NON-LINEAR ELECTRONICS

LAB MANUAL

GEL 472

Roy Samia

September 2025

TABLE OF CONTENT

1 EXPERIMENT1 – INVERTING CIRCUIT:

To study the voltage shunt feedback in operational amplifier (Inverting mode Op-Amp).

Aim: To obtain the closed loop voltage gain

Materials: Op-Amp 741, breadboard, regulated power supply ±15 V, resistors 100 K and 10 K, connecting wires, hookup wire, oscillator, multimeter, digital voltmeter (DVM)

Formula: Closed loop voltage gain $A_v = -\frac{R_f}{R_1}$

Where R_f is Feedback Resistor and

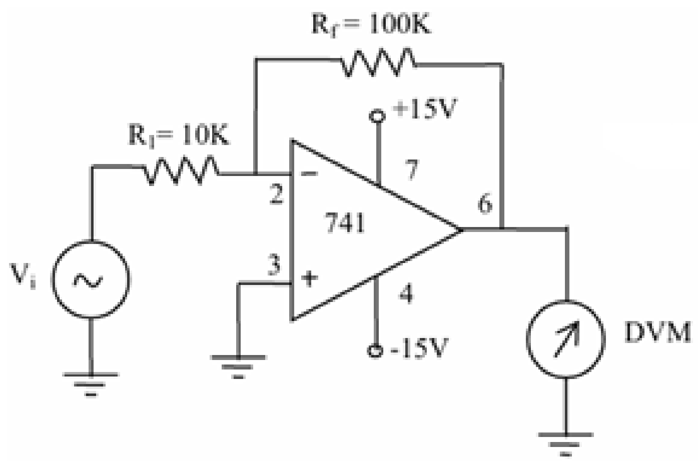
R_1 is the input series resistor

Procedure: This experiment can be performed in two ways.

(a) Either by changing the values of feedback resistor (R_f) and input resistor (R_1) and keeping input signal voltage fixed or

(b) by keeping the R_f and R_1 values fixed and changing amplitude of input signal.

In the first case—Make the connections as shown in below figure by selecting proper values of feedback and input resistors



Connect the dual regulated power supply having ±15 V as shown in above figure.

Connect 10 K potentiometer between pin no. 1 and 5 (of Op-Amp 741) with its extreme ends and variable terminal of potentiometer to –Vcc.

Connect pin no. 2 and 3 (of Op-Amp 741) to the ground, and adjust the potentiometer resistance so that the output voltage should be zero. This arrangement is called offset null. Keep this position as it is throughout the experiment. Do not change the resistance of the potentiometer.

The input signal source Vi may be either DC or AC signal. Apply fixed input signal Vi from the Function Generator to the pin no. 2 of Op-Amp 741 through resistor R_1 .

The pin no. 3 should be connected to ground terminal.

Preferably make the connections on breadboard so that the circuit can be assembled or de-assembled as per the requirement.

Select proper values of feedback resistor such as R_f = 100 K and R_1 =10K.

Switch on power supply and apply a fixed input signal V_i =0.1 Vdc

Note down the output voltage V_o by DVM in observation Table1.

For various values of R_f and R_1 calculate theoretical and experimental gain and verify

Sr. No	Resistors	Input voltage (V)	Output voltage (V)	Voltage gain $A_V = -\frac{V_0}{V_i}$	Theoretical Voltage gain $A_V = -\frac{R_f}{R_1}$
1	R_f = 100 K R_1 = 10 K				−10
2	R_f = 470 K R_1 = 47 K				−10
3	R_f = R_1 =				
4	R_f = R_1 =				

Table1

Second case—Make the connections as shown in the previous figure.

Connect R_f = 100 K and R_1 = 10 K.

Switch on the power supply.

Apply signal of fixed frequency or DC signal to the input terminal

Measure the corresponding output voltage by varying the amplitude of the input signal.

Note down the observations in Table 2.

Calculate the voltage gain A_v, and compare the theoretical gain with experimental gain.

Sr. No.	Input voltage V_i (V)	Output voltage V_0 (V)	Voltage gain $A_V = -\frac{V_0}{V_i}$
1	0.1		
2	0.2		
3	0.3		
4	0.4		
5	0.5		
6	0.6		
7	0.7		
8	0.8		
9	0.9		

Table2

Results and Discussions:

Theoretical and experimental gain were compared and were found to be identical. The output voltage is found to be negative which shows that there is 180° phase shift between input and output voltage

Precautions:

- Dual power supply must be regulated.
- Apply very small input voltage so that the Op-Amp should not be saturated, because of the restrictions of amplifier that the amplified output should not go beyond supply voltage.

- Measure all voltages with respect to ground.
- Use high input impedance voltmeter for voltage measurement to avoid the loading.
- Use proper values of Rf and R1.

Software Implementation:

Draw the circuit on Multisim CAD and apply the same steps conducted in above experimentation and fill the information as per table1 and Table2.

VIVA Questions:

1. What is called differential amplifier?
2. What are the typical characteristics of differential amplifier?
3. Why there is a need of two power supplies in differential amplifier?
4. What do you mean by single-ended and double-ended differential amplifier?
5. What are the properties of ideal Op-Amp?
6. List few applications of Op-Amp?
7. What do you mean by inverting and non-inverting mode of Op-Amp?
8. What is the gain of operational amplifier in inverting and non-inverting mode?

2 EXPERIMENT2 – NON INVERTING CIRCUIT:

To study the voltage series feedback in operational amplifier (Non-inverting mode Op-Amp)

Aim: To obtain the closed loop voltage gain

Materials: Op-Amp 741, breadboard, regulated power supply ±15 V, resistors 100 K and 10 K, connecting wires, hookup wire, oscillator, multimeter, digital voltmeter (DVM)

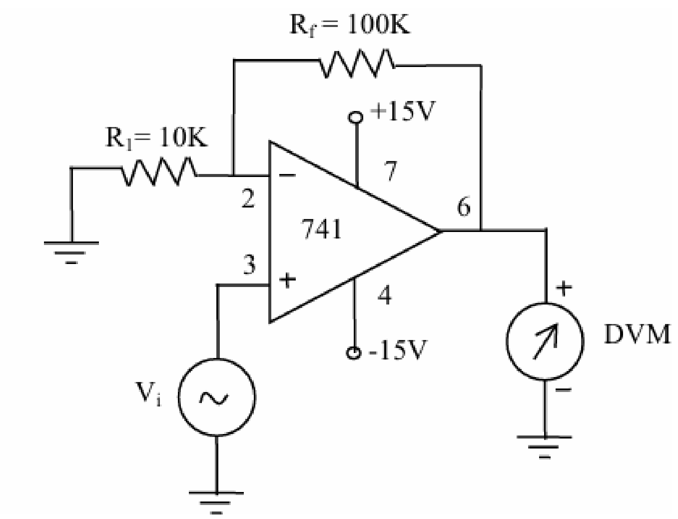
Formula: Closed loop voltage gain $A_v = 1 + \frac{R_f}{R_1}$

Where R_f is Feedback Resistor and

R_1 is the input series resistor

Procedure: The procedure of this experiment is same as that of inverting amplifier experiment no. 1. Instead of applying signal input to pin no. 2 of 741 Op-Amp, it is applied to pin no. 3. This experiment can also be performed in two ways as described in inverting amplifier experiment.

The observation table is also similar, only theoretical voltage gain $A_v = 1 + \frac{R_f}{R_1}$ should be mentioned, and accordingly the results should be written



Results and Discussions:

Theoretical and experimental gains were compared and were found to be identical. The output voltage is found to be positive which shows that there is no phase shift between input and output voltage. The voltage gain is always greater than one

Precautions:

- Dual power supply must be regulated.
- Apply very small input voltage so that the Op-Amp should not be saturated, because of the restrictions of amplifier that the amplified output should not go beyond supply voltage.
- Measure all voltages with respect to ground.
- Use high input impedance voltmeter for voltage measurement to avoid the loading.
- Use proper values of Rf and R1.

Software Implementation:

Draw the circuit on Multisim CAD and apply the same steps conducted in above experimentation and fill the information.

VIVA Questions:

1. What is the gain of differential amplifier in non-inverting mode?
2. On which factors the gain of Op-Amp depends?
3. Define input offset voltage and output offset voltage?
4. What is CMRR?
5. Define slew rate?
6. What will be the effect of slew rate on any application?
7. What is PSRR?
8. What do you know about temperature drift?

3 EXPERIMENT3 – ADDER / SUBTRACTOR CIRCUITS:

To study operational amplifier as adder/subtractor

Aim: To add the given voltages and verify the result

To subtract the given voltages and verify the result

Materials: Op-Amp 741, breadboard, regulated power supply ±15 V, resistors 100 K, 10 K, 1K Potentiometer, connecting wires, hookup wire, Function generator, multimeter, digital voltmeter (DVM), Oscilloscope.

Formula: For adder, output voltage $V_0 = -(\frac{R_f}{R_a} \times V_1 + \frac{R_f}{R_b} \times V_2 + \frac{R_f}{R_c} \times V_3)$

If $R_a = R_b = R_c = R_f = R$ then $V_o = -(V_1 + V_2 + V_3)$

Where V_1 , V_2 and V_3 are the input voltages to be added, and V_o is the output voltage

For subtractor, output voltage $V_0 = -(V_2 - V_1)$

Procedure: Adder or summing and subtractor or difference inverting amplifier. Make the connections as shown in below figures on breadboard. Use feedback resistor R_f and input resistors $R_a = R_b = R_c = R_f = R = 10K$. The input voltages V_1 , V_2 and V_3 should be made variable using function generator and/or using voltage divider arrangement. Switch on the power supply, apply input voltage to be added/subtracted (as the case may be), and note the corresponding output voltage using DVM/oscilloscope. Record the observations in observations table1 and table2

