

**SAIT JABALPUR**

**DEPARTMENT OF ELECTRONICS & COMMUNICATION ENGG.**

**EX-405**

**ELECTRONICS DEVICES & CIRCUITS - LAB**

**LIST OF EXPERIMENT**

- 1) TO MEASURE THE INPUT OFFSET VOLTAGE AND INPUT BIAS CURRENT OF OP-AMP USING IC-741.
- 2) TO MEASURE THE CMRR OF OP-AMP USING IC-741.
- 3) OP-AMP AS INVERTING AND NON-INVERTING AMPLIFIER USING IC-741.
- 4) OP-AMP AS ADDER AND SUBTRACTOR USING IC-741.
- 5) OP-AMP AS AN INTEGRATOR AND DIFFERENTIATOR USING IC-741.
- 6) DESIGN ASTABLE MULTI VIBRATOR USING IC-555.
- 7) DESIGN MONOSTABLE MULTI VIBRATOR USING IC-555.
- 8) DESIGN SCHMITT TRIGGER USING IC-555.
- 9) OP-AMP AS COMPARATOR USING IC-741.
- 10) TO MEASURE THE SLEW RATE OF OP-AMP USING IC-741.

S.NO.	NAME OF EXPERIMENT	PAGE NO.	DATE	GRADE/TOTAL	REMARK	SIGN
1.						
2.						
3.						
4.						
5.						
6.						
7.						
8.						
9.						
10.						

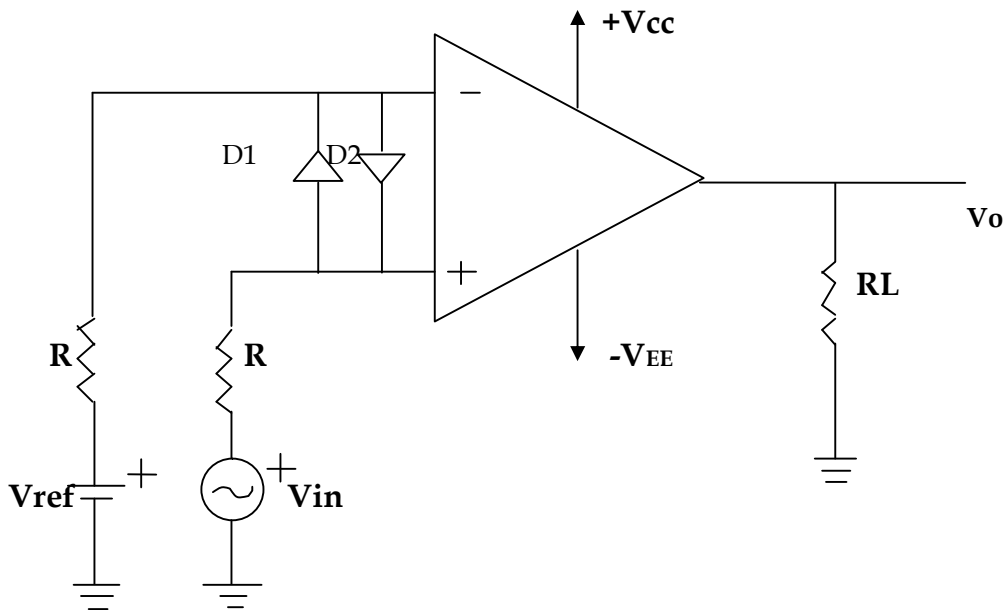
## EXPERIMENT NO:1

**OBJECTIVE:** - Study to design IC-741 as a Basic Comparator.

### **EQUIPMENTS REQUIRED:-**

1. OPT -01 Trainer kit.
2. Power supply.
3. Patch cord.
4. DSO and function generator.
5. Multimeter.

**THEORY:** - Comparator is basically a circuit which compares the signal applied to its inputs.



**Fig 1. Basic Comparator Using OPAMP**

Let the voltage applied at the non -inverting input be denoted by  $V_p$  & that applied at the inverting terminal be  $V_n$ .

Then the output voltage  $V_o$  is,

$$V_o = V_{oh} \text{ -----if } V_p > V_n$$

$$V_o = V_{ol} \text{ -----if } V_p < V_n$$

The voltage transfer characteristics of an ideal comparator is shown in fig.(b).It is basically a plot between differential voltage  $V_d$  V/S  $V_o$ . The above characteristics can be expressed as,

$$V_o = V_{oh} \text{ ----- for } V_d > 0$$

$$V_o = V_{ol} \text{ -----for } V_d < 0$$

OPAMP in open-loop mode is used as comparator.It compares a signal voltage applied to one of its input, with a known or reference voltage applied to the other input as shown below.

The output will be either  $+V_{sat}$  or  $-V_{sat}$  depending on  $V_i - V_{ref}$ . The OPAMP comparator can be either inverting or non-inverting type.

**NON- INVERTING COMPARATOR:** The schematic diagram of a non-inverting comparator is shown above .The reference voltage is applied to the inverting terminal & the signal to be compared is applied to non-inverting terminal. Because of the high input impedance of the OPAMP, the current through resistor  $R(-)$  &  $R(+)$  almost zero and hence the voltage across them will be nearly zero. Hence  $V_1 = V_{in}$  &  $V_2 = V_{ref}$ . The differential input voltage is,

$$V_d = V_1 - V_2 = V_{in} - V_{ref} \text{ -----(1)}$$

When  $V_{in} < V_{ref}$ , the inverting terminal is at higher potential than the non-inverting terminal. As the  $V_d$  is negative, OPAMP output will swing to  $-V_{sat}$ . Similarly for  $V_{in} > V_{ref}$  .  $V_d$  is positive and output will swing to  $+V_{sat}$ .

For OP-AMPS working with single power supply, the comparator output will be unipolar.

### **PROCEDURE:**

- 1) Make the connections on the OPT-01 board as shown in practical circuit.
- 2) Apply +ve 15v to pin 7 and -ve 15V to pin 4 of op-amp.

- 3) Apply 1 KHz, 4Vpp Sine wave as an input to Non-inverting input through R.
- 4) Apply + ve 1.4 v using VR1 as Vref to inverting input through R.
- 5) Switch on power supply.
- 6) Observe the waveform on CRO. Measure the output switching voltage.
- 7) Compare applied and observed Vref.
- 8) Repeat the all procedure with -ve 2.0 V using VR2 as Vref to inverting input through R.

**DESIGN:-**

Consider  $R_1=1K\Omega$  resistor,  $D_1$  &  $D_2 = 1N41007$ ,  $R_L = 10K\Omega$

**OBSERVATION TABLE:-**

Sr.No	Applied Vref input	Observed output
1		
2		

**WAVEFORMS:-**

**CONCLUSION:** We studied the design of Basic comparator using op-amp and observed the waveform of comparator.

**VIVA QUESTIONS**

**Q1. What is the Comparator amplifier?**

-----  
 -----

**Q2. Explain the operations?**

-----  
-----

**Q3. Derive the O/P expression of the Comparator?**

-----  
-----

**Q4. What are the properties of Comparator?**

-----  
-----

**Q5. What are the applications of Comparator?**

-----  
-----  
-----

## EXPERIMENT NO: 2

**OBJECTIVE:** - To measure the Input offset voltage ( $V_{io}$ ) & Input Bias current of op-amp.

**EQUIPMENTS REQUIRED:** -

1. OPT Trainer Kit
2. Power supply.
3. Patch cords.
4. Multimeter

**THEORY:-** 1) **Input offset voltage ( $V_{io}$ ):-**

An ideal op-amp will give o/p of 0 v if both its input are shorted together. A real world op-amp will have non-zero voltage, even if its inputs are shorted together. This is the result of I/P offset voltage which is slight voltage present at its inputs brought about its non-zero input offset voltage which is slight voltage present at its inputs brought about its non-zero input offset current. In essence, the input offset voltage is also the input voltage that's needs to be applied across the inputs of an OP-Amp to make the output voltage zero.

**2) Input offset current ( $I_{io}$ ):-**

It is defined as the difference b/w the separator currents entering the input terminal of a balanced amplifier.

$$I_{io} = (I_{B1}) - (I_{B2})$$

**3) Input bias current ( $I_B$ ):-**

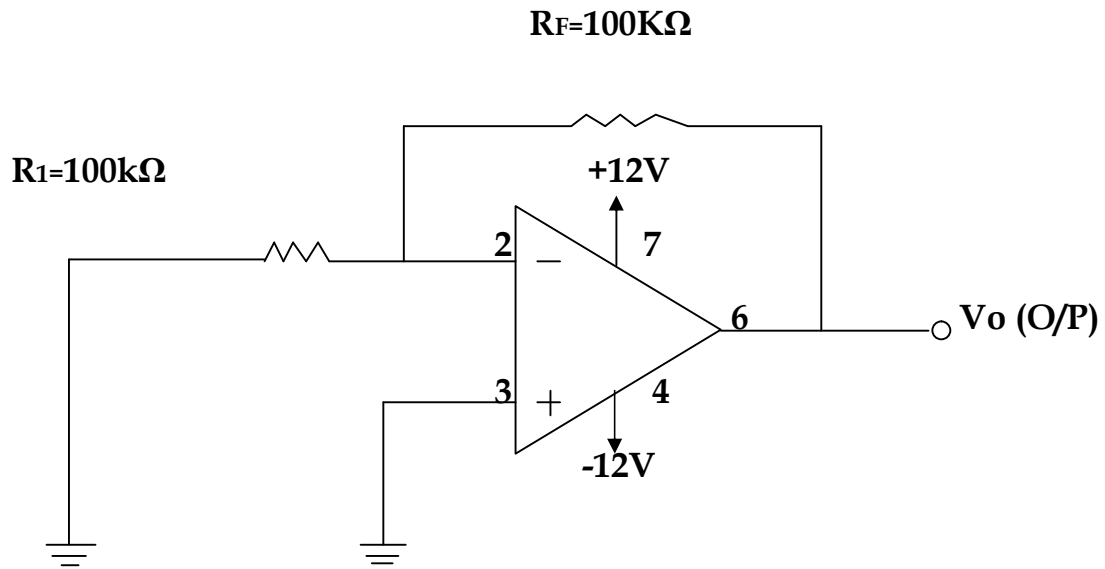
It is defined as the average of the currents that flow into the inverting and non-inverting input terminals of the OP-AMP.

