SAIT JABALPUR

DEPARTMENT OF ELECTRONICS & COMMUNICATION ENGG.

EC-404 **ELECTRONICS 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 SUSTRACTOR USING IC-741.

5) OP-AMP AS AN INTEGRATOR AND DIFFERENTIATOR USING IC-741.

6) DESIGN & CONSTRUCT A SHUNT & SERIES REGULATOR AND FINE LINE AND LOAD REGULATION.

7) PERFORMANCE EVALUATION OF FEEDBACK AMPLIFIERS (OSCILLATOR).

8) 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: -

To measure the Input offset voltage (Vio) & Input Bias current of op-amp.

EQUIPMENT: -

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

THEORY:-

1) Input offset voltage (Vio):-

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 (Iio):-

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

Iio= (IB1) – (IB2)

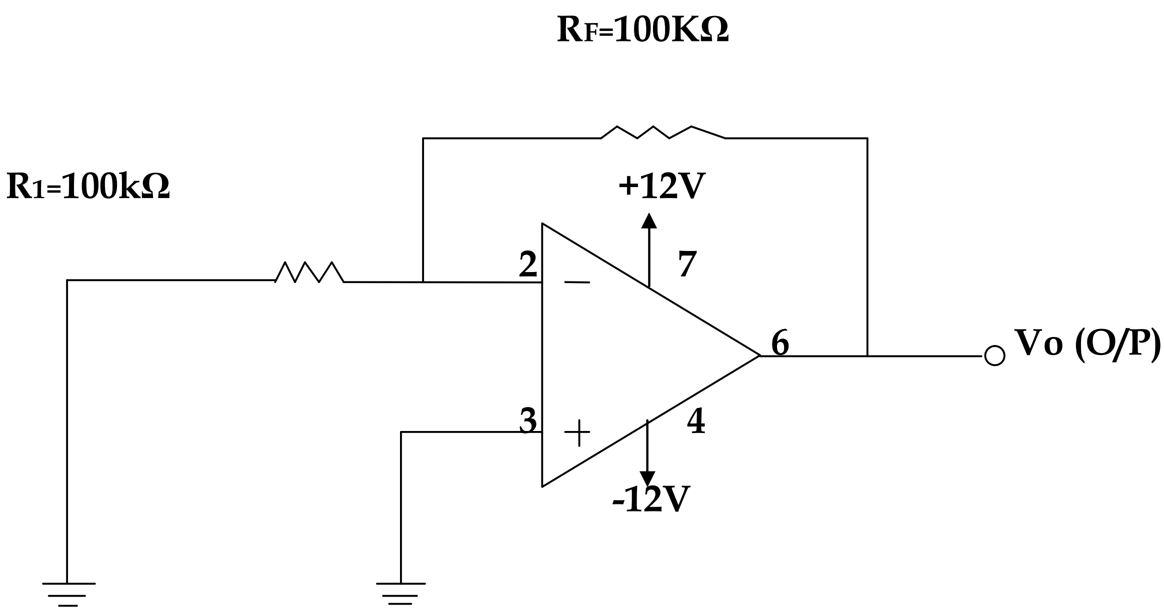
3) Input bias current (IB):-

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

$$IB = \frac{(IB1) - (IB2)}{2}$$

PROCEDURE:-

MEASUREMENTS OF I/P OFFSET VOLTAGE (Vio)



1) Make the connections as shown in the diagrams.

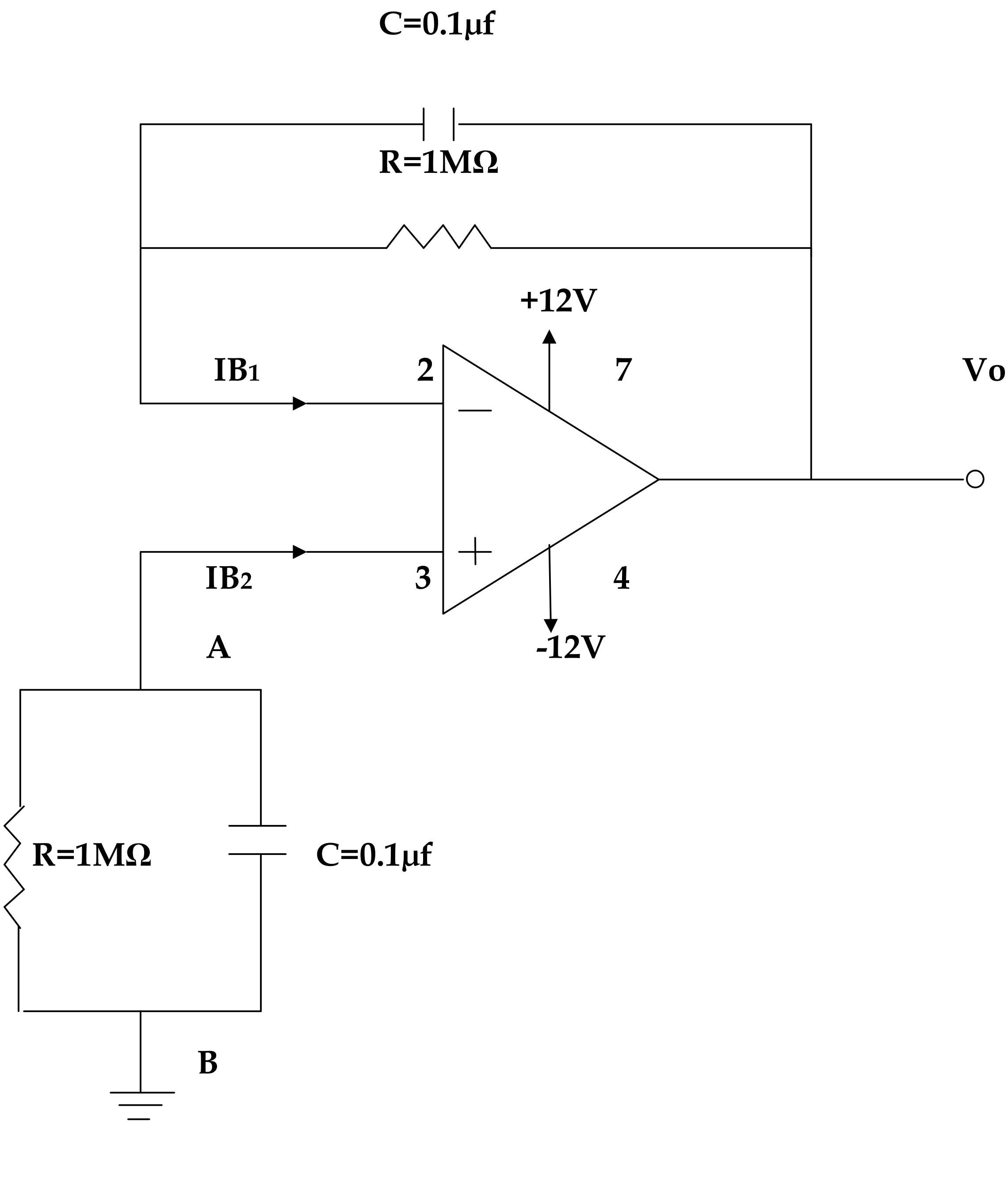
2) Apply +VCC and –VEE as + 12V.