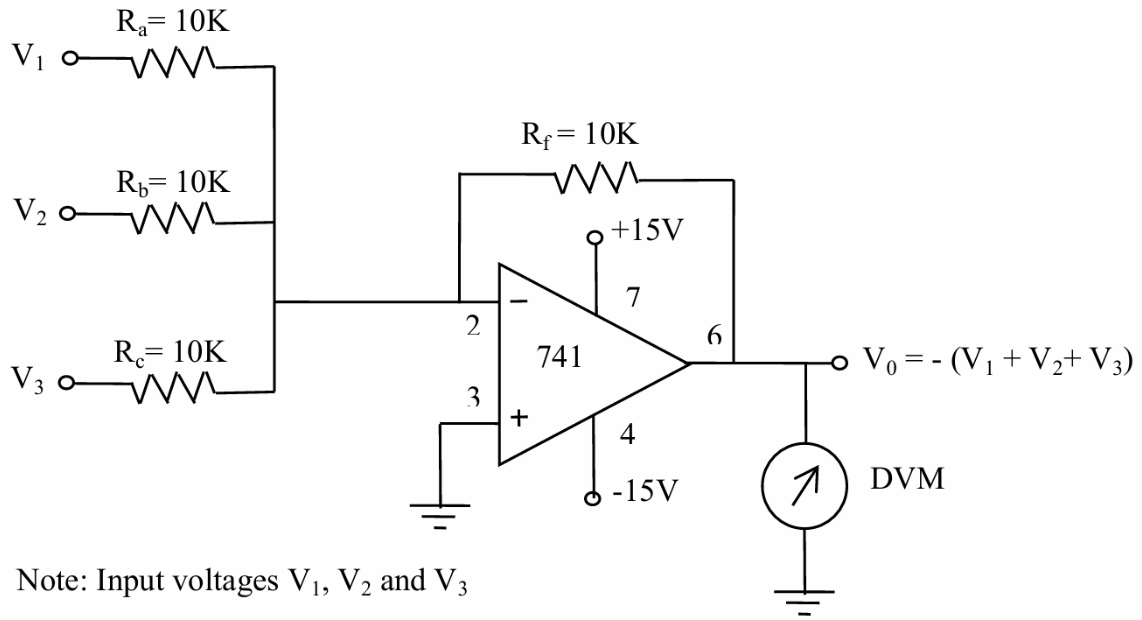


Figure1 – Adder/Summing amplifier Circuit

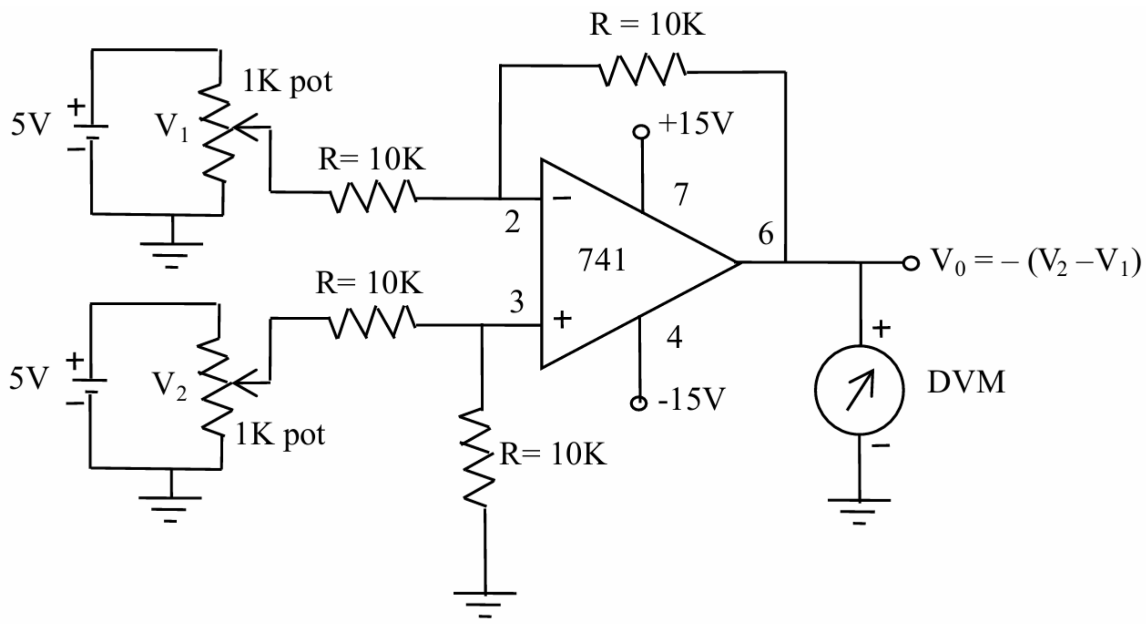


Figure2 – Subtractor/Difference amplifier Circuit

Sr. No.	Input voltages to be added (volts)			Output voltage (V_0) volts $V_0 = (V_1 + V_2 + V_3)$	Measured output voltage (V_0) V
	V_1	V_2	V_3		
1					
2					
3					
4					
5					
6					

Table1 – For adder amplifier

Sr. No.	Input voltages to be subtracted (volts) Apply negative voltage V_1 (i.e. $-V_1$)		Output voltage (V_0) volts $V_0 = (V_2 - V_1)$	Measured output voltage (V_0) V
	$-V_1$	V_2		
1				
2				
3				

Table2 – For Subtraction or difference amplifier

Results and Discussions:

In case of adder and subtractor, the measured output voltage is found to be identical as that of output voltage obtained from formula of addition or subtraction. Hence the result is verified

Precautions:

- Dual power supply must be regulated.
- Apply very small input voltage so that the Op-Amp should not be saturated, because of the restrictions of amplifier that the amplified output should not go beyond supply voltage.
- Measure all voltages with respect to ground.

- Use high input impedance voltmeter for voltage measurement to avoid the loading.
- Use proper values of Rf, Ra, Rb, Rc.

Software Implementation:

Draw the circuit on Multisim CAD and apply the same steps conducted in above experimentation and fill the information as per table1 and Table2.

VIVA Questions:

1. Why CMRR should be large in differential amplifier?
2. What do you mean by differential mode gain?
3. Give examples of linear circuits of operational amplifier?
4. Explain adder, summing amplifier and scalar?
5. Define input bias current?

4 EXPERIMENT4 – INTEGRATOR CIRCUIT:

Application of Op-Amp 741 as integrator

Aim: To study the Op-Amp as integrator

To observe the output waveform when square wave signal is applied at input

Observe the change in output voltage when variable frequency signal is applied at the input

Materials: Op-Amp 741, breadboard, regulated power supply ±15 V, resistors capacitors, connecting wires, hookup wire, Function generator, multimeter, digital voltmeter (DVM), Oscilloscope.

Formula: Output voltage $V_0 = \int \frac{1}{R \times C} V_i \; dt$

where V_i is the input signal voltage, R and C For subtractor, are the components connected in circuit as resistor and capacitor, selected as per $T \leq R \times C$ required condition for integration

If V_i is a sinusoidal wave as $V_i = V \sin \omega t$

Then $V_0 = \int \frac{1}{R \times C} V \sin \omega t \, dt = \frac{1}{R \times C \times \omega} V \cos \omega t$

This Equation shows that the amplitude of the output signal is inversely proportional to the frequency of the input signal

The operating frequency $f_a = \frac{1}{2\pi \times R_f \times C_f}$

The 0dB frequency $f_b = \frac{1}{2\pi \times R_1 \times C_f}$

Procedure: Make the connections as shown in below Figure1 on bread board. Select proper values of R_f, R_1 and C_f so that $T \leq R_1 \times C_f$ is satisfied.

By choosing proper value of C_f, calculate value of resistor R_1 from $f_b = \frac{1}{2\pi \times R_1 \times C_f}$, R_f is the resistor having high value, which provides feedback. f_b is the frequency below which the circuit can work as practical integrator (Figure2).

Connect the function generator at the input, and measure the amplitude of output signal keeping input signal amplitude constant for each frequency. Note down the output voltage for corresponding input frequency.

Draw the graph between output voltage against frequency. It should be observed that the amplitude of the output signal decreases with increasing frequency.

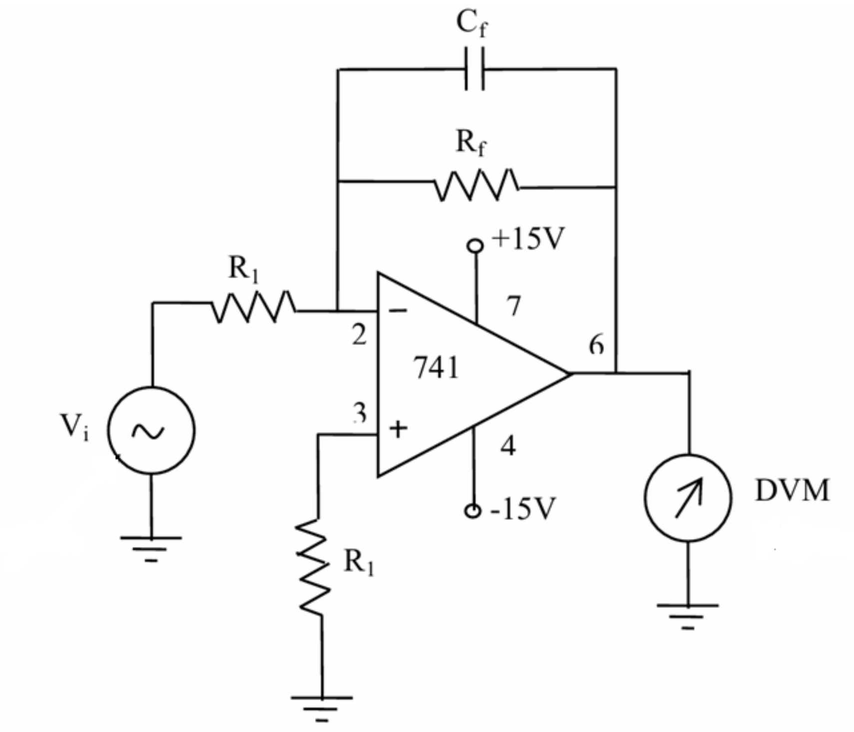


Figure1 – Integrator Circuit

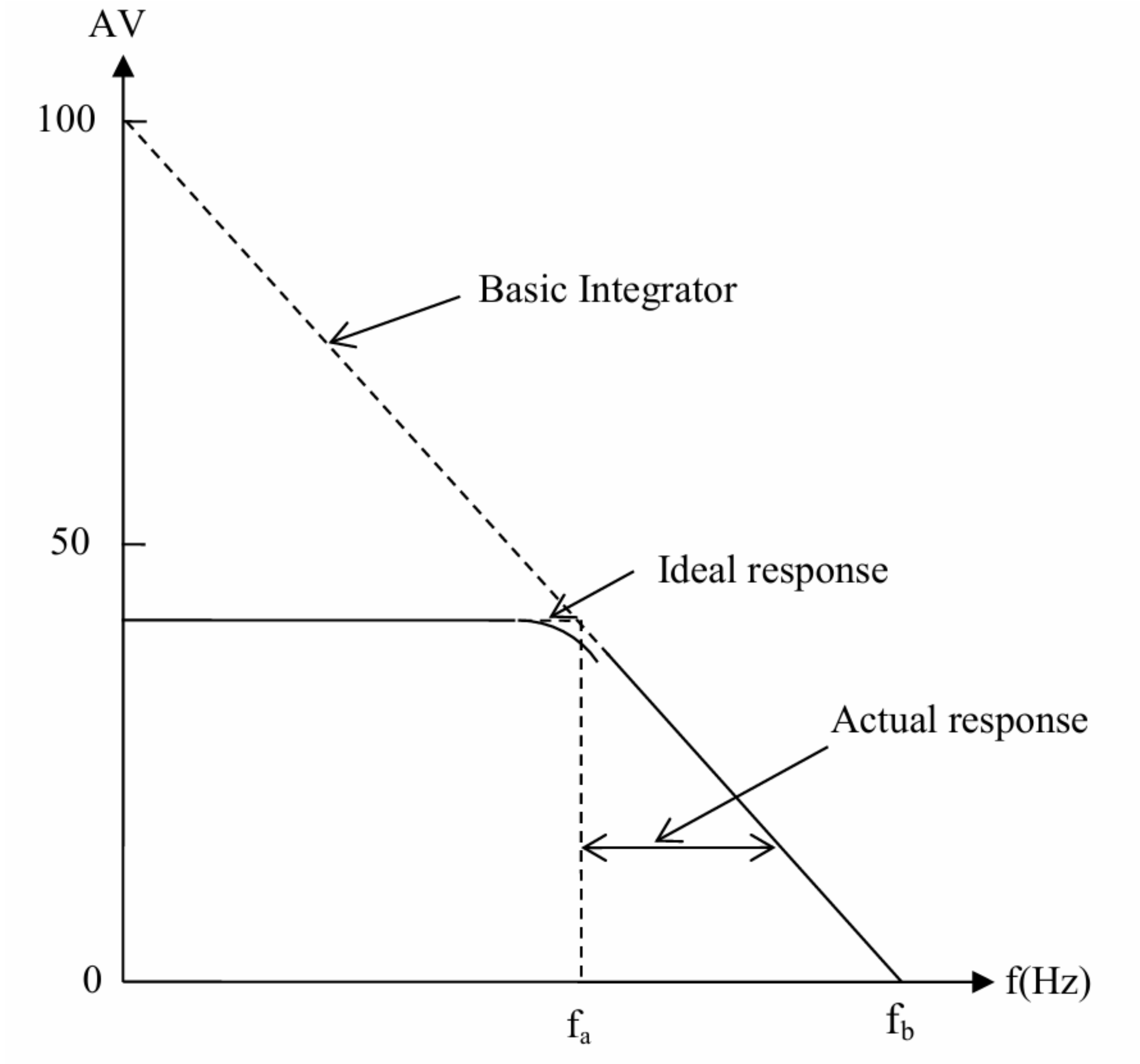


Figure2 – Frequency response of practical integrator

Observation Table:

$$R_1 = \_\_\_\_\_\_\_\_K, \quad C = \_\_\_\_\_\_\_\_\_uF, \quad V_i = \text{constant}$$

Sr. No.	Frequency of input signal (Hz)	Output voltage (V_0) (V)
1		
2		
3		
4		
5		
6		
-		
-		
-		
-		

Results and Discussions:

From the graph of output voltage against frequency it is observed that the output voltage decreases with increasing frequency. The square waveform is applied at the input, and output is observed which is ramp up and down, i.e. it generates the triangular wave. This shows that the circuit works as an integrator

Precautions:

- Dual power supply must be regulated.
- Apply very small input voltage so that the Op-Amp should not be saturated, because of the restrictions of amplifier that the amplified output should not go beyond supply voltage.
- Measure all voltages with respect to ground.
- Use high input impedance voltmeter for voltage measurement to avoid the loading.
- Selection of proper values of R_f, R_1 and C_f is necessary to satisfy $T \leq R_1 \times C_f$, time period of the input cycle

Software Implementation:

Draw the circuit on Multisim CAD and apply the same steps conducted in above experimentation and fill the information as per table1.

