Exp. No : 01 Date:

CHARACTERISTICS OF OPERATIONAL AMPLIFIERS

OBJECTIVE

The purpose of the experiment is to examine non-ideal characteristics of an operational amplifier. The characteristics that is investigated includes input offset current, offset voltage, input bias current, common-mode rejection ratio, slew rate and Gain-bandwidth product.

PRELAB

1. Read the data sheet for the LM741 Operational Amplifier and fill in the following parameters.

Input Offset Current:
Input Offset Voltage:
Input Bias Current:
Input Resistance:
Common-Mode Rejection ratio:
Bandwidth:
Slew Rate:
Power Consumption:
Output Short Circuit Current:
Supply Voltage Rejection ratio:

2. Design a suitable circuit to measure the output short circuit current. Simulate the circuit and determine the short circuit current in an op-amp. Comment on the results obtained.
3. Simulate the open loop differential amplifier using *Spice* tool. Comment on the results obtained. Repeat the simulation steps for open loop inverting and non-inverting amplifiers. State the limitations of open loop configurations of op-amp. Attach the Spice schematic with input and output waveforms.

EQUIPMENT REQUIRED

741 Op-amp
100Ω, 100kΩ, 220kΩ resistor, ¼ W
5 kΩ potentiometer
0-30 V dc dual regulated power supply
Digital Multimeter
Breadboard

THEORY

- The dc characteristics of op-amp that affect the steady state response are: Input bias current, Input offset current, Input offset voltage and thermal drift.
- The *input bias current* is the average of the two base currents in the input stage of an op-amp under no-signal conditions.
- The *input offset current* is the difference of the two base currents that results due to non-identical transistors.
- The *input offset voltage* is the input voltage needed to null or zero the quiescent output voltage.
- The *CMRR* of an op-amp is the ratio of differential voltage gain to common mode voltage gain.
- *Slew rate* is the maximum rate at which the output voltage can be distorted.

- An op-amp's *bandwidth* is the highest undistorted frequency an op-amp can deliver. It is directly proportional to slew rate and inversely proportional to amplitude.

FORMULA

Input bias current, $I_B = \dfrac{I_B^+ + I_B^-}{2}$

Input offset current, $|I_{OS}| = I_B^+ - I_B^-$

Input offset voltage, $V_{OS} = V_O \dfrac{R_i}{(R_f + R_i)}$

Slew rate

For sine input, SR= $\dfrac{2\pi f V_m}{10^6} V/\mu s$

For square input, SR= $\dfrac{\Delta V_o}{\Delta t} V/\mu s$

CMRR, ρ = $\dfrac{V_i}{V_o}\dfrac{R_f}{R_i}$ = 20 *log*(ρ) *dB*

FURTHER READING

1. Paul B. Zbar, Albert P. Malvino, Michael A. Miller "Basic Electronics A Text – Lab Manual", Tata McGraw-Hill, seventh edition, 1995.
2. Ramakand A. Gayakwad, "Op-amps and linear integrated circuits", PHI learning, 2009.

CIRCUIT DIAGRAM

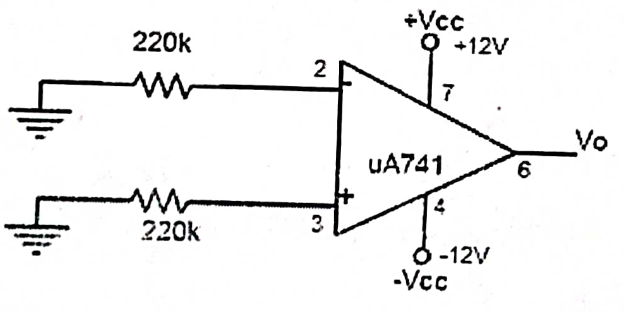


Figure1.Input bias current and offset current Measurement

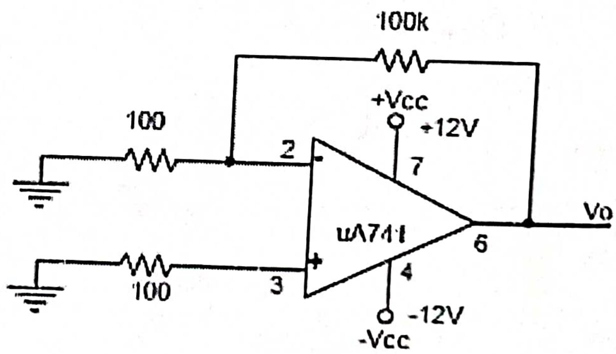


Figure2.Input and output offset voltage measurement

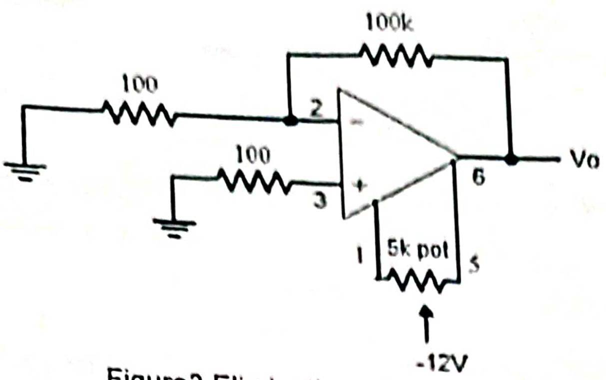


Figure3.Elimination of offset voltage

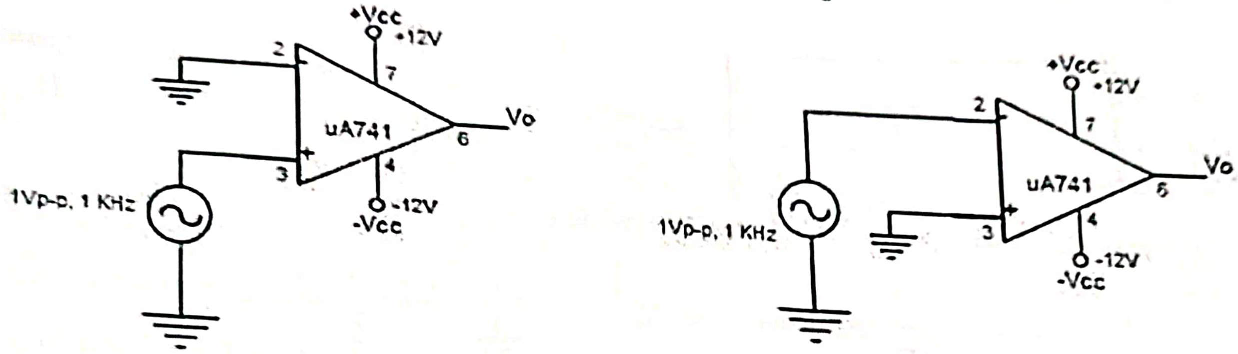


Figure4. Measurement of saturation voltages

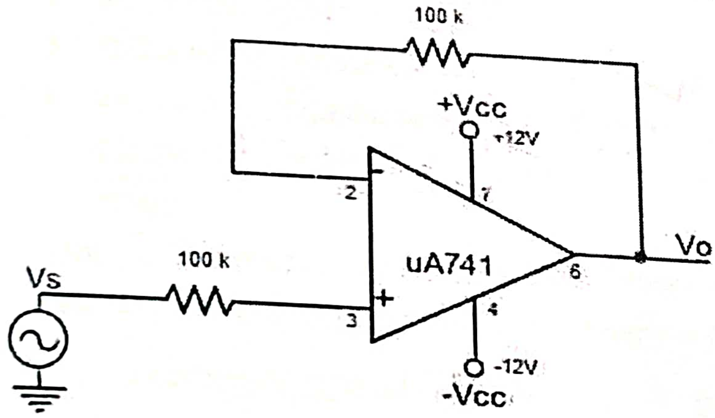


Figure5.1. Measurement of input voltage ranges

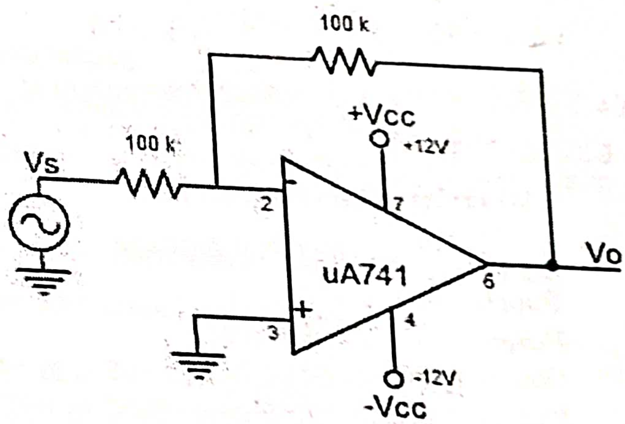


Figure5.2. Measurement of output voltage ranges

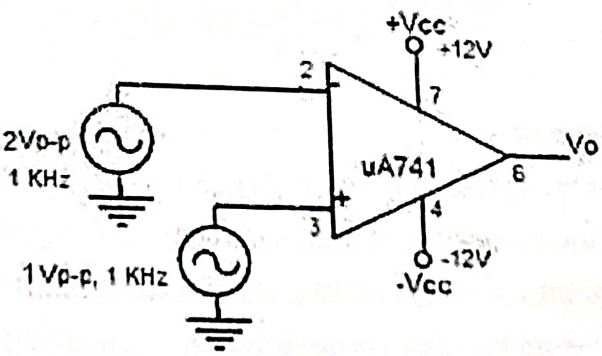


Figure6.1 Open loop differential amplifier

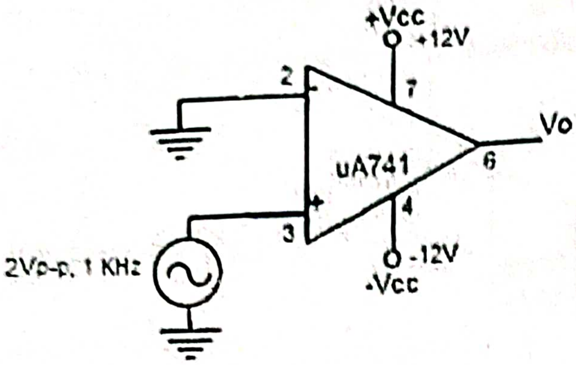


Figure6.2 Open loop non-inverting amplifier

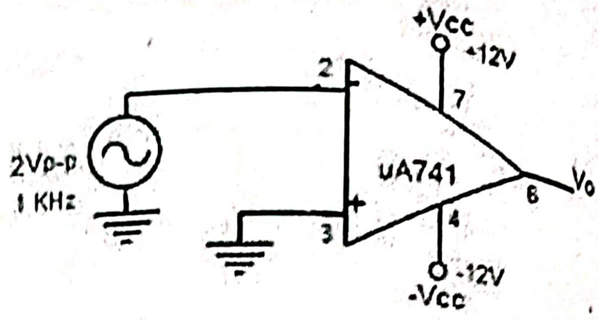


Figure6.3 Open loop inverting amplifier

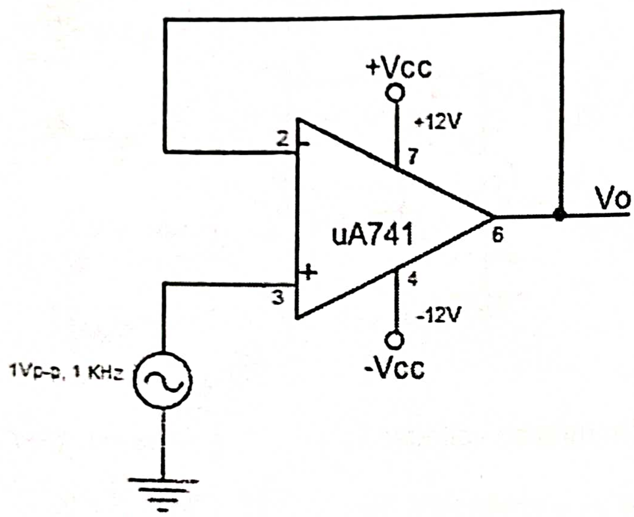


Figure7. Slew rate and Bandwidth measurement

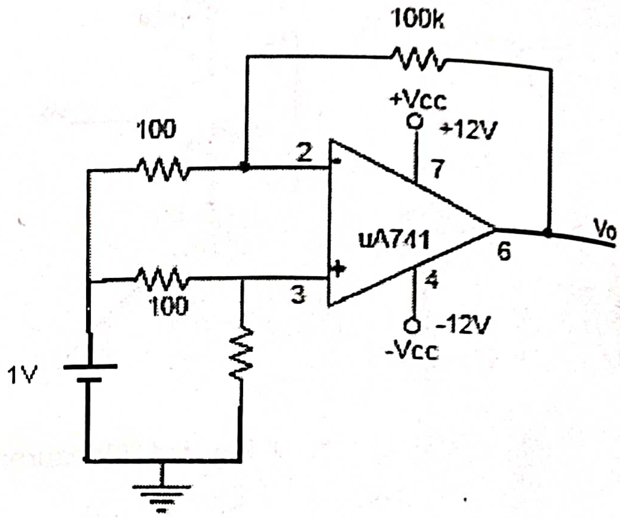


Figure8. CMRR measurement

SPECIFICATION

LM741 Operational Amplifier

8-pin dual-in-line package

Supply Voltage ±22V
Power Dissipation 500 mW
Operating Temperature Range −55°C to +125°C
Storage Temperature Range −65°C to +150°C
Junction Temperature 150°C
Large signal voltage gain: 50 V/mV

PRACTICE PROCEDURE

Input Bias Current and Offset Current

1. Construct the circuit as per the diagram shown in Figure1.
2. Measure the dc voltage at the inverting and non-inverting terminals of the op-amp.
3. Calculate the input bias current with the formula given in the table.
4. Calculate the difference between the bias currents to know the offset current.

Repeat the above steps with 100K resistors and then with 100Ω resistors. Report your results.

Input and output Offset Voltage

1. Construct the circuit as per the diagram shown in figure in Figure2.
2. Measure the dc output voltage. This is the output offset voltage.
3. Calculate the input offset voltage.

Replace the 100k resistor with a 220k resistor and repeat the above steps. Report your results.

Repeat the above steps with a 1K resistor and comment on the results.

4. To eliminate this offset voltage, connect the circuit as shown in Figure3.
5. Vary the pot until the output voltage becomes zero.

Note: Due to the pot sensitivity, you may not get a full zero on the output. A minimum voltage at the output in the range of mV will be sufficient.

Saturation Voltages

1. Construct the circuit as shown in figure in Figure4.
2. Measure the output voltage.
3. Measure the output voltage by applying an ac voltage of 1Vp-p, 1 kHz to the non-inverting terminal.

