LAB MANUAL

EC205PPC06

LIC & ITS APPLICATIONS LAB

Bachelor of Technology

in

Electronics & Communication Engineering

Department of Electronics & Communication Engineering

School of Studies of Engineering & Technology

Guru Ghasidas Vishwavidyalaya

Bilaspur-495009 (C. G.)

Website: www.ggu.ac.in

SCHOOL OF STUDIES OF ENGINEERING & TECHNOLOGY

GURU GHASIDAS VISHWAVIDYALAYA, BILASPUR (C.G.)

(A CENTRAL UNIVERSITY)

CBCS-NEW SYLLABUS

B. TECH. THIRD YEAR (Electronics and Communication Engineering)

Vision and Mission of the Institute

Vision		To be a leading technological institute that imparts transformative education to create globally competent technologists, entrepreneurs, researchers and leaders for a sustainable society
Mission	1	To create an ambience of teaching learning through transformative education for future leaders with professional skills, ethics, and conduct.
	2	To identify and develop sustainable research solutions for the local and global needs.
	3	To build a bridge between the academia, industry and society to promote entrepreneurial skills and spirit

Vision and Mission of the Department

Vision		The Department endeavours for academic excellence in Electronics & Communication Engineering by imparting in depth knowledge to the students, facilitating research activities and cater to the ever-changing industrial demands, global and societal needs with leadership qualities.
Mission	1	To be the epitome of academic rigour, flexible to accommodate every student and faculty for basic, current and future technologies in Electronics and Communication Engineering with professional ethics.
	2	To develop an advanced research centre for local & global needs.
	3	To mitigate the gap between academia, industry & societal needs through entrepreneurial and leadership promotion.

Program Educational Objectives (PEOs)

The graduate of the Electronics and Communication Engineering Program will

PEO1: Have fundamental and progressive knowledge along with research initiatives in the field of Electronics & Communication Engineering.

PEO2: Be capable to contrive solutions for electronic & communication systems for real world applications which are technically achievable and economically feasible leading to academia, industry, government and social benefits.

PEO3: Have performed effectively in a multi-disciplinary environment and have self-learning & self-perceptive skills for higher studies, professional career or entrepreneurial endeavors to be confronted with a number of difficulties.

PEO4: Attain team spirit, communication skills, ethical and professional attitude for lifelong learning.

Programme Outcomes: Graduates will be able to:

PO1: Fundamentals: Apply knowledge of mathematics, science and engineering.

PO2: Problem analysis: Identify, formulate and solve real time engineering problems using first principles.

PO3: Design: Design engineering systems complying with public health, safety, cultural, societal and environmental considerations

PO4: Investigation: Investigate complex problems by analysis and interpreting the data to synthesize valid solution.

PO5: Tools: Predict and model by using creative techniques, skills and IT tools necessary for modern engineering practice.

PO6: Society: Apply the knowledge to assess societal, health, safety, legal and cultural issues for practicing engineering profession.

PO7: Environment: Understand the importance of the environment for sustainable development.

PO8: Ethics: Apply ethical principles and commit to professional ethics, and responsibilities and norms of the engineering practice.

PO9: Teamwork: Function effectively as an individual and as a member or leader in diverse teams and multidisciplinary settings.

PO10: Communication: Communicate effectively by presentations and writing reports.

PO11: Management: Manage projects in multidisciplinary environments as member or a team leader.

PO12: Life-long learning: Engage in independent lifelong learning in the broadest context of technological change.

Programme Specific Outcomes:

PSO1: Identify, formulate and apply concepts acquired through Electronics & Communication Engineering courses to the real-world applications.

PSO2: Design and implement products using the cutting-edge software and hardware tools to attain skills for analyzing and developing subsystem/processes.

PSO3: Ability to adapt and comprehend the technology advancement in research and contemporary industry demands with demonstration of leadership qualities and betterment of organization, environment and society.

SubCode	L	T	P	Duration	IA	ESE	Total	Credits
EC205PPC06	-	-	2	2Hours	30	20	50	1

LIC & ITS APPLICATIONS LAB

Course Objectives:

- To develop basic operations of IC741.
- To design and implement different linear and non linear applications of OPAMP.
- To design different filter, oscillator, andwaveform generator circuits using OPAMP ICs.
- To design different multivibrator, modulator circuits using IC555.

Course Outcomes:

At the end of the course, students will be able to:

CO1Design and develop different linear and nonlinear application of OPAMP.

CO2 Implement different multivibrator circuits.