VIVA Questions:

1. How integrator circuit works?
2. Explain action of capacitor connected in feedback in integrator circuit.
3. What will be the output of integrator when sinusoidal wave is applied at input?
4. What are the characteristics of ideal operational amplifier?
5. Why is capacitor shunted by resistor in feedback?
6. Define slew rate?
7. Whether slew rate plays any role in integration? Explain

5 EXPERIMENT5 – DIFFERENTIATOR CIRCUIT:

Application of Op-Amp 741 as differentiator

Aim: To study the Op-Amp as differentiator

Observe the change in output voltage when variable frequency signal is applied at the input

Materials: Op-Amp 741, breadboard, regulated power supply ±15 V, resistors capacitors, connecting wires, hookup wire, Function generator, multimeter, digital voltmeter (DVM), Oscilloscope.

Formula: Output voltage $V_o = -R_f \times C_1 \times \frac{dV_i}{dt}$

where V_i is the input signal voltage, R_f is the resistance connected and C_1 is the capacitor connected in Input. The negative sign indicates phase change of 180° between input and output signal. If a sinusoidal signal is applied at the input of the differentiator say

$$V_i = V \times \sin \omega t$$

The output will be $V_o = -R_f \times C_1 \times V \times \omega \times \cos \omega t$

This Equation shows that the amplitude of the output signal is proportional to the frequency of the input signal

At low frequencies the capacitive reactance is high resulting in low gain or low output of the differentiator, whereas at high frequencies the output voltage increases with frequencies of input signal.

Procedure: Make the connections as shown in below Figure1 on bread board. Select proper of feedback resistor and capacitor as per the frequency response of the differentiator so that time period of the input signal $T \geq R_f \times C_1$ is satisfied.

Calculate values of R_f and C_1 from $f_a = \frac{1}{2\pi \times R_f \times C_1}$ and $f_b = \frac{1}{2\pi \times R_1 \times C_1}$.

$f_b = 20 \times f_a$ and $C_1 < 1\mu F$. Satisfying $R_1 \times C_1 = R_f \times C_f$, evaluate values of R_1 and C_f

f_a is the frequency at which gain of the differentiator is 0 dB

f_b is the gain limiting frequency

The components R_1 and C_f are used for stability of the circuit and high-frequency noise correction. By selecting value of f_a as highest frequency to be differentiated and assuming $C_1 > 1\mu F$ calculate value of R_1 and C_f

Connect these components in circuit as shown below, and switch on the power supply. Apply sinusoidal input to the differentiator through the function generator. Keeping amplitude of the input sine wave constant, change the frequency, and note the corresponding output voltage. Record the readings in observation below Table and plot the graph between voltage gains against frequency. It should be observed that the output voltage increases with increasing frequency of the input signal.

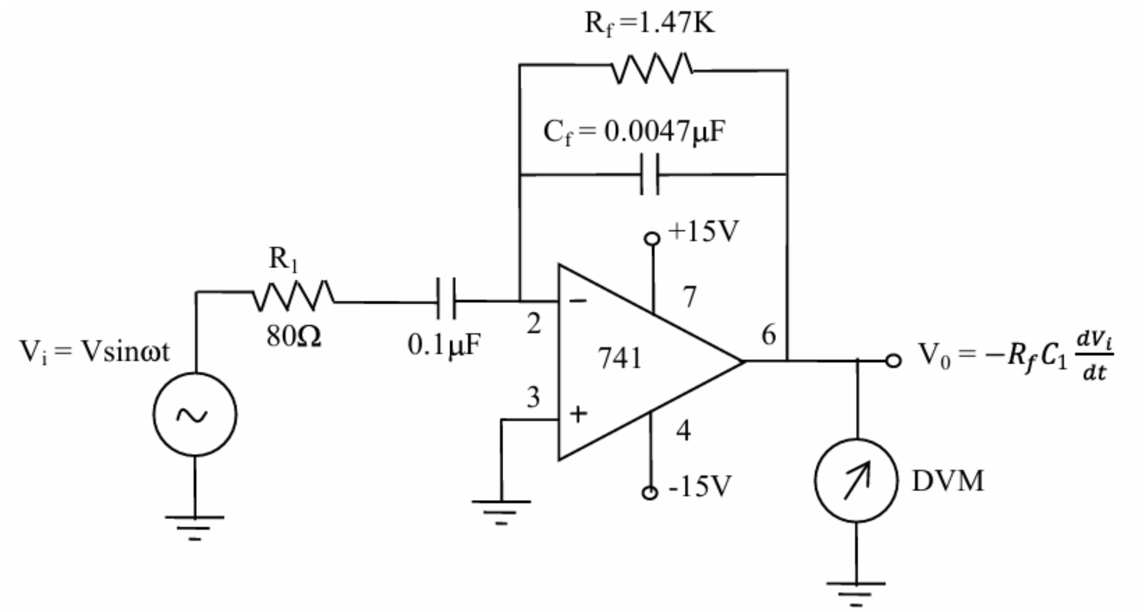


Figure1 – Differentiator Circuit

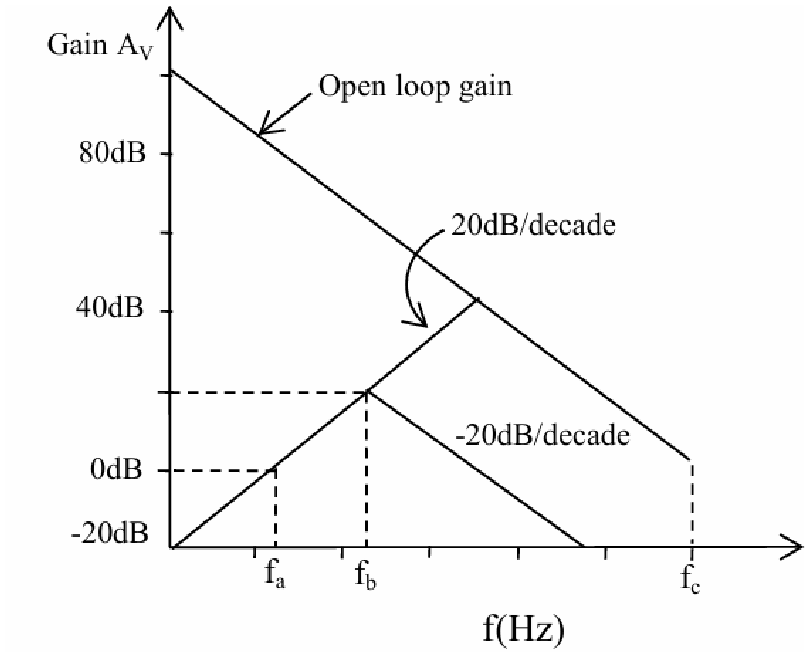


Figure2 – Frequency response of practical differentiator

Observation Table:

R_1 = _______K, C_1 = _____uF, R_f = _______K, C_f = ______uF

Sr. No.	Frequency of input signal (Hz)	Output voltage (V_0) (V)
1		
2		
3		
4		
5		
6		
7		
-		
-		
-		
-		

Results and Discussions:

The frequency f_a (at which gain of the differentiator is 0 dB) and f_b (the gain limiting frequency) is calculated. The plot of gain against frequency of input signal is drawn which is linear and gain increases with frequency of input signal. Various waveforms such as sinusoidal, square and triangular are feed at the input of the differentiator, and output waveforms such as cosine, spikes and square are drawn

Precautions:

- Dual power supply must be regulated.
- Apply very small input voltage so that the Op-Amp should not be saturated, because of the restrictions of amplifier that the amplified output should not go beyond supply voltage.
- Measure all voltages with respect to ground.
- Use high input impedance voltmeter for voltage measurement to avoid the loading.
- Selection of proper values of R_f and R_1

Software Implementation:

Draw the circuit on Multisim CAD and apply the same steps conducted in above experimentation and fill the information as per above table.

VIVA Questions:

1. Explain working of differentiator
2. $T \geq R_f \times C_1$ should be satisfied for working of differentiator, explain.
3. What will be the output of differentiator for sinusoidal waveform applied at input?
4. What are the different parameters of operational amplifier?
5. What is CMRR?

6 EXPERIMENT6 – ACTIVE FILTERS:

To design and study active filters using Op-Amp

Aim: First-order low pass Butterworth filter

Second-order low pass Butterworth filter

First-order high pass Butterworth filter

Second-order high pass Butterworth filter

Materials: Op-Amp 741, breadboard, regulated power supply ±15 V, resistors capacitors, connecting wires, hookup wire, Function generator, digital Multimeter (DMM), Oscilloscope.

Formula: The Higher cut-off frequency:

$$f_H = \frac{1}{2\pi RC}$$ first order low pass

$$f_H = \frac{1}{2\pi\sqrt{R_1\,R_3 C_1 C_2}}$$ Second order low pass

The Lower cut-off frequency:

$$f_L = \frac{1}{2\pi RC}$$ first order low pass

$$f_L = \frac{1}{2\pi\sqrt{R_1\,R_3 C_1 C_2}}$$ Second order low pass

Procedure: Initially design the filter circuit and calculate values of resistor R and capacitor C. Select the higher cut-off frequency f_H, value of capacitor C, and calculate the value of resistor R from the given formula in first-order low pass Butterworth filter.

Similarly select the cut-off frequency f_H for second-order low pass filter, select the proper values of C_1 and C_2 capacitors, and calculate the values of R_2 and R_3 from the given formula. Preferably choose the value of capacitor *C* less than 1 µF.

The gain of the amplifier can be adjusted by proper selection of feedback resistor R_F and R_1 as per our desired value of voltage gain $\left(1 + \frac{R_f}{R_1}\right)$.

Make the connections as shown in below figures on breadboard, and apply ± 15V power supply to Op-Amp 741. Connect function generator to the input and Oscilloscope at the output. Change the frequency of the function generator and measure the output voltage at corresponding frequency. Record the observations in observation table.

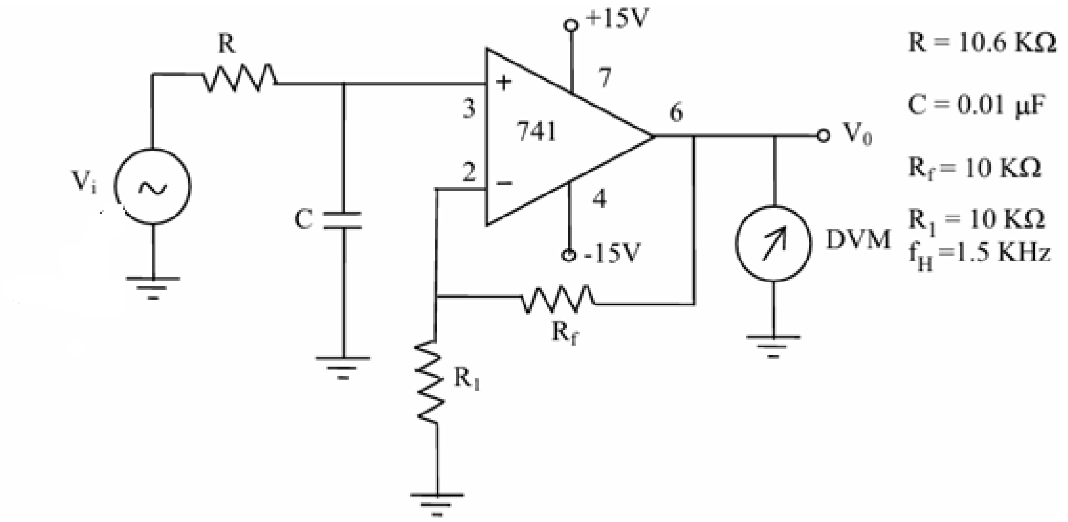


Figure1 – First-order low pass Butterworth filter

Observation Table:

$$V_i = \_\_\_\_\_\_\_\_\_ V$$

Sr. No.	Frequency (Hz)	Output voltage V_0 (V)	Voltage gain in dB	Cut-off frequency Hz from formula
1				
2				
3				
4				
5				
6				
7				
8				
9				