Repeat the above step applying a dc voltage of 1Vp-p, 1 kHz to the inverting terminal.

Input and output voltage ranges

1. Construct the circuit as shown in Figure5.1.
2. Apply +5V, 100Hz sinusoidal input.
3. Observe the voltages at the input and output simultaneously.
4. Increase the signal amplitude until distortion is observed at the peak value of the output.
5. Measure the positive and negative input voltage peak values. This gives the op-amp input voltage range.

Change the circuit as shown in Figure5.2. Repeat the above steps and measure the positive and negative output voltage peak values. This gives the op-amp output voltage range.

Open loop configurations of op-amp

Open loop differential amplifier

1. Construct the circuit as shown in Figure6.1.
2. Apply a sinusoidal voltage of 1Vp-p, 100Hz at the non-inverting terminal and 2Vp-p, 100Hz at the inverting terminal.
3. Observe and plot the input and output waveforms.

Open loop inverting amplifier

4. Change the circuit as given in Figure6.2.
5. Apply 2Vp-p, 100Hz at the inverting terminal and the non-inverting terminal is grounded.
6. Observe and plot the input and output waveforms.

Open loop non-inverting amplifier

7. Change the circuit as given in Figure6.3.
8. Apply 2Vp-p, 100Hz at the non-inverting terminal and the inverting terminal is grounded.
9. Observe and plot the input and output waveforms.

Slew rate and Bandwidth

1. Construct the circuit as shown in figure in Figure7.
2. Apply sinusoidal signal of 2Vp-p, 1 kHz.
3. Increase the frequency until the output gets distorted.
4. Measure the peak output amplitude and frequency.
5. Calculate the slew rate

Repeat the above steps for square wave input.

CMMR

1. Construct the circuit as shown in figure in Figure8.
2. Apply a sinusoidal signal of 1Vp-p, 1 kHz as input to the circuit.
3. Measure the output voltage.
4. Calculate CMRR

MODEL GRAPH

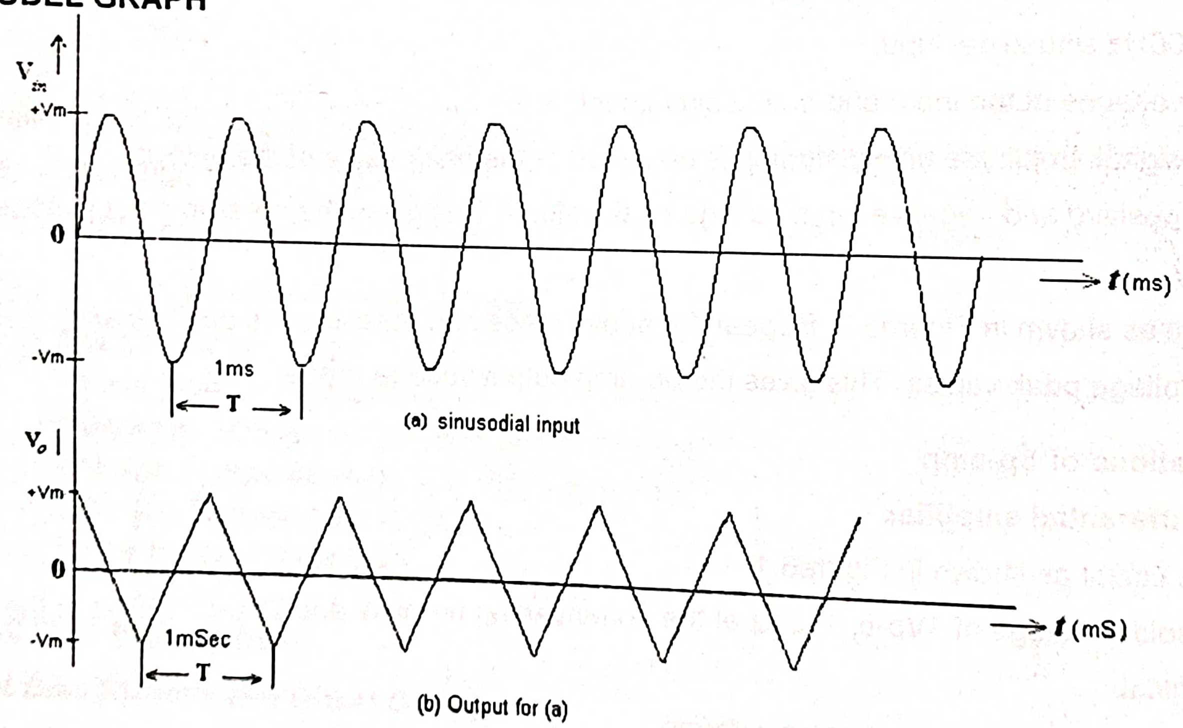


Figure9:Slew rate

OBSERVATION

Table 1: Input bias and offset current

	Resistor	DC Output voltage at inverting terminal, V^- Volts	DC Output voltage at non-inverting terminal, V^+ Volts	Input bias current, I_B^- nA	Input bias current, I_B^+ nA	Input bias current, I_B nA	Input offset current, I_{OS} nA
First 741 IC	220kΩ						
	100kΩ						
	100 Ω						
Second 741 IC	220kΩ						
	100kΩ						
	100 Ω						

Table 2: Input Offset voltages

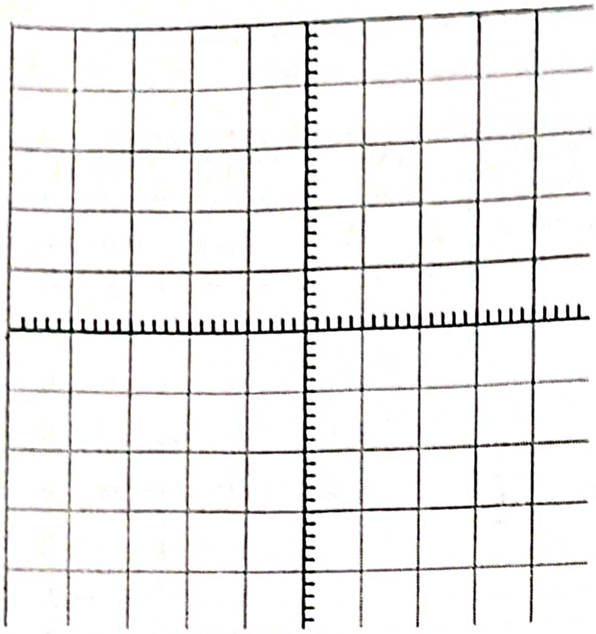
	Resistor	DC Output voltage, (outout offset) V_O Volts	Input offset voltage, V_{OS} mV
First 741 IC	100kΩ		
	220kΩ		
	1kΩ		
Second 741 IC	100kΩ		
	220kΩ		
	1kΩ		

Table 3: Output short circuit current

	Output short circuit current $I_{O(s.c)}$
Fig 5.1	
Fig 5.2	

Table4: Saturation voltages

	Input voltage, V_I Volts	Output voltage, V_O Volts
Fig 4.1		
Fig 4.2		



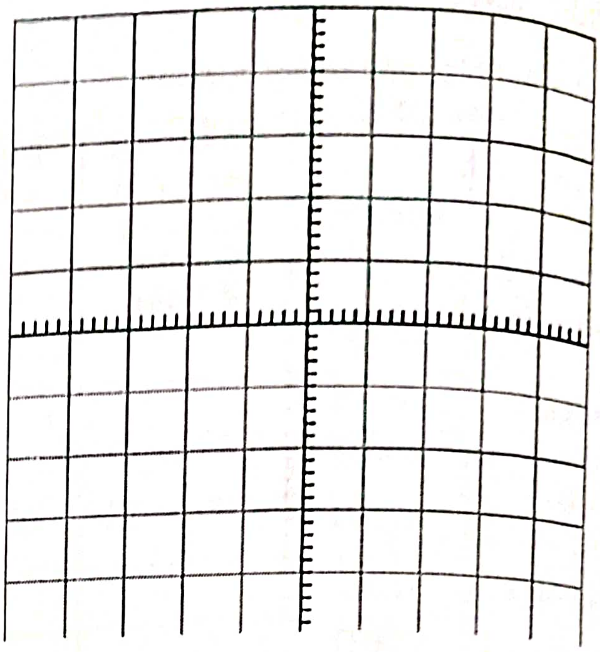
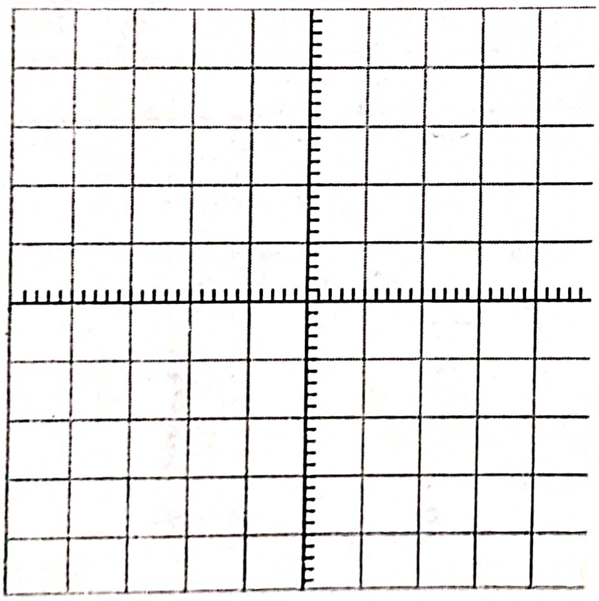
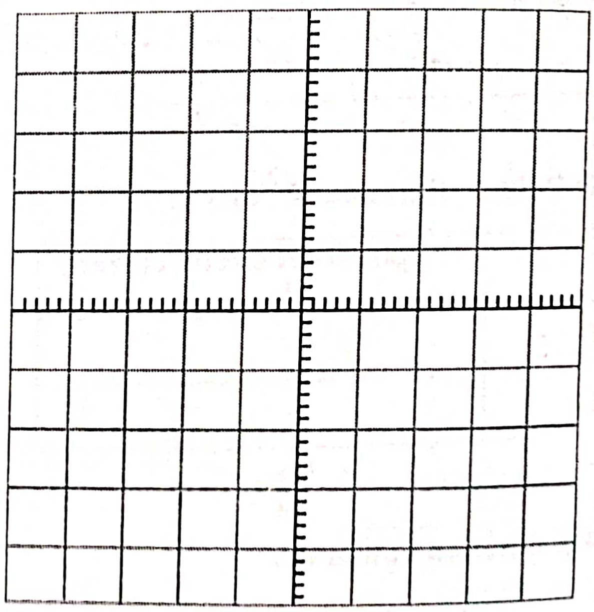


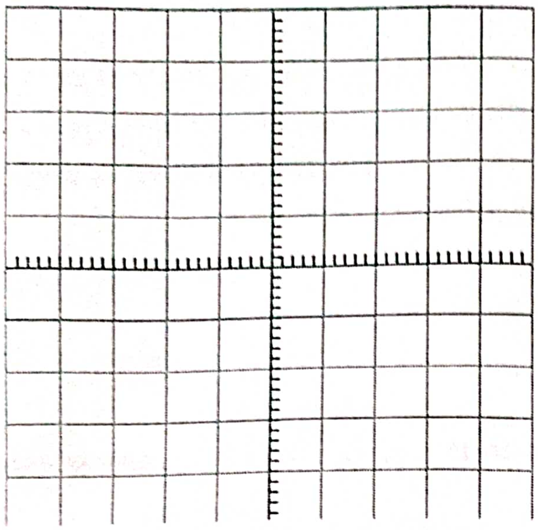
Table5: Input and output voltage range

	Input voltage range (volts)	Output voltage range (volts)
741 IC		

Open loop configurations of op-amp







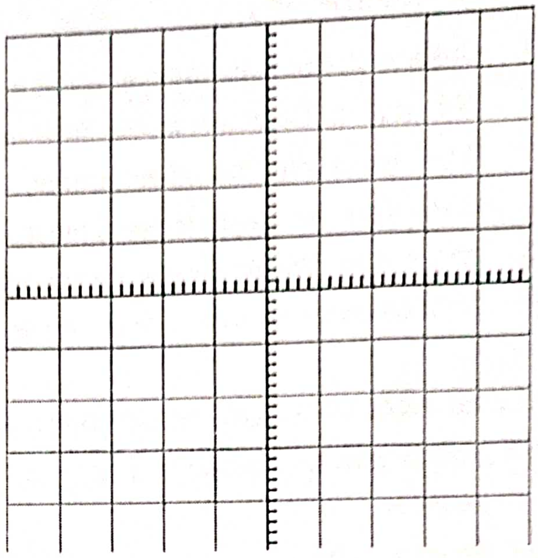
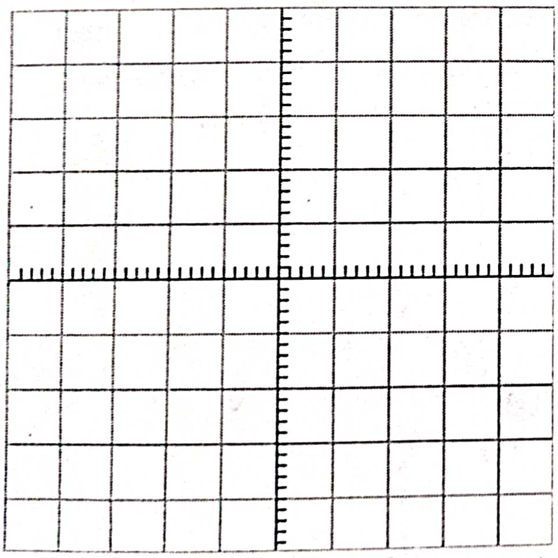


Table6.1: Slew Rate (for sine input)

	fmax	Vm	SR	Bandwidth
First 741 IC				
Second 741 IC				

Gain-Bandwidth Product, Gain-BW= ____________



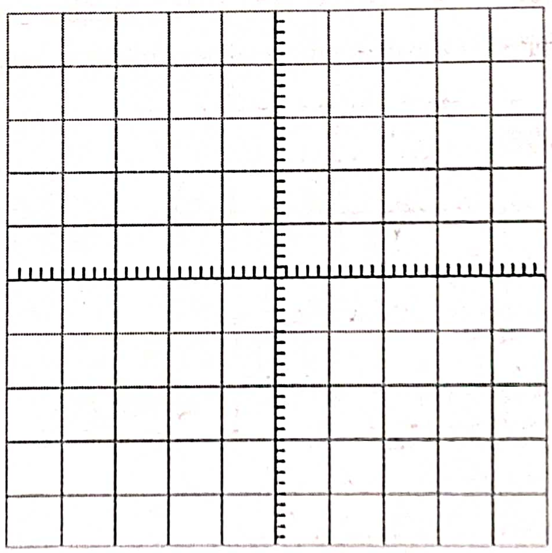


Table6.2: Slew Rate (for square input)

	ΔV	ΔT	SR
First 741 IC			
Second 741 IC			

Instructor ______________________

Date ____________

POST LAB INFERENCE

1. Observe the parameters measured in lab. Look up the same parameters on a data sheet for the 741 op-amp. Comment on the differences between your measurements and the specified values.
2. While measuring the offset current, you used resistors of 100Ω, 100k Ω and 220kΩ. In which cases were the measurements most accurate? Explain why this is so.
3. Why is it so difficult to measure the open loop gain of an op amp?
4. What causes slew rate? State the effect of slew rate in applications.