3) Measure the output voltage on DMM.

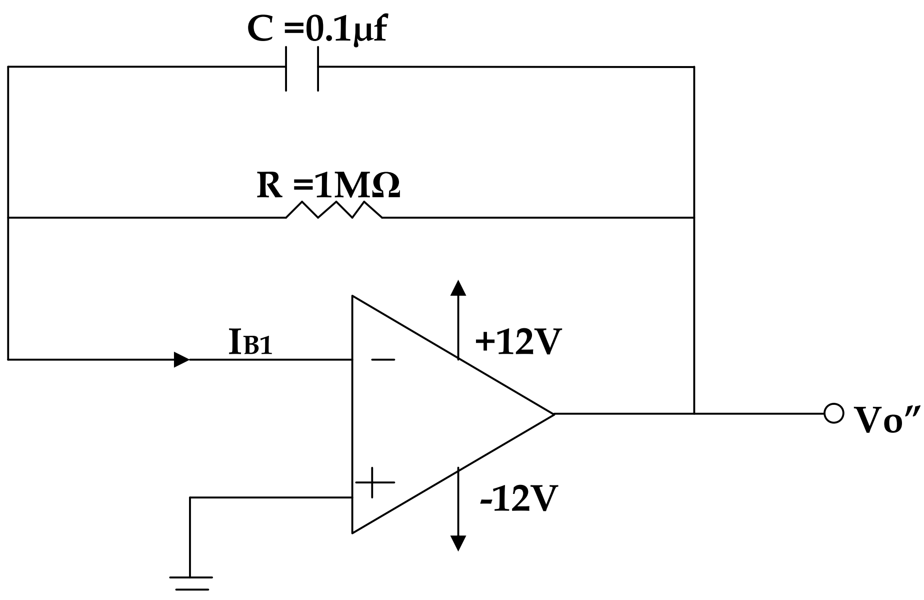
4) Find the input offset voltage using formula.

Vio = vo*R1/ (R1+Rf)

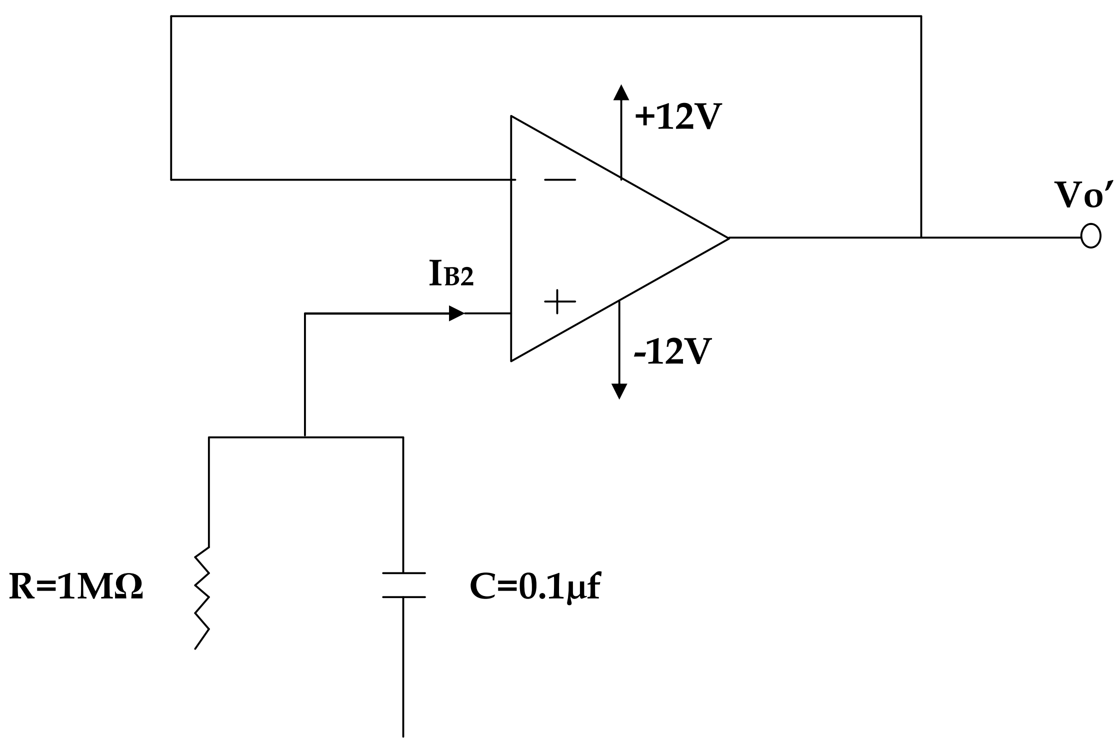
MEASUREMENT OF INPUT OFFSET CURRENT (Iio) AND INPUT BIAS CURRENT (Ib)

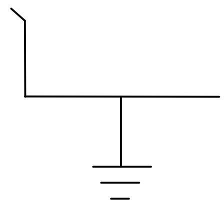


Measurement of IB1:-



Measurement of IB2:-





Measurement of input offset current (Iio) and input bias current (IB):-

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 IB1, short A & B, measure Vo as Vo". Calculate IB1 using formula, IB1= Vo'' /R.

5) To measure IB2, short C & D, measure Vo as Vo". Calculate IB2 using formula, IB2= Vo' /R.

6) Calculate input offset current Iio = (IB1) – (IB2).

7) Calculate input bias current, using IB = (IB1) – (IB2)/2.

DESIGNS:-

1) Input offset voltage (Vio):-

Vio = Vo*R1/ (R1+Rf)

2) Input bias current = IB = $\frac{(IB1) - (IB2)}{2}$

OBSERVATION TABLE:-

IC-741:-

1. I/p offset voltage (Vio) = Output voltage reading Vo = -1.225 V.
2. Vo" = 29 mV. Therefore IB1 = 29 nA.
3. Vo' = -33 mV. Therefore IB2 = 33 nA

Serial no.	OpAmp parameter	Ideal Value	Typical Value	Practical Value
1.	i/p offset voltage	O	2 mV	1.25 mV
2.	i/p offset current (Iio)	O	20 nA	4 nA
3.	i/p bias current(IB)	500nA	80 nA	31 nA

CONCLUSION:-

In this experiment, we studied different parameters like Vio, Iio & IB 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: 2

OBJECTIVE: -

To measure the OP-Amp parameter for IC-741.

