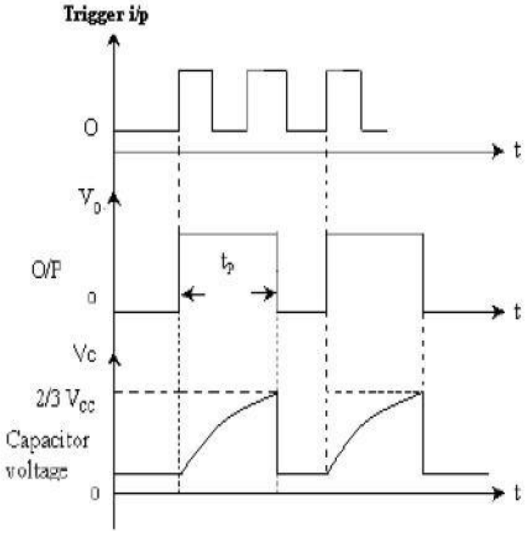


Fig8.2: expected waveforms of monostable multivibrator

Observations:

Trigger input

Amplitude = Time period =

Square wave Output signal

Amplitude = Time period =

Triangle wave Output signal

Amplitude = Time period =

Applications:

1. Frequency divider
2. Pulse width modulation
3. Linear ramp generator
4. Missing pulse detector

RESULT: Hence designed & studied 555 timer as a Monostable multivibrator and also theoretical & Practical of time period values of the output waveform are compared.

EXPERIMENT NO: 9 **Date:**

ASTABLE MULTIVIBRATOR USING 555 TIMER

AIM: To design an Astable Multivibrator using IC 555 timer to generate a square wave of 6.9 KHz with 52.38 % Duty Cycle.

APPARATUS:
1. C.R.O
2. Function generator
3. Regulated DC power Supply
4. CDS Board/ Bread Board.
5. Connecting patch chords.

COMPONENTS REQUIRED:

IC 555	1No
Resistor-------2.2kΩ	1No
3.3kΩ	1No
Capacitor-----0.1µf	1No
0.01µf	1No

CIRCUIT DIAGRAM:

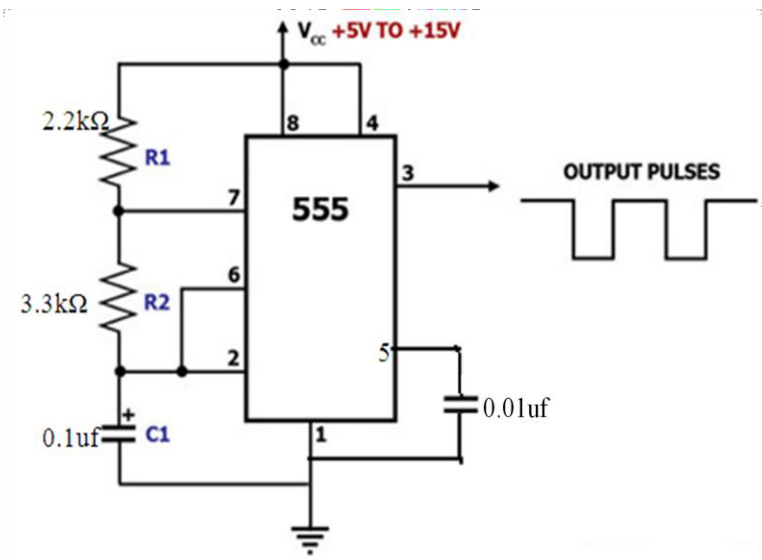


fig 9.1: circuit diagram for astable multivibrator

THEORY:

The 555 Timer is used in number of applications; it can be used as monostable, astable multivibrators, DC to DC converters, digital logic probes, analogy frequency meters, voltage regulators and time delay circuits. The IC 555 timer is 8-pin IC and it can operate in free-running (Astable) mode or in one-shot (Monostable) mode. The pin configuration of NE 555 Timer is as shown fig (1). It can produce accurate and highly stable time delays or oscillations.