POST LAB ANALYSIS QUESTIONS (OPTIONAL)

1. Create gain Vs frequency plot of an op-amp using Spice tool. Analyze the bandwidth of the amplifier.
2. Study the effect of variation in power supply voltages on offset voltage. Create the following plots: (i) supply voltage Vs input bias current and (ii) supply voltage Vs offset current.
3. Using spice tool obtain a plot between supply voltage and open loop voltage gain
4. Using Spice tool create a suitable circuit to determine the CMRR of the op-amp. Plot the curve between frequency and CMRR. Comment on the curve obtained.
5. Using Spice tool simulation determine the slew rate of an op-amp in (i) voltage follower and (ii) non-inverting amplifier. Comment on the differences between the results. Which circuit exhibits better performance and why?
6. Design a suitable circuit to measure op-amp input resistance, output resistance, input capacitance and supply voltage rejection ratio. Comment on the significance of these parameters.

INFERENCE

Electronics Lab.

EXPERIMENT NO. - 02 02

Object :- Study of operational amplifier (741) type as inverting and non-inverting amplifier.

Apparatus Required :- Op-amp. kit, connection wires, CRO with probe.

Theory and Formula used :- The op-amp is a versatile device that can be used to amplify dc as well as ac input signals and was originally designed for computing such mathematical functions as addition, subtraction, multiplication and integration. With the addition of suitable external feedback components the modern day op amp can be used for a variety of applications such as ac and dc signal amplification, active filters, oscillators, comparators, regulators and others.

An ideal op- amp exhibit the following characteristics-

I. Infinite voltage gain (A).

II. Infinite input resistance Ri so that almost any signal source can drive it and there is no loading of the preceding stage.

III. Zero output resistance RO so that output can drive an infinite number of other devices.

IV. Zero output voltage when output voltage is zero.

V. Infinite bandwidth so that any frequency signal can be amplitude without attenuation.

VI. Infinite common mode rejection ratio so that output common mode noise voltage is zero.

VII. Infinite slew rate so that output voltage changes occur simultaneously input voltage changes.

The gain of the inverting amplifier is simply given by

$$A = \frac{-R_f}{R_1}$$

So the output voltage

$$V_0 = -V_{in}\frac{R_f}{R_1}$$

And the gain of the non - inverting amplifier is given by

$$A = \frac{R1+Rf}{R1}$$

Hence the output voltage

$$V_0 = V_{in}\frac{R_1 + R_f}{R_1}$$

4. **Circuit Diagram**:-Basic circuit diagram for inverting and non-invertin amplifiers are schematically shown in fig. (1) and fig. (2) respectively.

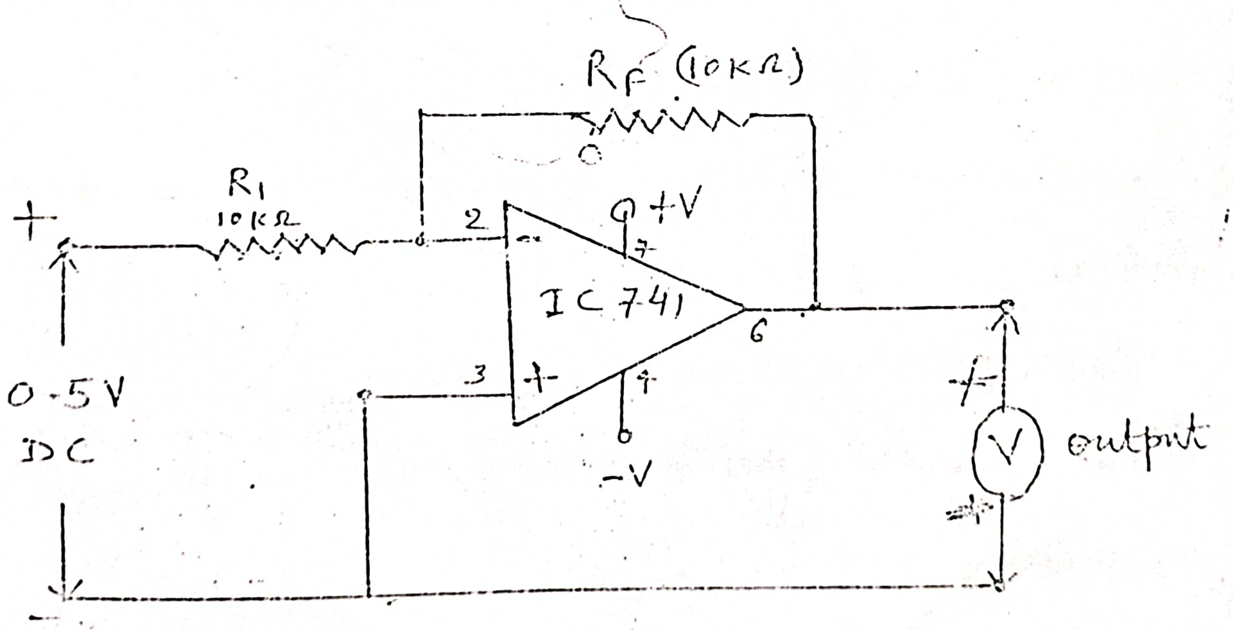
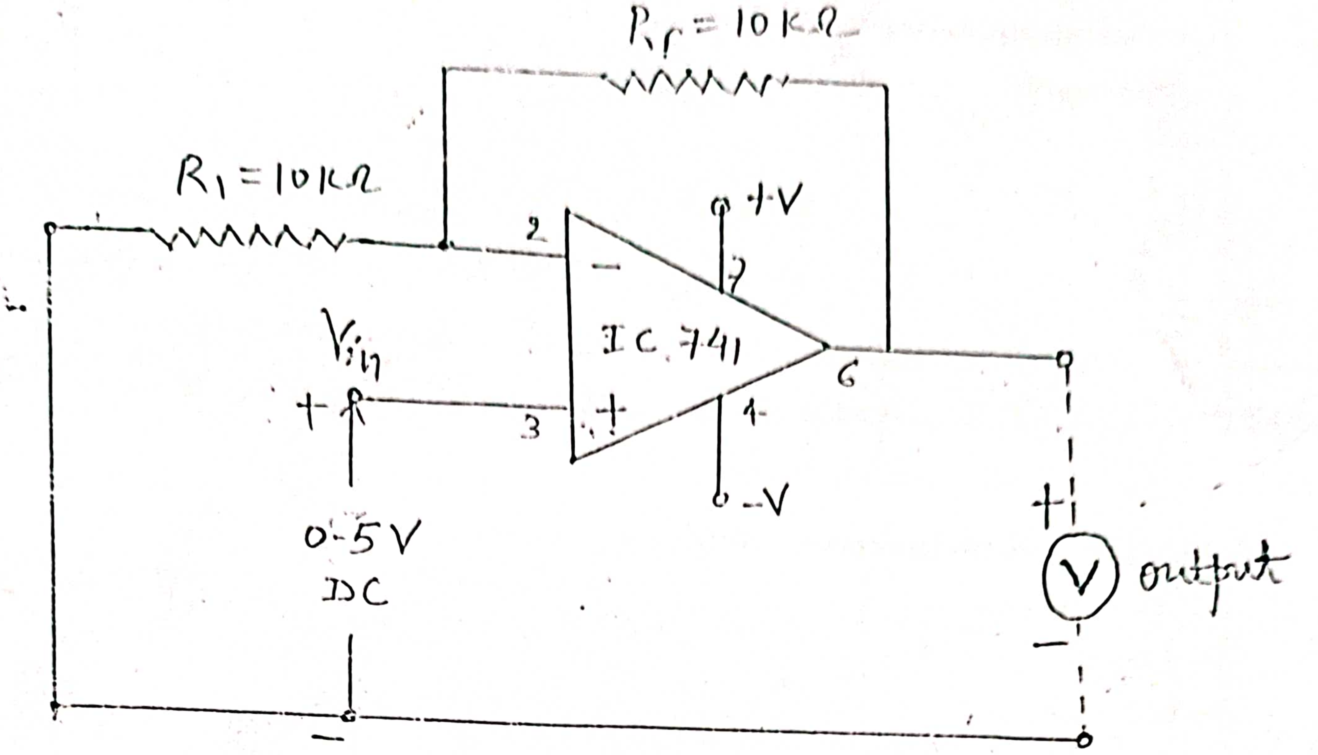


fig (1) Inverting Amplifier



fig(2) - Non- inverting amplifier

5. **Procedure :–**

(A) OP-AMP AS INVERTING AMPLIFIER-

(1) Connect the circuit as shown in fig.(1).

(2) Switch ON the instrument using ON/OFF toggle switch.

(3) Set the input supply voltage to 1 V DC and note down the output voltage using DC voltmeter.

(4) Repeat step – 3 for different input voltages.

(B) OP-AMP AS NON-INVERTING AMPLIFIER-

(1) Connect the circuit as shown in fig.(2).

(2) Switch ON the instrument using ON/OFF toggle switch.

(3) Set the input supply voltage to 1 VDC and note down the output voltage using DC voltmeter.

(4) Repeat step-3 for different input voltages.

6. Observation Table:-

Table-1. For Inverting amplifier

S. No.	V_{in}(volt)	V_{out} (Theoritical) (volt)	V_{out} (Practical)(volt)
1			
2			
3			
4			
5			

Table-2. For Non-Inverting amplifier

S. No.	V_{in}(volt)	V_{out} (Theoritical) (volt)	V_{out} (Practical) (volt)
1			
2			
3			
4			
5			

7. Calculations:-

The output voltage of the inverting amplifier

$$V_{out} = -V_{in} \, (R_f/R_1)$$

=................volt

The output voltage of the Non-inverting amplifier

$$V_{out} = +V_{in} \, [1 + (R_f/R_1)]$$

=................volt

8. Results:- It is clear from the table-1 and table-2 that the theoretical a practical values of output voltages are approximately equal. Hence the op-a works as an inverting and non – inverting amplifier.

9. Precautions :-

I. All connections should be tight.

II. Readings should be taken very carefully.

Jared Sen

Exp NO — 03

Expt. No. ....

Date ....

Page No. ....

Object — Study of operational Amplifier as adder and subtractor.

Apparatus Required → op-amp, trainer kit, connection wires. etc.

Theory and formula used →

The op-amp is a versatile device that can be used to amplify dc as well as ac input signals and was originally designed for computing such mathematical functions as addition, substraction, multiplication and integration. With the addition of suitable external feedback components, the modern day op-amp can be used for a variety of applications such as ac and dc signal amplification, active filters, oscillators, comparators, regulators and others.

Whenever we need to combine two or more analog signals into a single output, the summing amplifier or adder is a natural choice.

The summing amplifier or adder combines all the amplified input signals into a single output, given by

$$V_{out} = -\left(\frac{R_f}{R_1} V_1 + \frac{R_f}{R_2} V_2\right)$$

If $R_f = R_1 = R_2 = 1K\Omega$

then $V_{out} = -(V_1 + V_2)$

Teacher's Signature :

This means that the output voltage is equal to the negative sum of all the inputs times the gain of the circuit (R_f/R_1).

If $R_1 = R_2 = R_f$, then the output voltage is equal to the negative sum of all input voltages

i.e.

$$V_o = -(V_1 + V_2)$$

A subtractor is a basic differential amplifier. The output voltage of the differential amplifier is

$$V_o = -\frac{R_f}{R_1}(V_1 - V_2)$$

If $R_f = R_1$

then $V_o = -(V_1 - V_2)$

or $V_o = V_2 - V_1$

Procedure → (for Adder)

(1) Connect the circuit as shown in fig (1)

(2) Switch on the Instrument using ON/OFF toggle switch. ~~provided~~

(3) Apply ~~different~~ input voltages at 1V from both the ~~pin no. 2&3~~ supplies and note down the output voltages.

~~(4) Calculate the output voltage using formula~~

(4) Repeat step-3 for different input voltages.

(5) Calculate the output voltage using formula.

for Subtractor →

(1) Connect the circuit as shown in fig (2).

(2) Switch ON the instrument using ON/OFF toggle switch.

(3) Apply different input voltages at pin no. 2 & 3 and note down the output voltages.

(4) Calculate the output voltage using formula

Circuit Diagram →

off-set voltage.

Date

onn-set voltage

Expt. No.

Page No.