Common mode rejection ration (CMRR)

EQUIPMENT:-

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.

$$CMRR = Ad/Ac$$

$$Ad = Vo/Vd$$

And common mode voltage gain can be determined by

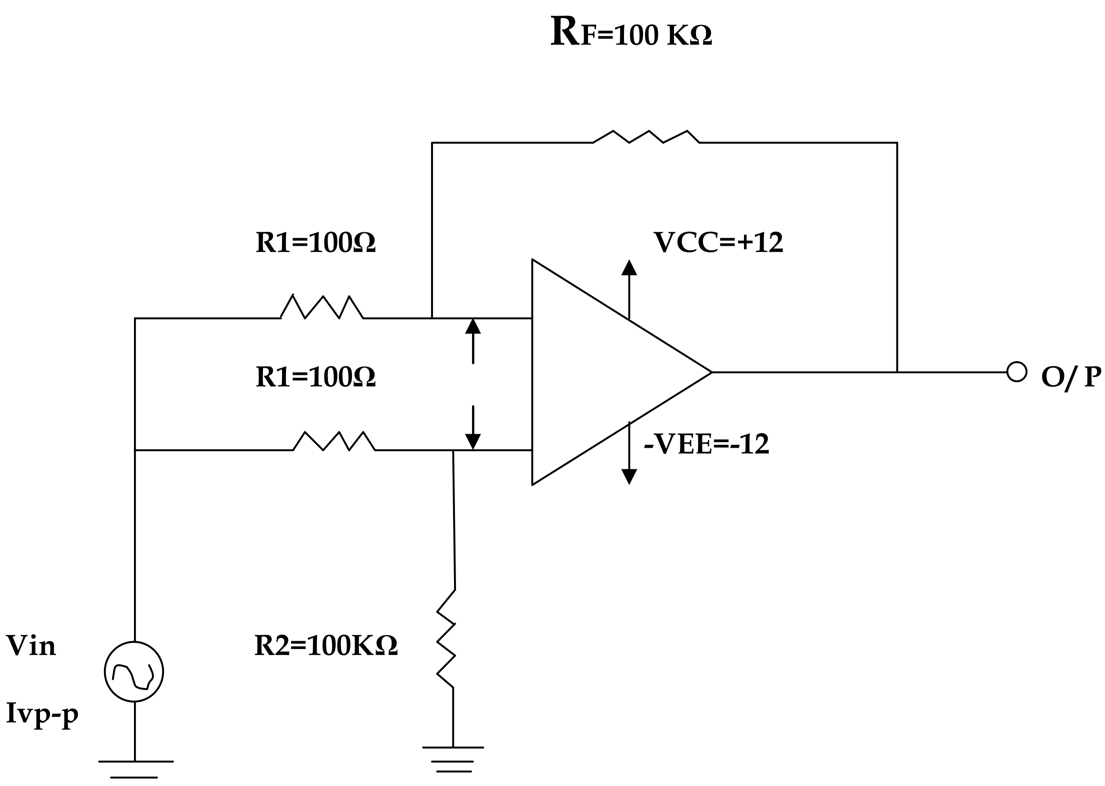
$$ACM = \frac{VocM}{VicM}$$

VicM is the common mode I/P voltage.

ACM is common mode voltage gain. The CMRR is very large hence CMRR is most often expressed in decibels.

PROCEDURE:-

MEASUREMENT OF CMRR:-



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

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

DESIGNS:-

1. 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	92.04 dB

CONCLUSION:-

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

Viva-Voce

Q1. What are the characteristics of OP-AMP?

EXPERIMENT NO: 3

OBJECTIVE: -

To measure the OP-Amp parameter for IC-741.

Slew Rate (SR)

EQUIPMENT:-

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

THEORY:-

SLEW RATE:-

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

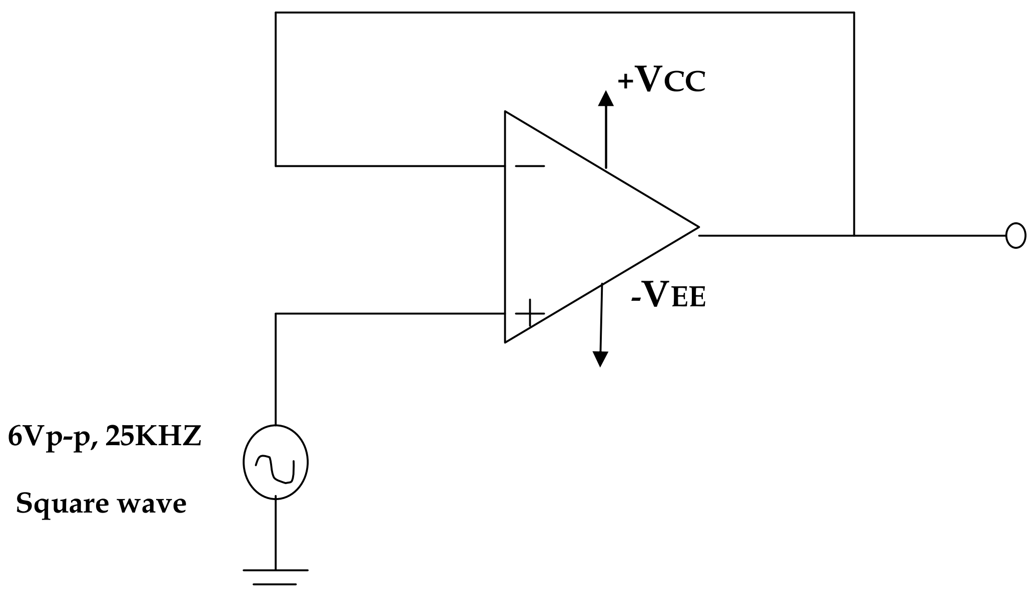
SR =? VO/? t

It is expressed in V/µ 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 Sr is commonly specified.

MEASUREMENT OF SLEW RATE:-

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



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? Vo 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/µsec	.73V/µsec

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: 4

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.

EQUIPMENT:-

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 RF, However an extra resistor R_1 is connected in series with the input signal source Vin. As shown in fig. 1. It is also known as voltage shunt feedback amplifier.