$$I_B = \frac{(I_{B1}) + (I_{B2})}{2}$$

2

**PROCEDURE:-**

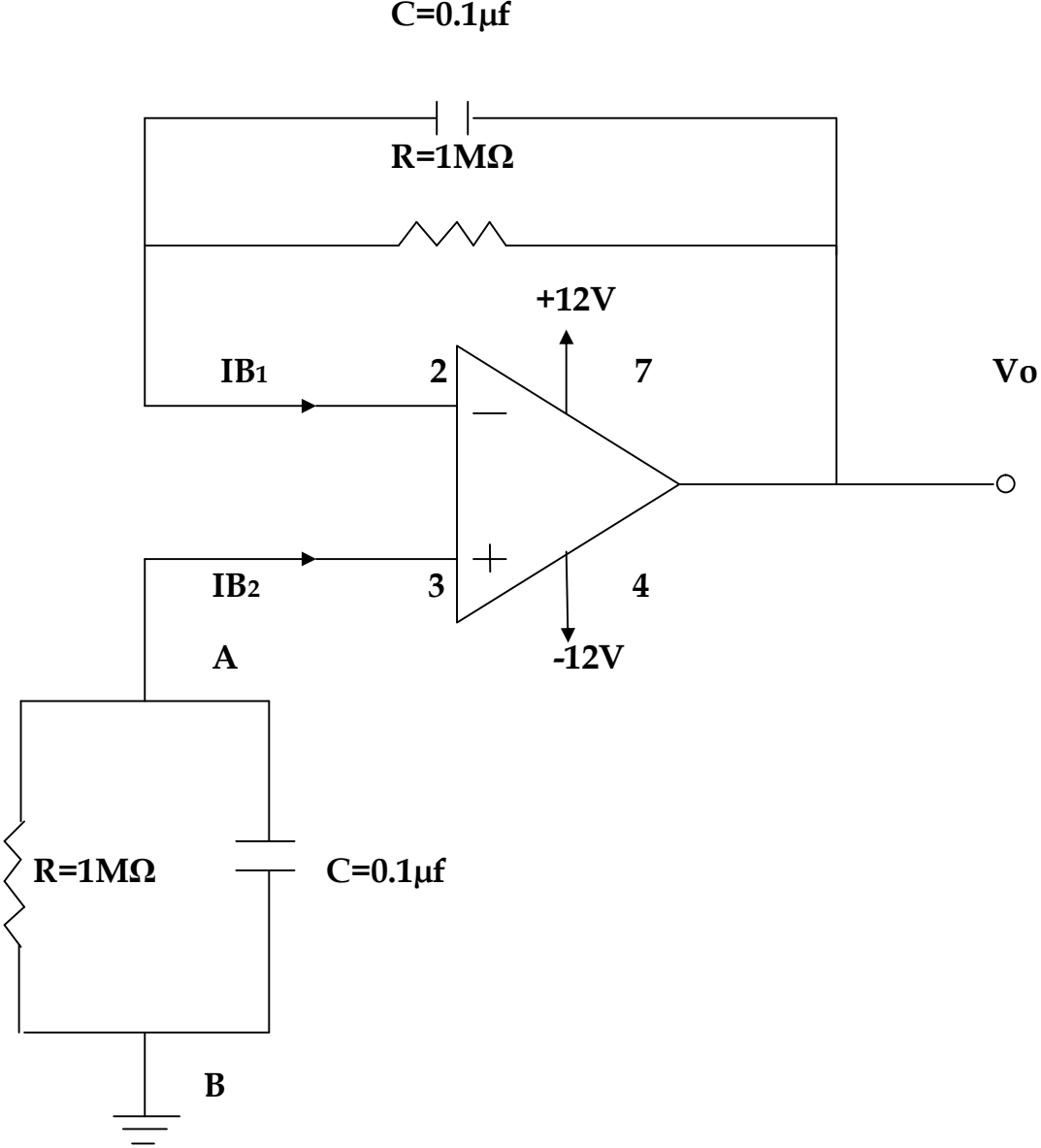
**MEASUREMENTS OF I/P OFFSET VOLTAGE ( $V_{io}$ )**



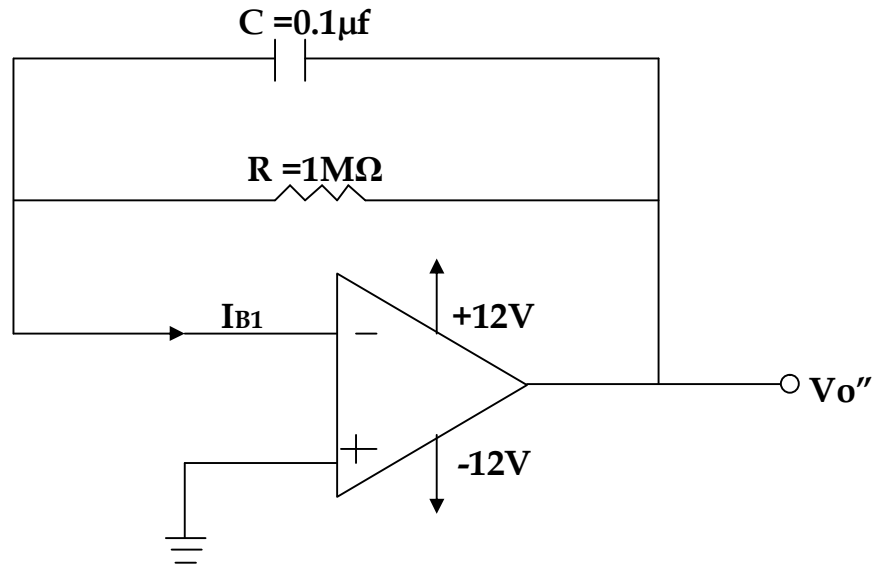
- 1) Make the connections as shown in the diagrams.
- 2) Apply +VCC and -VEE as + 12V.
- 3) Measure the output voltage on DMM.
- 4) Find the input offset voltage using formula.

$$V_{io} = v_o \cdot R_1 / (R_1 + R_f)$$

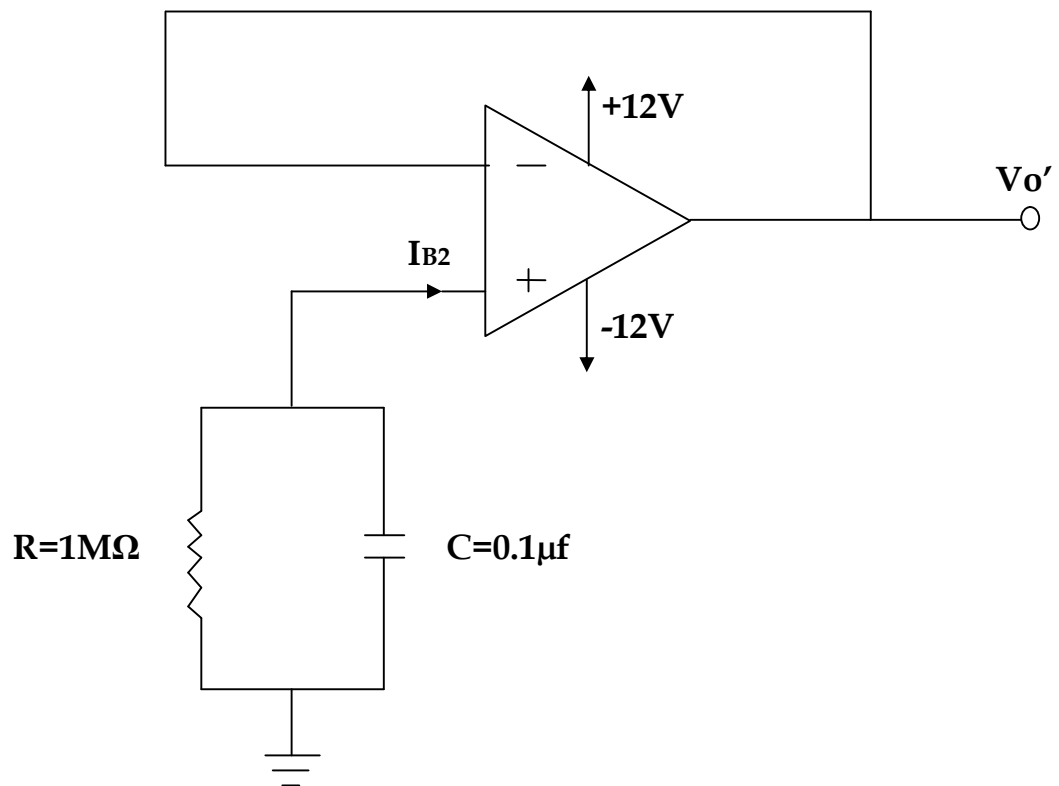
**MEASUREMENT OF INPUT OFFSET CURRENT ( $I_{io}$ ) AND INPUT BIAS CURRENT ( $I_b$ )**



Measurement of  $I_{B1}$ :-



Measurement of  $I_{B2}$ :-



### Measurement of input offset current (I<sub>io</sub>) and input bias current (I<sub>B</sub>):-

- 1) Make the connections as shown in the diagrams.
- 2) Apply +VCC and -VEE as + 12V.
- 3) Measure the output voltage on DMM.
- 4) To measure I<sub>B1</sub>, short A & B, measure V<sub>o</sub> as V<sub>o''</sub>. Calculate I<sub>B1</sub> using formula,  
$$I_{B1} = V_{o''} / R$$
- 5) To measure I<sub>B2</sub>, short C & D, measure V<sub>o</sub> as V<sub>o'</sub>. Calculate I<sub>B2</sub> using formula,  
$$I_{B2} = V_{o'} / R$$
- 6) Calculate input offset current I<sub>io</sub> = (I<sub>B1</sub>) - (I<sub>B2</sub>).
- 7) Calculate input bias current, using I<sub>B</sub> = (I<sub>B1</sub>) + (I<sub>B2</sub>)/2.

### DESIGNS:-

- 1) Input offset voltage (V<sub>io</sub>):-  $V_{io} = V_o * R_1 / (R_1 + R_f)$
- 2) Input bias current =  $I_B = \frac{(I_{B1}) + (I_{B2})}{2}$

### OBSERVATION TABLE:-

#### IC-741:-

1. I/p offset voltage (V<sub>io</sub>) = Output voltage reading V<sub>o</sub> = -1.225 V.
2. V<sub>o''</sub> = 29 mV. Therefore I<sub>B1</sub> = 29 nA.
3. V<sub>o'</sub> = -33 mV. Therefore I<sub>B2</sub> = 33 nA.

Serial no.	OpAmp parameter	Ideal Value	Typical Value	Practical Value
1.	i/p offset voltage	0	2 mV	1.25 mV
2.	i/p bias current(I <sub>B</sub> )	500nA	80 nA	31 nA

## CONCLUSION:-

In this experiment, we studied different parameters like  $V_{io}$ ,  $I_{io}$  &  $I_B$  for the IC 741. We observed that the IC 741 as its parameter values is verified with the ideal values.

## VIVA QUESTIONS

**Q.1. What is Input offset voltage?**

---

---

---

**Q2. What is Input Bias current?**

---

---

---

**Q3. What is the I/P and O/P impedance of an ideal OP-AMP?**

---

---

---

## EXPERIMENT NO: 3

**OBJECTIVE: -** To measure the OP-Amp parameter for IC-741. Common mode rejection ratio (CMRR)

### **EQUIPMENTS REQUIRED:-**

1. OPT- 01 Trainer kit.
2. Power supply.
3. Patch chords.
4. DSO
5. Multi-meter.

### **THEORY:-**

#### **COMMON MODE REJECTION RATIO (CMRR):-**

It is defined as the ratio of differential voltage gain to common mode voltage gain.

$$CMRR = A_d / A_c$$

$$A_d = V_o / V_d$$

And common mode voltage gain can be determined by

$$A_{CM} = \frac{V_{ocM}}{V_{icM}}$$

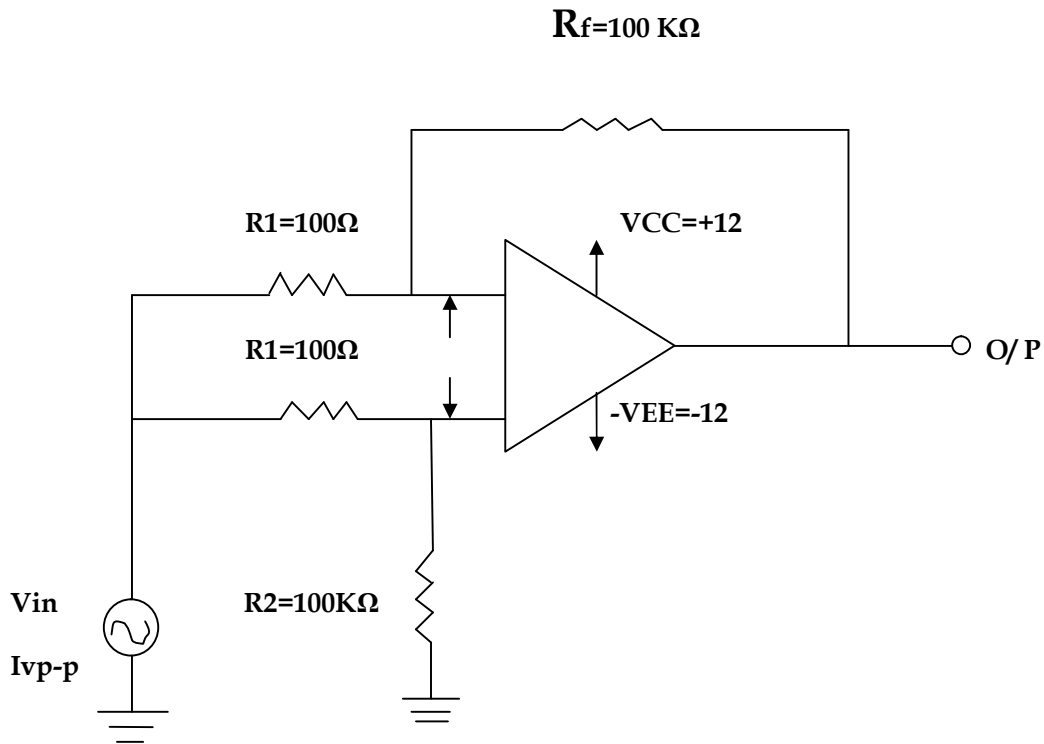
$$V_{icM}$$

$V_{icM}$  is the common mode i/p voltage.