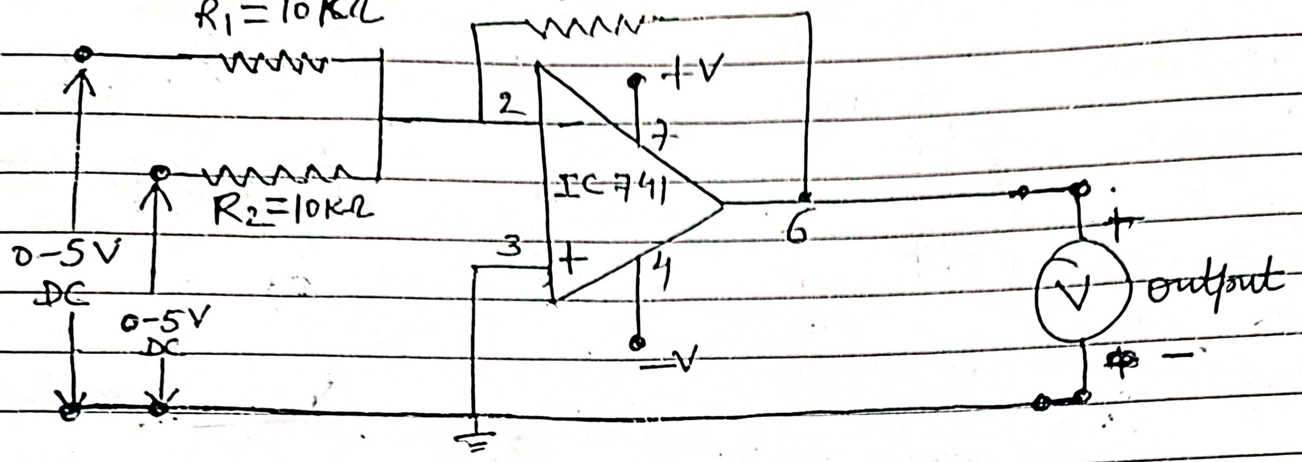


fig ① Op-amp as Adder

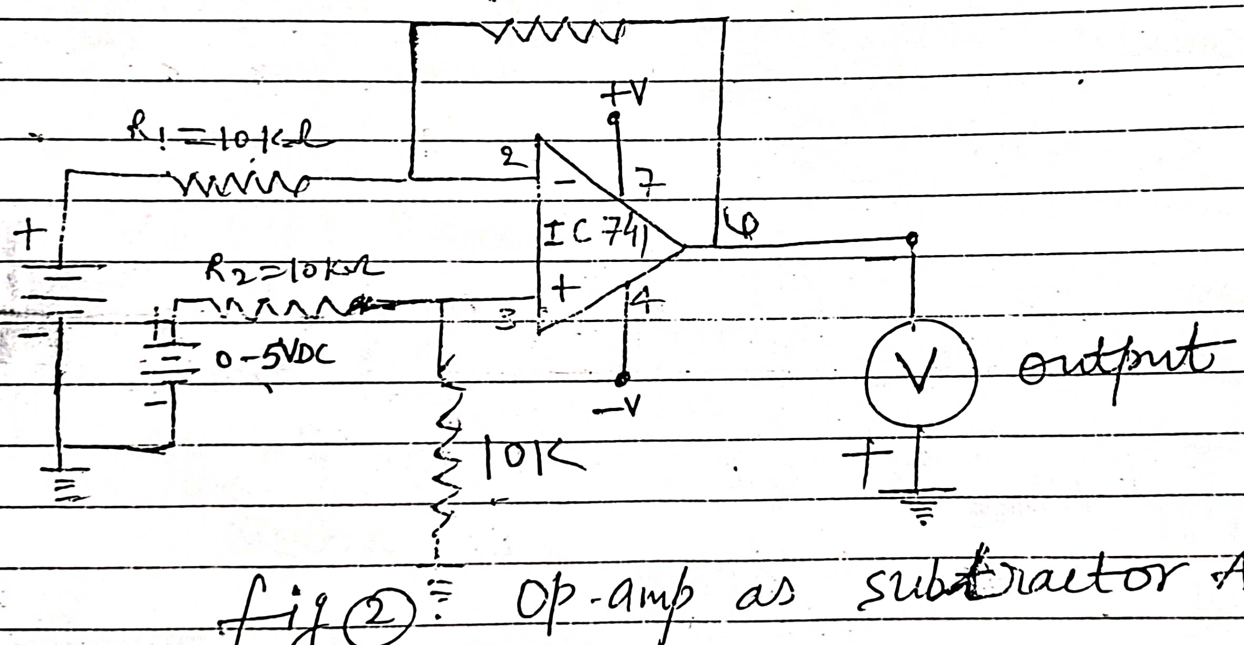


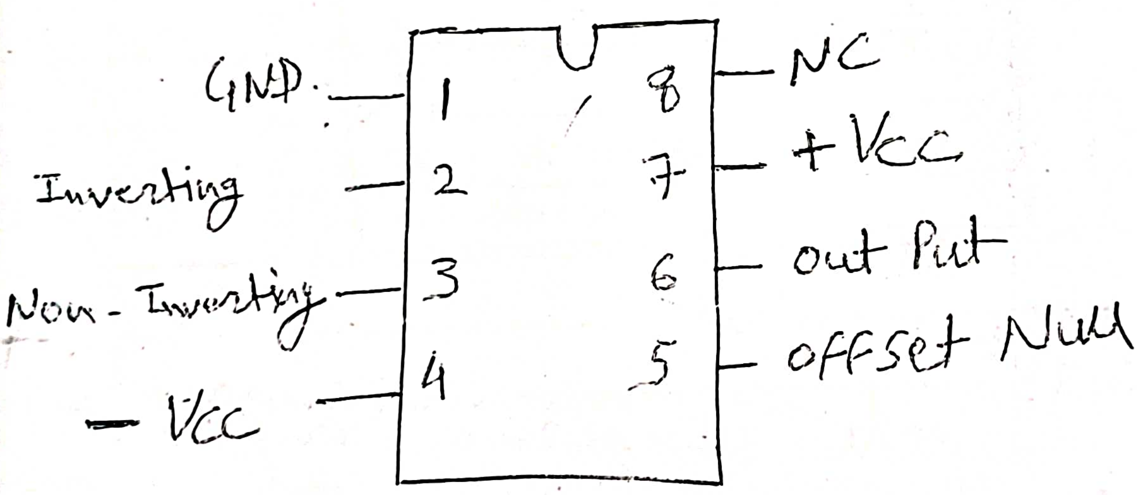
fig ② Op-amp as subtractor Amplifier

Precaution →

① The apparatus should not be kept on for long time

② The connection should be neat and tight.

Pin Diagram of op-amp. 741.



Observation Table → for Adder

S.N.	V_1 (volts)	V_2 (volts)	R_1	R_2	R	Theoritical $V_0 = (V_1 + V_2)$	practical $V_0 = (V_1 + V_2)$
1							
2							
3							
4							
5							

for subtractor →

S.N.	V_1 (volts)	V_2 (volts)	Theoritical $V_0 = V_2 - V_1$	practical $V_0 = V_2 - V_1$
1				
2				
5				

Result →

The operational amplifier as a adder and a subtractor are studied. It is clear from the tables that the theoritical and experimental values are same.

1. **Object :-** **Study of operational amplifier (741) as integrator.**

2. **Apparatus Required :-** Op-amp. kit, connection wires, CRO with probe.

3. **Theory and Formula used :-** The Op-amp is a versatile device that can be used to amplify dc as well as ac input signals and was originally designed for computing such mathematical functions as addition, subtraction, multiplication and integration. with the addition of suitable external feedback components the modern day. Op-amp can be used for a variety of applications such as dc and ac signal amplification, active filters, oscillators, comparators, regulators and others.

Integrator is a circuit in which the output voltage waveform is the integral of the input voltage waveform; such a circuit is obtained by using a basic inverting amplifier configuration if the feedback resistor R_f is replaced by a capacitor C_F.

The expression for the output voltage V_o can be obtained by writing Kirchhoff's current equation at node V_2:

$$I_1 = I_b + I_f$$

Since I_b is negligible small therefore $I_1 \approx I_f$

We know that the relationship between current through and voltage across the capacitor is

$$i_c = C\frac{dV_c}{dt}$$

Therefore,

$$\frac{V_{in} - V_2}{R_1} = C_f\left[\frac{d(V_2 - V_O)}{dt}\right]$$

However, $V_1 = V_2 = 0$, because A is very large.

Therefore,

$$\frac{V_{in}}{R_1} = -C_f\frac{dV_o}{dt}$$

The output voltage can be obtained by integrating both sides with respect to time

$$\int_0^t \frac{V_{in}}{R_1} = \int_0^t C_f \frac{d(-V_o)}{dt} dt$$

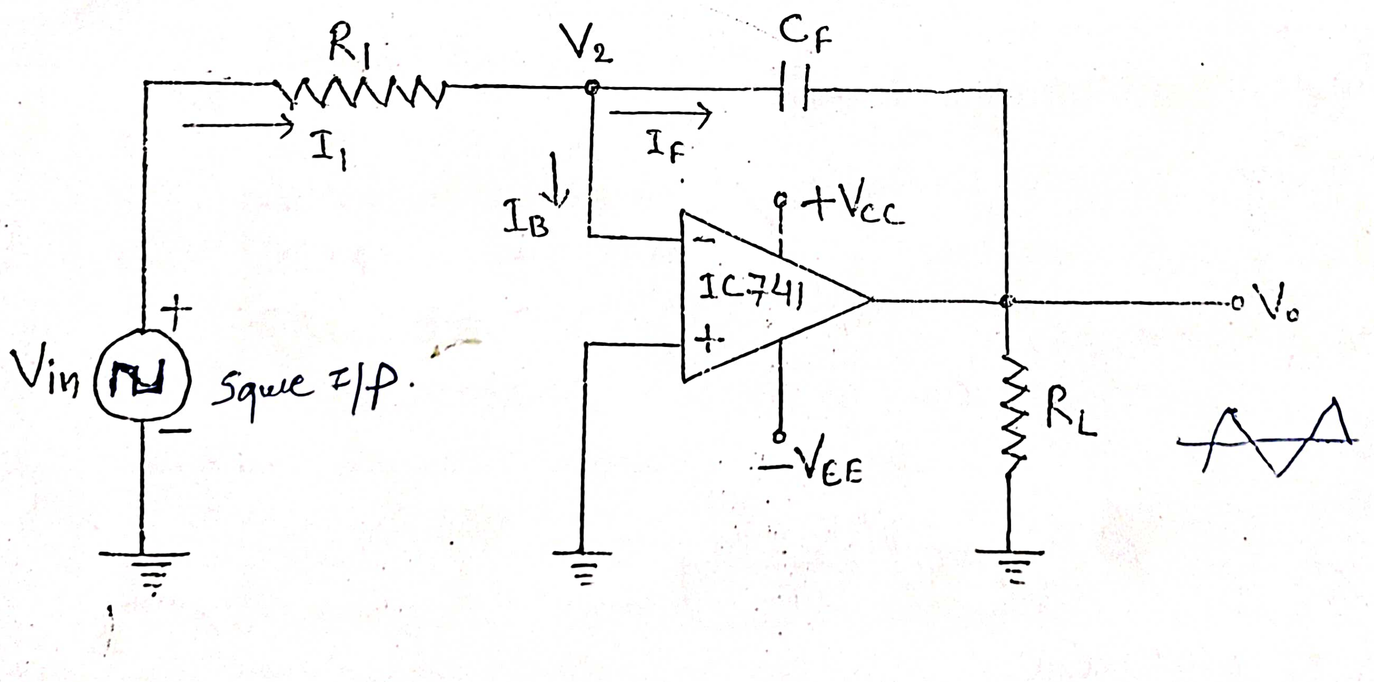
$$= C_f\,(-V_o) + V_{O\,(t=0)}$$

Therefore,

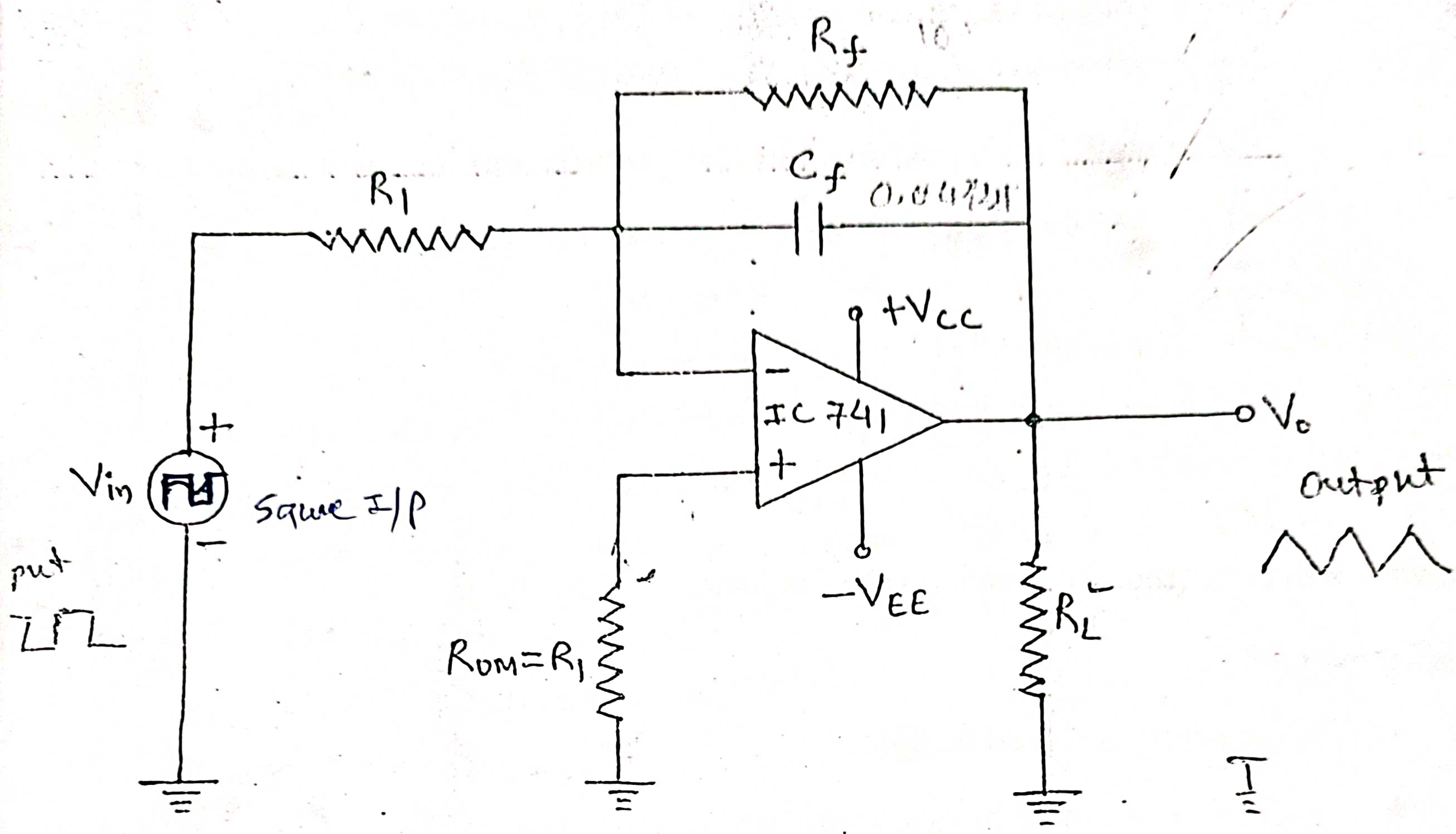
$$V_o = -\frac{1}{R_1 C_f}\int_0^t V_{in}dt + C \quad \text{.........................(A)}$$

Where C is the integration constant and is proportional to the value of the output voltage V_o time $t = 0$ seconds. When $V_{in} = 0$, the integrator works as an open loop amplifier, this is because the capacitor C_f acts as an open circuit $(X_{cf} = \infty)$ to the input offset voltage V_{io}. In other words the input offset voltage V_{io} and the parts of the input current charging capacitor C_f produce the error voltage at the output of the integrator. Therefore in the practical integrator shown in fig. (2) to reduce the error voltage at the output, a resistor R_f is connected across the feedback capacitor C_f. Thus R_f limits the low frequency gain and hence minimizes the variation in the output voltage.