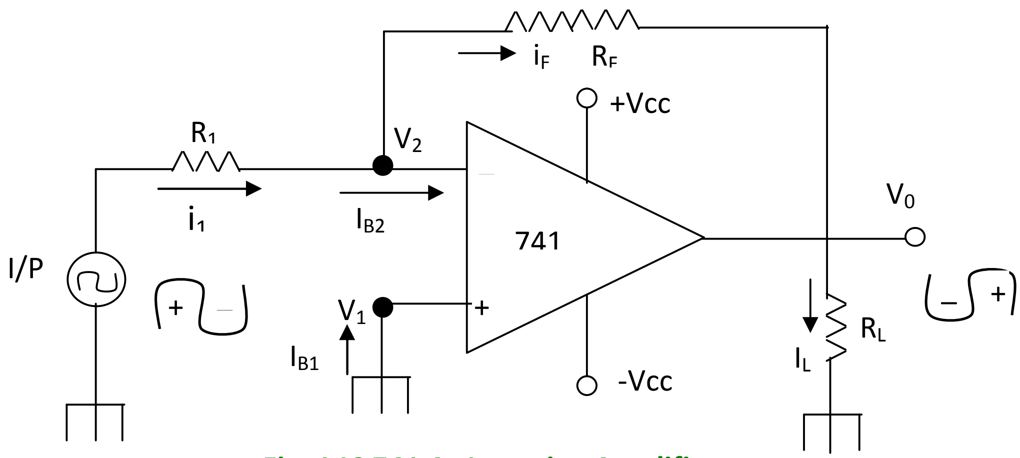


Fig. 1 IC 741 As Inverting Amplifier.

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.

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

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

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

From fig. 1 $$i_1 = \frac{Vin - V_2}{R_1}$$

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

$$\frac{Vin - V_2}{R_1} \qquad \frac{V_2 - V_0}{R_F}$$

$$= \quad \text{----------------(3)}$$

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

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

$$\therefore V_0 = -A\,V_2$$

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

Substituting the value of V_2 in equation (3)

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

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

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

We get,

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

$$A_F = \frac{V_0}{Vin} = \frac{-R_F}{R_1}$$

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

The negative sign in the above equation indicates the phase difference of 180° 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.
3. Select desired R_1 & R_F.
5. Connect dual trace CRO at input and output side to observe Vin & V_0 respectively.
6. Apply 100Hz, sine wave input Vin from signal generator. Adjust it's amplitude so that op-amp should not enter in saturation.
7. Observe & note input Vin amplitude & output V_0 amplitude on CRO calculate it's gain using $A_F = V_0/Vin$
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	Vin	V_0	Theoretical gain **$A_F = - R_F/ R_1$**	Practical gain **$A_F = V_0/Vin$**
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 RF, with R_1 is grounded form inverting terminal. However an extra resistor R_1 is connected in series with the input signal source Vin. As shown in fig. 1. It is also known as voltage series feedback amplifier.

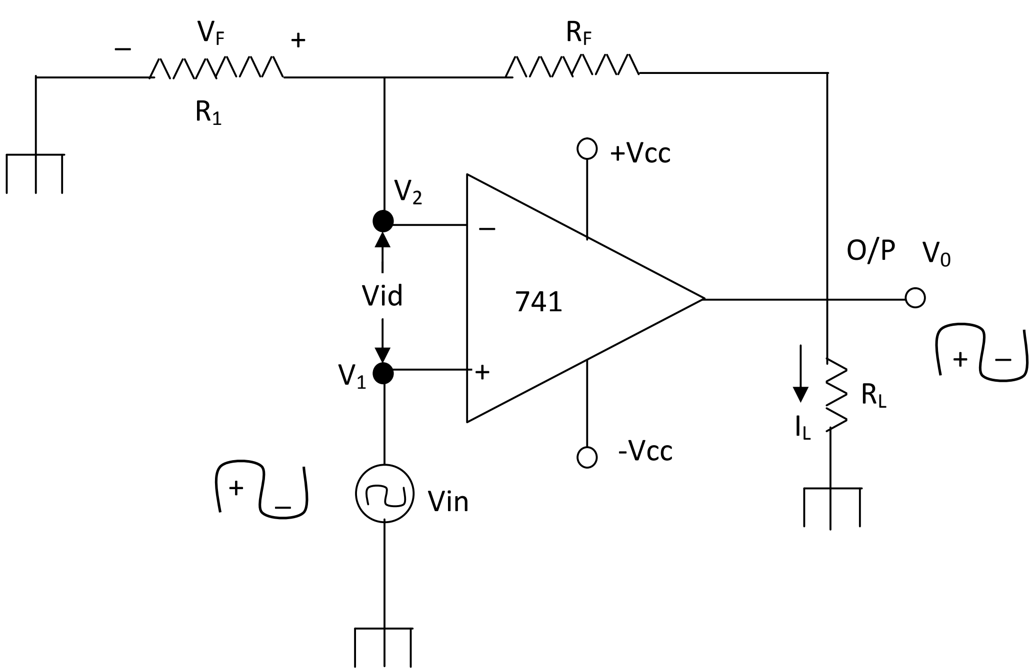


Fig. 1: IC 741 As Non-Inverting Amplifier.

Closed Loop Voltage Gain (AF)

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

We know, closed loop gain as defined as

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

Referring to Fig. 1: We get, $V_1 = Vin$

$$V_2 = V_F = \frac{R_1 V_0}{R_1 + RF} \quad \text{Since } Ri >> R_1$$

$$\therefore V_0 = A\left(Vin \quad \frac{R_1 V_0}{R_1 + RF}\right)$$

Finally we get, $V_0 = \dfrac{A(R_1+RF)\,Vin}{R_1+R_F+AR_1}$

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

RF)
+

$$A_F = \frac{V_0}{Vin} = 1 + \frac{R_F}{R_1} \quad \text{-----------(3)} \quad \text{---------------------(4)}$$

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

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 Vin & V_0 respectively.
6) Apply 100Hz, sine wave input Vin from signal generator. Adjust it's amplitude so that op-amp should not enter in saturation.
7) Observe & note input Vin amplitude & output V_0 amplitude on CRO calculate it's gain using $A_F = V_0/Vin$
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	Vin	V_0	Theoretical gain **$A_F = 1 + R_F / R_1$**	Practical gain **$A_F = V_0/Vin$**
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: 5

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 & AVERGER CIRCUIT.