CO3 Demonstrate and design filter using OPAMP ICs.

CO4 Demonstrate and design oscillator and waveform generator circuits using OPAMPI Cs.

CO5 Implement different multivibrator circuits.

Course Outcomes and their mapping with Program Outcomes & Program Specific Outcomes:

CO	PO												PSO		
	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12	PSO1	PSO2	PSO3
CO1	3	2	1	1	1	1			2			3	2		1
CO2	3	2	1	1	1	1			2			3	2		1
CO3	3	2	1	1	1	1			2			3	2		1
CO4	3	2	1	1	1	1			2			3	2		1
CO5	3	2	1	1	1	1			2			3	2		1

Weightage: **1-Sightly;2-Moderately;3-Strongly**

Experiment No. 1

INVERTING AMPLIFIER

Aim: To design and setup an inverting amplifier circuit with OP AMP 741C for a gain of 10, plot the waveforms, observe the phase reversal, measure the gain.

Objectives: After completion of this experiment, student will be able to design and setup an inverting amplifier using OP AMP. He/she will be able to design and implement OPAMP inverting amplifier circuit.

Equipments/Components:

Sl .No	Name and Specification	Quantity required
1	Dual power supply +/- 15V	1
2	Function generator (0 - 1MHz)	1
3	Oscilloscope	1
4	Bread board	1
5	IC 741C	1
6	Resistors	2
7	Probes and connecting wires	As required.

Theory

It is a closed loop mode application of opamp and employs negative feedback. The R_f and R_i are the feedback and input resistance of the circuit respectively. The input terminals of the opamp draws no current because of the large differential input impedance. The potential difference across the input terminals of an opamp is zero because of the large open loop gain. Due to these two conditions, the inverting terminal is at virtual ground potential. So the current flowing through Ri and Rf are the same.

$$I_i = I_f$$

That is $Vin/R_i = - Vo/R_f$

Therefore $V_o/V_{in} = A_v = - R_f/R_i,$

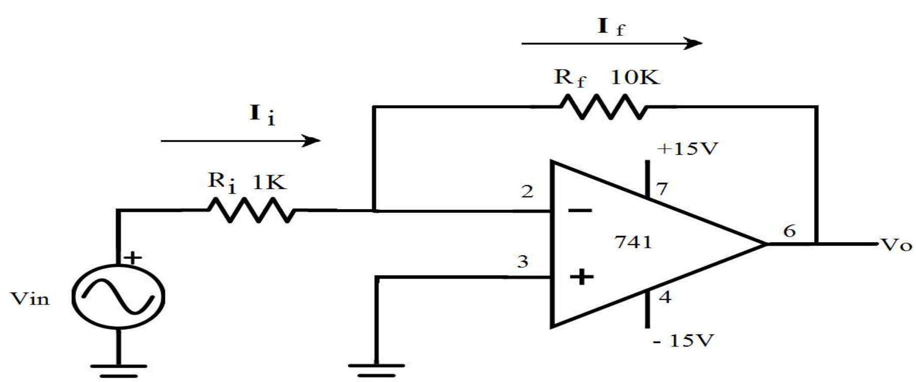
Here the –Ve sign indicates that the output will be an amplified wave with 180^0 phase shift (inverted output). By varying the R_f or R_i, the gain of the amplifier can be varied to any desired value.

Procedure

1. Check the components.
2. Setup the circuit on the breadboard and check the connections.
3. Switch on the power supply.
4. Give 1 Vpp / 1 KHz sine wave as input.
5. Observe input and output on the two channels of the oscilloscope simultaneously.
6. Note down and draw the input and output waveforms on the graph.
7. Verify the input and output waveforms are out of phase.
8. Verify the obtained gain is same as designed value of gain.

Circuit

NON- INVERTING AMPLIFIER



Aim: To design and setup a non-inverting amplifier circuit with OPAMP IC 741C for a gain of 11, plot the waveform, observe the phase reversal, measure the gain.

Objectives: After completion of this experiment, student will be able to design and setup a non-inverting amplifier using OP AMP. He/she will acquire skill to design and implement OPAMP non-inverting amplifier circuit.

Equipments/Components:

Sl .No	Name and Specification	Quantity required
1	Dual power supply +/- 15V	1
2	Function generator (0 - 1MHz)	1

3	Oscilloscope	1
4	Bread board	1
5	IC 741C	1
6	Resistors	2
7	Probes and connecting wires	As required.

Theory

It is a linear closed loop mode application of op-amp and employs negative feedback. The R_f and R_i are the feedback and input resistance of the circuit respectively. There will be no phase difference between the output and input. Hence it is called non-inverting amplifier.