4. Circuit Diagram:-



Fig(1). Basic Integrator circuit



Fig(2). A practical Integrator.

5. Procedure :-

I. Make the connection as shown in fig (2).

II. Set $R_1 = 10K\Omega$, $R_f = 10K\Omega$ $R_{om} = 10K\Omega$, $C_f = 0.1\mu F$.

III. Apply the square wave input voltage V_i (in-built) and observe its amplitude and frequency on CRO.

IV. Switch ON the unit and observe the output (triangular wave) amplitude and frequency on CRO.

V. Tabulate the amplitude and frequency of input and output by varying the frequency of the input by frequency pot.

VI. Verify the results with the help of equation (A).

6. Calculation :-

Input Voltage V_{in} (square wave) = …… volt (p − p)

Frequency of V_{in} = …… KHz

Output V_o (triangular wave) = …… volt (p − p)

Frequency of V_o = …… KHz

Now,

$$V_0 = \frac{1}{-R_1 C_f} \int_0^{t} V_{in}\, dt$$

$$= \cdots\cdots \; volt$$

7. Result :- It is clear from the calculation that equation (A) is verified.

8. Precautions :-

I. All the connections should be tight.

II. Readings of CRO should be taken very carefully.

EXPERIMENT NO. - 05

1. **Object :- Study of operational amplifier (IC741) as differentiator.**

2. **Apparatus Required :-** Op-amp. kit, connection wires, CRO with probe.

3. **Theory and Formula used :-** The Op-amp is a versatile device that can be used to amplify dc as well as ac input signals and was originally designed for computing such mathematical functions as addition, subtraction, multiplication and integration, with the addition of suitable external feedback components the modern day. Op-amp can be used for a variety of applications such as dc and ac signal amplification, active filters, oscillators, comparators, regulators and others.

Differentiator is a circuit that performs the mathematical operation of differentiation; that is the output voltage waveform is 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 expression for the output voltage can be obtained from Kirchhoff's current equation written at node V_2 as follows:

$$I_C = I_B + I_F$$

Since, $I_B = 0$ therefore $I_C = I_F$

$$C_1\left[\frac{d(V_{in} - V_2)}{dt}\right] = \frac{V_2 - V_0}{R_F}$$

But, $V_1 = V_2 = 0$ V (approx.), because A is very large.

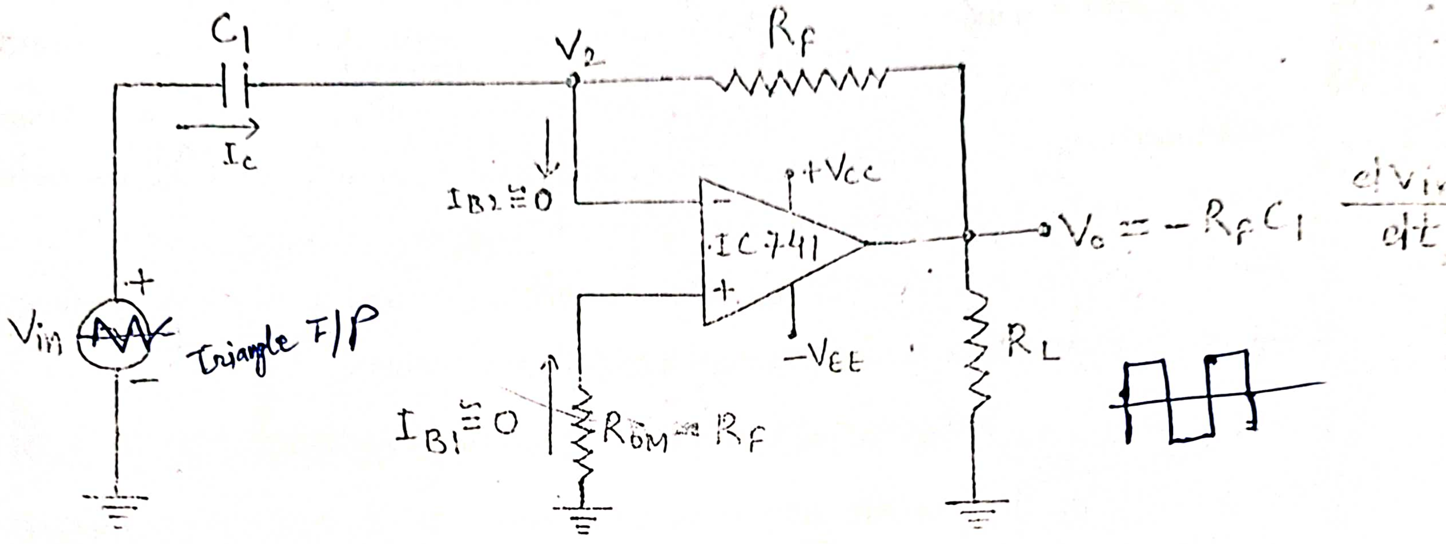
Therefore,

$$C_1\frac{dV_{in}}{dt} = -\frac{V_0}{R_F}$$

$$V_0 = -R_F C_1\frac{dV_{in}}{dt} \quad \text{.............................(A)}$$

Thus the output Vo is equal to the $R_F C_1$ times the negative instantaneous rate of change of the input voltage Vin with time. Since the differentiator performs the reverse of the integrators function, a cosine wave input will produce a sine wave output or a triangular input will produce a square wave output. However, the differentiator of fig.(1) will not do because it . practical problems. By the addition of two components R_1 and C_F problems can be corrected as shown in fig.(2). This circuit is a practical differentiator.

4. Circuit Diagram:-



Fig(1) Basic differentiator circuit

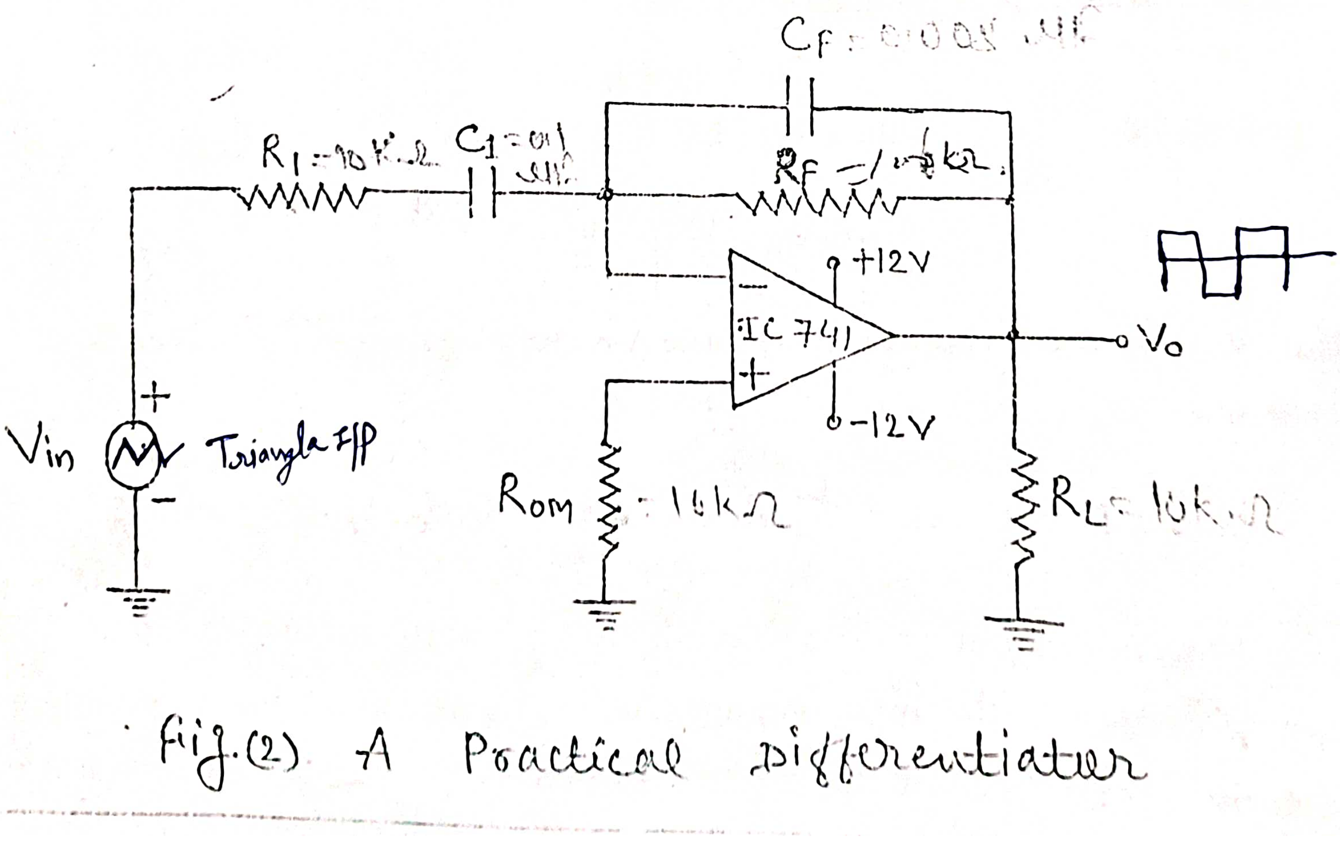


Fig.(2) A Practical Differentiator

Procedure :-

$C_1 = 0.1\ \mu F$
$C_F = 0.005\ \mu F$
$R_L = 10K\ \Omega$

I. Make the connection as shown in fig (2).

II. Set $R_1 = 270\Omega, R_f = 100K\Omega, R_{om} = 10K\Omega,\ C_1 = 0.1\mu F$

III. Apply the triangular wave input voltage Vin (in-built) and observe its amplitude and frequency on CRO.

IV. Switch on the unit and observe the output (square wave) amplitude and frequency on CRO.

V. Tabulate the amplitude and frequency of input and output by varying the frequency of the input by frequency.

VI. Verify the results with the help of equation (A).

6. Calculation :-

Input Voltage V_{in}(triangular wave) = ... volt (p – p)

Frequency of V_{in} = ... KHz

Output V_o(square wave) = ... volt (p – p)

Frequency of V_o = ... KHz

Now,

$$V_0 = -R_f C_1 \frac{dV_{in}}{dt}$$

$$= -(100 \times 10^3 \times 2.2 \times 10^{-6}) \frac{dV_{in}}{dt}$$

$$= \cdots\cdots volt$$

7. Result :- It is clear from the calculation that equation (A) is verified.

8. Precautions :-

I. All connections should be tight.

II. Readings of CRO should be taken very carefully.

Experiment no. – 06

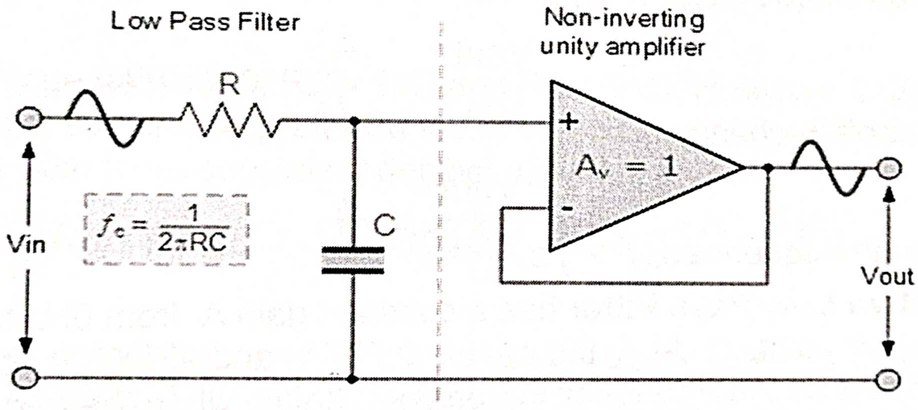
Aim:- designing of a first order low pass filter using op-amp.

Apparatus: - op-amp. (741) resistance, capacitance, connecting wires.

Theory :- Active Low Pass Filter

The most common and easily understood active filter is the **Active Low Pass Filter**. Its principle of operation and frequency response is exactly the same as those for the previously seen passive filter, the only difference this time is that it uses an op-amp for amplification and gain control. The simplest form of a low pass active filter is to connect an inverting or non-inverting amplifier, the same as those discussed in the Op-amp tutorial, to the basic RC low pass filter circuit as shown.

First Order Low Pass Filter



This first-order low pass active filter, consists simply of a passive RC filter stage providing a low frequency path to the input of a non-inverting operational amplifier. The amplifier is configured as a voltage-follower (Buffer) giving it a DC gain of one, Av = +1 or unity gain as opposed to the previous passive RC filter which has a DC gain of less than unity.

The advantage of this configuration is that the op-amps high input impedance prevents excessive loading on the filters output while its low output impedance prevents the filters cut-off frequency point from being affected by changes in the impedance of the load.

While this configuration provides good stability to the filter, its main disadvantage is that it has no voltage gain above one. However, although the voltage gain is unity the power gain is very high as its output impedance is

Gain of a first-order low pass filter

$$\text{Voltage Gain, } (Av) = \frac{Vout}{Vin} = \frac{A_F}{\sqrt{1+\left(\frac{f}{fc}\right)^2}}$$

- Where:
- A_F = the pass band gain of the filter, (1 + R2/R1)
- f = the frequency of the input signal in Hertz, (Hz)
- fc = the cut-off frequency in Hertz, (Hz)

Thus, the operation of a low pass active filter can be verified from the frequency gain equation above as:

- 1. At very low frequencies, $f < fc$ $\quad \frac{Vout}{Vin} \cong A_F$
- 2. At the cut-off frequency, $f = fc$ $\quad \frac{Vout}{Vin} = \frac{A_F}{\sqrt{2}} = 0.707\,A_F$
- 3. At very high frequencies, $f > fc$ $\quad \frac{Vout}{Vin} < A_F$

Thus, the **Active Low Pass Filter** has a constant gain A_F from 0Hz to the high frequency cut-off point, f_C. At f_C the gain is $0.707A_F$, and after f_C it decreases at a constant rate as the frequency increases. That is, when the frequency is increased tenfold (one decade), the voltage gain is divided by 10.

In other words, the gain decreases 20dB (= 20*log(10)) each time the frequency is increased by 10. When dealing with filter circuits the magnitude of the pass band gain of the circuit is generally expressed in *decibels* or *dB* as a function of the voltage gain, and this is defined as:

Magnitude of Voltage Gain in (dB)

$$Av(dB) = 20\log_{10}\left(\frac{Vout}{Vin}\right)$$

$$\therefore\ -3dB = 20\log_{10}\left(0.707\frac{Vout}{Vin}\right)$$

Active Low Pass Filter Example No1

Design a non-inverting active low pass filter circuit that has a gain of ten at low frequencies, a high frequency cut-off or corner frequency of 159Hz and an input impedance of 10KΩ.