If the different I/P voltage sources (Va, Vb, Vc) & 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 RF

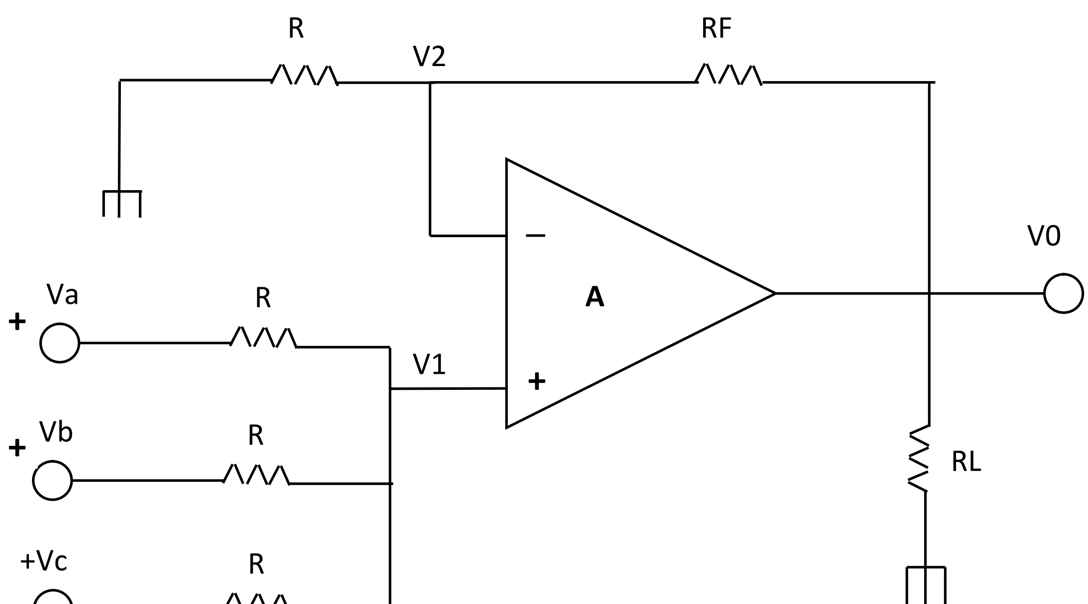


Fig. 1:- **Non-inverting adder averager circuit.**

As the input Resistance Rif of the Non-inverting amplifier is very large. Therefore, using the superposition theorem, the voltage V_1 at the Non-inverting terminal is

$$V1 = \frac{R/2}{R + R/2} Va + \frac{R/2}{R + R/2} Vb + \frac{R/2}{R + R/2}$$

OR

$$V1 = \frac{Va}{3} + \frac{Vb}{3} + \frac{Vc}{3} = \frac{Va + Vb + Vc}{3}$$

The gain of the Non-inverting amplifier is given by

$$V0 = \left(1 + \frac{RF}{R_1}\right) V_1$$

$$V0 = \left(1 + \frac{RF}{R_1}\right) \left(1 + \frac{Va + Vb + Vc}{3}\right) \quad \text{---------------- (1)}$$

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

If in equation (1)

$$\left(1 + \frac{R_F}{R_1}\right) = 3 \qquad \text{i.e.} \quad R_F = 2R_1$$

then

$$V_0 = Va + Vb + Vc$$

Hence the circuit will be adder/summing amplifier circuit.Where as if in equation (1).

$$1 + \frac{R_F}{R_1} = 1 \qquad \text{i.e.} \quad RF = 0 \quad \text{OR} \quad R1 = \infty$$

Then from equation (1),

$$V0 = \frac{Va + Vb + Vc}{3}$$

The circuit will be average 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.

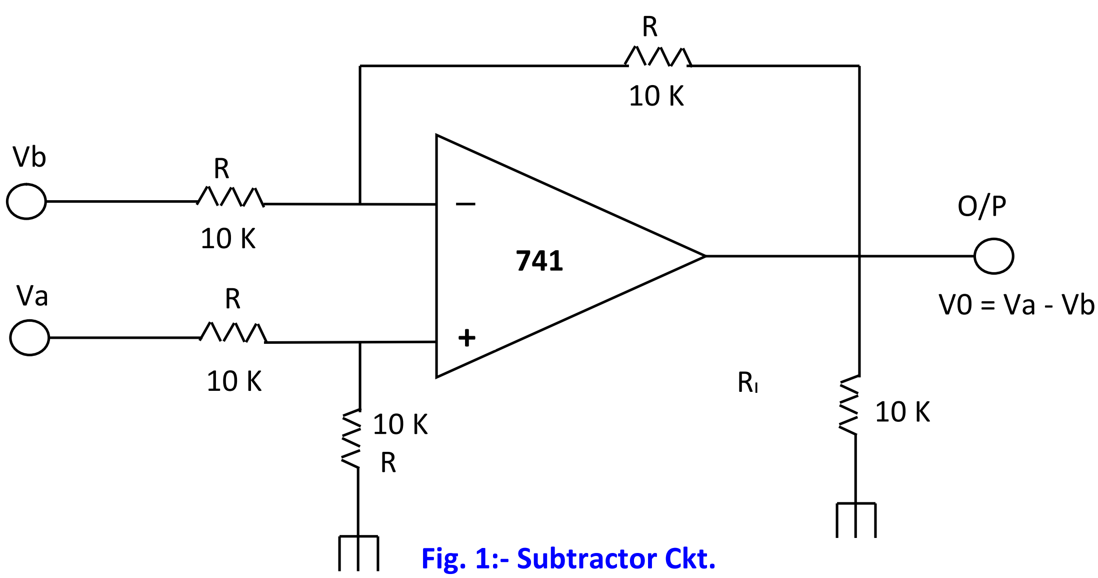


Fig. 1:- Subtractor Ckt.

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

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

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

$V_0 = V_b – Va$

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 Va, Va_1 & Va_2 to desired voltages V_1, V_2, V_3 or V_4, note its value, measured O/P voltage V_0.
Compare with theoretical $V_0 = Va + Vb + Vc$

(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 $V_0 = \frac{Va + Vb + Vc}{3}$

(6) For Subtraction then Connect OP-AMP I/Ps Va & Vb to the desired voltages of potential divider network (V1 to V4) & note it's values. Measure the O/P voltage V_0 & compare it with theoretical $V_0 = -(Va – Vb)$

(7) Repeat step (4) for different combinations of Va and Vb.

OBSERVATIONS:

For Adder Operations:-

I/Ps	Theoretical	Observed

Va	Vb	Vc	V0 = Va+Vb+Vc	V_0
V_1 V_2	V_2 V_3	V_3 V_4		

For Subtractor Operations:-

V1 = ----------V., V2= ----------V., V3 = ----------V, V4 = ----------V.