$A_{CM}$  is common mode voltage gain. The CMRR is very large hence CMRR is most often expressed in decibels.

### **PROCEDURE:-**

#### **MEASUREMENT OF CMRR:-**



**COMMON MODE REJECTION RATIO**

Make the connection as shown in fig.  
 Apply 1 Vpp sine wave at input.  
 Measure o/p and calculate CMRR as.

$$CMRR = 1 + (R2/R1) * (Vs/Vo)$$

**DESIGNS:-**

Common mode rejection ratio (CMRR).

$$CMRR = 20 \log \{ (1 + R2/R1) Vs/Vo \}$$

**OBSERVATION TABLE:-**

**IC-741:-**

Sr. No.	PARAMETER	IDEAL VALUE	PRACTICAL VALUE
1	CMRR	90 dB	

## CONCLUSION:-

In this experiment we studied parameter like CMRR for the IC-741. WE observed that the IC-741 as its parameters values are verified with the ideal values.

## VIVA QUESTIONS

Q1. What are the characteristics of OP-AMP?

-----  
-----  
-----

Q2. What is CMRR?

-----  
-----  
-----

## EXPERIMENT NO: 4

### OBJECTIVE: -

To measure the OP-Amp parameter for IC-741. Slew Rate (SR)

### EQUIPMENTS REQUIRED:-

1. OPT- 01 Trainer kit.
2. Power supply.
3. Patch chords.
3. DSO
4. Multi-meter.

### THEORY:-

#### SLEW RATE:-

This is the minimum rate of change of o/p Voltage/unit time.

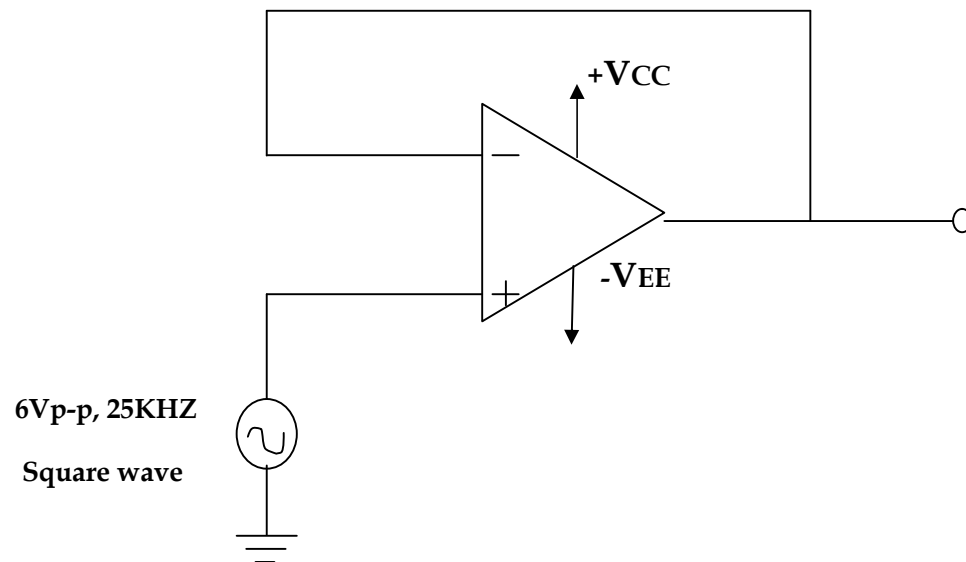
$$SR = \frac{VO}{t}$$

It is expressed in V/ $\mu$  second.

The SR is measured in the unity gain voltage follower ckt as shown. The SR is measured in the amplifier driven by high frequency wave of sufficient magnitude. The SR slope of the transition between o/p levels frequently +ve and -ve swings will have different SR and both must be examined. In such case lower  $S_R$  is commonly specified.

#### MEASUREMENT OF SLEW RATE:-

$$SR = \frac{\Delta V_o}{\Delta t} \text{ -VEE}$$



### SLEW RATE:-

1. Make the connections as shown in fig.
2. Apply 6 VPP, 25 khz square wave input at non-inverting terminal.
3. Measure?  $V_o$  w.r.t.?t and calculate SR as

$$SR = \frac{\Delta V_o}{\Delta t}$$

### DESIGNS:-

Slew rate(SR)

$$SR = \frac{\Delta V_o}{\Delta t}$$

### OBSERVATION TABLE:-

**IC-741:-**

Sr. No.	PARAMETER	IDEAL VALUE	PRACTICAL VALUE
1	SR	0.5 V/ $\mu$ sec	.73V/ $\mu$ sec

**CONCLUSION:-**

In this experiment we studied different parameter like SLEW RATE for the IC-741. WE observed that the IC-741 as its parameters values is verified with the ideal values.

**VIVA QUESTIONS**

**Q1.What is Slew Rate?**

-----  
-----  
-----

## EXPERIMENT NO: 5

### OBJECTIVE: -

- 1) To study IC 741 as Inverting Amplifier.
- 2) To observe & note effect of change in  $R_F$  &  $R_1$  value on O/P  $V_0$ .
- 3) To study IC 741 as Non-Inverting Amplifier.
- 4) To observe & note effect of change in  $R_F$  &  $R_1$  value on O/P  $V_0$ .

### EQUIPMENTS REQUIRED:-

Signal generator, CRO - dual channel, patch chord, DMM.

### COMPONENT VALUE:-

$R_{11} = 1K$ ,  $R_{12} = 10K$   $R_{13} = 4k7$  pot ,  $R_{F1} = 10K$  , $R_{F2} = 100K$ , $R_{F3} = 33K$

$R_1 = R_{11}, R_{12}, R_{13}$   $R_F = R_{F1}, R_{F2}, R_{F3}$   $R_L = 10K$

### THEORY:-

An op-amp can be used for number of application like amplifier, Adder's Subtractors, Rectifier, Multivibrators, Analog computer etc. IC 741 is used as a Inverting & Non-inverting amplifier.

### **1. IC 741 AS INVERTING AMPLIFIER.**

When the input is applied to the inverting input terminal (negative terminal) of op-Amp, then Op-Amp is said to be operated in inverting amplifier mode. The amplified as well as the inverted output signal is obtained from output pin 6 of Op-Amp. This output signal is applied to inverting input via feedback resistor  $R_F$ . It forms a negative feedback because any increase in the output signal results in a feedback signal into inverting causing a decrease in the output signal.

Note that non-inverting terminal is grounded. Feedback circuit uses anyone resistor  $R_F$ , However an extra resistor  $R_1$  is connected in series with the input signal source  $V_{in}$ . As shown in fig. 1. It is also known as voltage shunt feedback amplifier.

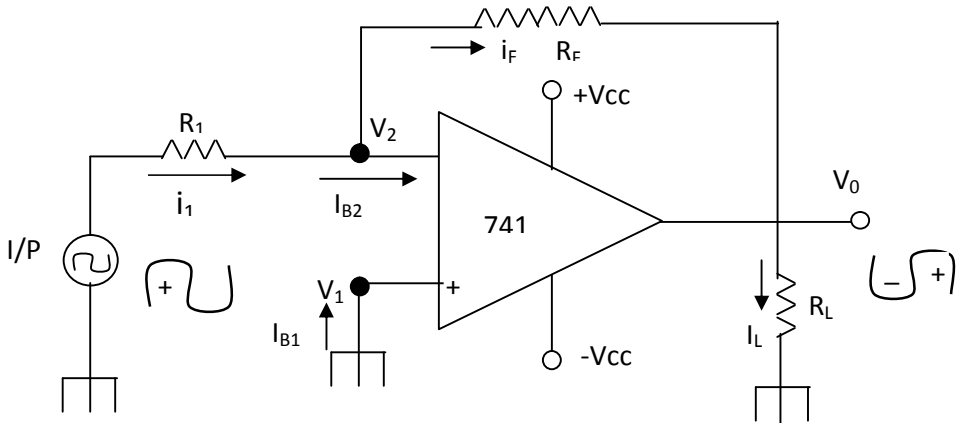


Fig. 1 IC 741 As Inverting Amplifier.

**Closed Loop Voltage Gain ( $A_F$ ) :**

The closed loop voltage gain -  $A_F$  of the amplifier can be obtained by applying KCL at node  $V_2$ .

$$i_1 = I_{B2} + i_F \text{ -----(1)}$$

Since input impedance  $R_i$  of Op-Amp is very large, so  $I_{B2} \approx 0$ . Hence neglecting  $I_{B2}$

we get ,  $i_1 = i_F \text{ -----(2)}$

$$i_1 = \frac{V_{in} - V_2}{R_1}$$

From fig. 1

From equation (2) we get  $i_F = \frac{V_2 - V_0}{R_F}$

$$\frac{V_{in} - V_2}{R_1} = \frac{V_2 - V_0}{R_F} \text{ -----(3)}$$

We know,  $V_0 = A (V_{id})$

$$V_0 = A (V_1 - V_2) \quad \therefore \text{As } V_1 = 0$$

$$\therefore V_0 = -A V_2$$

$$\therefore V_2 = \frac{-V_0}{A}$$

Substituting the value of  $V_2$  in equation (3)

$$\text{We get, } \frac{V_{in} + V_0/A}{R_1} = \frac{-V_0/A - V_0}{R_F} \quad \text{-----(4)}$$

$$A_F = \frac{V_0}{V_{in}} = \frac{AR_F}{R_1 + R_F + AR_1} \quad \text{-----(5)}$$

As gain  $A$  is very large (ideally infinity).  $AR_1 \gg R_1 + R_F$ . So neglecting  $(R_1 + R_F)$

We get,

$$A_F = - \frac{AR_F}{AR_1}$$

$$\underline{A_F} = \frac{V_0}{V_{in}} = \frac{-R_F}{R_1}$$

$$\therefore V_0 = - \frac{R_F}{R_1} V_{in} \quad \text{----- (6)}$$

The negative sign in the above equation indicates the phase difference of  $180^\circ$  between input and output. By changing the value of  $R_1$  &  $R_F$  the gain can be changes.