The voltage gain of a non-inverting operational amplifier is given as:

$$A_F = 1 + \frac{R_2}{R_1} = 10$$

Assume a value for resistor R1 of 1kΩ rearranging the formula above gives a value for R2 of:

$$R_2 = (10 - 1)\ x\ R_1 = 9\ x\ 1k\Omega = 9k\Omega$$

So for a voltage gain of 10, R1 = 1kΩ and R2 = 9kΩ. However, a 9kΩ resistor does not exist so the next preferred value of 9k1Ω is used instead. Converting this voltage gain to an equivalent decibel dB value gives:

$$Gain\ in\ dB = 20\log A = 20\log 10 = 20dB$$

The cut-off or corner frequency (fc) is given as being 159Hz with an input impedance of 10kΩ. This cut-off frequency can be found by using the formula:

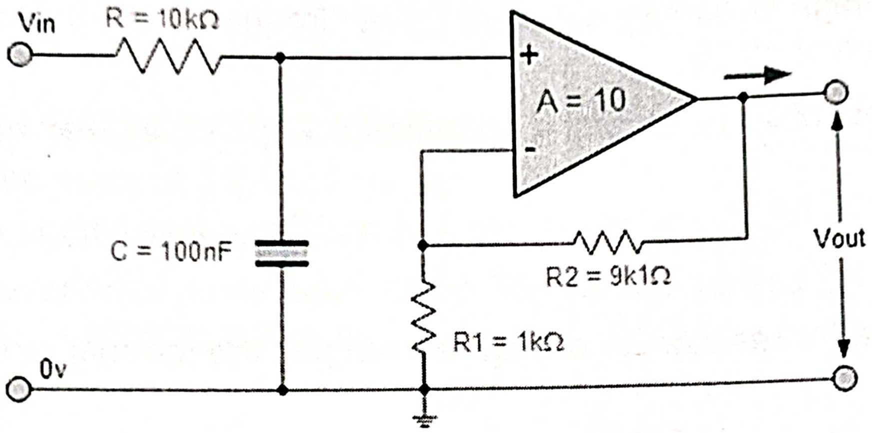
$fc = \frac{1}{2\pi RC}\ Hz$	where fc = 159Hz and R = 10kΩ.

By rearranging the above standard formula we can find the value of the filter capacitor C as:

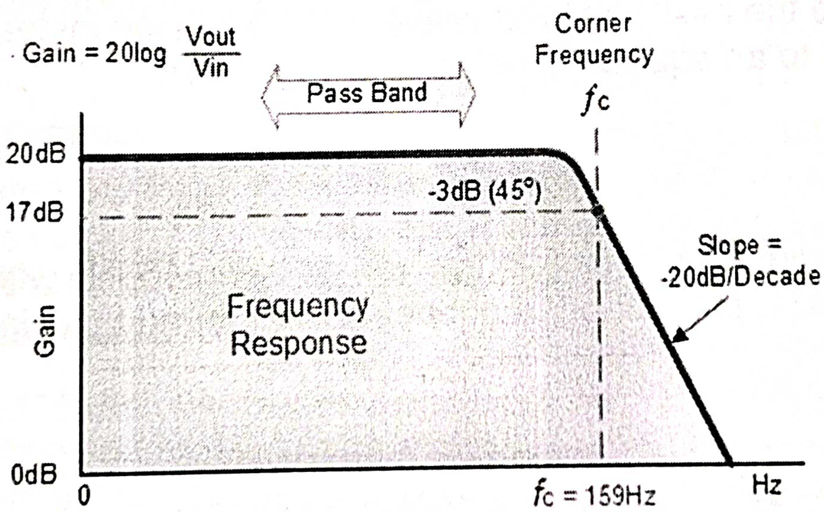
$$C = \frac{1}{2\pi fc R} = \frac{1}{2\pi \; x \; 159 \; x \; 10k\Omega} = 100nF$$

Thus the final low pass filter circuit along with its frequency response is given below as:

Low Pass Filter Circuit



Frequency Response Curve



If the external impedance connected to the input of the filter circuit changes, this impedance change would also affect the corner frequency of the filter (components connected together in series or parallel). One way of avoiding any external influence is to place the capacitor in parallel with the feedback resistor R2 effectively removing it from the input but still maintaining the filters characteristics.

However, the value of the capacitor will change slightly from being 100nF to 110nF to take account of the 9k1Ω resistor, but the formula used to calculate the cut-off corner frequency is the same as that used for the RC passive low pass filter.

$$fc = \frac{1}{2\pi C R_2} \ Hertz$$

Observation table :-

Fixed input volgae (Vin) = ...

s.no.	Frequency f (Hz)	o/p voltage	Voltage Gain (*Vo / Vin*)	Gain, (dB) 20log(*Vo / Vin*
1				
2				
3				
4				
5				
-				
-				

Result :-

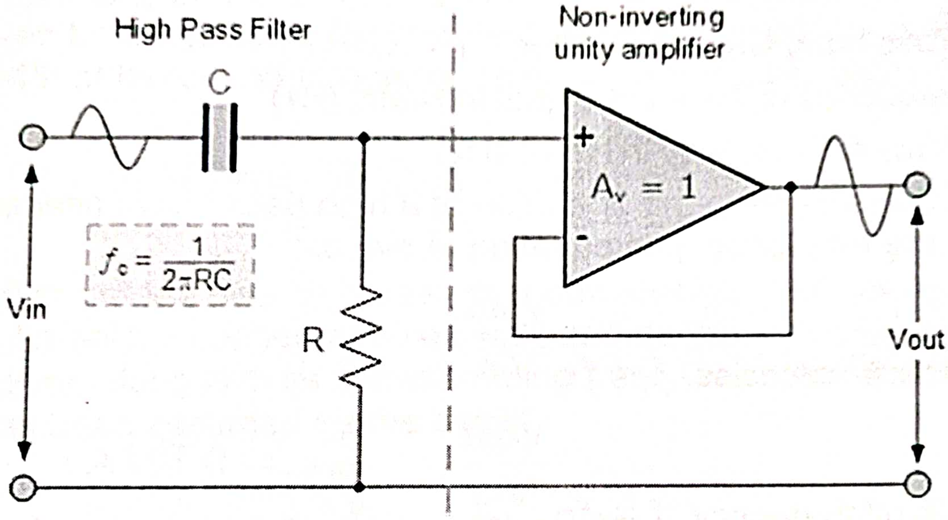
Precautions:-

Experiment no. - 07

Aim:- designing of a first order High pass filter using op-amp.

Apparatus: - op-amp. (741) resistance, capacitance, connecting wires.

Theory :- First Order High Pass Filter



Technically, there is no such thing as an **active high pass filter**. Unlike Passive High Pass Filters which have an "infinite" frequency response, the maximum pass band frequency response of an active high pass filter is limited by the open-loop characteristics or bandwidth of the operational amplifier being used, making them appear as if they are band pass filters with a high frequency cut-off determined by the selection of op-amp and gain.

In the Operational Amplifier tutorial we saw that the maximum frequency response of an op-amp is limited to the Gain/Bandwidth product or open loop voltage gain (A_v) of the operational amplifier being used giving it a bandwidth limitation, where the closed loop response of the op amp intersects the open loop response.

A commonly available operational amplifier such as the uA741 has a typical "open-loop" (without any feedback) DC voltage gain of about 100dB maximum reducing at a roll off rate of -20dB/Decade (-6db/Octave) as the input frequency increases. The gain of the uA741 reduces until it reaches unity gain, (0dB) or its "transition frequency" (*ft*) which is about 1MHz. This causes the op-amp to have a frequency response curve very similar to that of a first-order low pass filter and this is shown below.

Gain for an Active High Pass Filter

$$\text{Voltage Gain, } (Av) = \frac{Vout}{Vin} = \frac{A_F\left(\frac{f}{fc}\right)}{\sqrt{1+\left(\frac{f}{fc}\right)^2}}$$

- Where:
- A_F = the Pass band Gain of the filter, (1 + R2/R1)
- f = the Frequency of the Input Signal in Hertz, (Hz)
- fc = the Cut-off Frequency in Hertz, (Hz)

Just like the low pass filter, the operation of a high pass active filter can be verified from the frequency gain equation above as:

- 1. At very low frequencies, $f < fc$ $\quad \frac{Vout}{Vin} < A_F$
- 2. At the cut-off frequency, $f = fc$ $\quad \frac{Vout}{Vin} = \frac{A_F}{\sqrt{2}} = 0.707A_F$
- 3. At very high frequencies, $f > fc$ $\quad \frac{Vout}{Vin} \cong A_F$

Then, the **Active High Pass Filter** has a gain A_F that increases from 0Hz to the low frequency cut-off point, f_c at 20dB/decade as the frequency increases. At f_c the gain is 0.707*A_F, and after f_c all frequencies are pass band frequencies so the filter has a constant gain A_F with the highest frequency being determined by the closed loop bandwidth of the op-amp.

When dealing with filter circuits the magnitude of the pass band gain of the circuit is generally expressed in *decibels* or *dB* as a function of the voltage gain, and this is defined as:

Magnitude of Voltage Gain in (dB)

$$Av(dB) = 20\log_{10}\left(\frac{Vout}{Vin}\right)$$

$$\therefore\ -3dB = 20\log_{10}\left(0.707\frac{Vout}{Vin}\right)$$

For a first-order filter the frequency response curve of the filter increases by 20dB/decade or 6dB/octave up to the determined cut-off frequency point which is always at -3dB below the maximum gain value. As with the previous filter circuits, the lower cut-off or corner frequency (fc) can be found by using the same formula:

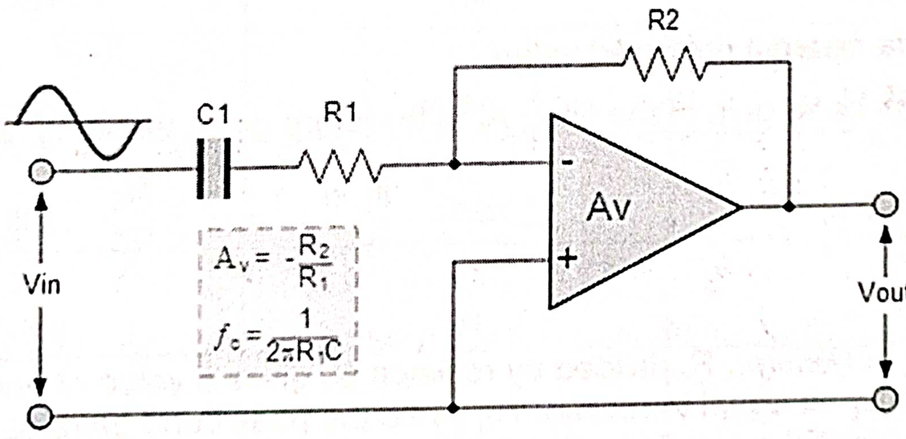
$$fc = \frac{1}{2\pi RC} \, Hz$$

The corresponding phase angle or phase shift of the output signal is the same as that given for the passive RC filter and **leads** that of the input signal. It is equal to **+45°** at the cut-off frequency fc value and is given as:

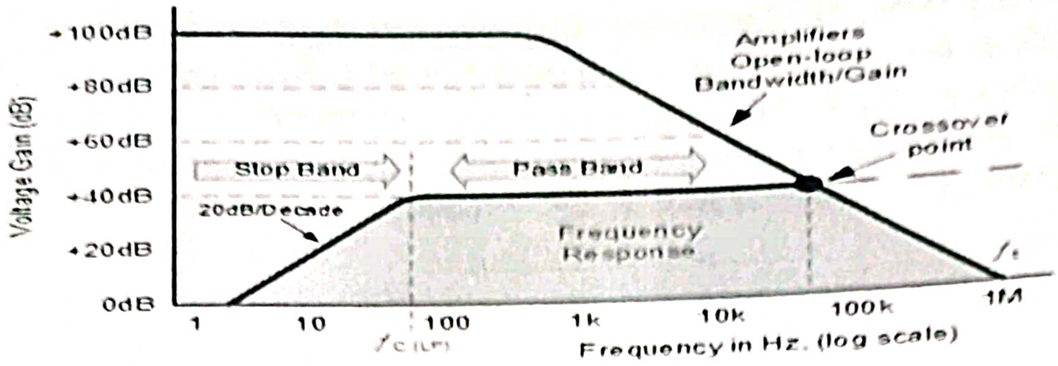
$$\text{Phase Shift } \phi = \tan^{-1}\left(\frac{1}{2\pi fRC}\right)$$

A simple first-order active high pass filter can also be made using an inverting operational amplifier configuration as well, and an example of this circuit design is given along with its corresponding frequency response curve. A gain of 40dB has been assumed for the circuit.

Inverting Operational Amplifier Circuit



Frequency Response Curve



Active High Pass Filter Example No1

A first order active high pass filter has a pass band gain of two and a cut-off corner frequency of 1kHz. If the input capacitor has a value of 10nF, calculate the value of the cut-off frequency determining resistor and the gain resistors in the feedback network. Also, plot the expected frequency response of the filter.

With a cut-off corner frequency given as 1kHz and a capacitor of 10nF, the value of R will therefore be:

$$R = \frac{1}{2\pi f_C C} = \frac{1}{2\pi.1000.10\times10^{-9}} = 15.92\text{k}\Omega$$

or 16kΩ to the nearest preferred value.

Thus the pass band gain of the filter, A_F is therefore given as being: 2.

$$A_F = 1 + \frac{R_2}{R_1}, \quad \therefore 2 = 1 + \frac{R_2}{R_1} \quad \text{and} \quad \frac{R_2}{R_1} = 1$$

As the value of resistor, R_2 divided by resistor, R_1 gives a value of one. Then, resistor R_1 must be equal to resistor R_2, since the pass band gain, $A_F = 2$. We can therefore select a suitable value for the two resistors of say, 10kΩ each for both feedback resistors.