Astable Multivibrator often called a free-running Multivibrator. External Trigger input is not required to operate the 555 as an Astable Configuration. However, the time during which the output is either high or low is determined by two external components Resistor & Capacitor. Fig shows the 555 as Astable Multivibrator. Initially, when the output is high, capacitor C starts charging towards Vcc through resistor Ra and Rb. As soon as voltage across the capacitor equals to 2/3 Vcc, comparator-1 triggers the flip-flop, and the output is low. Now capacitor discharges through Rb and transistor Q1. When the voltage across capacitor C equals to 1/3Vcc, comparator-2"s output triggers the flip-flop, and the output goes high. Then the cycle repeats. The output voltage waveforms are as shown in fig (3).In this way capacitor periodically charges and discharges between 2/3Vcc and 1/3Vcc respectively.

The time during which the capacitor charges from 1/3Vcc to 2/3 Vcc is equal to the ON time of the timer (i.e. the output is HIGH) and is given by

$$tc = 0.69(R_1+R_2)C$$

The time during which the capacitor discharges from 2/3 Vcc to 1/3Vcc is equal to the OFF time of the timer, during which the output is LOW and is given by

$$td = 0.69(R_2)C$$

The total time period of the output is the sum of charging time(tc)and discharging time(td) and is given by

$$T = tc + td = 0.69(R_1 + 2R_2)\,C$$

Therefore the frequency of oscillations of Astable multivibrator is given by

$$F = 1/T = 1.45/\,(R_1 + 2R_2)\,C$$

DUTY CYCLE:

This term is in conjunction with Astable Multivibrator. The duty cycle is the ratio of the ON time, tc during which the output is high to the total time period T. It is generally expressed as a percentage.

Duty cycle, $D = (T_{ON} / T_{ON} + T_{OFF}) = tc / T = (R_1 + R_2) / (R_1 + 2R_2)$

DESIGN:

Step1: Choose C=0.01 µF

Step2: using the formula, $F = 1.45/ (R_1 + 2R_2) C$, Get a relation between R_1 & R_2.

Step3: Consider the expression for duty cycle, $D = (T_{ON} / T_{ON} + T_{OFF}) = (R_1 + R_2) / (R_1 + 2R_2)$ & obtain a relation between R_1 & R_2.

Step4: Using the relations between R_1 & R_2., obtained in step2 & step3, solve for R_1 & R_2.

PROCEDURE:

1. Connect the IC 555 timer in Astable mode as shown in fig.
2. Connect the C.R.O at the output terminal (pin 3) and observe the output.
3. Record the waveforms at pin3, across the capacitor & compare them with the sample output waveforms as shown in fig (3)
4. Measure the charging time (tc), discharging time (td) and total time period/ Frequency from the output waveform.
5. Calculate tc, td, time period (T), frequency (f) of the square wave output and percentage duty cycle theoretically.

Compare the theoretical values charging time (tc), discharging time (td) total time period/ Frequency & % Dutycyclewiththepracticalvalues

OBSERVATION TABLE:

S.NO	Theoretical Values					Practical Values				
	t_c (m.sec)	t_d (m.sec)	T (m.sec)	f (in Hz)	D	t_c (m.sec)	t_d (m.sec)	T (m.sec)	F (inHz)	D

EXPECTED WAVEFORMS:

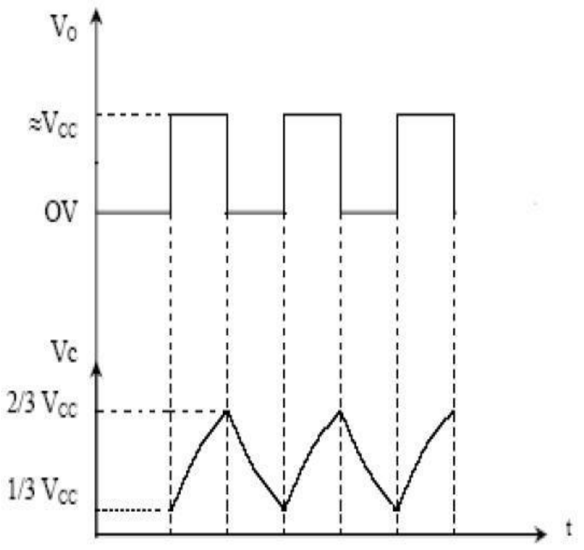


Fig 9.2: expected output waveforms of astable multivibrator

Observations:

Square wave Output signal

Amplitude = Time period =

Triangle wave Output signal

Amplitude = Time period =

Applications:

1. square wave generator
2. Voltage controlled oscillator
3. FSK(frequency shift keying) generator

RESULT: Hence designed & studied IC 555 timer as an Astable multivibrator and also calculated the frequency of oscillations & time period of output waveform.