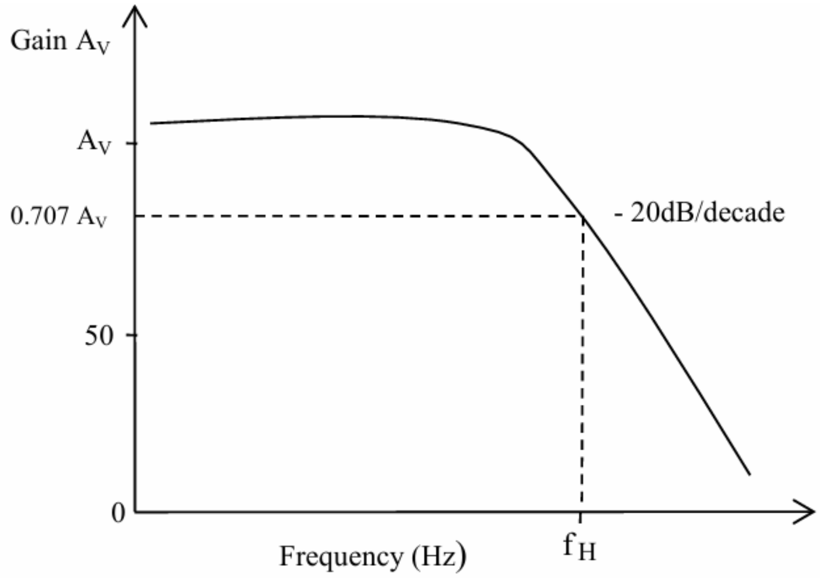


Figure2 – Frequency response of first-order low pass Butterworth filter

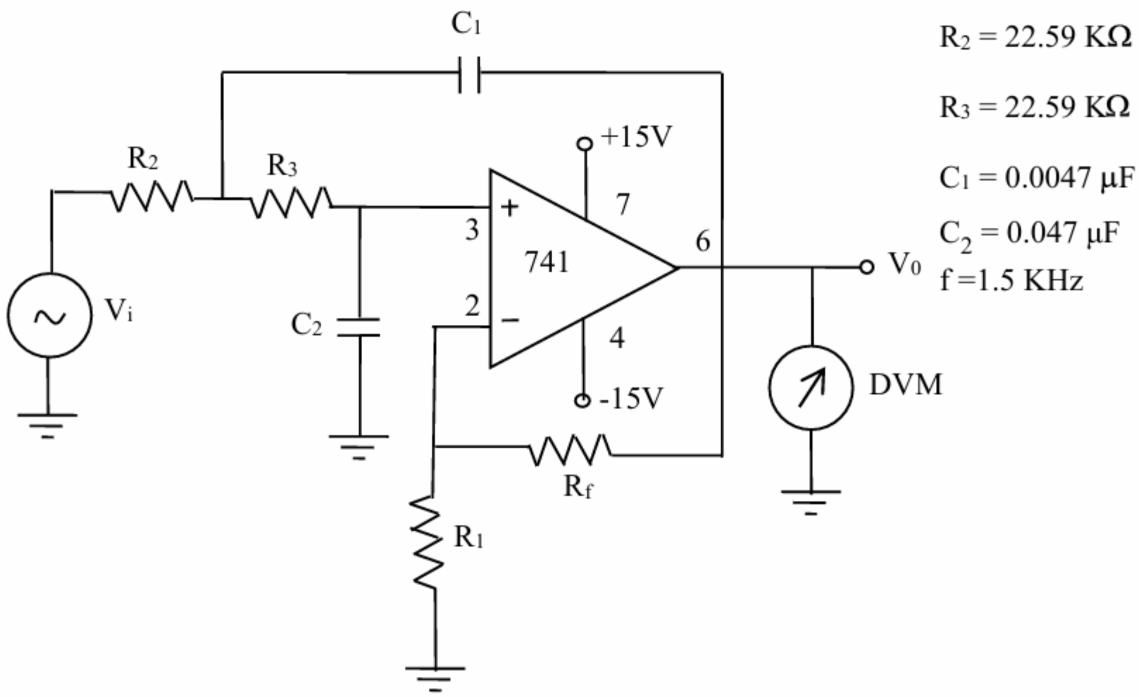


Figure3 – Second-order low pass Butterworth filter

Observation Table:

$$V_i = \_\_\_\_\_\_\_\_\_ V$$

Sr. No.	Frequency (Hz)	Output voltage V_0 (V)	Voltage gain in dB	Cut-off frequency Hz from formula
1				
2				
3				
4				
5				
6				
7				
8				
9				

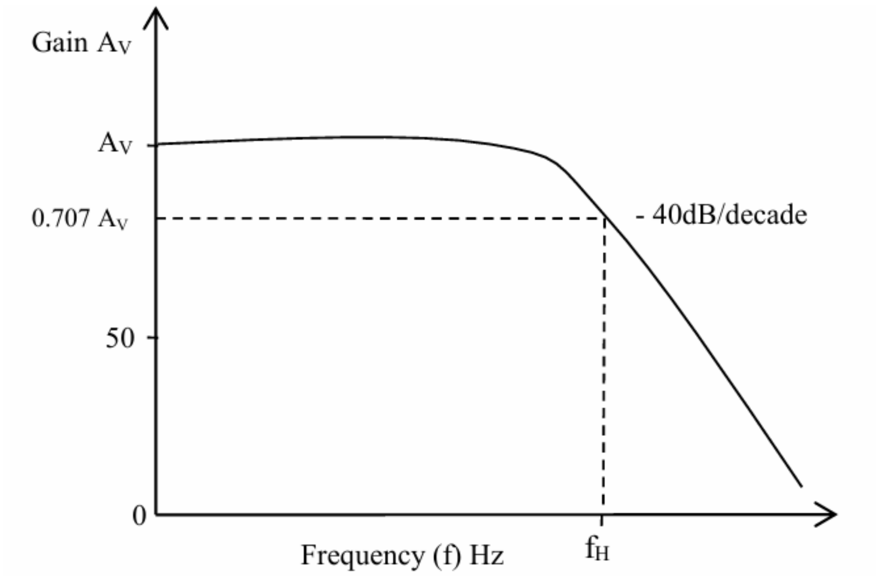


Figure4 – Frequency response of Second-order low pass Butterworth filter

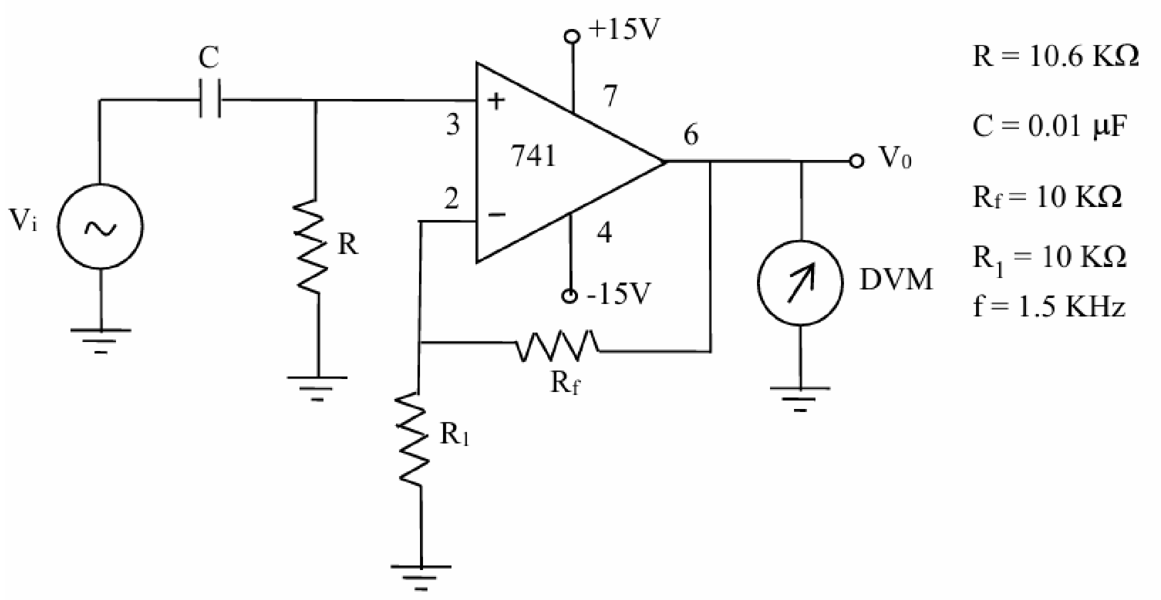


Figure5 – First-order High pass Butterworth filter

Observation Table:

$$V_i = \_\_\_\_\_\_\_\_\_\_ V$$

Sr. No.	Frequency (Hz)	Output voltage V_0 (V)	Voltage gain in dB	Cut-off frequency Hz from formula
1				
2				
3				
4				
5				
6				
7				
8				
9				

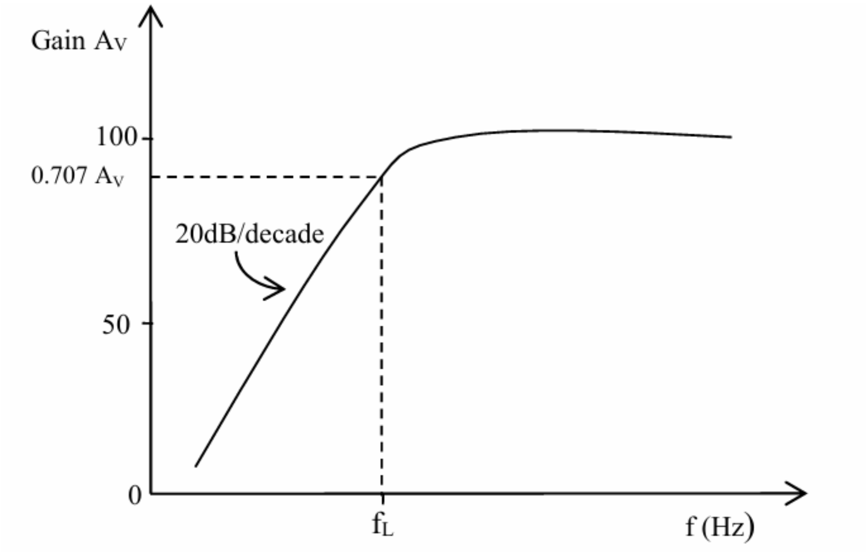


Figure6 – Frequency response of first-order high pass Butterworth filter

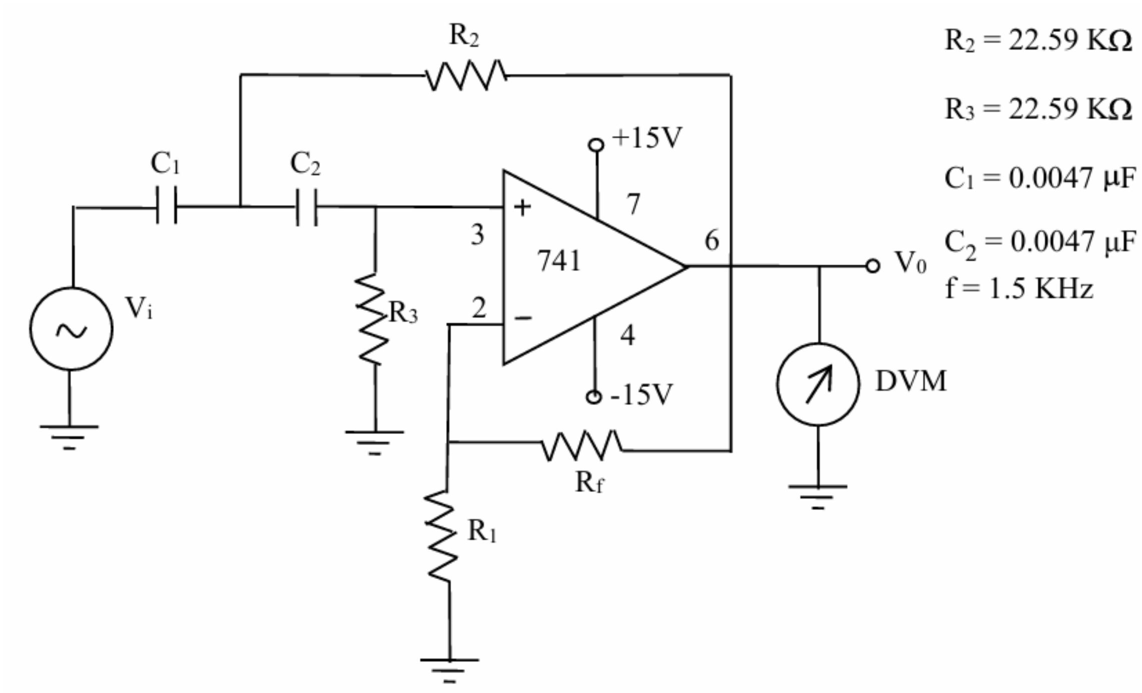


Figure7 – Second-order High pass Butterworth filter

Observation Table:

$$V_i = \_\_\_\_\_\_\_\_\_ V$$

Sr. No.	Frequency (Hz)	Output voltage V_0 (V)	Voltage gain in dB	Cut-off frequency Hz from formula
1				
2				
3				
4				
5				
6				
7				
8				
9				