### PROCEDURE:-

#### For AC I/P:

1. Study the circuit provided on front panel of the kit.
2. Make the circuit as shown in above figure.
3. Switch on the power supply.
4. Select desired  $R_1$  &  $R_F$ .

5. Connect dual trace CRO at input and output side to observe  $V_{in}$  &  $V_0$  respectively.
6. Apply 100Hz, sine wave input  $V_{in}$  from signal generator. Adjust it's amplitude so that op-amp should not enter in saturation.
7. Observe & note input  $V_{in}$  amplitude & output  $V_0$  amplitude on CRO calculate it's gain using  $A_F = V_0/V_{in}$
8. Calculate theoretical gain of Op-amp using selected  $R_1$  &  $R_F$  values.
9. Compare the above results.
10. Vary the input frequency observe change in output.
11. Draw the waveforms on graph.
12. Repeat the above procedure for different combination of  $R_1$  &  $R_F$ .

**OBSERVATION TABLE:-**

$R_{11} =$  \_\_\_\_\_       $R_{12} =$  \_\_\_\_\_       $R_{13} =$  \_\_\_\_\_

$R_{F1} =$  \_\_\_\_\_       $R_{F2} =$  \_\_\_\_\_       $R_{F3} =$  \_\_\_\_\_

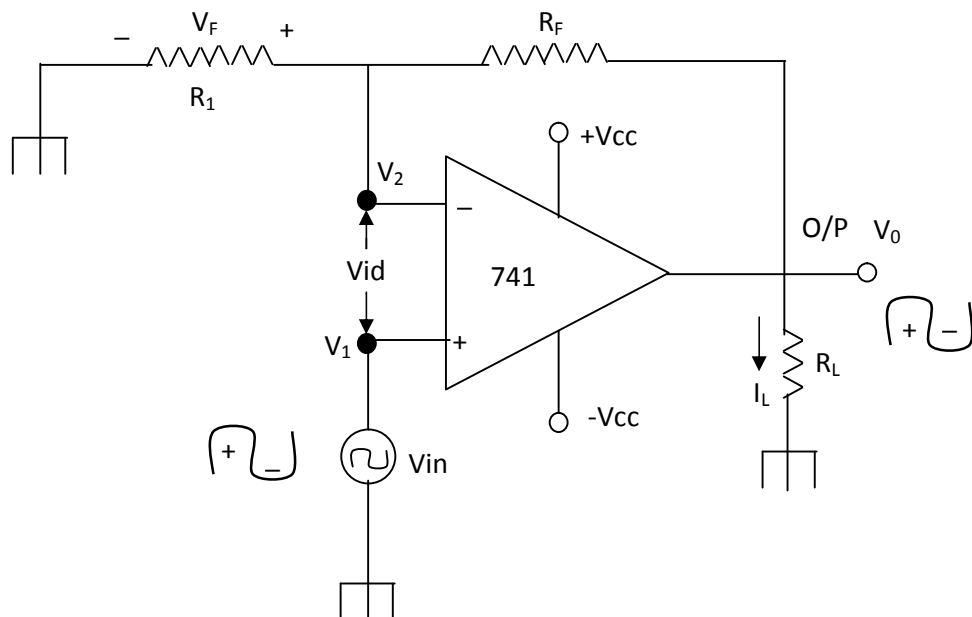
Sr. No.	$R_1$	$R_F$	$V_{in}$	$V_0$	Theoretical gain $A_F = - R_F / R_1$	Practical gain $A_F = V_0 / V_{in}$
1	$R_{11}$	$R_{F1}$				
		$R_{F2}$				
		$R_{F3}$				
2	$R_{12}$	$R_{F1}$				
		$R_{F2}$				
		$R_{F3}$				
3	$R_{13}$	$R_{F1}$				
		$R_{F2}$				
		$R_{F3}$				

## RESULT:

The input signal is amplified & inverted at the output of inverting amplifier.

## 2. IC 741 AS NON-INVERTING AMPLIFIER.

When the input is applied to the non-inverting input terminal (positive terminal) of op-Amp, then Op-Amp is said to be operated in non-inverting amplifier mode. The amplified in phase output signal is obtained from output pin 6 of Op-Amp. This output signal is applied to inverting input via feedback resistor  $R_F$ . It forms a negative feedback because any increase in the output signal results in a feedback signal into inverting causing a decrease in the output signal. Note that inverting terminal is connected with O/P  $V_0$  through Feedback resistor  $R_F$ , with  $R_1$  is grounded from inverting terminal. However an extra resistor  $R_1$  is connected in series with the input signal source  $V_{in}$ . As shown in fig. 1. It is also known as voltage series feedback amplifier.



**Fig. 1: IC 741 As Non-Inverting Amplifier.**

## Closed Loop Voltage Gain (AF)

$$AF = \frac{V_0}{V_{in}} \text{-----(1)}$$

We know, closed loop gain as defined as

$$\text{Also, } V_0 = A (V_1 - V_2) \text{----- (2)}$$

Referring to Fig. 1: We get,  $V_1 = V_{in}$

$$V_2 = V_F = \frac{R_1 V_0}{R_1 + R_F} \quad \text{Since } R_i \gg R_1$$

$$\therefore V_0 = A \left( V_{in} - \frac{R_1 V_0}{R_1 + R_F} \right)$$

$$\text{Finally we get, } V_0 = \frac{A(R_1 + R_F) V_{in}}{R_1 + R_F + AR_1}$$

$$\text{Thus, } A_F = \frac{V_0}{V_{in}} = \frac{A(R_1 + R_F)}{R_1 + R_F + AR_1} \text{-----(3)}$$

Generally A is very large,  $AR_1 \gg (R_1 + R_F)$  &  $R_1 + R_F + AR_1 \cong AR_1$

$$A_F = \frac{V_0}{V_{in}} = 1 + \frac{R_F}{R_1} \quad \text{-----(4)}$$

$$V_0 = \left( 1 + \frac{R_F}{R_1} \right) V_{in} \quad \text{-----(5)}$$

By changing the value of  $R_1$  &  $R_F$ , the closed loop gain can be changed. The phase difference between I/P & O/P is zero, so it is called as Non-inverting amplifier.

### PROCEDURE:-

#### For AC I/P:

- 1) Study the circuit provided on front panel of the kit.
- 2) Make the circuit as shown in above figure.
- 3) Switch on the power supply.
- 4) Select desired  $R_1$  &  $R_F$ .
- 5) Connect dual trace CRO at input and output side to observe  $V_{in}$  &  $V_0$  respectively.
- 6) Apply 100Hz, sine wave input  $V_{in}$  from signal generator. Adjust it's amplitude so that op-amp should not enter in saturation.
- 7) Observe & note input  $V_{in}$  amplitude & output  $V_0$  amplitude on CRO calculate it's gain using  $A_F = V_0/V_{in}$
- 8) Calculate theoretical gain of Op-amp using selected  $R_1$  &  $R_F$  values.
- 9) Compare the above results.
- 10) Vary the input frequency observe change in output.
- 11) Draw the waveforms on graph.
- 12) Repeat the above procedure for different combination of  $R_1$  &  $R_F$ .

### OBSERVATION TABLE:-

$R_{11} = \underline{\hspace{2cm}}$        $R_{12} = \underline{\hspace{2cm}}$        $R_{13} = \underline{\hspace{2cm}}$

$R_{F1} = \underline{\hspace{2cm}}$        $R_{F2} = \underline{\hspace{2cm}}$        $R_{F3} = \underline{\hspace{2cm}}$

Sr. No.	$R_1$	$R_F$	$V_{in}$	$V_0$	Theoretical gain $A_F = 1 + R_F/R_1$	Practical gain $A_F = V_0/V_{in}$
1	$R_{11}$	$R_{F1}$				
		$R_{F2}$				
		$R_{F3}$				
2	$R_{12}$	$R_{F1}$				
		$R_{F2}$				
		$R_{F3}$				
3	$R_{13}$	$R_{F1}$				
		$R_{F2}$				
		$R_{F3}$				

**RESULT:-**

The IC 741 as a Non-inverting amplifier provided amplified and in phase O/P when input is applied to non-inverting I/P. By changing  $R_1$  &  $R_F$  gain can be changed.

**VIVA QUESTIONS**

**Q1. What is OP-AMP?**

-----  
 -----  
 -----

Q2.What is inverting & non-inverting terminal of OP-AMP and why it is called so?

---

---

---

Q3.What is the name of the IC used in OP-AMP?

---

---

Q4.IC 741 is ----- pin IC.

Q5.What is IC?

---

---

Q6. What are the type of IC`s?

---

---

Q7. What is the gain of inverting amplifier?

---

---

Q8.What is the gain on non-inverting amplifier?

---

---

## EXPERIMENT NO: 6

### OBJECTIVE: -

- (1) To study IC 741 as an Adder circuit.
- (2) To study IC 741 as a Subtractor circuit.

### EQUIPMENT:-

Digital Multimeter, Patch Cords.

### COMPONENT USED:-

$R_1 = 10K, R = 10K, R_L = 10K, R_F = 10K, 20K, IC 741, \pm 12V.$

### THEORY:

An op-amp can be used for number of application like amplifier, Adder's Subtractors, Rectifier, Multivibrators, Analog computer etc.

### 1. IC 741 AS ADDER & AVERAGER CIRCUIT.

If the different I/P voltage sources ( $V_a, V_b, V_c$ ) & resistors are connected to the non-inverting terminal as shown in fig. 1, then it forms Non-inverting adder, averager circuit, depending on the selection of resistor  $R_1$  and  $R_F$

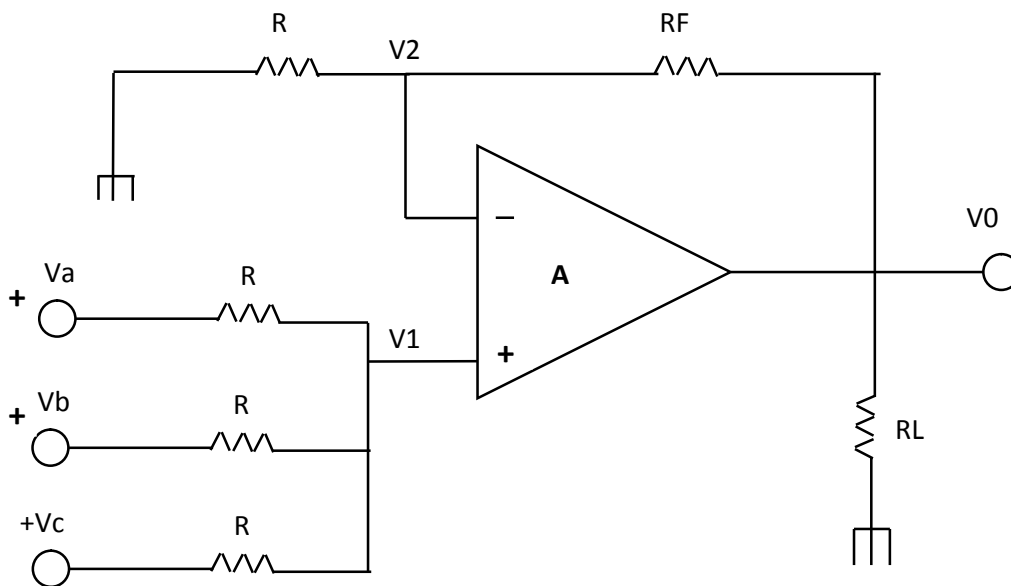


Fig. 1:- Non-inverting adder averager circuit.