EXPERIMENT NO: 10 **Date:**

Schmitt Trigger Circuits – using IC 741 and IC 555

AIM: To construct and study the Schmitt Trigger using IC741 and IC 555 Operational Amplifiers

APPARATUS REQUIRED:

1. Function Generator
2. Regulated DC power Supply
3. Dual Channel Oscilloscope(CRO)
4. Digital Multimeter
5. CDS Board / Bread Board
6. Connecting wires

COMPONENTS REQUIRED:

IC 555	1No
Resistor ------10kΩ	1No
1kΩ	2No
100kΩ	2No
Capacitor-----0.1µf	1No
0.01µf	1No

CIRCUIT DIAGRAMS:

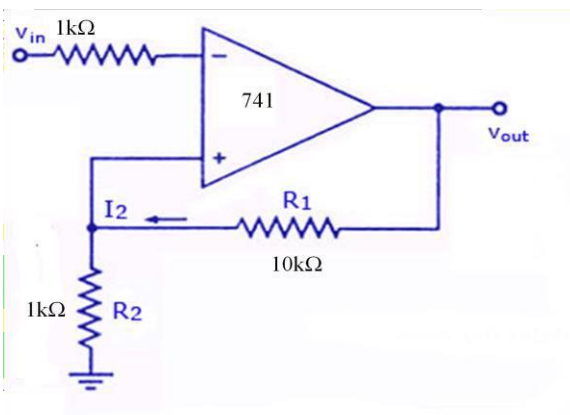


Fig 10.1: circuit diagram of schmitt trigger using IC 741

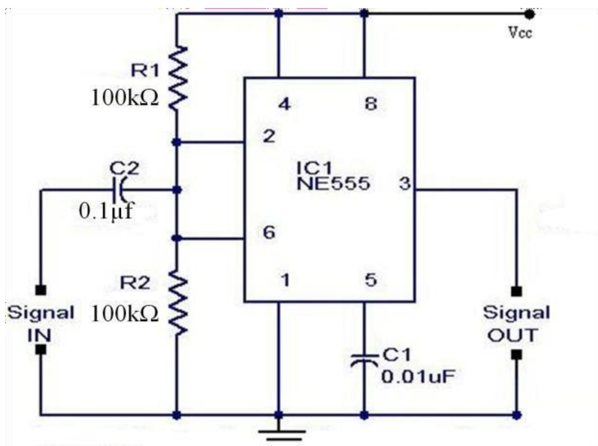


Fig 10.2: circuit diagram of schmitt trigger using IC 555

THEORY:

In Schmitt Trigger two internal comparators are tied together and externally biased at VCC/2 through R_1 & R_2. Since the upper comparator will trip at (2/3) VCC and lower comparator at (2/3) VCC the bias provided by R1 & R2 is centered within these two thresholds. Thus a sine wave of sufficient amplitude (>VCC/6 = 2/3 VCC – VCC/2) to exceed the reference levels causes the internal flip –flop to alternately set and reset providing a square wave output.

PROCEDURE for IC 741

1. Connect the circuit as shown Fig.
2. Set Function Generator output for sine wave signal of Amplitude at 1V(p-p) & frequency 1KHz.
3. Set R1 and R2 values at fixed positions and note down the values in tabular column. Calculate theoretical values of Vut and Vlt and note down the values in tabular column. (+Vsat = 14V,- Vsat = -14V).
4. Apply Function Generator output at input terminals Vi, connect C.R.O- CH2 at output terminals Vo, C.R.O-CH1 at input terminals Vi.
5. Observe square wave output on C.R.O for the given input sine wave & compare them with the sample waveform as shown in fig.2.
6. Note down the practical V_{ut} , V_{lt} and VH values in tabular column.
7. Compare the theoretical and practical values of Vut,V_{lt} and V_H

PROCEDURE for IC 555:

1. Connect the circuit as shown in fig (10.2).
2. Apply the input sine wave 5V (P-P) using function generator at 1KHZ frequency.
3. Observe the output waveform at Pin No: 3.