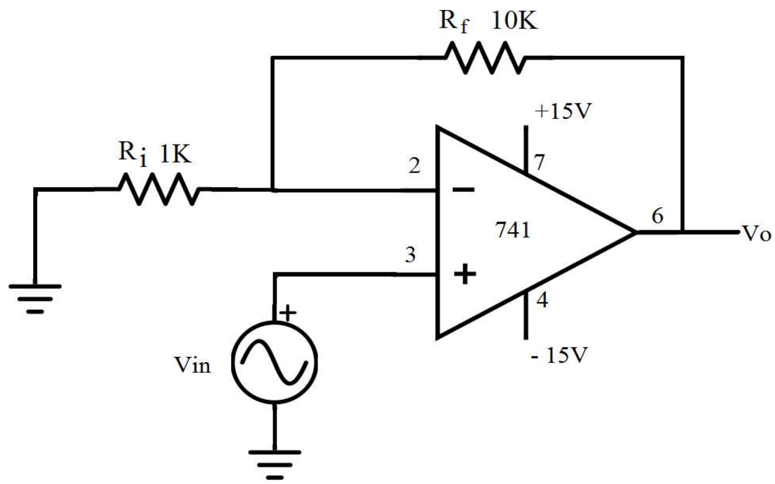
$$Av = V_o / V_{in} = 1 + R_f / R_i ,$$

Here the +Ve sign indicates that the output will be an amplified wave in phase with the input. By varying the R_f or $R_{i,}$ the gain of the amplifier can be varied to any desired value.

Procedure

1. Check the components.
2. Setup the circuit on the breadboard and check the connections.
3. Switch on the power supply.
4. Give 1 Vpp / 1 KHz sine wave as input.
5. Observe input and output on the two channels of the CRO simultaneously.
6. Note down and draw the input and output waveforms on the graph.
7. Verify the input and output waveforms are in phase.
8. Verify the obtained gain is same as designed value.

Circuit Diagram



Design:

Gain of an inverting amplifier $Av = V_o/V_{in} = - R_f / R_i$

The required gain = 10,

That is $Av = - R_f/ R_i = 10$

Let $R_i = 1K\Omega$, Then $R_f = 10K\Omega$

Observations:

Vin = 1 Vpp

Vo=?

Gain, Av = Vo/Vin =?

Observed phase difference between the input and the output on the CRO =?

Graph

Design:

Gain of an inverting amplifier $Av=V_o/V_{in} = 1+R_f/ R_i$,

Let the required gain be 11,

Therefore $Av= 1+R_f/ R_i= 11$

$R_f/ R_i = 10$

Take $R_i= 1K\Omega$, Then $R_f = 10K\Omega$

Observations:

Vin = 1Vpp

Vo = ?

Gain Av = Vo/Vin =?

Observed phase difference between the input and the output on the CRO =?

Graph:

Experiment No 2

ADDER (SUMMING) AMPLIFIER

Aim: To design and setup a summing amplifier circuit with OP AMP 741C for a gain of 2 and verify the output.

Objectives: After completion of this experiment, student will be able to design and setup a summing amplifier using OP AMP.

Equipments/Components:

Sl .No	Name and Specification	Quantity required
1	Dual power supply +/- 15V	1
2	DC power source 1.5V	2
3	Function generator (0- 1MHz)	1
4	Oscilloscope	1
5	Bread board	1
6	IC 741C	1
7	Resistor	3
8	Probes and connecting wires	As required.

Theory:

Op-amp can be used to design a circuit whose output is the sum of several input signals. Such a circuit is called a summing amplifier or an adder. Summing amplifier can be classified as inverting & non-inverting summer depending on the input applied to inverting & non-inverting terminals respectively. Circuit Diagram shows an inverting summing amplifier with 2 inputs. Here the output will be amplified version of the sum of the two input voltages with 180^0 phase reversal.

$$V_o = - (R_f / R_i)(V_1+V_2)$$

Procedure

1. Check the components.
2. Setup the circuit on the breadboard and check the connections.
3. Switch on the power supply.
4. Give $V_1 = V_2 = +1.5V$ DC with polarity as shown in fig.1.
5. Make sure that the CRO selector is in the D.C. coupling position**.**

6. Observe input and output on two channels of the oscilloscope simultaneously.
7. Note down and draw the input and output waveforms on the graph.
8. Verify that the output voltage is -6V DC
9. Repeat the procedure with V_1 =1Vpp / 1 KHz sine wave and V_2 = +1.5Vdc as shown in fig2.