As the input Resistance  $R_{if}$  of the Non-inverting amplifier is very large. Therefore, using the superposition theorem, the voltage  $V_1$  at the Non-inverting terminal is

$$V_1 = \frac{R/2}{R + R/2} V_a + \frac{R/2}{R + R/2} V_b + \frac{R/2}{R + R/2}$$

OR

$$V_1 = \frac{V_a}{3} + \frac{V_b}{3} + \frac{V_c}{3} = \frac{V_a + V_b + V_c}{3}$$

The gain of the Non-inverting amplifier is given by

$$V_0 = \left( 1 + \frac{R_F}{R_1} \right) V_1$$

$$V_0 = \left( 1 + \frac{R_F}{R_1} \right) \left( 1 + \frac{V_a + V_b + V_c}{3} \right) \text{-----(1)}$$

The equation (1) shows the scaling O/P.

If in equation (1)

$$\left( 1 + \frac{R_F}{R_1} \right) = 3$$

i.e.  $R_F = 2R_1$

then

$$V_o = V_a + V_b + V_c$$

Hence the circuit will be adder/summing amplifier circuit. Where as if in equation

(1).  $1 + \frac{R_F}{R_1} = 1$  i.e.  $R_F = 0$  OR  $R_1 = \infty$

then from equation (1),

$$V_o = \frac{V_a + V_b + V_c}{3}$$

the circuit will be averager amplifier.

## 2. IC 741 AS SUBTRACTOR CIRCUIT

A basic differential amplifier can be used as a subtractor as shown in fig.1. In this figure, input signal can be scaled to the desired values by selecting appropriate values from the external resistors.

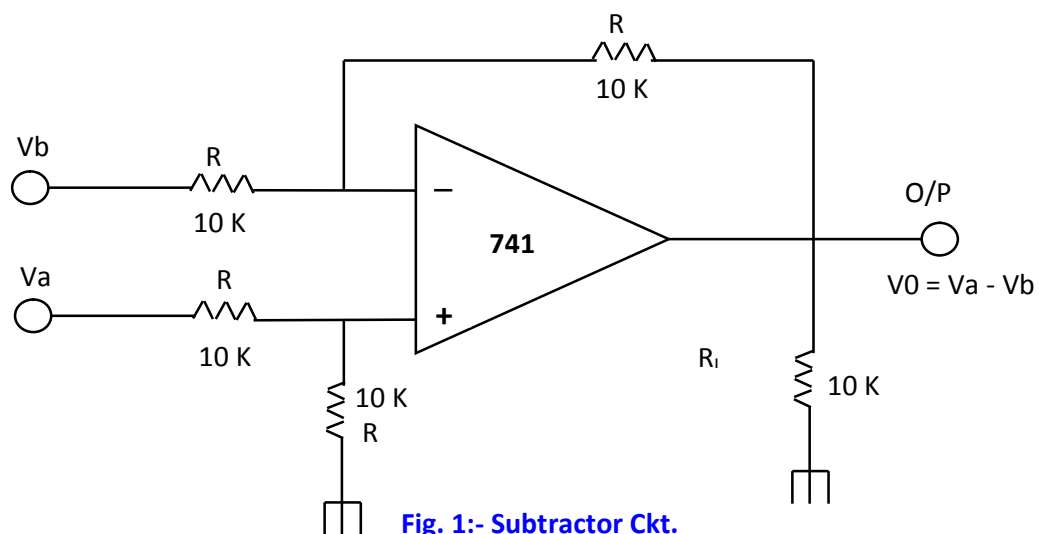


Fig. 1:- Subtractor Ckt.

When this is done, the circuit is referred as scaling amplifier. However if all the external resistors are equal in value, then the gain of the amplifier is equal to 1.

From fig.1, the O/P voltage of the differential amplifier with a gain of 1 is

$$V_0 = - \frac{R}{R} (V_a - V_b)$$

$$\therefore V_0 = - (V_a - V_b)$$

$$V_0 = V_b - V_a$$

As the O/P voltage is difference of the I/P voltages applied. Hence, the circuit is called as subtractor.

### **PROCEDURE:-**

- (1) Study the circuit provided on the front panel of the kit.
- (2) Switch ON the power supply and Note the Voltages formed by Network. Select Adder operation by Patch Cord shown in circuit.
- (3) Connect I/Ps of OP-AMP  $V_a$ ,  $V_{a1}$  &  $V_{a2}$  to desired voltages  $V_1$ ,  $V_2$ ,  $V_3$  or  $V_4$ , note its value, measured O/P voltage  $V_0$ .  
Compare with theoretical  $V_0 = V_a + V_b + V_c$
- (4) Repeat step (3) for different combinations of I/Ps.
- (5) Now select averager operation by using Patch Cord. Then repeat step (3). Compare the result with theoretical  $V_0 = \frac{V_a + V_b + V_c}{3}$
- (6) For Subtraction then Connect OP-AMP I/Ps  $V_a$  &  $V_b$  to the desired voltages of potential divider network ( $V_1$  to  $V_4$ ) & note it's values. Measure the O/P voltage  $V_0$  & compare it with theoretical  $V_0 = - (V_a - V_b)$
- (7) Repeat step (4) for different combinations of  $V_a$  and  $V_b$ .

**OBSERVATIONS:**

**For Adder Operations:-**

I/Ps			Theoretical $V_0 = V_a + V_b + V_c$	Observed $V_0$
Va	Vb	Vc		
V <sub>1</sub>	V <sub>2</sub>	V <sub>3</sub>		
V <sub>2</sub>	V <sub>3</sub>	V <sub>4</sub>		

**For Subtractor Operations:-**

V<sub>1</sub> = -----V., V<sub>2</sub> = -----V., V<sub>3</sub> = -----V, V<sub>4</sub> = -----V.

I/Ps		Theoretical $V_0 = V_b - V_a$	<u>Observed</u> <u>V<sub>0</sub></u>
Va (volts)	Vb (volts)		
Va = -----	Vb = -----		
Va = -----	Vb = -----		

**RESULT:-**

By selecting the proper values of resistors circuit can be operates as an adder, or subtractor circuit.

**VIVA QUESTIONS**

**Q1. What is the O/P voltage of non-inverting adder circuit?**

---

---

**Q2. What is the O/P voltage of inverting adder circuit?**

---

---

**Q3. What is the O/P voltage of non-inverting subtractor circuit?**

---

---

**Q4. What is the O/P voltage of inverting amplifier subtractor circuit?**

---

---

**Q5. Why IC741 is called 741 IC?**

---

---

## EXPERIMENT NO:

### OBJECTIVES:-

IC 555 as an Astable Multivibrator.

INSTRUMENTS REQUIRED: - CRO, Patch Chords 6 No.

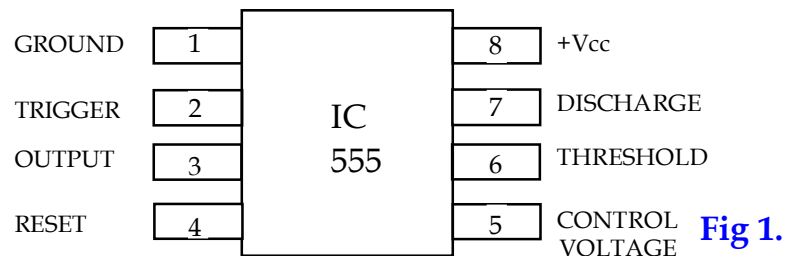
### THEORY:-

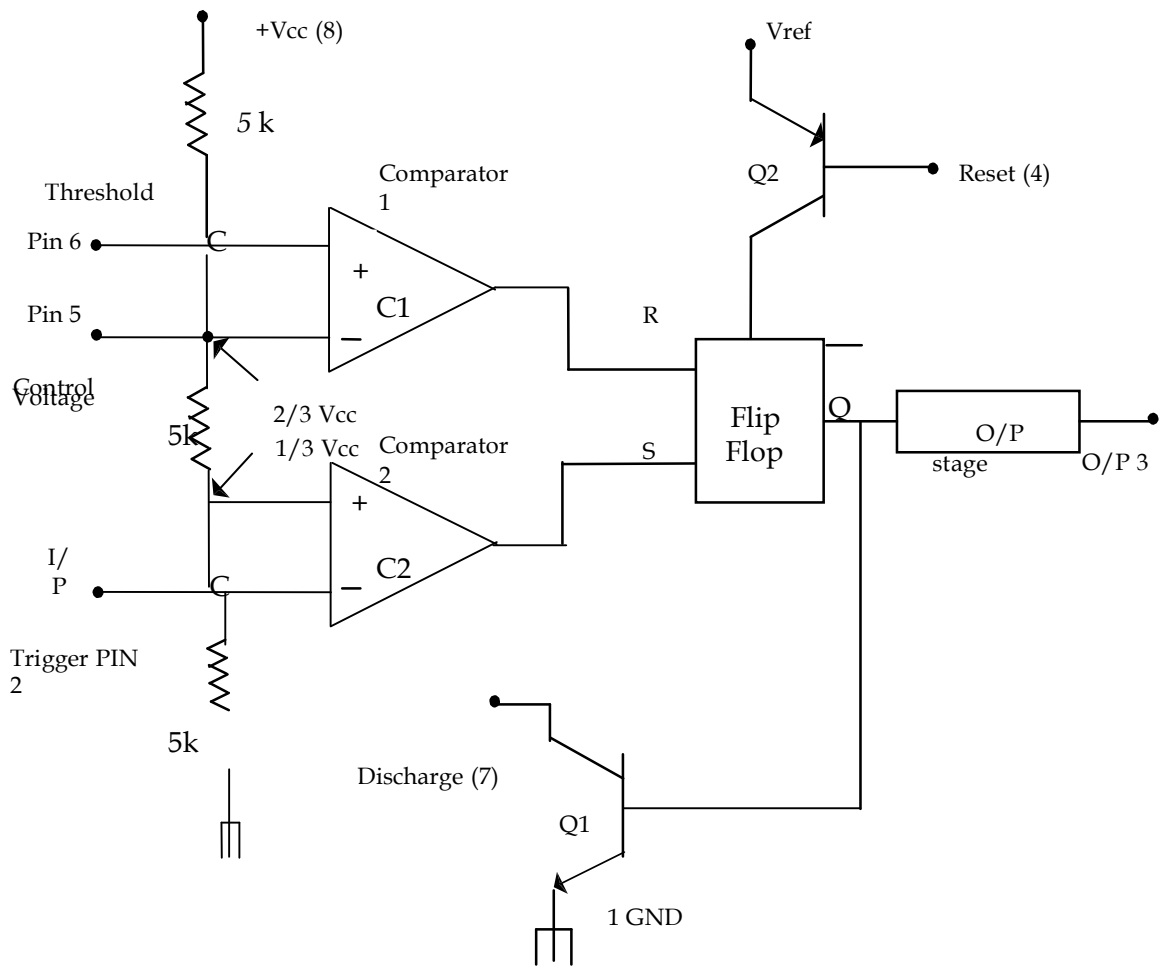
The timer IC 555 is the most versatile linear IC. It was introduced by signatic corporation as SE /NE 555. IC 555 used for following applications. Monostable, Astable multivibrator , Dc - Dc converter, Digital logic probes, waveform generator etc. The timer 555 is available as an 8 pin mini DIP, or a 14 pin mini DIP. The SE 555 is designed for the operating temperature range from 55 to + 125 , while NE 555 operates over a temperature range of 0 to 70 c.