INPUT AND OUTPUT WAVE FORMS OF SCHMITTH TRIGGER:

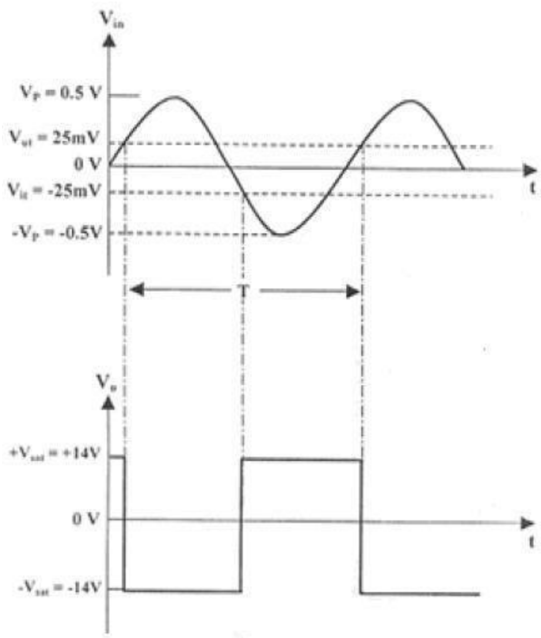


Fig 10.3: expected input and output waveforms of Schmitt trigger

<u>Observations for Schmitt trigger using IC 741:</u>

Input signal

Amplitude = Time period =

Output signal

Amplitude = Time period =

<u>Observations for Schmitt trigger using IC 555:</u>

Input signal

Amplitude = Time period =

Output signal

Amplitude = Time period =

Applications:

1. on/off controllers
2. Used as a comparator

RESULT: Hence constructed and studied Schmitt trigger using IC 741 and IC 555

EXPERIMENT NO: 11 **Date:**

PHASE LOCKED LOOP (PLL) USING IC 565

AIM: To calculate free running frequency, capture range and lock range of PLL System.

APPARATUS REQUIRED:

1. C.R.O
2. Function Generator
3. DC power supply
4. CDS board / Bread Board
5. Connecting wires

COMPONENTS REQUIRED:

1.	LM 565 IC		: 1No
2.	Resistors	10K	: 1No
		680Ω	: 2No
3.	Capacitors	0.1µ F	: 1No
		1µ F	: 1No
		0.01µ F	: 1No

CIRCUIT DIAGRAM:

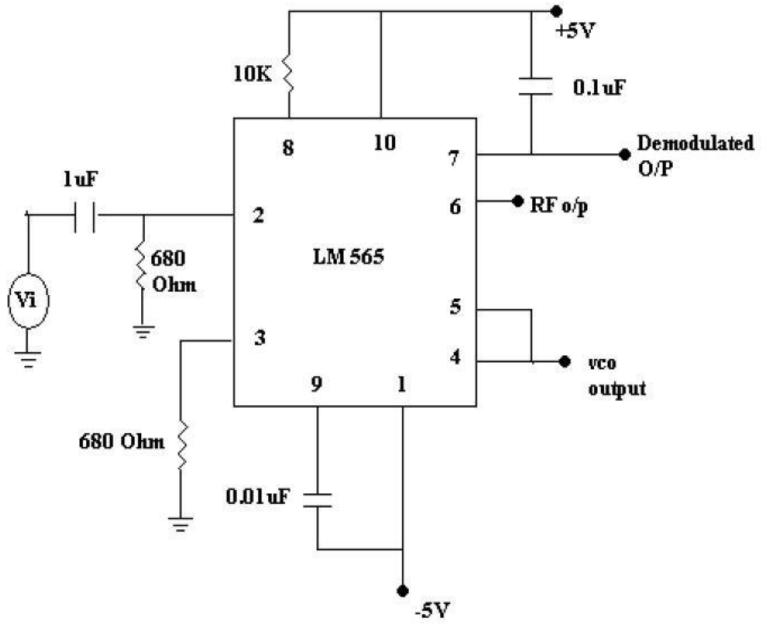


Fig 11.1:circuit diagram of PLL using IC 565

THEORY:

The fig.11.1 shows the phase-locked loop (PLL) in its basic form. The PLL consists of i) a phase detector ii) a low pass filter and iii) a voltage controlled oscillator as shown.