RESULT

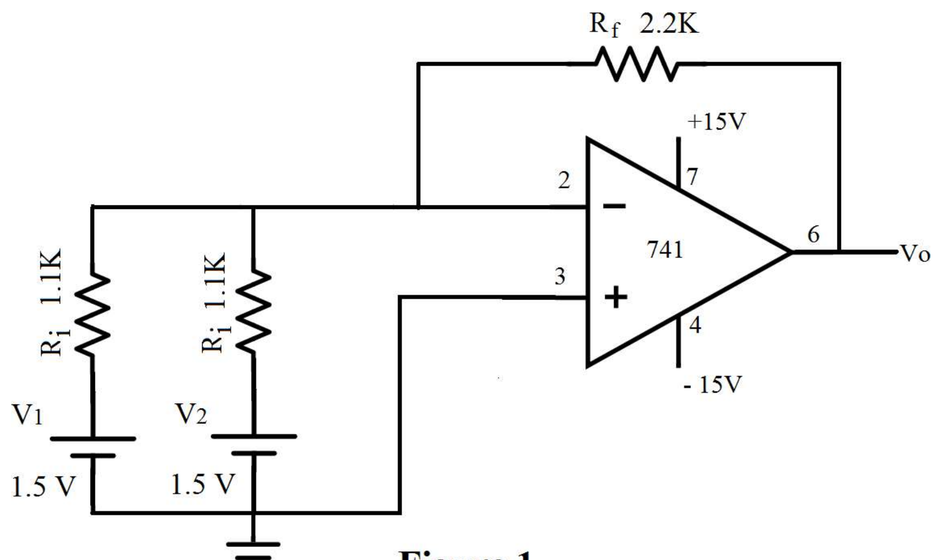


Figure 1

Circuit diagram

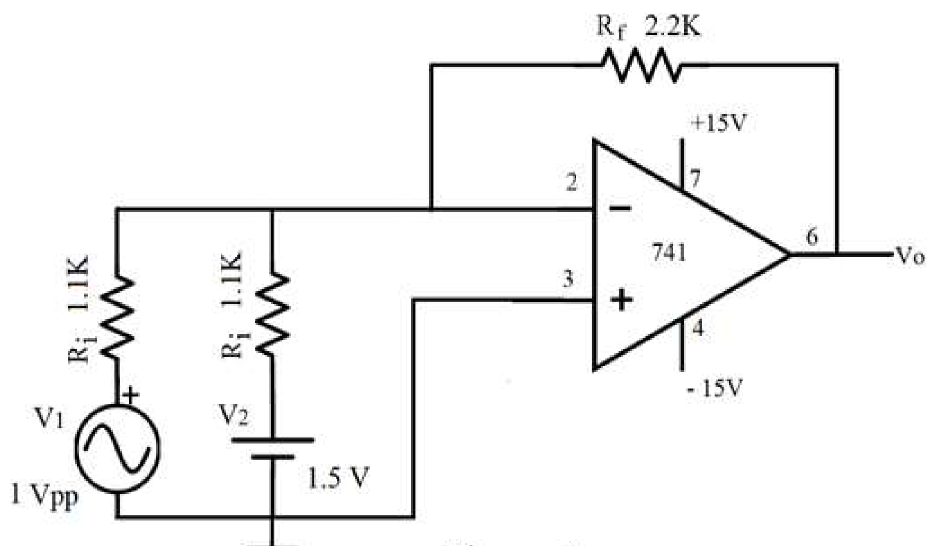


Figure 2

Design:

The output voltage of an inverting summing amplifier is given by $V_o = -(R_f / R_i)(V_1+V_2)$

Let $R_i = 1.1K\Omega$

Then $R_f = 2.2K\Omega$

Then $V_o = -2(V_1+V_2)$

Observations:

Part1:

V_1= 1.5 DC

V_2= 1.5 DC

Then Vo=?

Part 2:

V_1= 1Vpp sine wave

V_2= 1.5 DC

Then Vo=

Graph:

SUBSTRACTOR (DIFFERENCE) AMPLIFIER

Aim: To design and setup a difference amplifier circuit with OPAMP IC 741C for a gain of 2 and verify the output.

Objectives: After completion of this experiment, student will be able to design and setup a difference amplifier using OP AMP.