So for a high pass filter with a cut-off corner frequency of 1kHz, the values of R and C will be, 10kΩ and 10nF respectively. The values of the two feedback resistors to produce a pass band gain of two are given as: $R_1 = R_2 = 10\text{k}\Omega$

The data for the frequency response bode plot can be obtained by substituting the values obtained above over a frequency range from 100Hz to 100kHz into the equation for voltage gain:

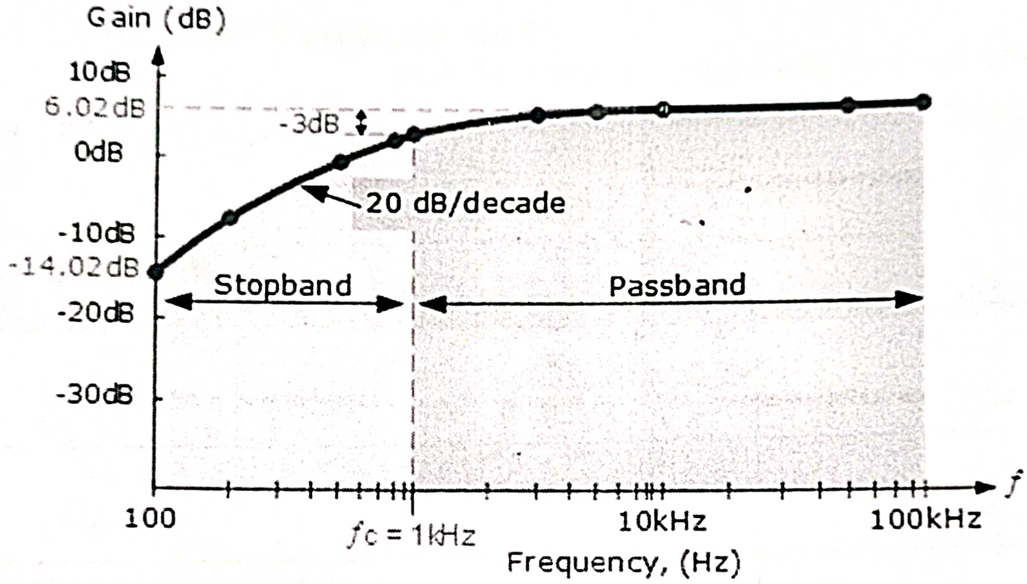
$$\text{Voltage Gain, (Av)} = \frac{\text{Vout}}{\text{Vin}} = \frac{A_F\left(\frac{f}{fc}\right)}{\sqrt{1+\left(\frac{f}{fc}\right)^2}}$$

This then will give us the following table of data.

Frequency, f (Hz)	Voltage Gain (*Vo / Vin*)	Gain, (dB) 20log(*Vo / Vin*)
100	0.20	-14.02
200	0.39	-8.13
500	0.89	-0.97
800	1.25	1.93
1.000	1.41	3.01
3.000	1.90	5.56
5.000	1.96	5.85
10.000	1.99	5.98
50.000	2.00	6.02

The frequency response data from the table above can now be plotted as shown below. In the stop band (from 100Hz to 1kHz), the gain increases at a rate of 20dB/decade. However, in the pass band after the cut-off frequency, f_c = 1kHz, the gain remains constant at 6.02dB. The upper-frequency limit of the pass band is determined by the open loop bandwidth of the operational amplifier used as we discussed earlier. Then the bode plot of the filter circuit will look like this.

The Frequency Response Bode-plot for our example



Applications of **Active High Pass Filters** are in audio amplifiers, equalizers or speaker systems to direct the high frequency signals to the smaller tweeter speakers or to reduce any low frequency noise or "rumble" type distortion. When used like this in audio applications the active high pass filter is sometimes called a "Treble Boost" filter.

Result :-

Precautions:-

INSTRUCTION MANUAL
FOR
ACTIVE FILTERS USING OPERATIONAL AMPLIFIER

Active Filter Circuits have been designed to study the characteristics of a Active Filter.

The instrument comprises of the following built in parts :-

1. Fixed output DC regulated power supply of ±15V.
2. One operational amplifier IC 741 is placed behind the front panel & important connections are brought on sockets.
3. Different combinations of resistance and capacitors are provided on the front panel.

THEORY

By using operational amplifier IC and reactive elements we can build active filters. Active filters have several advantages over passive filters. Active filters can also have variable voltage gain, allow easy tuning of cutoff frequencies, etc.

CLASSIFICATION OF ACTIVE FILTER :-

We can classify the active filter in three catagories

A. First order low pass active filter.
B. Second order low pass active filter. (Band Pass)
C. Second order high pass active filters. (High Pass)

A. First Order Low Pass Filter :-

Figure (1) shows one way to build an active first order low pass filter. At low frequencies the capacitor appears open and the circuit acts like an inverting amplifier with a voltage gain of -R2/ R1. As the frequency increases, the capacitive reactance decreases causes the voltage agin to dropoff. As the frequency approaches infinity, the capacitor appears shorted and the voltage gain approaches zero.

B. Second Order Low Pass Filter (Band Pass Filter) :-

Second order low pass filter is similar to first order low pass filter except for minor modifications. In first order low pass filter the gain decreases 6db/octave beyond the cutoff frequencies. A second order low pass filter is one that decreases 12 db/octave beyond the cut-off frequency.

C. Second Order High Pass Filter:-

A High Pass Filter is that which passes high frequencies but stops lower ones. So at higher frequencies output will be maximum but as we decrease the frequency output will also decrease.

PROCEDURE

A. First Order Low Pass Filter :-

1. Connect the circuit as shown in Fig. (1).
2. Connect the Audio Frequency Function Generator across the input of the circuit and CRO across output.
3. Switch ON all the instruments one by one.
4. Set the output of Audio Frequency Function Generator at sine wave signal of 100Hz frequency. Adjust the signal level to get 1V peak to peak at the output of the filter on CRO
5. Now increase the frequency in small steps towards 1kHz & note down the observation in observation table.
6. Plot a graph Frequency vs Output gain.

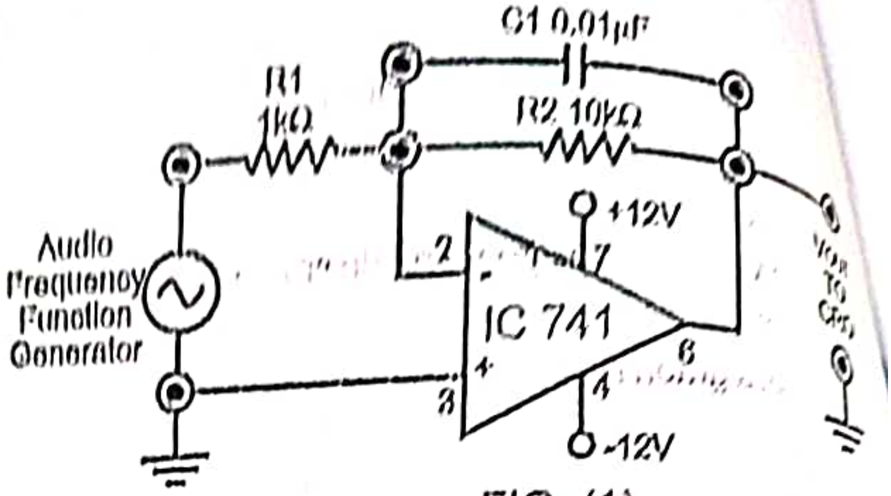


FIG. (1)

B. Second Order Low Pass Filter (Band Pass Filter) :-

1. Connect the circuit as shown in Fig. (2).
2. Connect the Audio Frequency Function Generator across the input of the circuit and CRO across output.
3. Switch ON all the instruments one by one.
4. Set the output of Audio Frequency Function Generator at sine wave signal of 100Hz frequency. Adjust the signal level to get 1V peak to peak at the output of the filter on CRO
5. Now increase the frequency in small steps towards 10kHz & note down the observation in observation table.
6. Plot a graph Frequency vs Output gain.

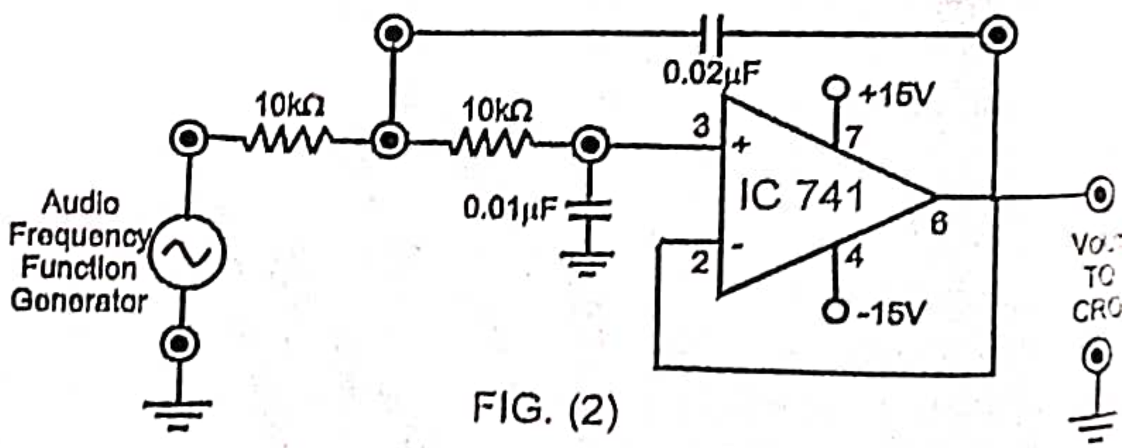


FIG. (2)

C. Second Order High Pass Filter:-

1. Connect the circuit as shown in Fig. (3).
2. Connect the Audio Frequency Function Generator across the input of the circuit and CRO across output.
3. Switch ON all the instruments one by one.
4. Set the output of Audio Frequency Function Generator at sine wave signal of 1kHz frequency.
5. Now increase the frequency in small steps towards 100kHz & note down the observation in observation table.
6. Plot a graph Frequency vs Output gain.

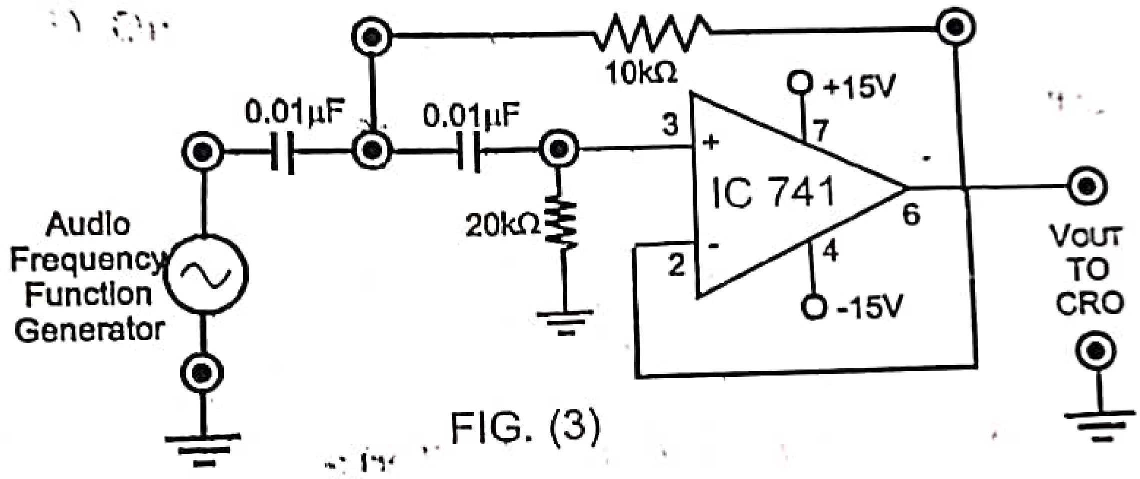


FIG. (3)

Sr. No.	Input Frequency	Input Amplitude	Output Amplitude	Gain = Output Amplitude / Input Amplitude
1.				
2.				
3.				
4.				
5.				

OBSERVATION TABLE

Experiment No: 9

ASTABLE MULTIVIBRATOR USING IC 555

AIM

To design and set up astable multivibrator of 1000 Hz frequency and 60% duty cycle using IC 555

THEORY

IC 555 timer is an analog IC used for generating accurate time delay or oscillations. The entire circuit is usually housed in an 8-pin package as specified in figures 1 & 2 below. A series connection of three resistors inside the IC sets the reference voltage levels to the two comparators at $\frac{2}{3}V_{CC}$ and $\frac{1}{3}V_{CC}$, the output of these comparators setting or resetting the flip-flop unit. The output of the flip-flop circuit is then brought out through an output buffer stage. In the stable state the $\bar{Q}$ output of the flip-flop is high (ie Q low). This makes the output (pin 3) low because of the buffer which basically is an inverter.The flip-flop circuit also operates a transistor inside the IC, the transistor collector usually being driven low to discharge a timing capacitor connected at pin 7. The description of each pin s described below,

Pin 1: (Ground): Supply ground is connected to this pin.

Pin 2: (Trigger): This pin is used to give the trigger input in monostable multivibrator. When trigger of amplitude greater than (1/3)Vcc is applied to this terminal circuit switches to quasi-stable state.

Pin 3: (Output)

Pin 4 (Reset): This pin is used to reset the output irrespective of input. A logic low at this pin will reset output. For normal operation pin 4 is connected to Vcc.

Pin 5 (Control): Voltage applied to this terminal will control the instant at which the comparator switches, hence the pulse width of the output. When this pin is not used it is bypassed to ground using a 0.01μF capacitor.

Pin 6 (Threshold): If the voltage applied to threshold terminal is greater than (2/3)V_{CC}, upper comparator switches to +Vsat and flip-flop gets reset.