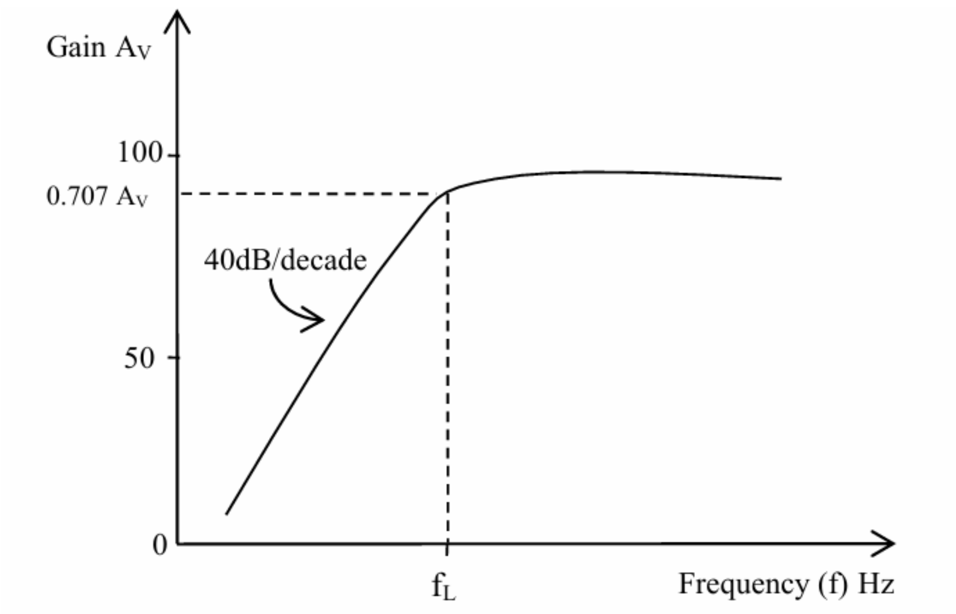


Figure8 – Frequency response of Second-order high pass Butterworth filter

Results and Discussions:

The graph is plotted between voltage gains against frequency of the input signal, and cut-off frequency is calculated from the graph and formula. Both are found to be identical.

Precautions:

- Dual power supply must be regulated.
- All connections should be clean and tight.
- Apply very small input voltage so that the Op-Amp should not be saturated, because of the restrictions of amplifier that the amplified output should not go beyond supply voltage.
- Do not use operational amplifier in open loop condition.
- Measure all voltages with respect to ground.
- Use high input impedance voltmeter for voltage measurement to avoid the loading.
- Selection of proper values of R_f and R_1

Software Implementation:

Draw the circuit on Multisim CAD and apply the same steps conducted in above experimentation and fill the information as per above table.

VIVA Questions:

1. What do you mean by active filter?
2. What is the difference between active and passive filters?
3. What do you mean by cut-off frequency?
4. What is roll-off?
5. How roll-off is important in active filters?
6. What is the difference between digital and analog filters?
7. What is first-order and second- order Butterworth filter?

7 EXPERIMENT7 – BAND PASS AND BANS STOP FILTERS:

To design and study active band pass and band stop or notch filter

Aim: Design of active band pass filter and study frequency response

Design of active band stop or notch filter and study the frequency response

Materials: Op-Amp 741, breadboard, regulated power supply ±15 V, resistors capacitors, connecting wires, hookup wire, Function generator, digital Multimeter (DMM), Oscilloscope.

Formula: Band Pass Filter:

$$f_C = \sqrt{f_H f_L} \quad \text{Center Frequency}$$

$$Q = \frac{f_C}{f_H f_L} \quad \text{Quality Factor}$$

Where f_C is called center frequency, f_H is higher cut-off and f_L is the lower cut-off frequency

BW = $f_H - f_L$ is the band Width

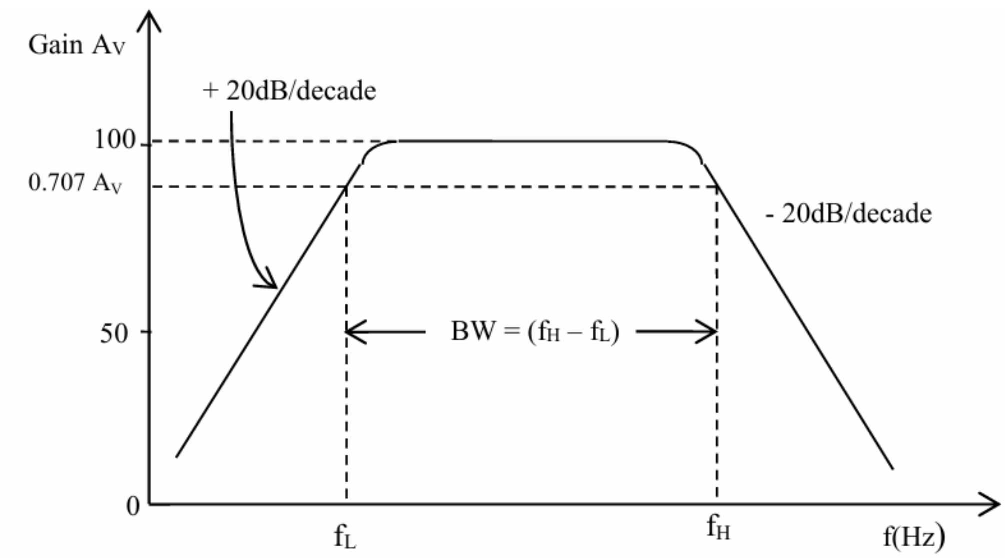


Figure1 – Frequency response of first order band pass filter

Band stop or Notch Filter:

The same formula for centre frequency and bandwidth is used for wide band stop filter, but for narrow band reject filter, the frequency at which maximum attenuation occurs is called notch frequency f_N, which is given by

$$f_N = \frac{1}{2\,\pi\,R\,C}$$

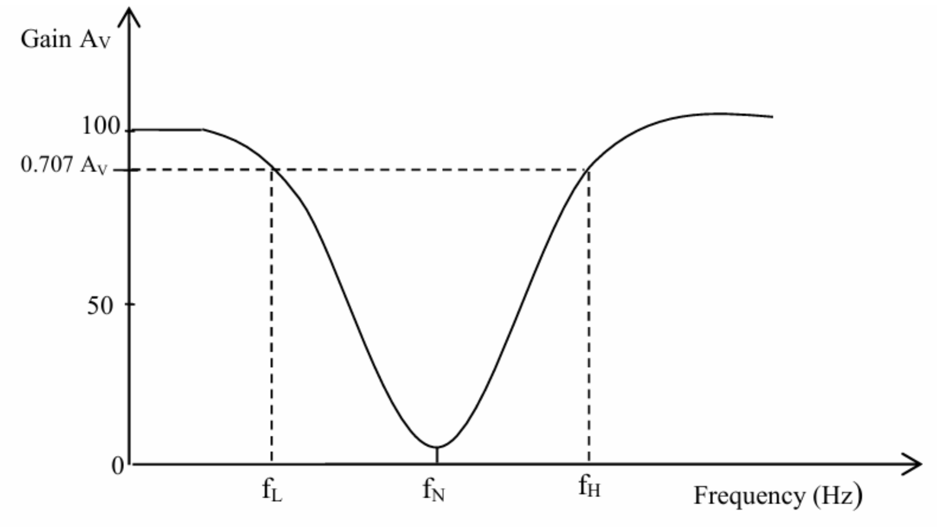


Figure2 – Frequency response of Notch filter

Procedure: By cascading high pass and low pass filters together a band pass filter can be formed.

Make the connection as shown in below figures for respective filters on breadboard. In case of band pass filter for obtaining ±20 dB band pass, cascade first-order high pass and first order low pass filters. To obtain ±40 dB band pass, cascade second-order high pass and second order low pass filters in series. Design filter circuit with proper component values of R and C

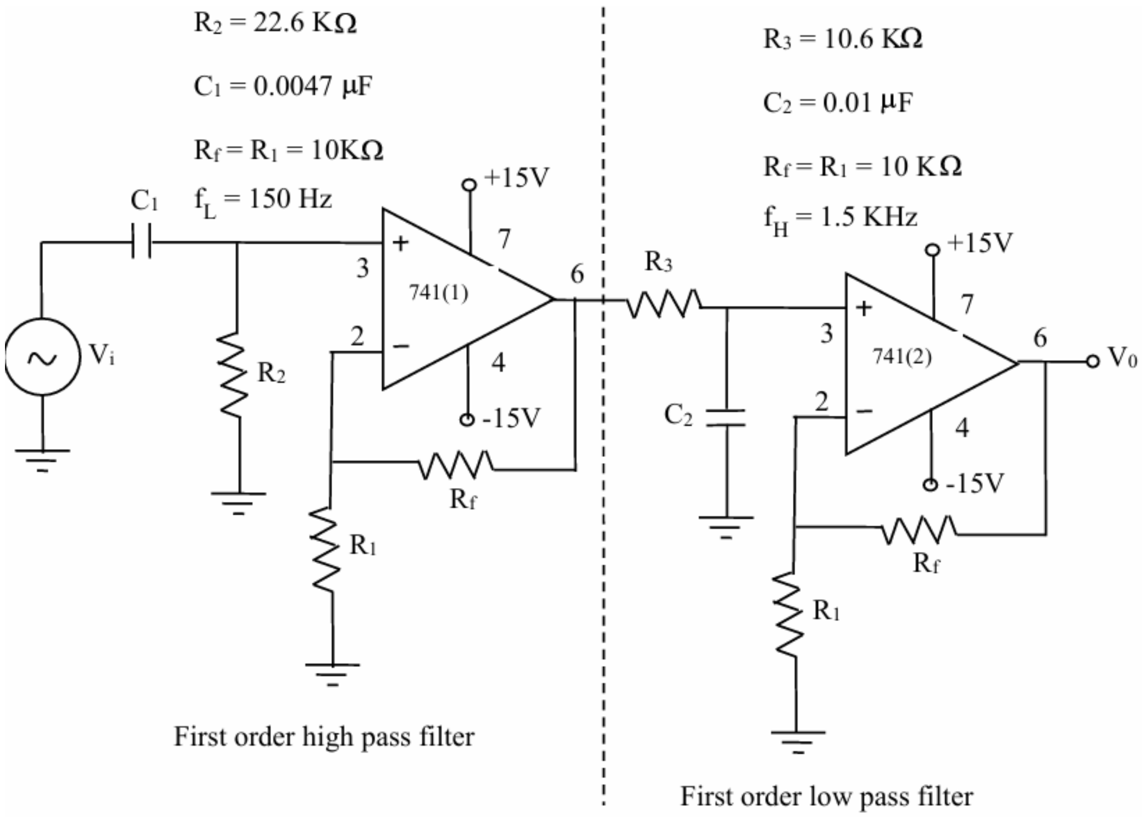


Figure3 – Band pass filter

Connect function generator at the input of the filter and DMM/Oscilloscope at the output. Change the frequency of the input signal keeping input amplitude constant for each frequency, and measure the corresponding output. Note down the readings in observations table. Calculate the gain of the circuit. Plot the graph between voltage gains

Observation Table:

$R =$ ________K, $C =$ ________uF, $V_i =$ ________ V

$f_H =$ _______KHz, $f_L =$ ________KHz, $BW =$ ________KHz

Sr. No.	Frequency of input signal (Hz)	Output voltage V_0 (V)	Voltage gain A_V (dB)
1			
2			
3			
4			
5			
6			
7			