Equipments/Components:

Sl. No	Name and Specification	Quantity required
1	Dual power supply +/- 15V	1
2	DC power source 1.5V	1
3	Function generator (0-1MHz)	1
4	Oscilloscope	1
5	Bread board	1
6	IC 741C	1
7	Resistor	3
8	Probes and connecting wires	As required.

Theory:

A difference amplifier is a circuit that gives the amplified version of the difference of the two inputs, Vo =A(V1-V2), Where V1 and V2 are the inputs and A is the voltage gain. Here input voltage V1 is connected to non-inverting terminal and V2 to the inverting terminal. This is also called as differential amplifier. Output of a differential amplifier can be determined using super position theorem. When V_1=0, the circuit becomes an inverting amplifier with input V_2 and the resulting output is V_{02}= -Rf /Ri (V_2). When V_2=0, the circuit become a non-inverting amplifier with input V_1 and the resulting output is V_{01}= Rf/Ri(V_1). Therefore the resulting output according to super position theorem is

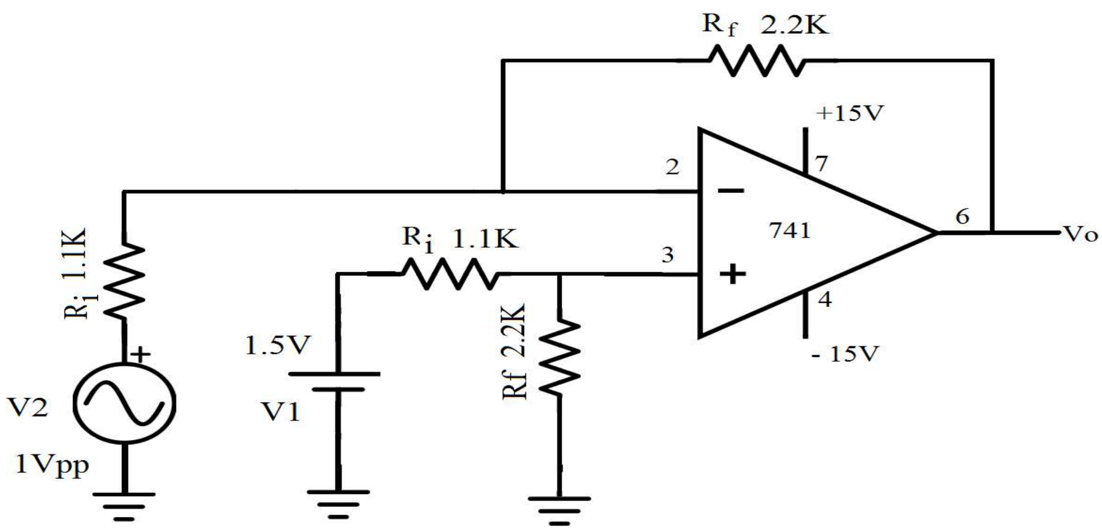
$$Vo = V_{01} + V_{02} = Rf/Ri(V1-V2)$$

Procedure

1. Check the components.
2. Setup the circuit on the breadboard and check the connections.
3. Switch on the power supply.
4. Give V_1 = +1.5V DC with polarity as shown.

5. Give V_2= 1Vpp/ 1 KHz sine wave.
6. Make sure that the oscilloscope coupling selector is in the D.C. position.
7. Observe input and output on oscilloscope simultaneously.
8. Note down and draw the input and output waveforms on the graph.

Circuit Diagram



Design:

Given the gain = 2

Vo = V_{01}+ V_{02} = Rf/Ri(V1-V2)

That is Rf / Ri = 2

Let Ri = 1.1KΩ

Then Rf = 2.2KΩ

Observations:

V_1= 1.5 DC

V_2= 1Vpp sine wave

Then Vo = ?

Graph:

Experiment No 3

INTEGRATOR

Aim: To design and setup an integrator circuit using OP AMP 741C and plot its pulse response.

Objectives: After completion of this experiment, student will be able to design and setup an integrator circuit using OP AMP.

Equipments/Components:

Sl. No	Name and Specification	Quantity required
1	Dual power supply +/- 15V	1
2	Function generator (0-1MHz)	1
3	Oscilloscope	1
4	Bread board	1
5	IC 741C	1
6	Resistor	1
7	Capacitor 0.01µF	1
8	Probes and connecting wires	As required.