Pin 7: (Discharge): When the output is low, the external capacitor is discharged through this pin

Pin 8 (V_{CC}): The power supply pin

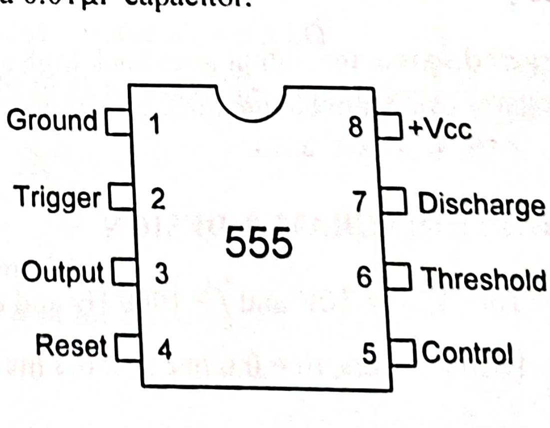


Figure 2: IC 555 pin diagram

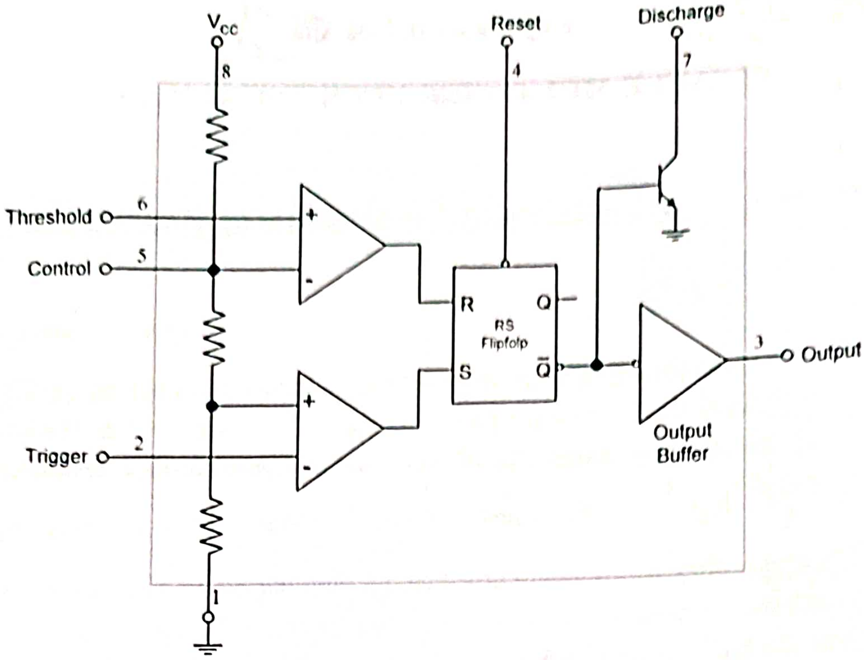


Figure 1: IC 555 Functional block diagram

Astable multivibrator using IC 555

One popular application of the 555 timer IC is as an astable multivibrator or clock Circuit. Figure 3 shows an astable circuit built using 2 external resistors and a capacitor to set the timing interval of the output signal. Capacitor C charges toward V_{CC} through external resistors R_A and R_B. Referring to figure, the capacitor voltage rises until it goes above $\frac{2}{3}V_{CC}$. This voltage is the threshold voltage at pin 6, which drives comparator 1 to trigger the flip-flop(Q low $\bar{Q}$ high) so that the output at pin 3 goes low. In addition, the discharge transistor is driven on, causing the output at pin 7 to discharge the capacitor through resistor R_B. The capacitor voltage then decreases until it drops below the trigger level $\frac{1}{3}V_{CC}$. The flipflop is triggered so that the output goes back high and the discharge transistor is turned off, so that the capacitor can again charge through resistors R_A and R_B towards V_{CC}.

CIRCUIT DIAGRAM & DESIGN

Take $V_{CC} = 10V$ and f= 1000 Hz and duty cycle = 60 %

Then t = 1 ms, t_H = 0.6 ms, t_L = 0.4 ms

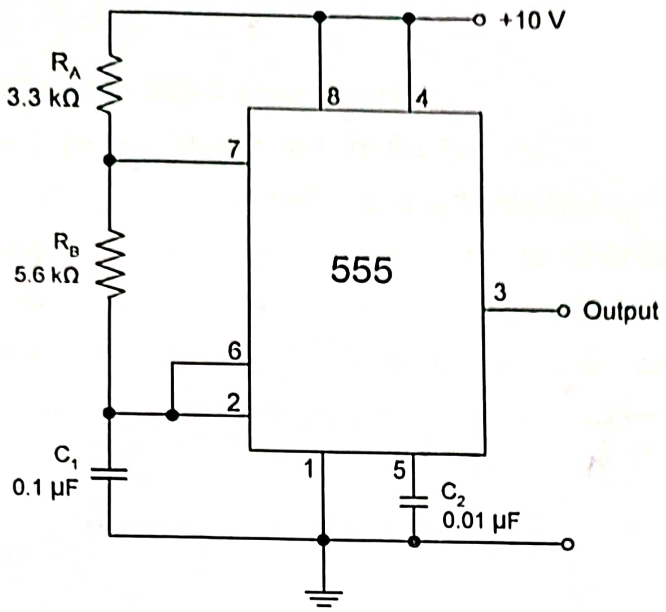


Figure 3 Astable multivibrator circuit using IC 555

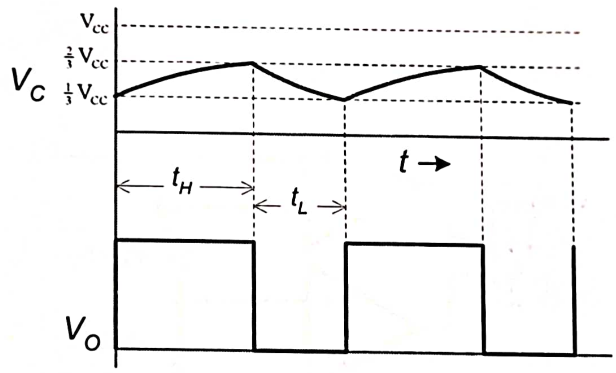


Figure 4 Waveforms of voltage across the capacitor and output voltage

Assume C = 0.1 μF

$t_L = 0.693 \times R_B \times C$ then $R_B = 5.77$ kΩ take $R_B = 5.6$ kΩ

$t_H = 0.693 \times (R_A + R_B) \times C$ then $R_A = 3.06$ kΩ take $R_A = 3.3$ kΩ

The resistance R_A and R_B should be in the range of 1k to 10k to limit the collector current of the internal transistor.

PROCEDURE

1. Set up the circuit after verifying the condition of IC
2. Observe the waveforms at pin number 3 and 6 of the IC

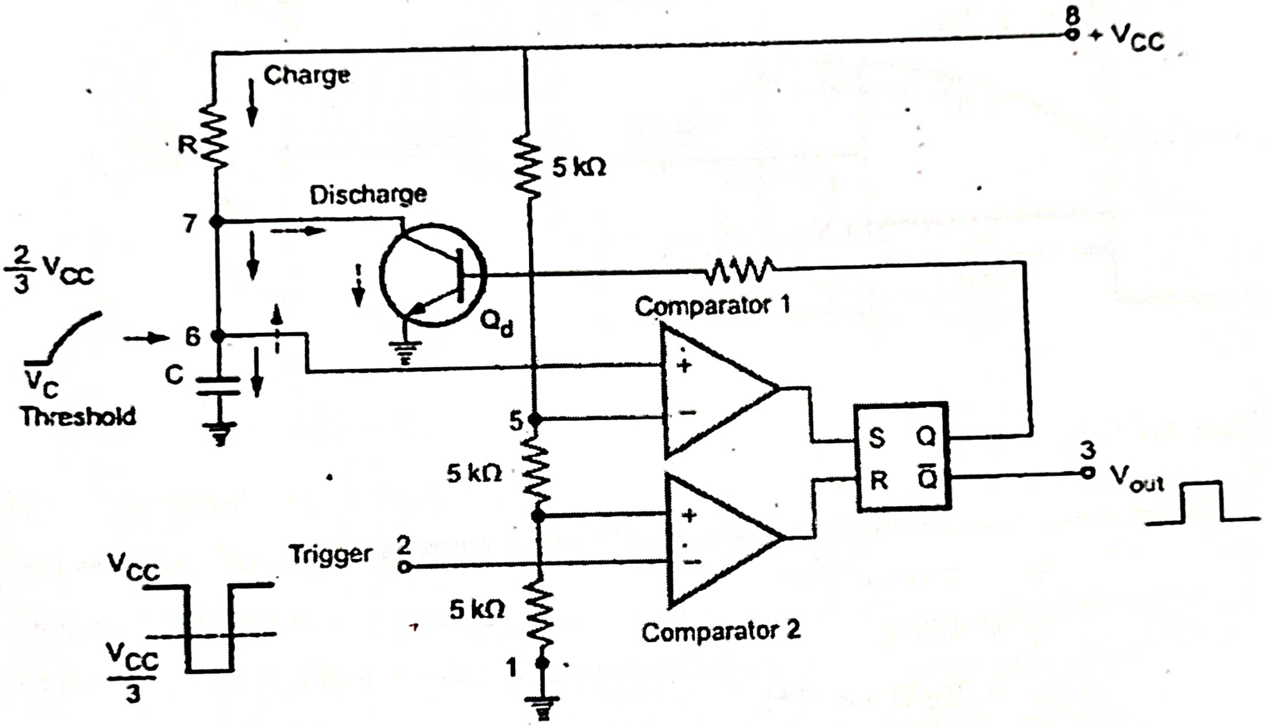
RESULT

Astable multivibrator using timer IC 555 is designed and setup, and the waveforms are obtained.

Exp. NO-10

Monostable Multivibrator using 555 timer.

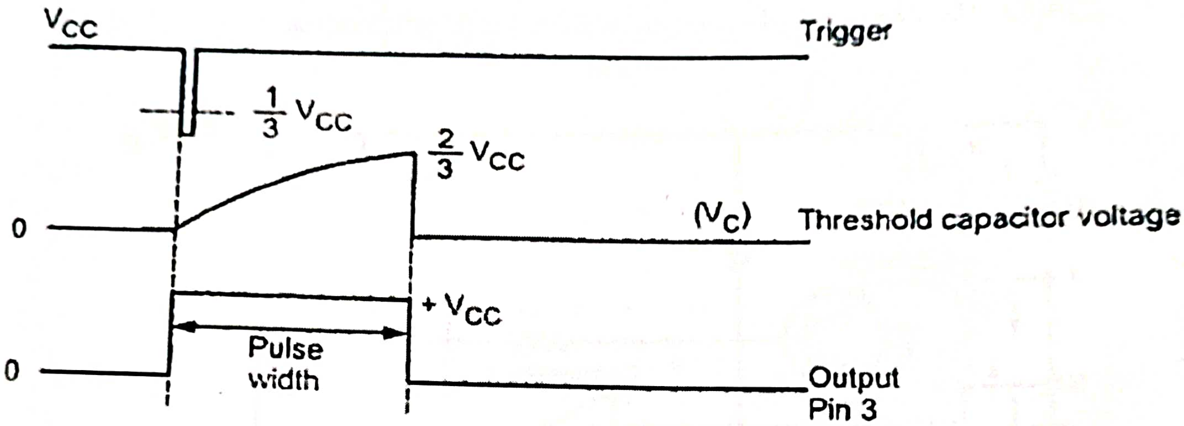
A monostable multivibrator often called a one-shot multivibrator, is a pulse generator circuit in which the duration of the pulse is determined by the R-C network, connected externally to the 555 timer. In such a vibrator, one state of output is stable while the other is quasi-stable (unstable). For auto-triggering of output from quasi-stable state to stable state energy is stored by an externally connected capacitor C to a reference level. The time taken in storage determines the pulse width. The transition of output from stable state to quasi-stable state is accomplished by external triggering. The functional block of monostable mode of operation is shown in figure.



Operation

Initially, when the output at pin 3 is low i.e. the circuit is in a stable state, the transistor is on and capacitor- C is shorted to ground. When a negative pulse is applied to pin 2, the trigger input falls below +1/3 V_{CC}, the output of comparator goes high which resets the flip-flop and consequently the transistor turns off and the output at pin 3 goes high. This is the transition of the output from stable to quasi-stable state, as shown in figure. As the discharge transistor is cut-off, the capacitor C begins charging toward $+V_{CC}$ through resistance R with a time constant equal to RC. When the increasing capacitor voltage becomes slightly greater than +2/3 V_{CC}, the output of comparator 1 goes high, which sets the flip-flop. The transistor goes to saturation, thereby discharging the capacitor C and the output of the timer goes low. Thus the output returns back to stable state from quasi-stable state.

The output of the Monostable Multivibrator remains low until a trigger pulse is again applied. Then the cycle repeats. Trigger input, output voltage and capacitor voltage waveforms are shown in Figure



Derivation for Pulse Width

The voltage across capacitor increases exponentially and is given by

$$V_c = V\,(1 - e^{-t/CR})$$

If

$$V_c = 2/3\ V_{cc}$$

then

$$\frac{2}{3} V_{cc} = V_{cc}\,(1 - e^{-t/CR})$$

$$\frac{2}{3} - 1 = -e^{-t/CR}$$

$$\frac{1}{3} = e^{-t/CR}$$

$$\therefore \quad -\frac{t}{CR} = -1.0986$$

$$\therefore \quad t = +1.0986\ CR$$

$$\therefore \quad t \approx 1.1\ CR$$

where C in farads, R in ohms, t in seconds.

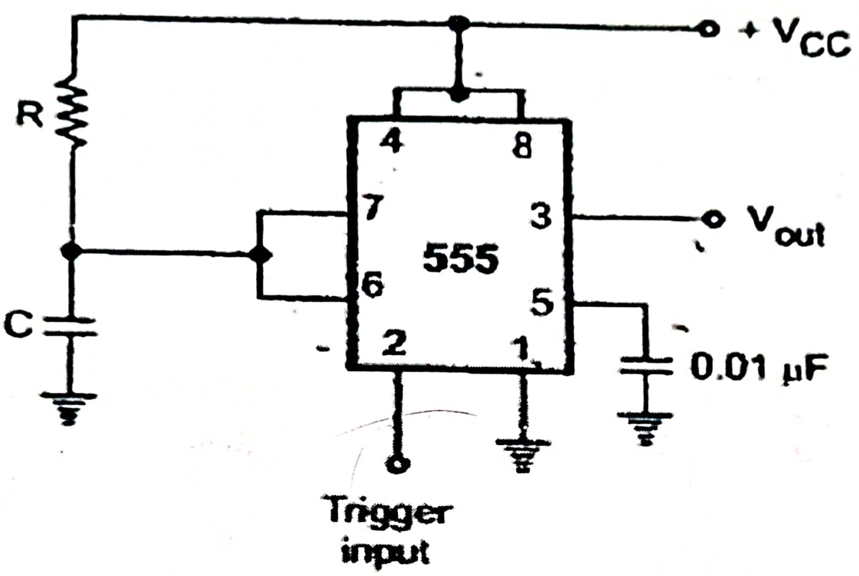
Key points : Thus, we can say that voltage across capacitor will reach 2/3 V_{CC} in approximately 1.1 times, time constant i.e 1.1 RC.

Thus the pulse width denoted as W is given by,

$$W = 1.1 R.C.$$

Schematic Diagram of IC 555 in Monostable mode

The schematic diagram shows only the external components to the timer.



Pin 1 is grounded. Trigger input is applied to pin 2. In quiescent condition of output this input is kept at + V_{CC}. To obtain transition of output from stable state to quasi-stable state, a negative-going pulse of narrow width and amplitude of greater than + 2/3 V_{CC} is applied to pin 2. Output is taken from pin 3. Pin 4 is usually connected to the supply to avoid accidental reset. Pin 5 is grounded through a 0.01 µF capacitor to avoid noise problem. Pin 6 (threshold) is shorted to pin 7. A resistor R is connected between pins 6 and 8. At pins 7 a discharge capacitor is connected while pin 8 is connected to supply V_{CC}.

Applications of 555 in monostable mode

The applications of IC 555 in monostable operation include

1. Linear Ramp Generator
2. Pulse Width Modulator
3. Frequency Divider
4. Missing Pulse Detector