### Features of IC 555:-

- 1) It can operate on + 5v to + 18v supply voltage.
- 2) It has adjustable duty cycle.
- 3) Timing can be adjusted from sec to hours.
- 4) High current output.
- 5) Output can drive TTL.
- 6) Capacity to source or sink current of 22mA.
- 7) Reliable, easy to use, and low cost like general purpose OP-Amp.

### PIN DIAGRAM OF IC 555:-





**Fig. 2 Functional Diagram**

Fig (1) shows the pin diagram and fig(2) shows the functional diagram of IC 555 IC timer. In functional diagram three 5k internal resistors acts as voltage divider giving bias voltage of  $(\frac{2}{3} V_{cc})$  to comparator  $C_1$  and  $\frac{1}{3} V_{cc}$  to lower comparator  $C_2$ . Where  $V_{cc}$  is the Supply voltage.

1) **Pin 1 Ground** :- All voltage are measured with respect to this terminal.

2) **Pin 2 Trigger** :- The output of timer depends on the Amplitude of the external trigger pulse applied to this pin. The O/p is low if the voltage at this pin is greater than  $2/3 V_{cc}$ . However when negative going pulse of amplitude larger than  $1/3 V_{cc}$  is applied to this pin, the comparator 2 output goes low, which in turn switches output of the timer high. The output remains high as long as the trigger is held at a low voltage.

3) **Pin 3 Output** :- There are two ways a load can be connected at output terminal either between pin 3 and ground [pin 1] or between pin 3 and supply voltage  $+V_{cc}$ . When the output is low load current flows through the load connected between pin 3 and  $+V_{cc}$  into the output terminal and is called the sink current. However the current through the grounded load is zero when the output is low. For this reason, if the load is connected between  $+V_{cc}$  and pin 3 there it is called the normally off load. On the other hand, when the output is high the current through the load connected between pin3 and  $+V_{cc}$  (normally on load ) is zero. However the output terminal supplies current to the normally off load. This current is called the source current. The maximum value of sink or source current is 200 m A.

4) **Pin 4 :- Reset** :-By applying negative pulse to this Pin IC 555 can be reset. When this pin is not in used then it is connected to  $+V_{cc}$  to avoid any possibility of false triggering.

5) **Pin 5 :- Control Voltage** :- An external voltage applied to this terminal arrange the threshold as well as trigger voltage. Due to this voltage pulse width can be changed. When this pin is not in used then ground through capacitor of 0.01  $\mu$  f to avoid any noise problem.

6) **Pin 6 Threshold** :- This is the non-inverting input terminal of comparator 1 which monitors the voltage across the external capacitor connected to this pin and ground when the voltage is greater than or equal to threshold voltage  $2/3 V_{cc}$ , the output of comp 1 goes high, which in turn switches the output of timer to low.

7) **Pin 7 Discharge** :- This pin is internally connected to the collector of Q1. When output is high Q1 is off and acts as an open circuits to the external capacitor C

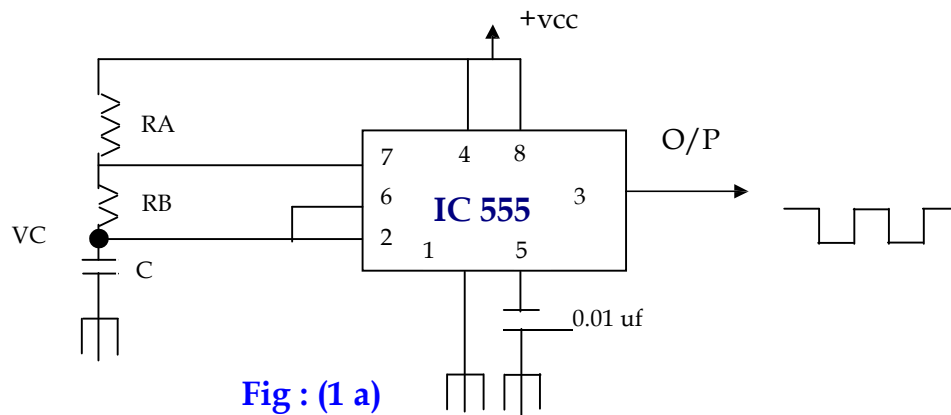
connected across it. Whereas if output is low Q1 is saturated and acts as a short circuit, Shorting out the external capacitor C to ground.

8) **Pin 8 +Vcc** :- The supply voltage + 5v to +18 v is applied to this pin w.r.t. ground.

### Applications of timer IC 555 :-

- 1) Astable Multivibrator.
- 2) Monostable Multivibrator .
- 3) Bistable Multivibrator.
- 4) Schmitt trigger.
- 5) Voltage controlled oscillator.

## IC 555 as an Astable Multivibrator



**Fig : (1 a)**

Fig (1a) shows the circuit diagram for Astable multivibrator using IC 555. Initially, if output is high, then capacitor C starts charging towards vcc through  $R_A$  &  $R_B$ . So voltage across capacitor  $V_c$  increases. As soon as voltage across capacitor  $V_c$  equal to  $2/3 V_{cc}$ , then comparator - 1 triggers the flip-flop and the output switches low (as shown in waveform) . So, capacitor 'C' starts discharging through  $R_B$  & internal transistor Q1. Hence  $V_c$  decreases. When voltage across capacitor  $V_c$  equals to  $1/3 V_{cc}$  then comparator-2 output trigger the flip-flop, and the output goes high. Then the complete cycle repeats as shown in waveform fig (1 b).

In waveform, the capacitor is periodically charged & discharged between  $2/3 V_{cc}$  and  $1/3 V_{cc}$  respectively. The time during which capacitor charges from  $1/3 V_{cc}$  to  $2/3 V_{cc}$  is equal to the time for which output is high. It is given by

$$T_c = 0.69 (R_A + R_B).C$$

Where  $R_A, R_B$  are in ohms &  $C$  is Farads

Similarly the time during which the capacitor discharges from  $2/3 V_{cc}$  to  $1/3 V_{cc}$  is equal to the time for which output is low and it is given by

$$T_d = 0.69 R_B.C$$

Thus the Total Time is given by ,

$$\begin{aligned} T &= T_c + T_d \\ &= 0.69 (R_A + 2R_B)C. \end{aligned}$$

Therefore the frequency of oscillation is given by

$$F = \frac{1}{T} = \frac{1.45}{(R_A + 2 R_B) C} \text{ ----- (1)}$$

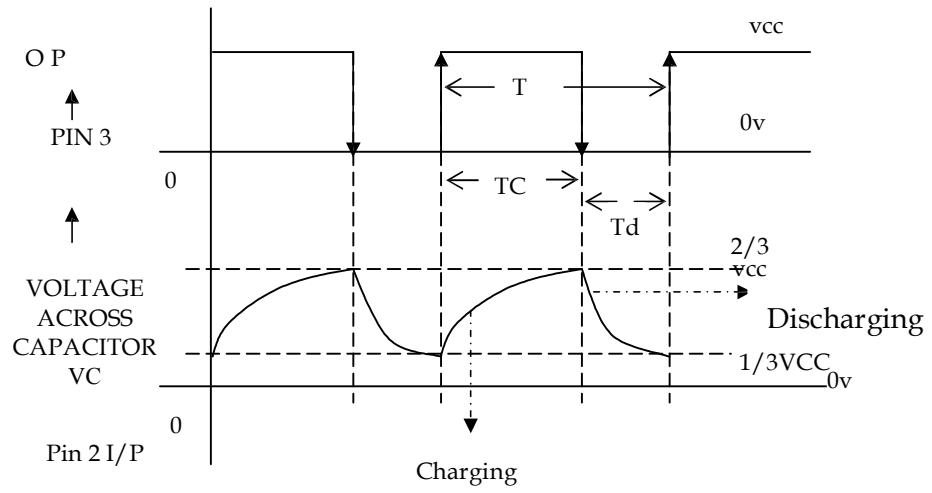
This frequency of oscillation is also called as free-running frequency. Above equation shows that frequency of oscillation is independent of supply voltage  $V_{cc}$ .

The percentage Duty cycle of AMV is given by

$$\begin{aligned} \% \text{ Duty Cycle} &= \frac{T_c}{T} \times 100 \\ &= \frac{R_A + R_B}{R_A + 2 R_B} \times 100 \% \text{ ----- (2)} \end{aligned}$$

**Voltage across capacitor (Trigger I/P pin 2) and O/P voltage waveform.**

$V_0$



**PROCEDURE:**

**FIG :1 (b)**

- (1) Study the Operation of above circuit diagram.
- (2) Make the connection as shown in fig. 1(a).
- (3) Select desired value of  $R_A$ ,  $R_B$ ,  $C$  from components bank.
- (4) Switch on the power supply. Measure the supply voltage + 12V at pin 8 with respect to ground pin 1.
- (5) Note the value of  $R_A$ ,  $R_B$ ,  $C$ . Find it's Theoretical frequency as per equation(1).
- (6) Now connect the CRO at output pin 3 and note the practical time period  $T_c$  &  $T_d$  & calculate Practical O/P frequency. As shown in fig. (1b)
- (7) Repeat the above steps for different combinations of components .
- (8) Calculate Duty Cycle using equation(2).

**OBSERVATIONS TABLE:-**

Capacitor	Resistors RB	Resistors RA	Theoretical freq.	Observed Freq.

## VIVA QUESTIONS

Q.1 Give some application of 555 timers IC

---

---

---

---

---

Q.2 What are the basic elements of a 555 timer IC?

---

---

---

---

---

Q.3 Give important features of 555 timer IC.

---

---

---

---

---

Q.4 What do you mean by the term duty cycle of an astable multivibrator?

---

---

---

---

---

Q.5 For monostable operation of 555 timer IC, what nature of input trigger pulse is needed?

---

---

---

---

---

## EXPERIMENT NO:

### OBJECTIVES:-

IC 555 as a Monostable Multivibrator.

INSTRUMENTS REQUIRED:- CRO, Patch Chords 6 No.

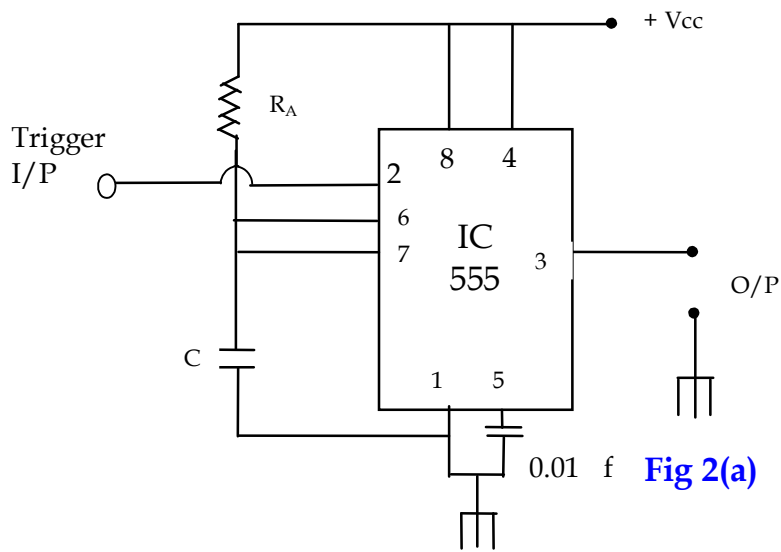
### **IC 555 as a Monostable Multivibrator**

Fig.2(a) shows circuit diagram & fig.2(b) shows waveform. In this when O/p is low that the ckt is in a stable state transistor Q1 is ON. So capacitor connected is shorted to ground. However upon application of negative trigger pulse at Pin 2 transistor Q1 turned off, which release the short Ckt across the external capacitor