The phase detector, or comparator compares the input frequency f_{IN} with the feedback frequency f_{OUT}. The output of the phase detector is proportional to the phase difference between f_{IN} and fout. The output voltage of a phase detector is a dc voltage and therefore is often referred to as the error voltage. The output of the phase detector is then applied to the low-pass filter, which removes the high-frequency noise and produces a dc level. This dc level, in turn, is the input to the voltage-controlled oscillator (VCO). The filter also helps in establishing the dynamic characteristics of the PLL circuit. The output frequency of the VCO is directly proportional to the input dc level. The VCO frequency is compared with the input frequencies and adjusted until it is equal to the input frequencies. In short, the phase-locked loop goes through three states: free running, capture, and phase lock.

Before the input is applied, the phase-locked loop is in the free-running state. Once the input frequency is applied, the VCO frequency starts to change and the phase-locked loop is said to be in the capture mode. The VCO frequency continues to change until it equals the input frequency, and the phase-locked state. When phase locked, the loop tracks any change in the input frequency through its repetitive action.

Lock Range: The range of frequencies over which the PLL can maintain lock with incoming signal is called the " Lock Range" or "Track Range"

FL= 8f0/V where V= + V –(–V), where f0 is free running frequency. Capture range: The range of frequencies over which the PLL can acquire lock with an input signal is called the capture range.

PROCEDURE:

1. Apply +5v to pin 10 and –5v to pin 1 of LM565
2. Connect R1= 10KΩ resistor from pin 8 to10 and C1 =0.01µ F capacitor from pin 9 to 1.
3. Connect 680Ω resistor from pin 2 & pin 3 to ground.
4. Connect pin 4(VCO o/p) to CRO and measure its frequency. This frequency is called the free running frequency, fo.
5. Calculate f0 theoretically using the formula f0 = 1.2 /4R1C1 and compare it with practical value.
6. Connect the circuit as shown in fig.
7. Apply square wave at the input with amplitude of 2Vpp and also connect it to channel 1 of CRO.
8. Connect pin 4(VCO o/p) to channel 2 of CRO.
9. Vary the input signal frequency in steps and measure its corresponding o/p frequency.
10. Find the lock range and capture range from the obtained data.

TABULAR COLUMN:

S.No	Input frequency in HZ	Output frequency in HZ	F_c in HZ	F_L in HZ

Applications:
1. Frequency multiplier
2. Frequency synthesizer
3. FM demodulator
4. FSK demodulator
5. AM detection
6. Frequency translation

RESULT: Free running frequency, lock range and capture range of PLL are measured practically and compared with theoretical values.

EXPERIMENT NO: 12 **Date:**

VOLTAGE CONTROLLED OSCILLATOR (IC 566)

AIM: To construct and study the voltage controlled oscillator-using IC 566.

APPARATUS REQUIRED:
1. Function Generator.
2. C.R.O.
3. CDS Board/ Bread Board
4. Connecting Patch chords.

COMPONENTS REQUIRED:

IC 566		1No
Resistor	10kΩ	1No
	1.5kΩ	1No
	20kΩ	1No
Capacitor	1µf	1No
	0.1µf	1No
	0.01µf	1No

CIRCUIT DIAGRAM:

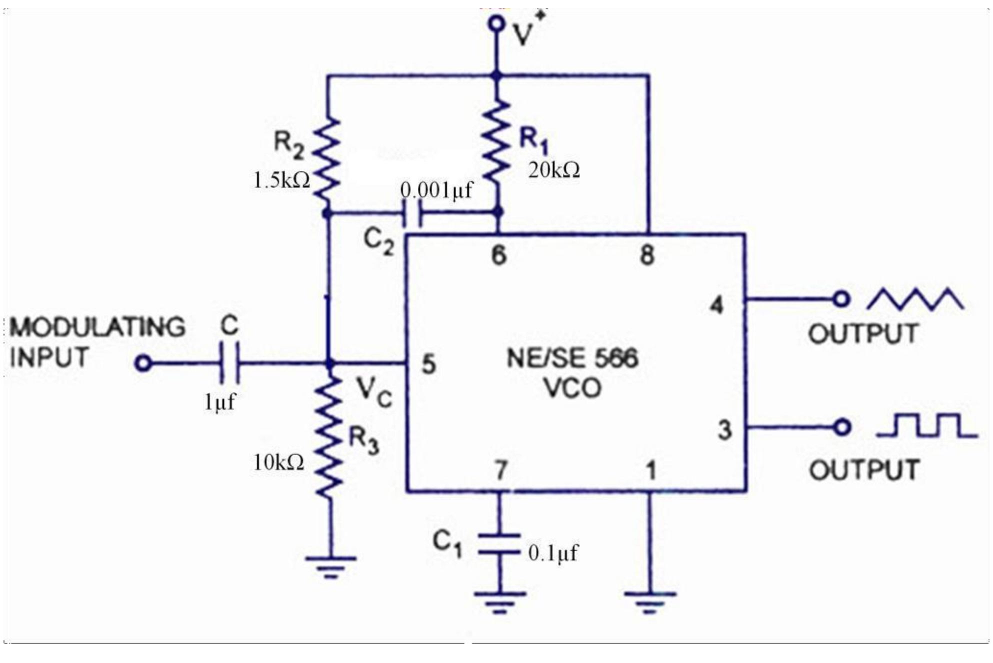


Fig 12.1: circuit diagram of VCO using IC 566

THOERY:

Fig.12.1 shows the circuit diagram of VCO. This arrangement R1C1 combination determines the free running frequency and the control voltage Vc at terminal 5 is sent by the voltage divider formed with R2 and R3. The initial voltage Vc at terminal is

$\frac{3}{4} V_{cc} \leq V_c \leq V_{cc}$ Where +V is the total supply voltage.

PROCEDURE:

1. Connect the circuit as shown in fig.
2. Switch on the power supply of 12V DC & observe square wave output at pin no.3 & triangular wave output at pin no.4.
3. Keep the product of R1 C1 as constant.
4. By varying the control voltage Vc, between ¾(Vcc) and Vcc observe the output frequency.

WAVEFORMS:

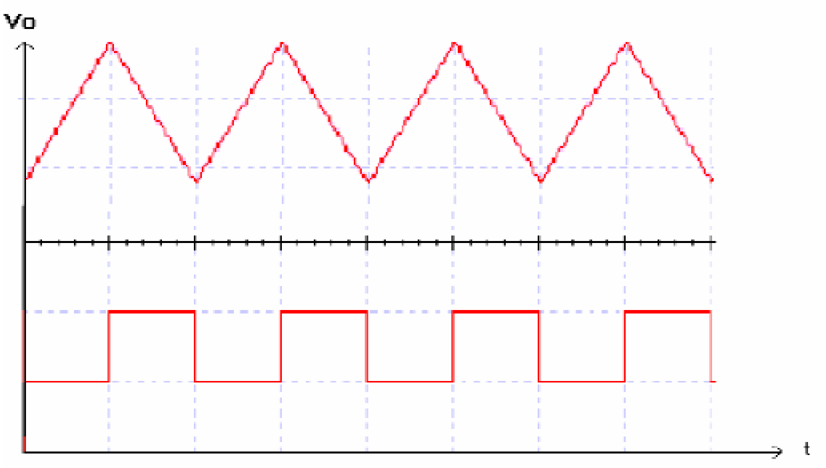


Fig 12.2: expected output waveforms of VCO

Observations:

Square wave Output signal

Amplitude = Time period =

Triangle wave Output signal

Amplitude = Time period =

Applications:

1. Function generators,
2. The production of electronic music, to generate variable tones,
3. Phase-locked loops,
4. Frequency synthesizers used in communication equipment.

RESULT: Voltage controlled oscillator-using IC 566 is constructed and the frequency of oscillations is estimated. Also observed the variations in the output signal frequency in accordancewiththemodulatingorcontrolvoltage,Vc

EXPERIMENT NO: 13 **Date:**

DIGITAL TO ANALOG (D/A) CONVERTERS

AIM: To obtain analog output voltages for the digital input data using 4-bit R-2R ladder type D/A converter.

APPARATUS REQUIRED:

1. 4 – Bit D/A converter (R-2R) Trainer Kit.
2. Multimeter.