I/Ps		Theoretical V_0 = Vb – Va	**Observed V0**
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: 6

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,

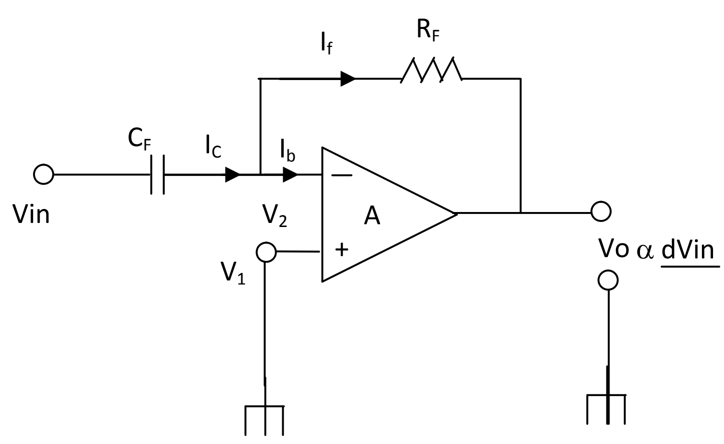
Components value: R = 1.5K, C_1 = 0.01μf, C_2 = 0.5μf, R_F =1.5K,

C_F = 0.001μf, 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 Ri = ∞, $I_b \cong 0$

$\therefore I_C = I_f$

$$\therefore C1 \frac{d}{dt} (Vin-V2) = \frac{V_2 - V_0}{R_F}$$

As V1 = 0, so by virtual ground V2 ≅ 0.

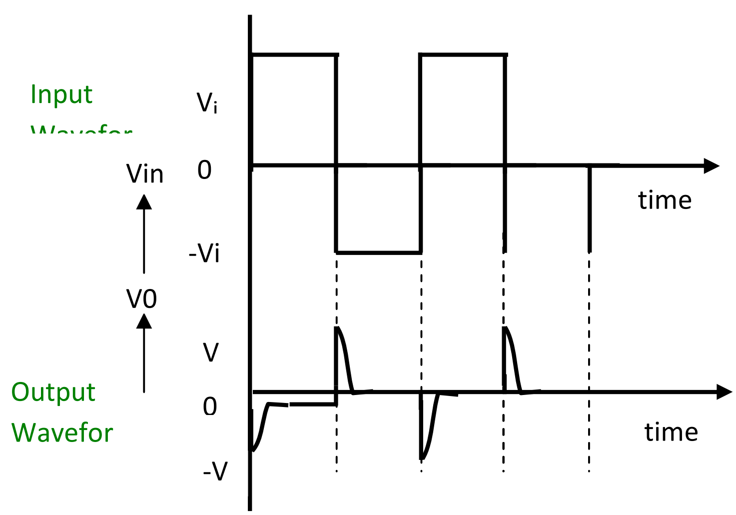
$$\therefore V0 = -R_F \frac{d}{dt} V\ in$$

R1 & CF 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_1C_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:-

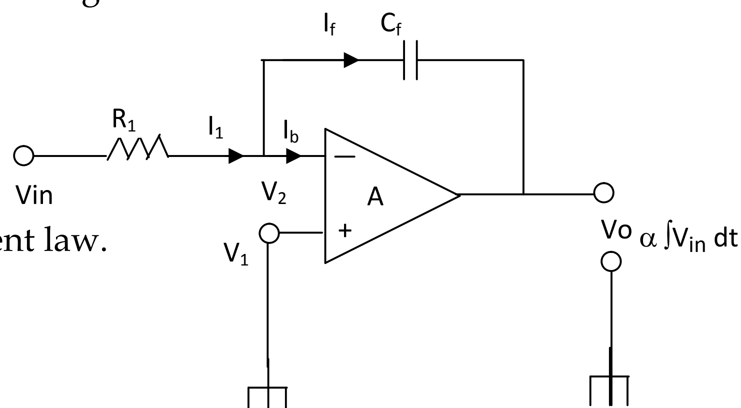
CRO (dual channel) function generator, Patch chords.

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.



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 Vc = voltage across capacitor.

$$\therefore\ \frac{V_{in} - V_2}{R1} = C_F \frac{d}{dt}(V_2 - V_0) \text{ ---------------(2)}$$

As V1 = 0, by virtual ground V2 = 0

From equation (2) $\dfrac{V_{in}}{R1} = C_f \dfrac{d}{dt}(-V_o)$

Now integrating both sides we get

$$t \ \frac{V_{in}}{R_1} \qquad t \qquad \frac{d}{dt}$$

$\int_0$ $dt_0 = \int C_f$ (-Vo) dt

∴ finally we get

$$V0 = -\frac{1}{R_1C_f}\int_0^t Vin\,dt + C$$

Where C is constant.

The proper response of integrator is with a particular range of input frequency wh... ...in by formula as given bellow

$$f_b = \frac{1}{2\Pi R_1C_f}$$

Where F_b is the frequency at which gain is zero dB.

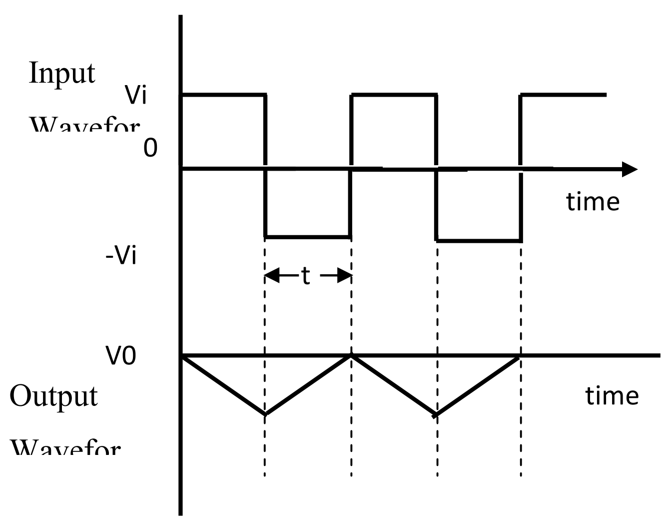
$$f_a = \frac{1}{2\Pi R_fC_f}$$

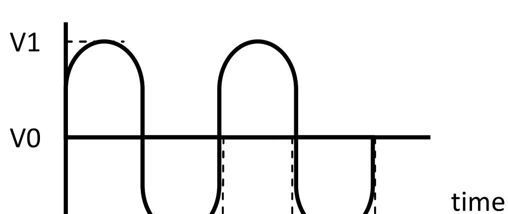
Where f_a = gain limiting frequency.