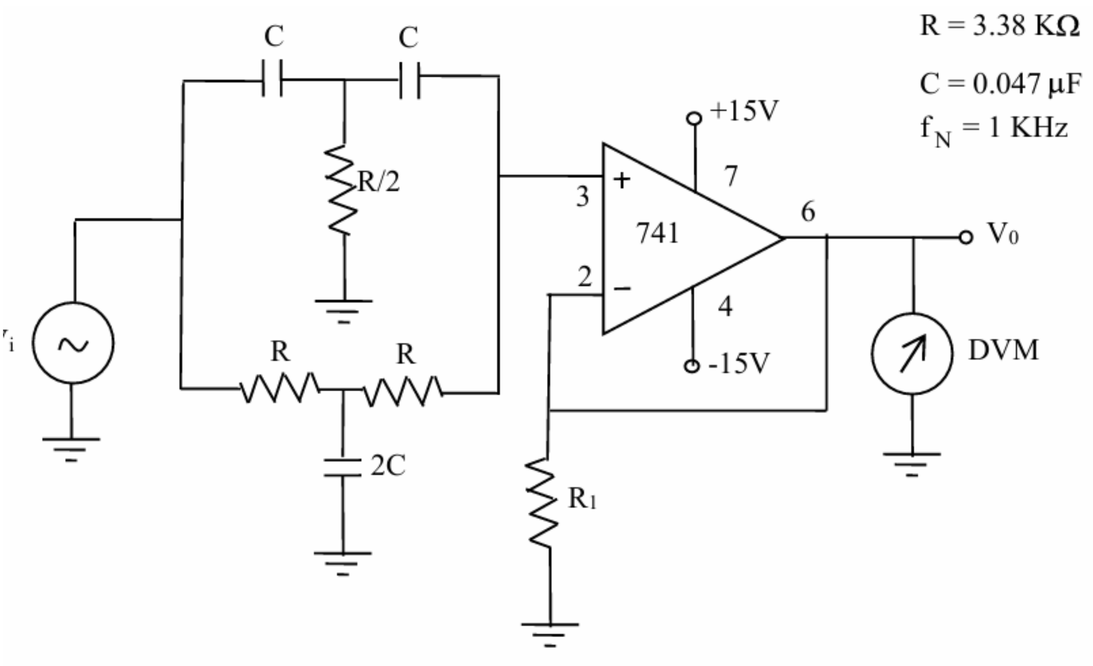


Figure4 – Band stop filter

Observation Table:

$$R = \_\_\_\_\_\_\_\_ \mathrm{K}, C = \_\_\_\_\_\_\_\_ \mathrm{uF}, V_i = \_\_\_\_\_\_\_\_ V$$

$$f_{\mathrm{H}} = \_\_\_\_\_\_\_ \mathrm{KHz}, f_L = \_\_\_\_\_\_\_\_ \mathrm{KHz}, BW = \_\_\_\_\_\_\_\_ \mathrm{KHz}$$

Sr. No.	Frequency of input signal (Hz)	Output voltage V_0 (V)	Voltage gain A_V (dB)
1			
2			
3			
4			
5			
6			
7			

Results and Discussions:

(a) Band pass filter:

The band pass filter is designed by considering higher cut-off (f_H) = _____ KHz and lower cut-off (f_L) = _______ KHz frequencies. Proper values of resistors and capacitors are connected in the filter circuit. The frequency response is studied, and f_H and f_L values are evaluated from the graph of voltage gain versus frequency plot. Also bandwidth is calculated as ($f_H - f_L$) which is found to be ________ KHz. The quality factor is found to be ________. It is found that the values of f_H and f_L match with the theoretical values evaluated from formula, which were predefined.

(b) Band stop or Twin-T notch filter

The pass band and the stop band of frequencies are determined. Also notch frequency is calculated from the graph of voltage gain versus frequency plot. The notch frequency (f_N) evaluated from the graph and the predefined f_N from formula are found to be identical

Precautions:

- Dual power supply must be regulated.
- All connections should be clean and tight.

- Apply very small input voltage so that the Op-Amp should not be saturated, because of the restrictions of amplifier that the amplified output should not go beyond supply voltage.
- Do not use operational amplifier in open loop condition.
- Measure all voltages with respect to ground.
- Use high input impedance voltmeter for voltage measurement to avoid the loading.
- Selection of proper values of R_f and R_1

Software Implementation:

Draw the circuit on Multisim CAD and apply the same steps conducted in above experimentation and fill the information as per above table.

VIVA Questions:

1. Define band pass filter and notch filter
2. Explain working of band pass and notch filter.
3. What are the applications of band pass and notch filters?
4. In what way these filters are playing important role in bass and treble circuits of audio system?

8 EXPERIMENT8 – VOLTAGE TO CURRENT & CURRENT TO VOLTAGE CONVERTERS:

Study of Op-Amp as voltage to current converter and current to voltage converter

Aim: (i) Voltage to current converter

(ii) Current to voltage converter

Materials: Op-Amp 741, breadboard, regulated power supply ±15 V, resistors capacitors, connecting wires, hookup wire, Function generator, digital Multimeter (DMM), Oscilloscope.

Formula: (i) Voltage to current converter:

$$I_{Out} = \frac{V_i}{R}$$

(ii) Current to voltage converter:

$$V_O = V_i \times R$$

Procedure: Make the connections as shown in below Figs. Switch on the power supply. Vary input in the circuits. Note down the input voltage and corresponding output current in case of voltage to current converter, and note down the current (I) and the output voltage in case of current to voltage converter. Record the observations in observation table. Plot the graph between voltage and current in case of V to I converter and current versus voltage in case of I to V converter

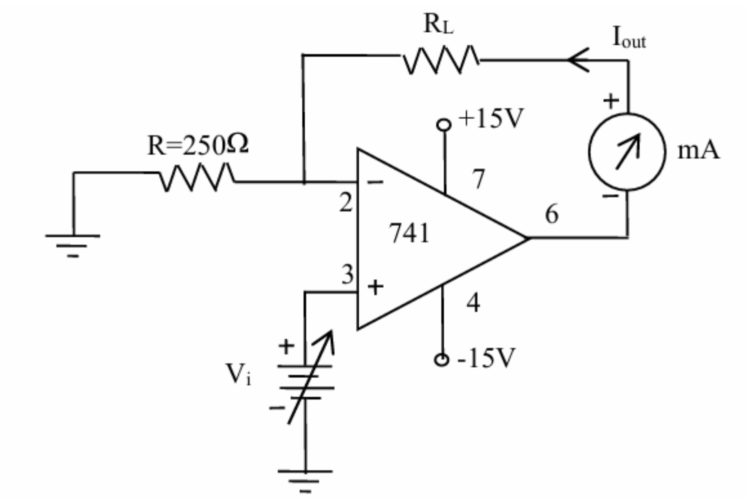


Figure1 – Voltage to current converter using Op-Amp

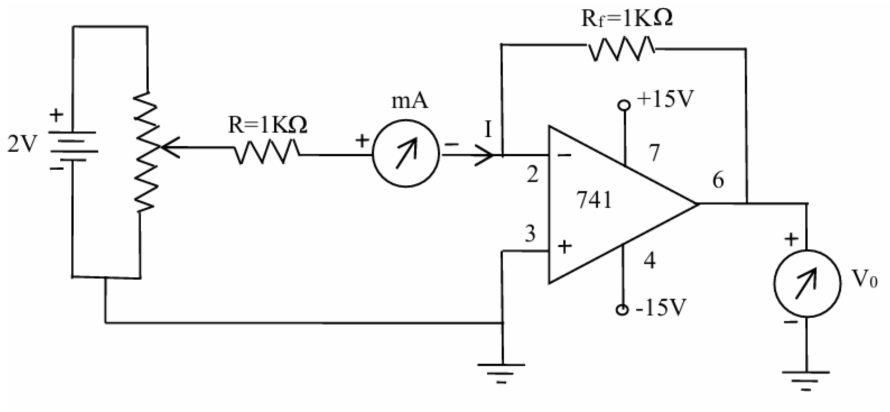


Figure2 – Current to voltage converter using 741 Op-Amp

Observation Table:

(a) Voltage to current converter

Sr. No.	Input voltage V_i (V)	Output current I (mA)
1		
2		
3		
4		
5		
6		

(b) Current to voltage converter

Sr. No.	Input current I (mA)	Output voltage V_0 (V)
1		
2		
3		
4		
5		
6		

Results and Discussions:

As we change the input voltage, the output current will also change, and it is directly proportional. The V-I plot is linear.

Similarly change in the input current will change output voltage, and I-V plot is linear. The input current is directly proportional to output voltage V_O

Precautions:

- Dual power supply must be regulated.
- All connections should be clean and tight.
- Apply very small input voltage so that the Op-Amp should not be saturated, because of the restrictions of amplifier that the amplified output should not go beyond supply voltage.
- Measure all voltages with respect to ground.
- Use high input impedance voltmeter for voltage measurement to avoid the loading.

Software Implementation:

Draw the circuit on Multisim CAD and apply the same steps conducted in above experimentation and fill the information as per above table.

VIVA Questions:

1. What is voltage to current converter?
2. What are the applications of voltage to current converter?
3. Explain working of current to voltage converter?

9 EXPERIMENT9 – PRECISION RECYIFIER AND PEAK DETECTOR:

Study of operational amplifier as precision rectifier and peak detector

Aim: (i) Half wave rectifier

(ii) Peak detector

Materials: Op-Amp 741, breadboard, regulated power supply ±15 V, resistors capacitors, connecting wires, hookup wire, Function generator, digital Multimeter (DMM), Oscilloscope.

Procedure: Precision rectifiers rectify very small AC voltage using Op-Amp. Make the connections as shown in below Figs. Switch on the power supply, apply very small AC signal say 20 mV of 1 KHz frequency to the rectifiers/detector, and observe the output on the oscilloscope. Measure the output voltage using DMM. Record the output voltages for 30, 40, 50, 60……. 100 mV up to 1 V, and note down corresponding output in observation table. Trace the input waveform as well as output waveform in both experiments.

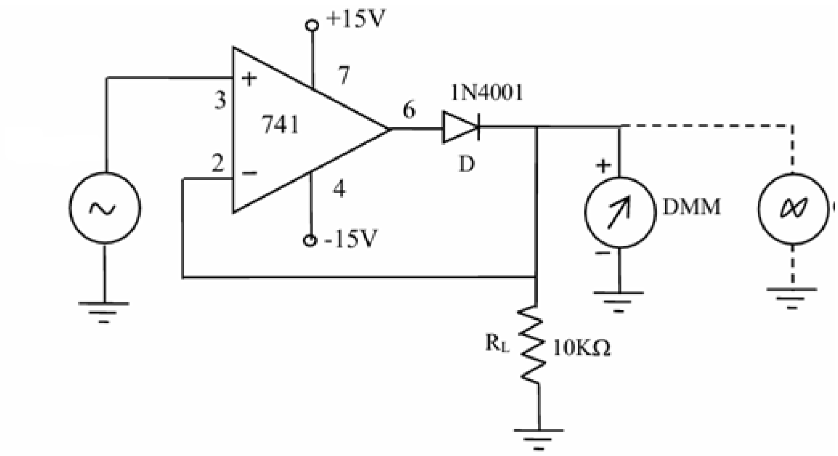


Figure1 – Precision rectifier or small signal half wave rectifier

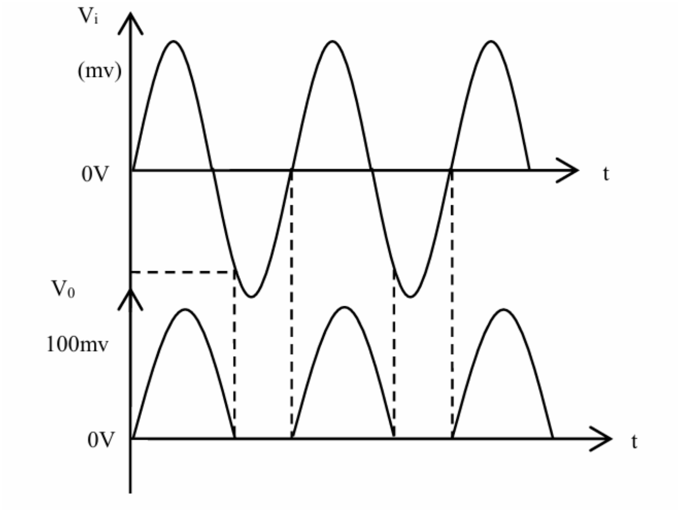


Figure2 – Input and corresponding output waveforms

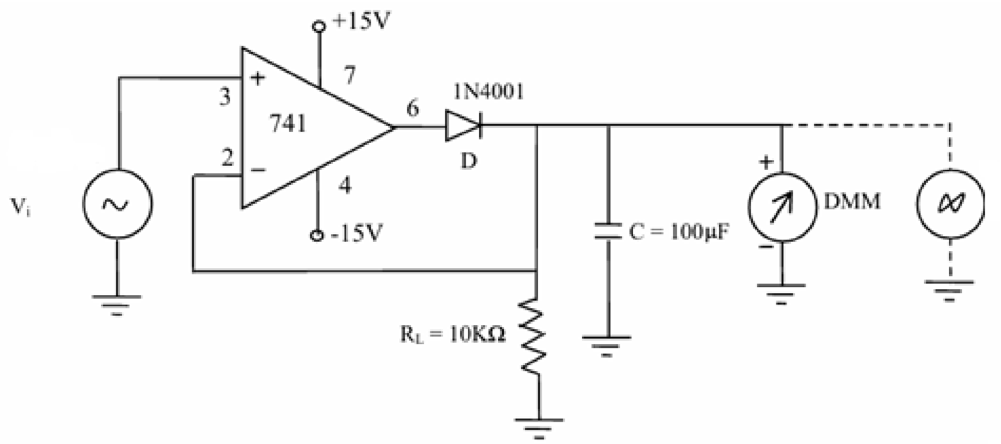


Figure3 – Peak detector using Op-Amp

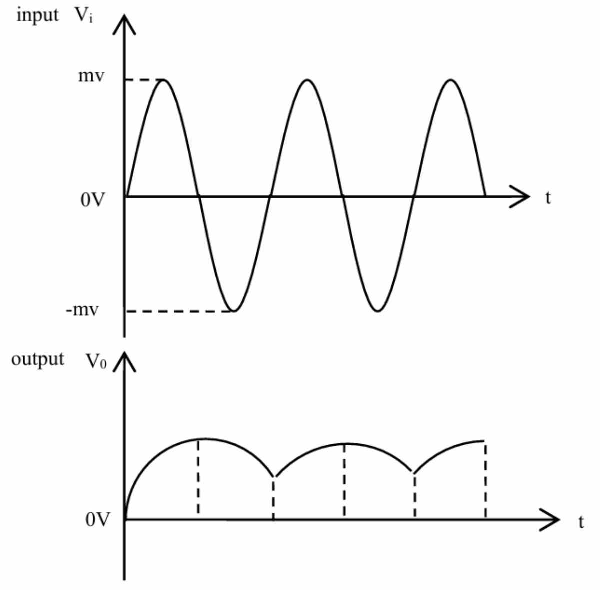


Figure4 – Input and output waveforms

Observation Table: (f=1KHz)