Theory:

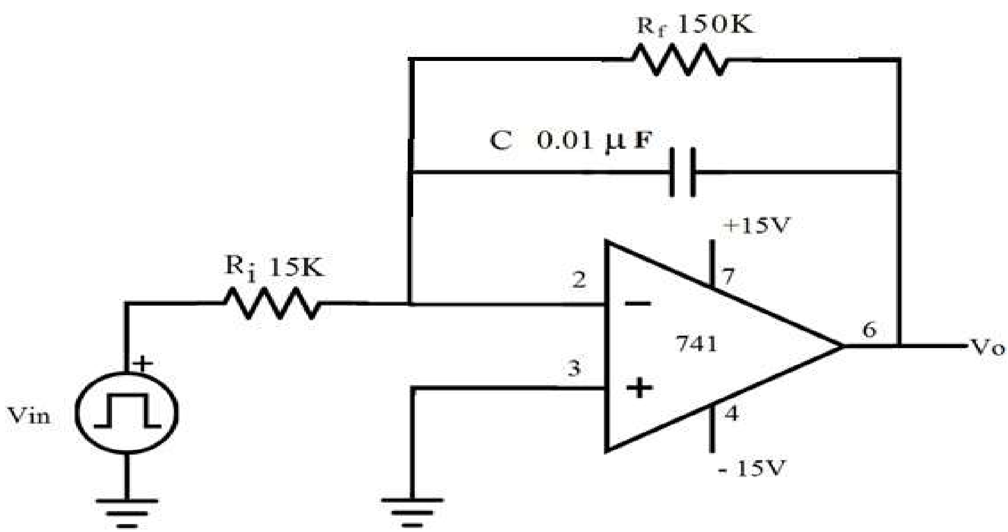
It is a closed loop op-amp circuit which performs the mathematical operation of integration. That is the output waveform is the integral of the input voltage and is given by $V_o = (-1/R_fC) \int V_{in}dt$. The integrator circuit is constructed from basic inverting amplifier by replacing the feedback resistance R_f with capacitor C. This circuit also works as low pass filter.

Procedure:

1. Check the components.
2. Setup the circuit on the breadboard and check the connections.
3. Switch on the power supply.
4. Give V_i= 2Vpp, 1KHz square wave.
5. Keep the oscilloscope in AC coupling mode.
6. Observe input and output on two channels of the oscilloscope simultaneously.

7. Draw the input and output waveforms on the graph.

Circuit Diagram:



Design:

Given f =1 KHz

So T = 1/f = 1ms

Design equation is $T = 2\pi R_i C$

Let $C = 0.01\mu F$

Then $R_i = 15K\Omega$

Take $R_f = 10R_i = 150K\Omega$

Graph:

Result:

DIFFERENTIATOR

Aim: To design and set up a Differentiator circuit using OP AMP 741C and plot their pulse response.

Objectives: After completion of this experiment, student will be able to design and setup a differentiator circuit using OP AMP.

Equipments/Components:

Sl. No	Name and Specification	Quantity required
1	Dual power supply +/- 15V	1
2	Function generator (0- 1MHz)	1
3	Oscilloscope	1
4	Bread board	1
5	IC 741C	1
6	Resistor	1
7	Capacitor	1
8	Probes and connecting wires	As required.

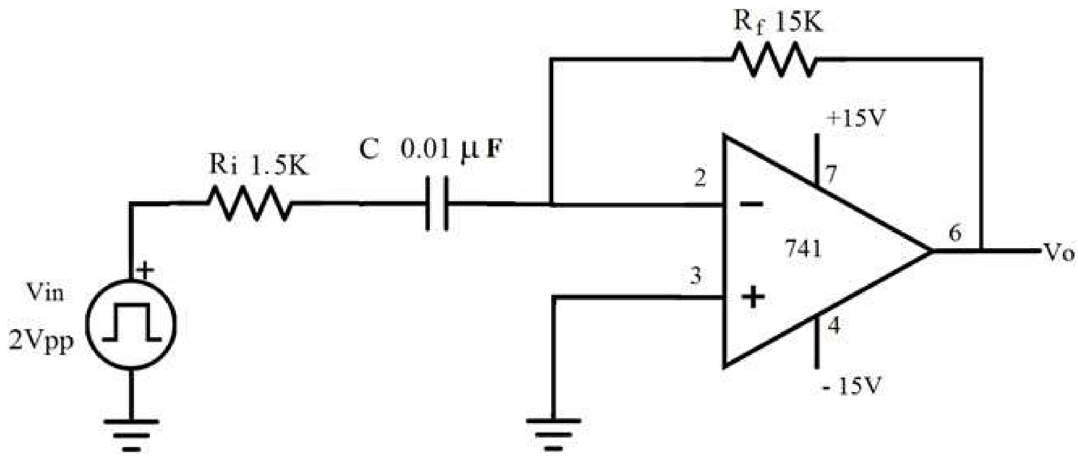
Theory:

It is an opamp circuit which performs the mathematical operation of differentiation. That is the output waveform is the derivative or differentia l of the input voltage. That is $V_o = -R_fCd(V_{in})/dt$. The differentiator circuit is constructed from basic inverting amplifier by replacing the input resistance R_i with capacitor C. This circuit also works as high pass filter.