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



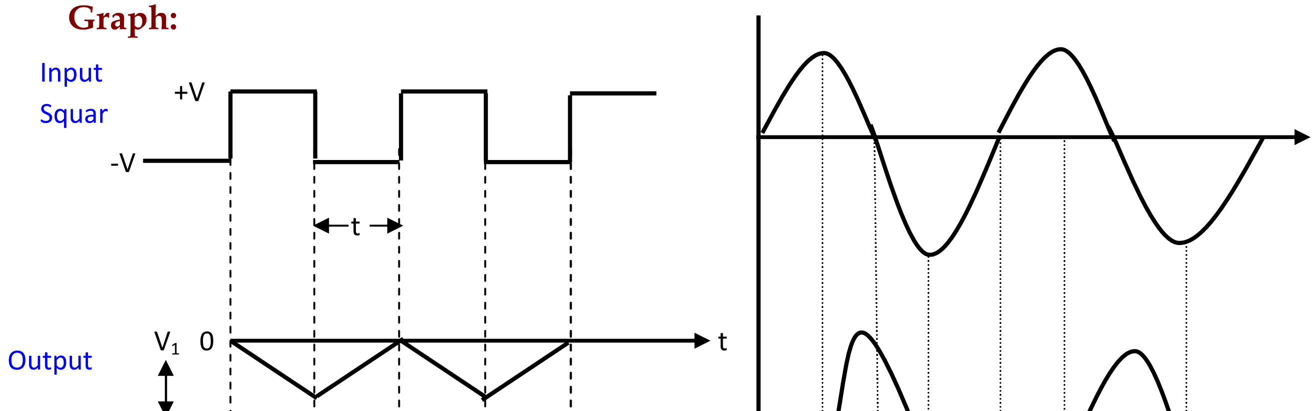


In practical circuit feedback resistor R_{f} is used to reduce the error in output voltage

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 C1 by C2 and repeat step (6)

Graph:



Draw the graph with C_f as C1 or C2

Now increase the frequency of input signal (T< RF.CF) 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?

EXPERIMENT NO: 7

OBJECTIVES:-

To Study operation of Zener Diode as a Shunt Voltage Regulator.

INSTRUMENT:-

Voltmeter (0 – 15V), milliammeter (0– 25mA) optional., Digital Multimeter.

Components used:-

R = 380Ω, RL= 10k pot, Zener Diode Vz = 6.2V & 6.8V

THEORY:-

A Zener diode shunt regulator is shown in Fig 1. Since the Zener is connected in parallel (or shunt) with the load, the circuit is said to be a shunt regulator. A resistance (RS) is connected in a series to the zener Diode which limit the current in the circuit. Hence it is also known as series current limiting resistor. The output voltage VL is taken across the load resistance RL. Which is also the voltage across Zener diode. To observe the proper operation, the input voltage (Vi) must be greater

than the Zener Voltage (Vz). Thus ensures that Zener diode operates in the Reverse breakdown Region. The Input Current (I) through Limiting resistor R is given by the relation.

$$I = \frac{V_i - V_z}{R_S} \qquad \text{———————} \quad (1)$$

Where Vi = Unregulated DC I/P Voltage to the circuit.

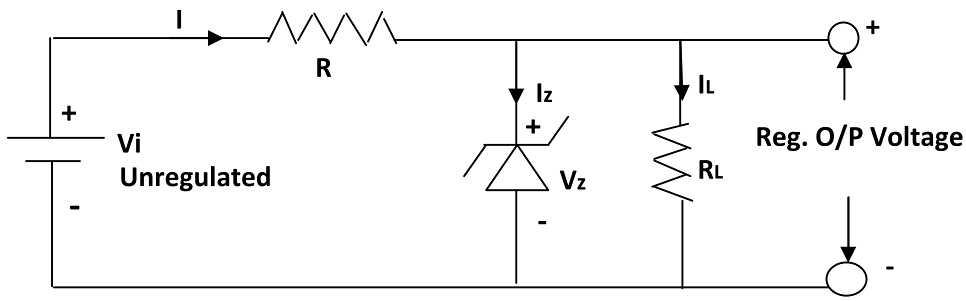


Fig 1:- Zener Diode Shunt Voltage Regulator

Vz = Zener Voltage

The ideal Zener diode acts as a constant voltage source of voltage (Vz). But a practical zener diode has a finite value of resistance called zener resistance (Rz). Because of the zener resistance, there is a voltage drop across it which is equal to IzRz. Therefore the voltage across the zener diode is given by expression.

VL = Vz = IzRz ------------------------- (2)

Let the zener resistance is negligible, and then the load voltage is

VL = Vz ------------------ (3)

And the current through the load resistance is given by the relation.

IL = VL / RL -----------------------(4)

From the circuit it is clear that

I = Iz + IL

& Iz = I – IL ----------------- (5)

<u>OPERATION</u>:-

The operation of the circuit can be discussed under following two heads.

1). Line Regulation with Varying input Voltage:-

The Load RL is constant & if I/P voltage Vi varies within the limits then the circuit provides constant O/P voltage. As RL is constant. So IL is constant, hence from equation (5), Iz = I. As the I/P voltage increase, the current I increases so Iz increases within the limits Izmin – Izmax. So zener diode remains in Reverse breakdown state. But the voltage drop across series Resistance R increases therefore keeping the O/P voltage constant at Vz value & Vice Versa. So As the I/P voltage changes within limits O/P remains constant.

2). Load Regulation with varying load Resistance (RL)

Consider the operation with Load RL Changes & input voltage Vi constant. As I/P voltage Vi constant so from equation (3) I = Iz + IL, current I is constant so as Iz decreases / increases then IL increases / decreases respectively. If Load Resistance RL decreases then current IL increases, current IZ decreases within limits IZmin – Izmax . So Zener diodes remains in breakdown thereby keeping a constant O/P voltage equals to Vz value. As the circuit provides a constant O/P voltage nearly equal to Vz if input voltage & Load RL changes so it is called as Zener diode shunt voltage regulator.

PROCEDURE:-

1) Study the circuit provided on the front panel of the kit.
2) Connect Saner diode ZD1 of Vz = 6.2V in the circuit.
3) Connect all the voltmeters at the input Vi & O/P Vo side.
4) Connect all the milliammeter at their respective places (Optional or short the terminals by patch cords)
5) Switch ON the Power Supply.
6) Keep Load Resistance RL constant vary the input voltage Vi from 0 to 12V in steps & note down the corresponding O/P voltage Vo for each step.
7) Now, keep I/P voltages vi constant at the voltage greater than Vz value say Vi = 10V constant, vary the Load RL & note O/P voltage V0 in steps.
8) Keep input Vi = 10v, note RL & R value using digital multimeter.
9) Find current I, I_L & Iz from equation (1), (4) & (5) respectively.
10) Repeat the above steps for the other Diode.

OBSERVATION TABLE:-

R = ---------Ω, Vz_1 = ---------V, Vz_2 = ---------V.

1). Line Regulation with RL constant:-

RL = ___________ KΩ

Unregulated I/P (Vi)	Regulated (Vo)

2). **Load Regulation with Input Vi constant:-**

Vi = ______________V.