Sr. No.	Input signal strength (mV)	Output voltage V_0 (V)
1	20	
2	30	
3	40	
4	50	
5	60	
6	70	
–	–	
–	–	
–	–	
–	100 mV	

Results and Discussions:

In ordinary rectifier sufficient input voltage is required to conduct the diode. Small magnitude AC signals cannot be rectified by ordinary diode alone. But if it is used in active rectifier circuits then very small AC signals can be rectified or detected.

Similarly change in the input current will change output voltage, and I-V plot is linear. The input current is directly proportional to output voltage V_O

Precautions:

- Dual power supply must be regulated.
- All connections should be clean and tight.
- Apply very small input voltage so that the Op-Amp should not be saturated, because of the restrictions of amplifier that the amplified output should not go beyond supply voltage.
- Do not operate operational amplifier under open loop condition; otherwise it will go into saturation
- Measure all voltages with respect to ground.
- Use high input impedance voltmeter for voltage measurement to avoid the loading.

Software Implementation:

Draw the circuit on Multisim CAD and apply the same steps conducted in above experimentation and fill the information as per above table.

VIVA Questions:

1. What do you mean by rectification?
2. Which elements are generally used as a rectifier?
3. What is the difference between half wave and full wave rectifier?
4. What is ripple factor?
5. Explain action of precision rectifier
6. What is peak detector?
7. What are applications of peak detector?

10 EXPERIMENT10 – PHASE SHIFT OSCILLATOR:

Design and study of operational amplifier as phase shift oscillator

Aim: (i) Design phase shift oscillator

(ii) Measure frequency of output signal

Materials: Op-Amp 741, breadboard, regulated power supply ±15 V, resistors capacitors, connecting wires, hookup wire, Function generator, digital Multimeter (DMM), Oscilloscope.

Formula: (i) Voltage Gain:

$$A_V = \left|\frac{R_f}{R_1}\right|$$

(ii) Frequency of oscillation:

$$f_o = \frac{1}{2\pi\sqrt{6}RC}$$

Procedure: Make the connections as shown in below Fig. Select frequency of oscillator f_o and value of capacitor C, and then calculate value of resistor R from the given formula $f_o = \frac{1}{2\pi\sqrt{6}RC}$. Maintain the gain of the amplifier as $R_f = 29\ R_1$ by selection. Generally, 1MOhm potentiometer is used. Connect oscilloscope at the output to observe the waveform. Switch on the power supply and adjust the voltage gain of the amplifier to 29 by adjusting the potentiometer resistance to 29 R_1, so that oscillation can start. Calculate the frequency of oscillation from Oscilloscope by measuring the time of the cycle. Note down the reading in the observation table. Verify the experimental frequency with the selected frequency f_o of oscillation. Repeat this experiment for various values of R or C combination and verify the frequency of oscillation.

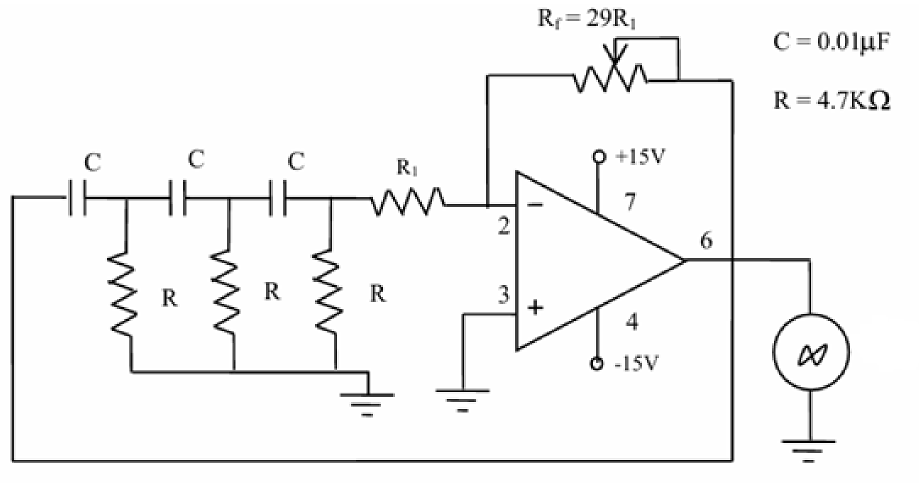


Figure1 – Phase shift oscillator

Observation Table:

Sr. No.	Value of resistor (KΩ)	Value of capacitor (μF)	Experimental frequency observed on CRO (KHz)	Frequency selected from formula f_0 (KHz)
1				
2				
3				
4				
5				

Results and Discussions:

The phase shift oscillator is designed, and waveform is observed on Oscilloscope which is sinusoidal. The frequency is calculated from Oscilloscope by measuring the time of the cycle. The value of frequency of oscillation f_o is verified with the theoretical value selected initially. It Is found that both frequencies are identical

Precautions:

- As feedback is small, starting oscillations is difficult
- The output is also small

- Each RC section should provide 60°phase shift
- Adjust $R_f = 29\ R_1$ so that oscillations will be produced
- Dual power supply must be regulated.
- All connections should be clean and tight.
- Apply very small input voltage so that the Op-Amp should not be saturated, because of the restrictions of amplifier that the amplified output should not go beyond supply voltage.
- Do not operate operational amplifier under open loop condition; otherwise it will go into saturation
- Measure all voltages with respect to ground.
- Use high input impedance voltmeter for voltage measurement to avoid the loading.

Software Implementation:

Draw the circuit on Multisim CAD and apply the same steps conducted in above experimentation and fill the information as per above table.

VIVA Questions:

1. Which feedback is used in oscillator?
2. Why above circuit is called phase shift oscillator?
3. What is a Barkhausen criterion?
4. Why it is necessary to adjust the gain of the amplifier as 29?
5. On which factors the frequency of oscillation depends?
6. How much phase shift is added by each RC network?

11 EXPERIMENT11 – WIEN BRIDGE OSCILLATOR:

To design and construct the Wien bridge oscillator using Op-Amp

Aim: (i) To design Wien bridge oscillator

(ii) To measure the frequency of output signal

Materials: Op-Amp 741, breadboard, regulated power supply ±15 V, resistors capacitors, connecting wires, hookup wire, Function generator, digital Multimeter (DMM), Oscilloscope.

Formula: (i) Voltage gain of the non-inverting amplifier is given by:

$$A_V = 1 + \frac{R_f}{R_1} = 3 \quad \text{or } R_f = 2R_1$$

(ii) Frequency of oscillation:

$$f_o = \frac{1}{2\pi RC}$$

Procedure: Experimental procedure is same as given in phase shift oscillator experiment no. 10

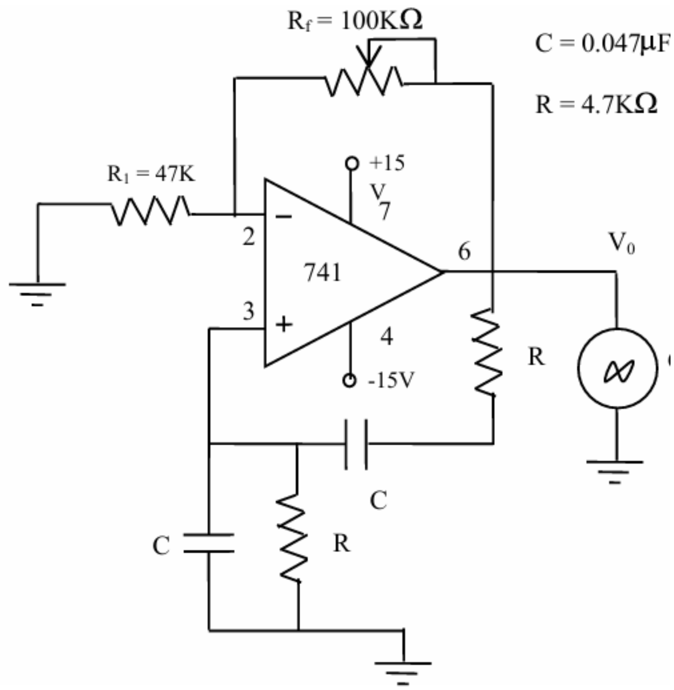


Figure1 – Wien bridge oscillator using Op-Amp

Observation Table:

Sr. No.	Value of resistor (KΩ)	Value of capacitor (μF)	Experimental frequency observed on CRO (KHz)	Frequency selected from formula f_0 (KHz)
1				
2				
3				
4				
5				

Results and Discussions:

The Wien Bridge oscillator is designed, and waveform is observed on Oscilloscope. The frequency is calculated from Oscilloscope by measuring the time of the cycle. The value of frequency of oscillation f_o is verified with the theoretical value selected initially. It Is found that both frequencies are identical

Precautions:

- Wien bridge oscillator offers zero phase shift, and hence non-inverting mode of Op-Amp is preferred
- While resonating the voltages at non-inverting and inverting terminals should be equal and in phase with each other
- Gain should be greater than 3 to start the oscillation
- There is a frequency limit of 1 MHz
- Dual power supply must be regulated.
- All connections should be clean and tight.
- Apply very small input voltage so that the Op-Amp should not be saturated, because of the restrictions of amplifier that the amplified output should not go beyond supply voltage.
- Do not operate operational amplifier under open loop condition; otherwise it will go into saturation
- Measure all voltages with respect to ground.

- Use high input impedance voltmeter for voltage measurement to avoid the loading.

Software Implementation:

Draw the circuit on Multisim CAD and apply the same steps conducted in above experimentation and fill the information as per above table.

VIVA Questions:

1. Why gain is adjusted to 3?
2. On which factors the frequency of oscillations depends?
3. What will be the frequency range of Wien bridge oscillator?
4. Explain the working of the oscillator?

12 EXPERIMENT12 – SQUARE WAVE GENERATOR:

Design and study of astable multivibrator using Op-Amp (Square wave generator)

Aim: (i) Design of astable multivibrator

(ii) Measure on and off time of the square wave

Materials: Op-Amp 741, breadboard, regulated power supply ±15 V, resistors capacitors, connecting wires, hookup wire, Function generator, digital Multimeter (DMM), Oscilloscope.

Formula: Frequency of oscillator:

$$f = \frac{1}{2RC} = \frac{1}{T}$$

Where T is time of one cycle

$$T = T_{ON} + T_{OFF}$$

Procedure: The astable multivibrator is called free-running multivibrator or square wave generator. The resistor R and capacitor C decide the frequency of oscillation. For proper oscillations the potentiometer of 22 Kohm is used. By adjusting the value of resistor, the proper feedback is provided. Make the connections as shown in below Fig. Switch on the power supply, and adjust the resistance R_1 to get the proper feedback so that oscillations will be produced at the output. Connect the oscilloscope at the output, and observe the square wave generated. Measure on and off time of the pulse, and calculate the frequency of oscillation. Record the observations in observation table. Change R and C values or any one component will change the frequency of oscillation. Perform this experiment for various values of R and C, and record the observation for calculation of frequency.