Procedure:

1. Check the components.
2. Setup the circuit on the breadboard and check the connections.
3. Switch on the power supply.
4. Keep the oscilloscope in AC coupling mode.
5. Give V_i= 2Vpp, 1KHz square wave.
6. Observe input and output on two channels of the oscilloscope simultaneously.
7. Note down and draw the input and output waveforms on the graph.

Circuit Diagram:



Design:

Given f = 1 KHz

So T = 1/f = 1ms

Design equation is $T = 2\pi R_f C$

Let C = 0.01µF

Then R_f = 15KΩ

Let Ri = Rf/10 = 1.5KΩ

Graph

EXPERIMENT 4

Log and Antilog Amplifiers

Objectives: To understand the behavior of logarithmic and antilogarithmic amplifiers.

Equipments/Components:

Sl. No	Name and Specification	Quantity required
1	Resistors (100K)	2
2	Diodes	2
3	Transistor	1
4	Bread board	1
5	IC 741C	2
6	Multimeter	1
7	Probes and connecting wires	As required.

Theory:

Log amplifiers are widely used for analog signal compression applications. When a diode used in the feedback loop of an operational amplifier is forward biased by

a constant current of magnitude

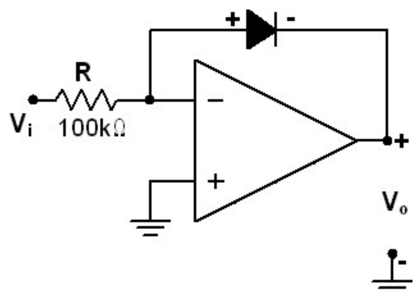
Note that the input voltage and diode voltage are related in a logarithmic fashion. If we take the diode voltage as an output voltage then the input and output will be related in a logarithmic fashion.

The base emitter junction of a bipolar junction transistor can be used as diode when collector and base are shorted. So a transistor can also be used in the feedback loop of an op-amp.

Antilog is inverse operation of log operation so, antilog amplifiers can be designed by reversing the arrangement of diodes and resistors in the log amplifiers.

It is important to note that a single polarity of current can only forward bias the diode. That means the log operation or antilog operation is single quadrant operation.

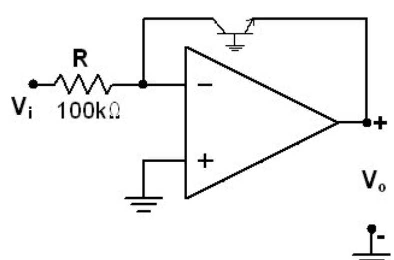
Log Amplifier using Diode



Procedure:

1. Set the supply voltage at ±12V.
2. Set the input voltage to 1V.
3. See the voltage across the diode. Note the negative sign.
4. Increase the input voltage in the step of 1V up to 20V.
5. Plot the characteristics of input voltage and output voltage.
6. Reverse the polarity of the diode and see the effect for positive input voltage.

Log Amplifier Using a BJT



Procedure

1. Use an NPN type BJT in place of diode as shown in fig 2.
2. Set the input voltage to 1V.
3. See the voltage across the output terminal. Note the negative sign.

5. Plot the characteristics of input voltage and output voltage.
6. Compare the characteristics with that of diode based log amplifier.

Anti-log Amplifier

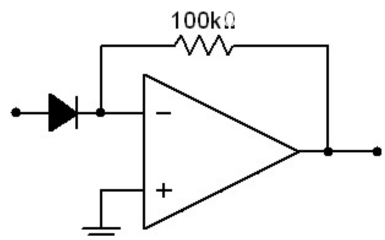
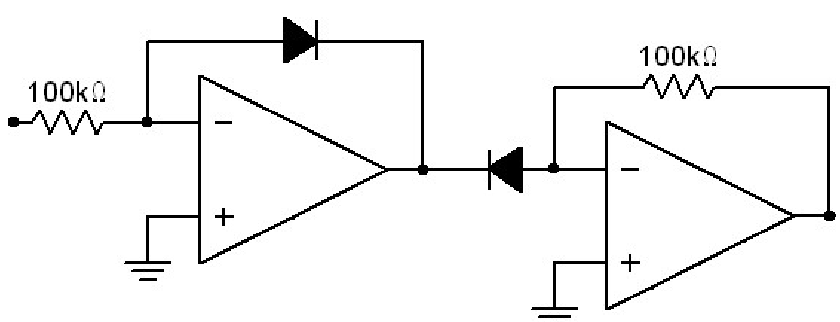