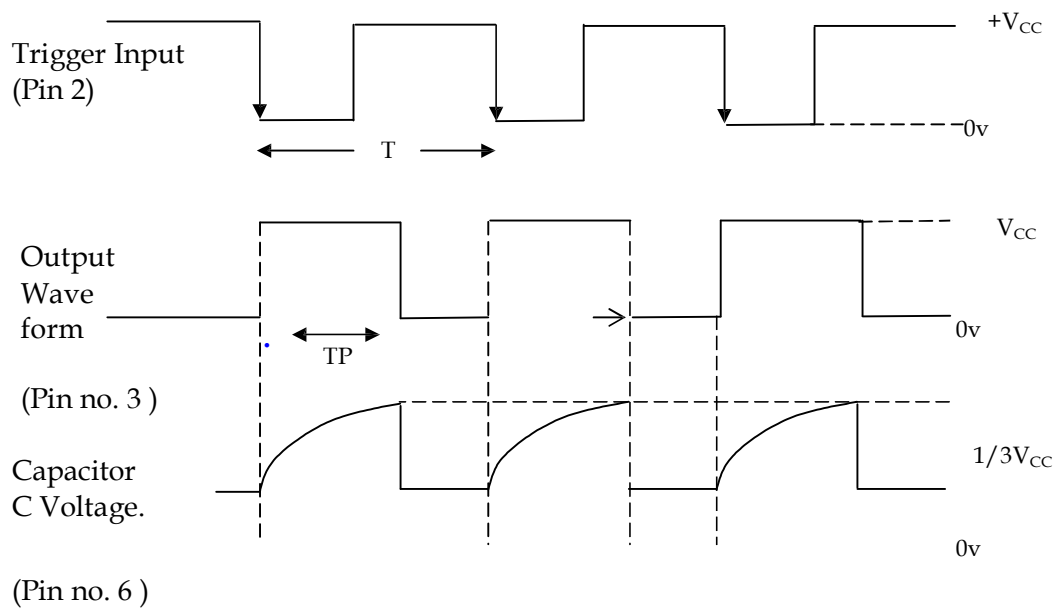
connected between 7,6 & gnd capacitor now starts charging up toward Vcc through RA.

At the same time O/P of flip flop turns Q1 ON , & hence capacitor C rapidly discharged through the transistor . The O/P of monostable remains low untill trigger pulse is again applied . Then the same cycle repeats again.

The connection for monostable are shown below.



**Fig 2(a)**



**FIG.2(b). Input Waveforms .**

Once triggered, the Ckt. Output will remain in the high state.

The Time for which O/P remains High is given by  $T_P = 1.1 R_A.C$  seconds

**PROCEDURE:-**

- (1) Study the Operation of above circuit diagram.
- (2) Make the connection as shown in fig. 2(a).

- (3) Select desired value of  $R_A$  ,  $C$  from components bank.
- (4) Switch on the power supply. Measure the supply voltage + 12V at pin 8 with respect to ground pin 1.
- (5) Note the value of  $R_A$ ,  $C$ . Find it's Theoretical frequency as per equation  $T_P = 1.1 R_A.C$
- (6) Apply Square I/P from signal generator at pin no.2 . Observe & Note practical time  $T_P$  for which O/P remains. Then goes to zero until next -ve triggers Pulse is applied.[as shown in fig.2(b).]
- (7) Repeat the above steps for different combinations of components.
- (7) Draw the Input & Output Waveforms as shown in fig.2 (b).

**OBSERVATION TABLE:-**

Capacitor $C$	Resistors $R_A$	Theoretical Time $T_P$	Practical Time $T_P$

## EXPERIMENT NO:

### OBJECTIVES:-

IC 555 as a Schmitt Trigger.

INSTRUMENTS REQUIRED: - CRO, Patch Chords 6 No.

### **IC 555 as a Schmitt Trigger.**

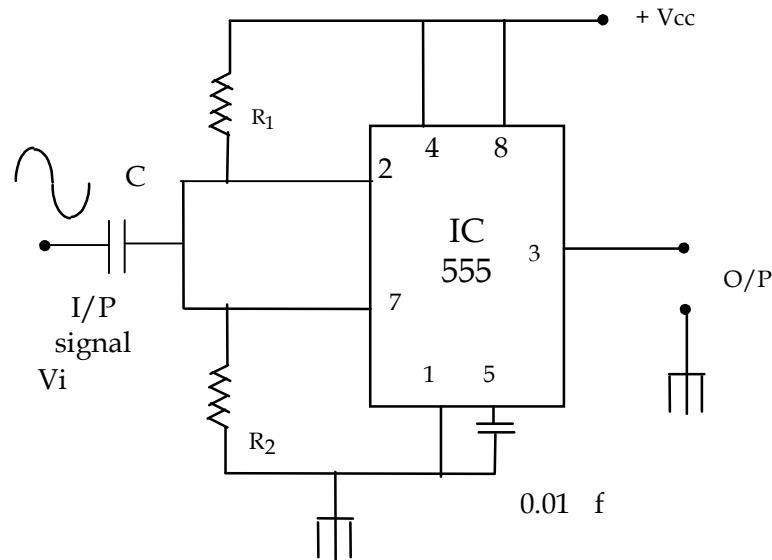
The circuit which converts an irregular shaped waveform to a square wave or pulse is called the Schmitt trigger or squaring circuit. The input voltage  $V_i$  triggers (changes the state of) the output every time it exceeds certain voltage levels called the upper threshold voltage (UTP) & lower threshold voltage (LTP).

Fig. 4(a) shows circuit diagram & Fig. 4(b) shows waveforms of Schmitt trigger. In Schmitt trigger the pin no. 2 is shorted with Pin 6 & rest of the connections are remaining same. The sine wave I/P is applied at pin 2 & 6.

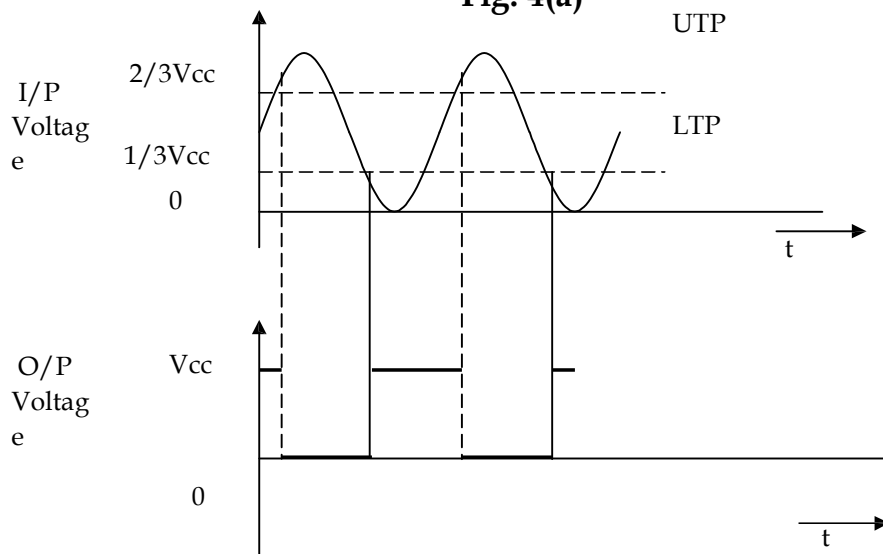
If the sine wave of sufficient amplitude is applied to the input terminals to exceed. The reference level ( $1/3 V_{cc}$  &  $2/3 V_{cc}$ ) cause the internal flip-flop to be set and reset. In this way, it generates a square wave at the output terminal. So long as  $R_1 = R_2$ , the 555 will be automatically biased correctly for almost any supply voltage. The output waveform is  $180^\circ$  out of phase with the applied sine wave. The circuit can be used as a signal shaper or buffer with advantages of availability of high output currents. By modifying the input time constant of the circuit reducing the value of  $C_1$  so that the input pulses get differentiated, the arrangement can also be used either as a bistable device or to invert pulse waveforms. The fast time constant of the combination

of C with  $R_1$  &  $R_2$  causes only the edges of the input pulse or rectangular waveform to be passed.

From the input and output waveform it is clear that when the input voltage exceeds the reference voltage ( $2/3 V_{cc}$ ) the output become zero, where as when the input voltage goes below the reference voltage ( $1/3 V_{cc}$ ) the output goes high i.e.  $V_{cc}$  and the cycle repeats.



**Fig. 4(a)**



**Fig. 4(b). Input & Output Waveforms.**



## **PROCEDURE:-**

- (1) Study the Operation of above circuit diagram.
- (2) Make the connection as shown in fig. 4(a).
- (3) Select desired value of  $R_1$  ,  $R_2$  ,  $C$  from components bank.
- (4) Switch on the power supply. Measure the supply voltage + 12V at pin 8 with respect to ground pin 1.
- (5) Apply a sine wave of 1KHz with 5Vpp at the input terminals.
- (6) Observe & Note O/P as shown in fig. 4(b). Note the level of UTP & LTP.
- (7) Change the frequency and amplitude of the input signal & observe the effect on O/P.
- (8) Draw the waveforms as shown in fig. 4(b).

## **OBSERVATIONS TABLE: -** Amplitude

of Sine wave I/P =-----V Frequency of

Sine wave I/P =-----Hz

Voltage Level of UTP. =-----V

Voltage Level of LTP. =-----V

## EXPERIMENT NO:

### OBJECTIVE:-

- 1) Study of Differentiator Circuit using Op-Amp IC 741.
- 2) Study of Integrator Circuit using Op-Amp IC741.

**INSTRUMENTS REQUIRED:** - CRO (dual channel) function generator,

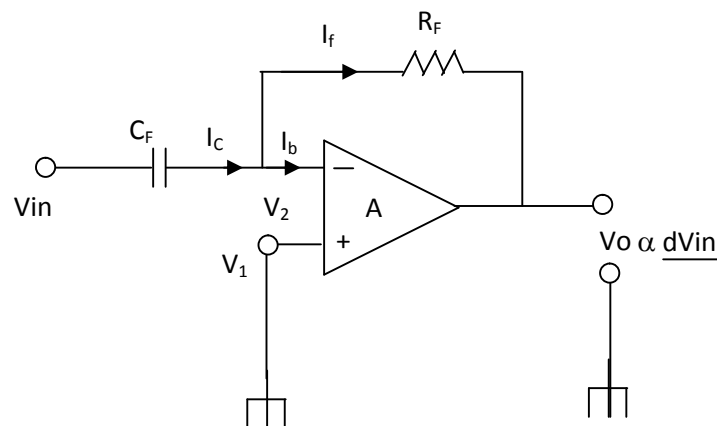
**Components value:**  $R = 1.5K$ ,  $C_1 = 0.01\mu f$ ,  $C_2 = 0.5\mu f$ ,  $R_F = 1.5K$ ,

$C_F = 0.001\mu f$ ,  $R_L = 10K$ , IC 741.

### THEORY:

The output waveform of differentiator is derivative of input signal.

By interchanging the capacitor & resistor in integrator circuit, differentiator circuit can be formed. The basic circuit of differentiator is shown in fig.



By KCL

$$I_C = I_b + I_f$$

As  $R_i = \infty$ ,  $I_b \cong 0$

$$\therefore I_C = I_f$$

$$\therefore C \frac{d}{dt} (V_{in} - V_2 - V_0)$$

As  $V_2 = V_0$  by virtual ground  $V_2 - V_0 = 0$ .