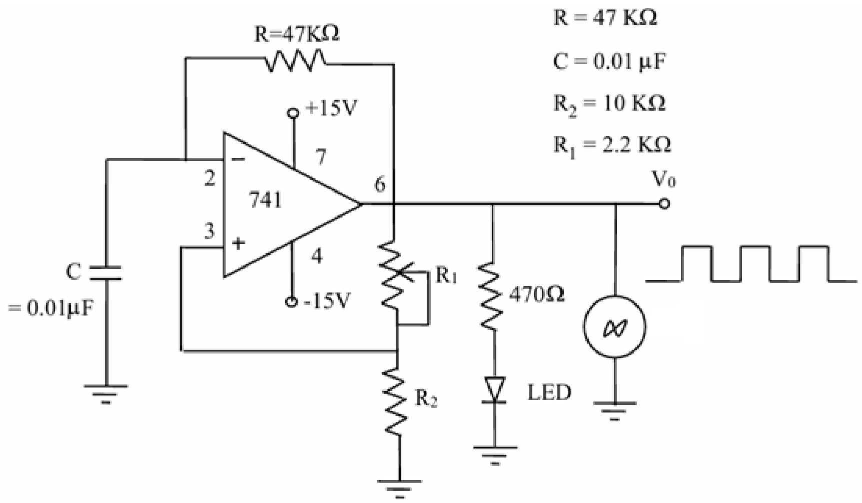


Figure1 – Square wave generator using Op-Amp

Observation Table:

Sr. No.	Value of resistor R (KΩ)	Value of capacitor C (μF)	On time T_{ON} (s)	Off time T_{OFF} (s)	Time period T (s)	Frequency (Hz)
1						
2						
3						
4						
5						

Results and Discussions:

The square wave is observed on the oscilloscope, and on and off time are measured. Frequency of oscillation of the wave form is found to be_________Hz. By changing the R and C values experiment is repeated.

Precautions:

- Adjust value of R1 to get proper feedback
- Dual power supply must be regulated.
- All connections should be clean and tight.

- Apply very small input voltage so that the Op-Amp should not be saturated, because of the restrictions of amplifier that the amplified output should not go beyond supply voltage.
- Measure all voltages with respect to ground.
- Use high input impedance voltmeter for voltage measurement to avoid the loading.

Software Implementation:

Draw the circuit on Multisim CAD and apply the same steps conducted in above experimentation and fill the information as per above table.

VIVA Questions:

1. What are non-sinusoidal oscillators?
2. How square wave is generated?
3. What do you mean by symmetrical and unsymmetrical multivibrators?
4. Describe working of astable multivibrator in brief

13 EXPERIMENT13 – TRIANGULAR WAVE GENERATOR:

Design and study of triangular wave generator using Op-Amp

Aim: (i) Design of triangular waveform generator

(ii) Measure the frequency of the oscillations

Materials: Op-Amp 741, breadboard, regulated power supply ±15 V, resistors capacitors, connecting wires, hookup wire, Function generator, digital Multimeter (DMM), Oscilloscope.

Formula: Frequency of oscillator:

$$f = \frac{R_3}{4\, R_1 C_1 R_2}$$

Procedure: Make the connections as shown in below Fig. Connect proper values of R_1, R_2, R_3 and C_1 in the circuit. Switch on the power supply. Connect the oscilloscope at the output to observe the waveform. The first Op-Amp will act as comparator, i.e. zero crossing detector because inverting terminal of the Op-Amp is connected to ground. Second Op-Amp acts as integrator which converts square wave into triangular waveform. Adjust the resistance of the potentiometer R_3 to get square wave output at the first Op-Amp. The frequency of oscillations can be adjusted by changing the value of R_3. Record the observations for various values of R_3 in observation table, and calculate the frequency of oscillations f.

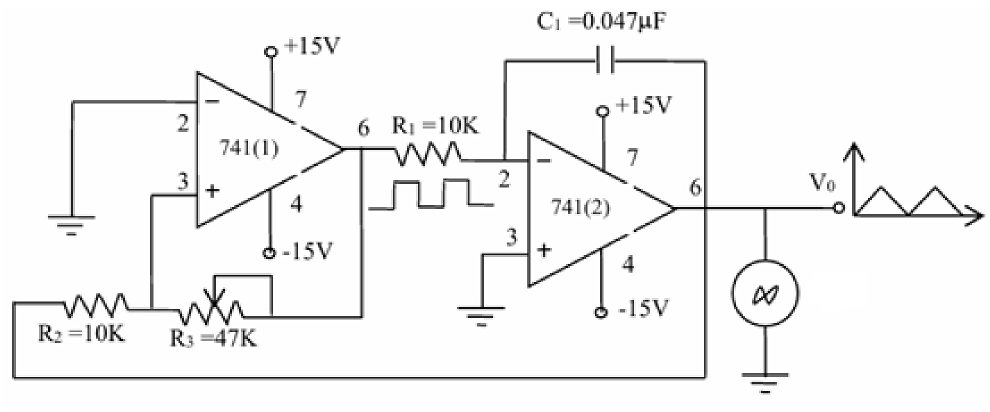


Figure1 – Triangular waveform generator

NOTE: If the on time of the square wave is adjusted to be small, the oscillator will work as sawtooth waveform generator. Rest of the procedure and experiment is same as triangular waveform generator.

Observation Table:

Sr. No.	Resistor values (KΩ)			Value of capacitor C (μF)	Frequency f (Hz) from formula	Frequency measured on CRO (Hz)
	R_1	R_2	R_3			
1						
2						
3						
4						
5						
6						

Results and Discussions:

By changing the values of R_1, R_2 and R_3 the frequency of oscillation is changed. The frequency calculated from formula and measured frequency on Oscilloscope are found to be identical.

Precautions:

- Measure accurate time period of cycle on oscilloscope
- Select proper values of resistors and capacitors
- To start the oscillations adjust value of R_3
- Dual power supply must be regulated.
- All connections should be clean and tight.
- Apply very small input voltage so that the Op-Amp should not be saturated, because of the restrictions of amplifier that the amplified output should not go beyond supply voltage.
- Measure all voltages with respect to ground.
- Use high input impedance voltmeter for voltage measurement to avoid the loading.

Software Implementation:

Draw the circuit on Multisim CAD and apply the same steps conducted in above experimentation and fill the information as per above table.

VIVA Questions:

1. How integrator converts square wave into triangular wave?
2. What is the necessity of adjusting the resistor value of R_3?
3. Explain the working of comparator
4. What is duty cycle?
5. What will be the effect of change in duty cycle on output waveform?
6. Explain working of integrator
7. How ramp up and ramp down voltage is generated?
8. List the factors which decide the frequency of oscillation of triangular wave

14 EXPERIMENT14 – FREQUENCY TO VOLTAGE CONVERTER:

To study the frequency to voltage (F/V) converter using Op-Amp

Aim: (i) Design and construct the F/V converter

(ii) Calculate the change in output with input frequency change

Materials: Op-Amp 741, breadboard, regulated power supply ±15 V, resistors capacitors, connecting wires, hookup wire, Function generator, digital Multimeter (DMM), Oscilloscope.

Procedure: The first stage Op-Amp is working as Schmitt trigger and second stage as integrator. The sinusoidal wave is converted into square waveform by the Schmitt trigger, and integrator produces a ramp. Make the connections as shown in below Fig. The oscillator is connected at the input, and the sinusoidal signal is directly connected to the inverting terminal of the first stage Op-Amp. Change the frequency of the input signal and adjust the amplitude to get square wave output of the Schmitt trigger. Note down the output voltage for corresponding input signal frequency. Record the observations in observation table. Plot the graph between input signal frequency and output voltage. A linear plot will give you change in output voltage per frequency of input signal.

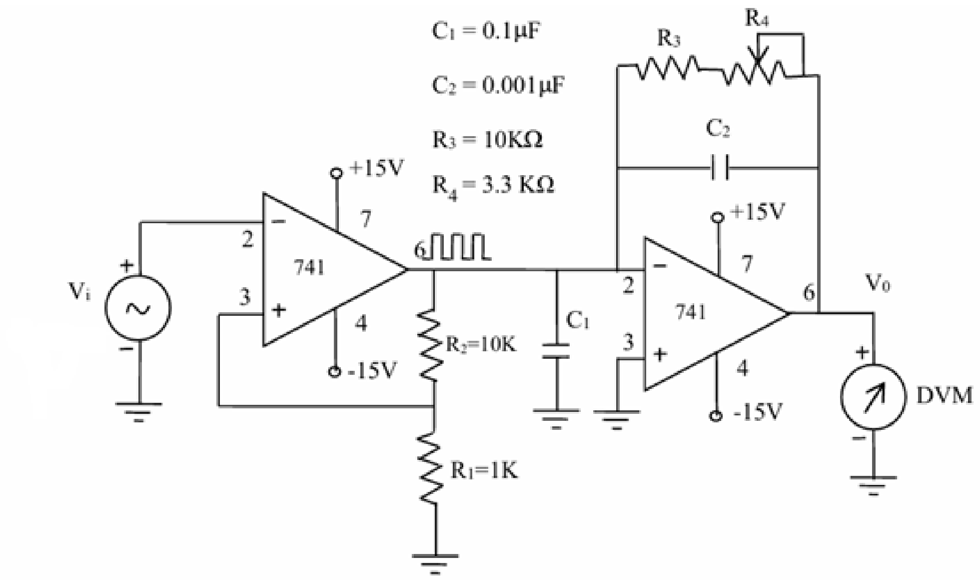


Figure1 – Frequency to voltage converter

Observation Table:

$$V_i = \_\_\_\_\_\_\_\_\_ V$$

Sr. No.	Frequency of input signal (Hz)	Output voltage V_0 (V)
1		
2		
3		
4		
5		
6		

Results and Discussions:

The output voltage varies linearly with increasing frequency of the input signal. The change in output voltage with change in input frequency of the signal is found to be _______.

Precautions:

- Dual power supply must be regulated.
- All connections should be clean and tight.
- Apply very small input voltage so that the Op-Amp should not be saturated, because of the restrictions of amplifier that the amplified output should not go beyond supply voltage.
- Measure all voltages with respect to ground.
- Use high input impedance voltmeter for voltage measurement to avoid the loading.

Software Implementation:

Draw the circuit on Multisim CAD and apply the same steps conducted in above experimentation and fill the information as per above table.

VIVA Questions:

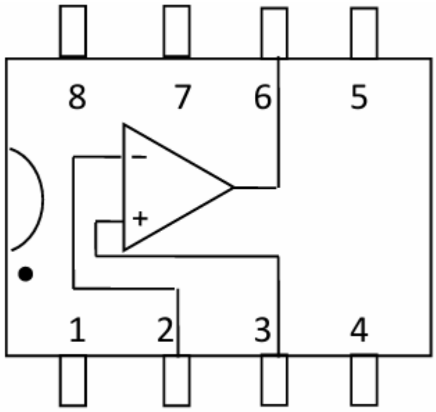
1. What is the function of integrator?
2. How IC 741 operational amplifier converts sinusoidal waveform into square waveform?
3. Are there any restrictions on integrator?
4. What is the resolution of the circuit?

15 APPENDIX:

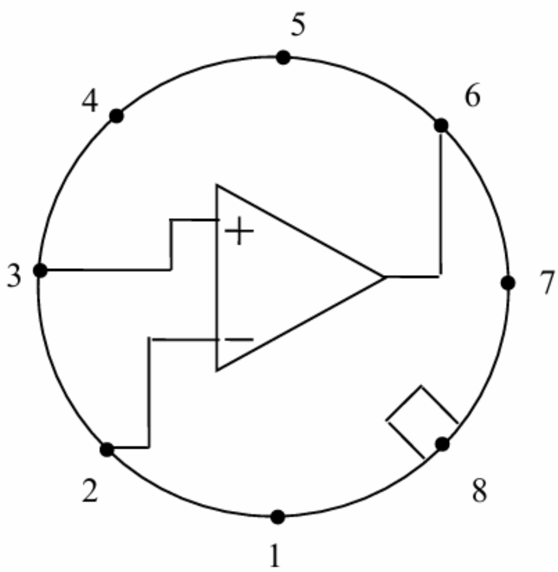
The IC 741 is an operational amplifier designed by Fair Child semiconductors.

It is a versatile amplifier used as a multipurpose device in many electronic instruments/equipments.

The pin configuration of this amplifier is given in below figures:



a) DIP



b) TO Package

Pin No.		Function
1	-	Offset null
2	-	Inverting input
3	-	Non-inverting input
4	-	- V_{CC}
5	-	Offset null
6	-	Output
7	-	+ V_{CC}
8	-	Not connected