COMPONENTS REQUIRED:

IC 741		1No
Resistor	10kΩ	3No
	22kΩ	6No

CIRCUIT DIAGRAM:

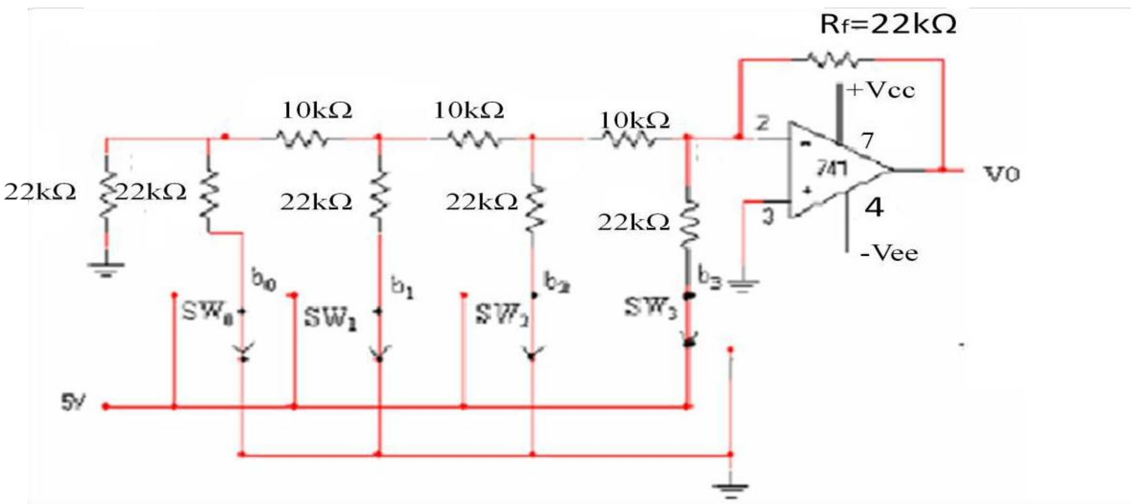


Fig 15.1: circuit diagram of D/A converter

PROCEDURE:

1. Connect the trainer to the mains and switch on the power supply.
2. Measure the supply voltages of the circuit as +12V & -12V.
3. Calculate theoretically Vo for all digital Input data using formula.

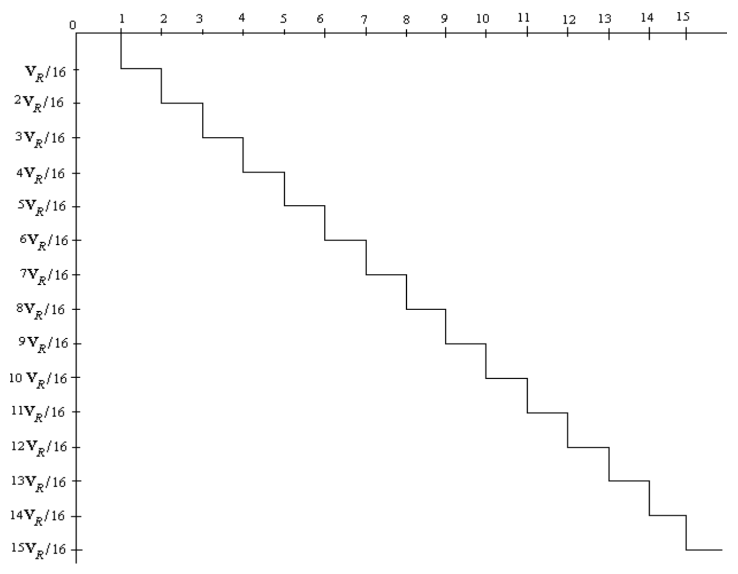
$$V_o = -R_f\left(\frac{b_0}{R} + \frac{2b_1}{R} + \frac{4b_2}{R} + \frac{8b_3}{R}\right)$$

4. In this experiment Rf = 22kΩ & R= 10kΩ.
5. Note down Output voltages for different combinations of digital inputs and compare it with theoretical values.

OBSERVATION TABLE:

Digital Input Data				Theoretical Value of the output, V_o (in volts)	Practical Value of the output, V_o (in volts)
b1	b2	b3	b4		
0	0	0	0		
0	0	0	1		
0	0	1	0		
0	0	1	1		
0	1	0	0		
0	1	0	1		
0	1	1	0		
0	1	1	1		
1	0	0	0		
1	0	0	1		
1	0	1	0		
1	0	1	1		
1	1	0	0		
1	1	0	1		
1	1	1	0		
1	1	1	1		

Model graph:



Applications:

1. Audio

Most modern audio signals are stored in digital form (for example MP3s and CDs) and in Order to be heard through speakers they must be converted into an analog signal.

2. Video

Video signals from a digital source, such as a computer, must be converted to analog form If they are to be displayed on an analog monitor.

RESULT: Obtained analog output voltages for the given digital input data using 4- bit R-2R ladder network D/A converter.