Load Resistance RL	Regulated O/P Vo
Min.	
Med.	
Max.	

Repeat the above for the other Zener Diode

<u>RESULT</u>:-

The Circuit provides a constant O/P voltage for Zener DiodeZD1, Vo = _________V. If the I/P voltage Vi is changes from ______ V to ______V & if RL changes from _____ to ______KΩ.

EXPERIMENT NO: 8

OBJECTIVE:-

Study to design IC741 as a Wein Bridge Oscillator.

EQUIPMENTS:-

1. OPT-1 Trainer Kit
2. Power Supply
3. Patch cords
4. DSO and Function Generator.
5. Multimeter.

THEORY:-

Wein bridge oscillator in which the wein bridge circuit is connected between the amplifiers input terminals and output terminal. The bridge has a series RC network in one arm and a parallel RC network in the adjoining arm. I the remaining two arms of the bridge resistors R1 and Rf are connected. See the practical figure1.

The phase angle criterion for oscillation is that the total phase shift around the circuit must be 0^0.This condition occurs only when the bridge is balanced.i.e.At resonance. The frequency of oscillation fo is exactly the resonant frequency of the balanced Wien bridge and is given by

$$f_o = \frac{1}{2\pi \times R \times C}$$

$$f_o = 0.159/RC$$

Assuming that the resistors are equal in value and capacitors are equal in value in the reactive leg of the Wien bridge. At this frequency the gain required for sustained oscillation is given by

$$A_v = 1/B = 3$$

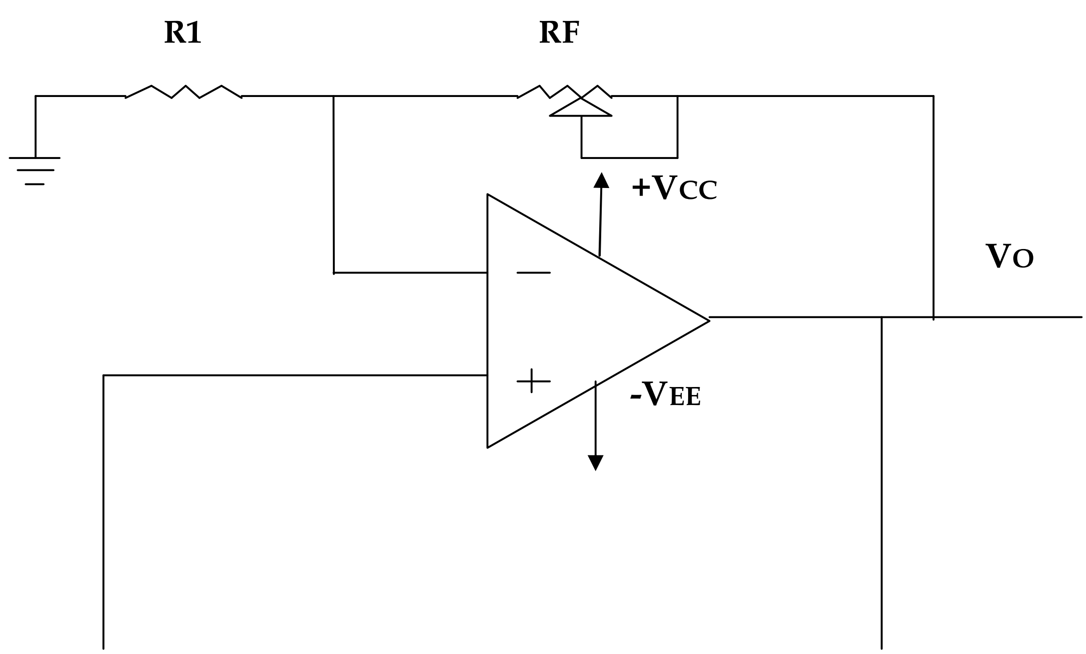
$$1 = R_f/R_1 = 3$$

Or

$$R_F = 2R_1$$

Wien bridge oscillator is mostly used as audio frequency oscillator.

PROCEDURE:-



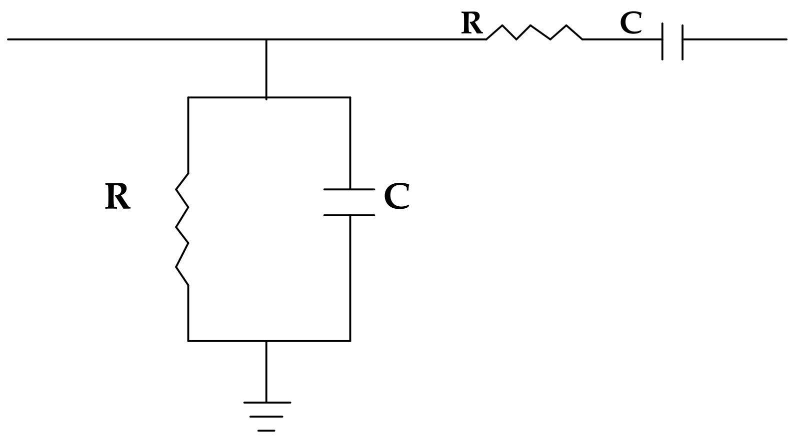


Figure 1: Wien Bridge Oscillator Using OP-AMP

1. Make the connections on OPT-01 board as shown in practical figure 1 as per your design.
2. Apply +ve 15V to pin 7 and –ve 15V to pin 4 of op-amp.
3. Switch on power supply.
4. Observe the waveform on oscilloscope.
5. Adjust the RF to required value to get the proper output.
6. Measure the frequency and amplitude output signal.

DESIGN: - Consider

R=1KΩ resistor

C=0.01μf capacitor

R1=10KΩ resistor

RF = must be at least 20 K.(here put RF =18.03K)

Do the calculations using formulas.

OBSERVATION TABLE:-

Sr.No.	Descriptions	Calculated	Observed (from

			waveform 1)
1	**Frequency**	**15.9KHZ**	**11.65KHZ**

WORK ASSIGNMENT:-

For Fo = 500hz find R=? When C=0.01µf capacitor, R1=10KΩ resistor. Observe the output of circuit.

DESIGN FORMULAS

1. Oscillator frequency is given by

$$fo = \frac{1}{2\pi \times R \times C}$$

$$fo = 0.159/RC$$

2. Gain requirement: - RF=2R1 minimum.

CONCLUSION:-

We studied the design of Wien bridge oscillator circuit using op-amp. Calculated and observed readings are almost same.

VIVA QUESTIONS

Q1. What is Oscillator?

Q2. Which type of feedback is provided in Oscillators?

Q3. What is balance condition?

Q4. What is lead – lag compensation circuit?

Q5. What is frequency of oscillation for Wien bridge oscillator?