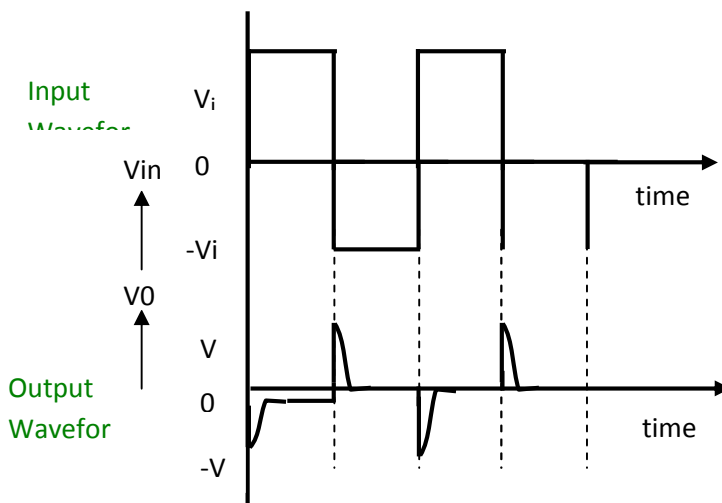
$$\therefore V_0 = -R_f \frac{d}{dt} V_{in}$$

$R_1$  &  $C_f$  are added in practical differentiator to compensate problem of stability & high frequency noise.

The input signal will be differentiated properly if the time period of input signal applied ( $T$ ) is larger than  $R_1 C_1$ .

$$T > R_f C_1$$

Waveforms for Differentiator Circuit



### PROCEDURE:-

- 1) Study the circuit given on the panel kit.
- 2) Connect function generators output to the circuit as input source and CRO to observe output waveform.
- 3) Apply input signal ( $T > R_f C_1$ ) and observe the output waveform as CRO.

- 4) Now replace C1 by C2 and repeat step (3).
- 5) Vary I/P frequency and observe the change in O/P waveform.

**OBSERVATION TABLE:-**

(1)

Input signal	Output signal
Square wave	-----
Sine wave	-----

**RESULT:-**

The input to the circuit of differentiator is integrated and differentiated. From observation it is seen that the circuit gives proper response and this is depend on I/P signal frequency.

## OP-AMP IC741 AS AN INTEGRATOR

### INSTRUMENTS REQUIRED:-

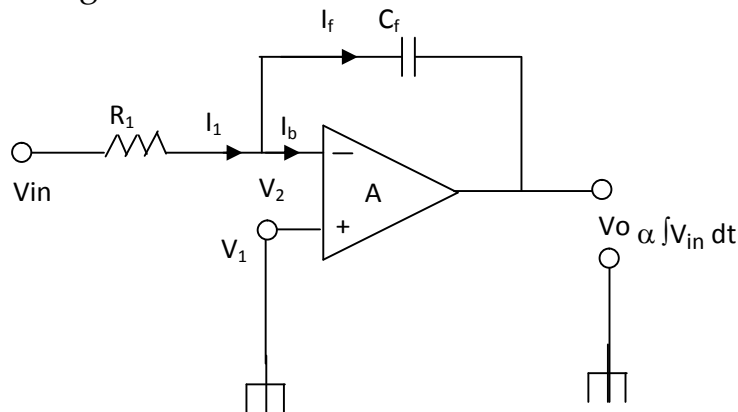
CRO (dual channel) function generator, Patch chords.

### THEORY:-

Integrator circuit are used in analog computer, A to D convertors and signal wave shaping. The main reason for using active components (741) with integrator is to give response for low level input signals.

### DEFINATION:-

The circuit in which the output is a integral of input voltage / signal is known as Integrator Circuit. Integrator acts as a low pass filter. The basic circuit of integrator is shown in fig below.



**Fig.1**

By Kirchoff's current law.

$$I_1 = I_b + I_f$$

Since input impedance of Op-Amp is very high,  $I_b \cong 0$ .

$$\therefore I_1 = I_f \text{ -----(1)}$$

Current through capacitor is given by

$$I_f = I_c$$

$$I_f = C_f \frac{dV_c}{dt}$$

Where  $V_c$  = voltage across capacitor.

$$\therefore \frac{V_{in} - V_2}{R_f} = C_f \frac{d}{dt} (V_2 - V_0) \quad (2)$$

As  $V_1 = 0$ , by virtual ground  $V_2 = 0$

From equation  $\frac{V_{in}}{R_f} = C_f \frac{d}{dt} (-V_0)$

Now integrating both sides we get

$$\int_0^t \frac{V_{in}}{R_f} dt = \int_0^t C_f \frac{d}{dt} (-V_0) dt$$

$\therefore$  finally we get

$$V_0 = -\frac{1}{R_f C_f} \int_0^t V_{in} dt + C$$

Where  $C$  is constant.

The proper response of integrator is with a particular range of input frequency which can be obtained by formula as given below

$$f_b = \frac{1}{2\pi R_f C_f}$$

Where  $f_b$  is the frequency at which gain is zero dB.

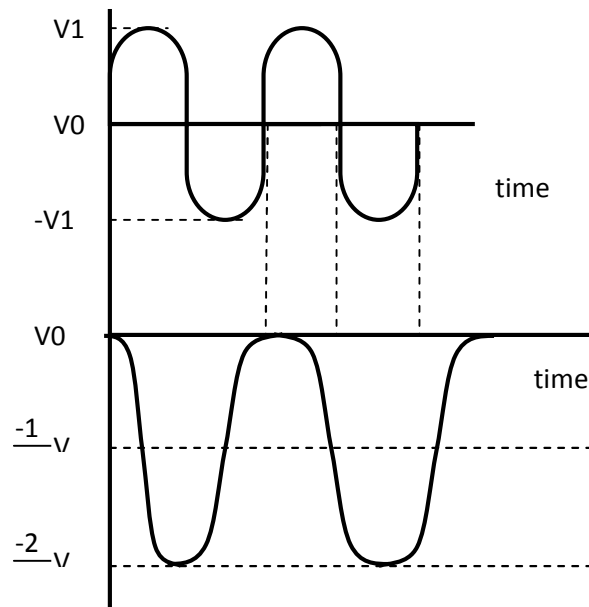
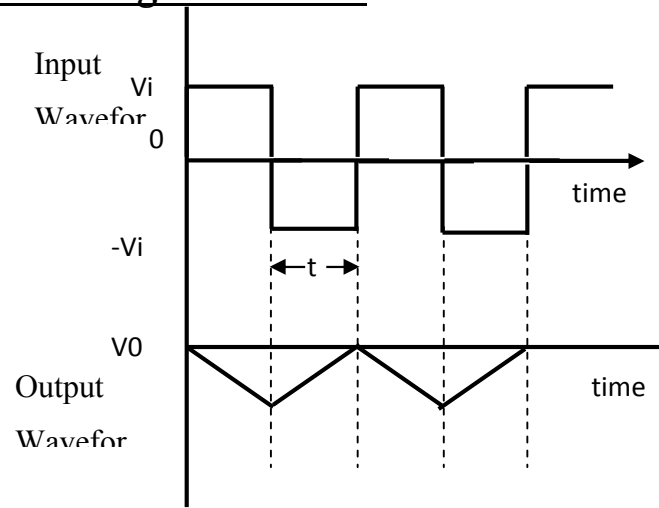
$$f_a = \frac{1}{2\pi R_f C_f}$$

Where  $f_a$  = gain limiting frequency.

Input signal will be integrated properly if the time period of input signal is larger than product of  $R_f C_f$ , i.e.  $T > R_f C_f$

Where  $T$ , is time period of input signal.

## Waveforms for Integrator Circuit

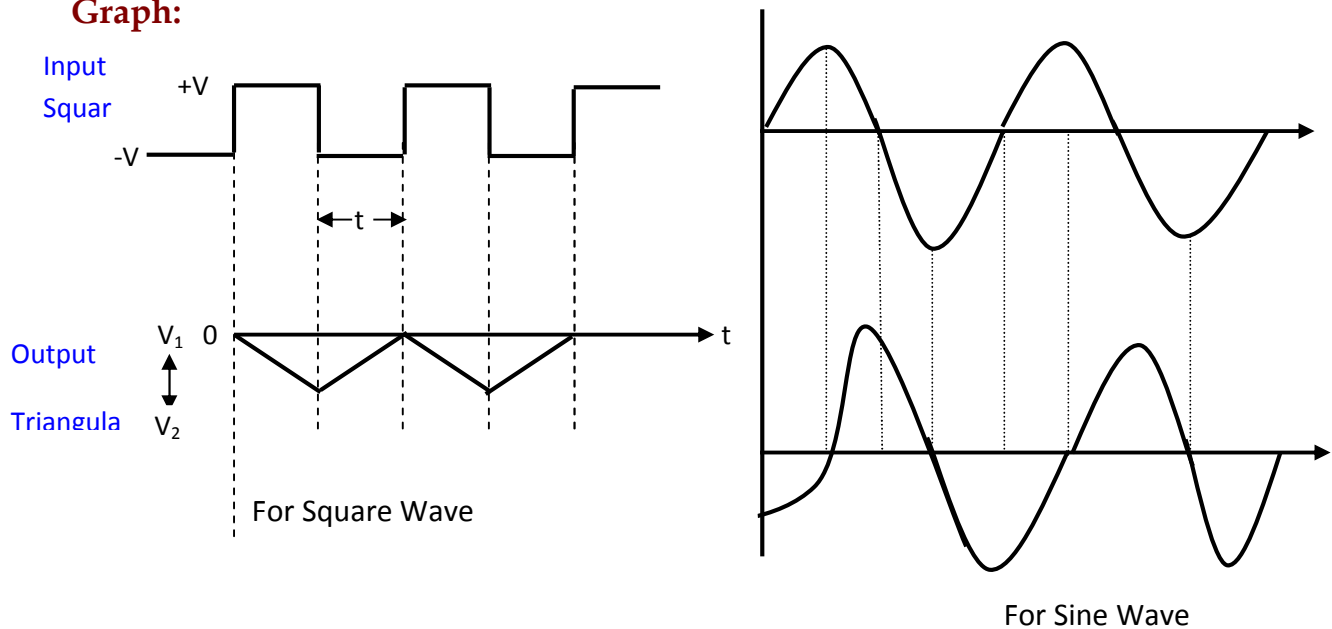


In practical circuit feedback resistor  $R_f$  is used to reduce the error in output voltage

**PROCEDURE:-**

- 1) Study the circuit given on the front panel of the kit.
- 2) Switch on the power supply.
- 3) Apply input from function generator i.e. sine or square wave (The input should be such that  $T > R_f C_f$ ).
- 4) Connect  $C_1$  in parallel with  $R_f$  by patch chords.
- 5) Connect output of integrator to CRO by probe and observe the waveform.
- 6) Vary input frequency and observe the change in output waveform.
- 7) Now replace  $C_1$  by  $C_2$  and repeat step (6)

**Graph:**



Draw the graph with  $C_f$  as  $C_1$  or  $C_2$

Now increase the frequency of input signal ( $T < R_f.C_f$ ) and see the effect on output waveform.

**OBSERVATION TABLE:-**

Input signal	Output signal
(1) Sine wave	-----
(2) Square wave	----- -

## **RESULT:-**

The O/P of the integrator is the integrator waveform of the I/P wave applied. From the observation it is seen that the circuit given repose and this is depends on the I/P signal applied.

## **VIVA QUESTIONS**

**Q1. Draw the circuit diagram of Integrator circuit?**

**Q2.What is the O/P voltage equation of Integrator?**

---

---

**Q3.What is the voltage across capacitor?**

---

---

**Q4. What is the current across capacitor?**

---

---

**Q5. Draw the circuit diagram of Differentiator circuit**

**Q6. What is the O/P voltage equation of Differentiator?**

---

---