Fig 3

Procedure

1. Set the input voltage to 100mV.
2. See the voltage across the Resistor. Note the negative sign.
3. Increase the input voltage in the step of 50mV up to 500mV.
4. Plot the characteristics of input voltage and output voltage.
5. Reverse the polarity of the diode and see the effect for positive input voltage.

Log - Antilog Amplifier



RESULT-

Experiment No. 5

ASTABLE MULTIVIBRATORS USING OP AMP

Aim: To design and setup symmetrical and asymmetrical astable multivibrators using Op-amp 741, plot the waveforms and measure the frequency of oscillation

Objectives: After completion of this experiment, student will be able to design and setup an astable multivibrators circuit using OP AMP.

Equipments/Components:

Sl. No	Name and Specification	Quantity required
1	Dual power supply +/- 15V	1
2	Function generator (0- 1MHz)	1
3	Oscilloscope	1
4	Bread board	1
5	IC 741C	1
6	Resistor	5
7	Capacitor 0.1µF	1
8	Diode 1N4001	2
9	Probes and connecting wires	As required.

Theory:

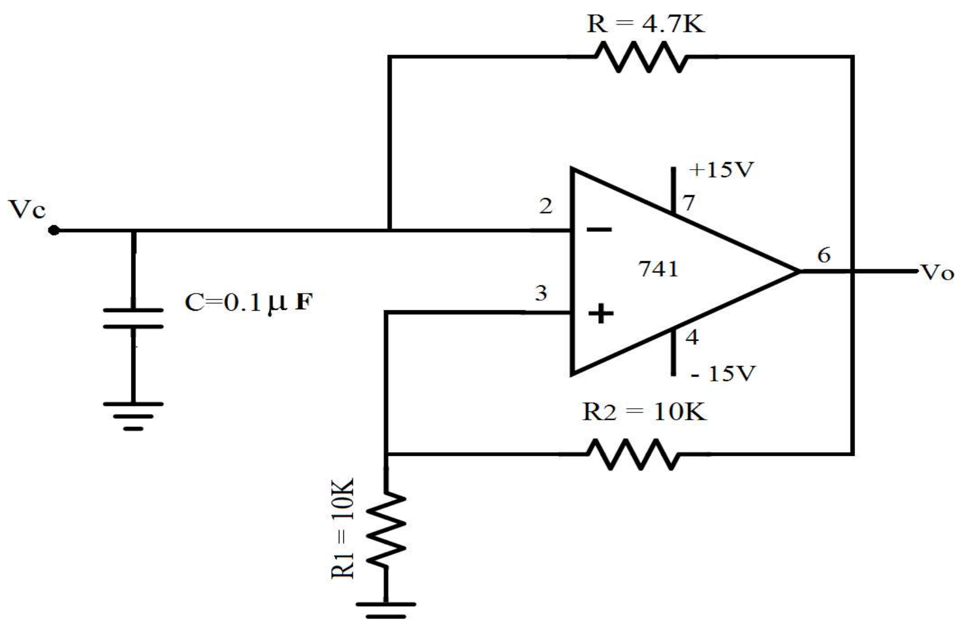
In this circuit, the opamp is operated in saturation mode and the output swings between +Vsat and –Vsat giving square wave output. This circuit is also called free running oscillator or square wave generator . A positive feedback with feedback factor $\beta = R_1 / (R_1+R_2)$ is provided to the non-inverting terminal. When Vo= +Vsat, the capacitor C starts to charge to + Vsat through R. when the capacitor voltage crosses +βVsat, output switches from +Vsat to –Vsat. Now the voltage appearing at the non-inverting terminal is –βVsat and capacitor discharges through R towards -Vsat. When the capacitor voltage crosses –βVsat, the output switches from –Vsat to + Vsat and this process continues to generate square wave output with time period $T=T_{on} + T_{off}= 2RC \ln[(1+\beta)/(1-\beta)]$. In asymmetrical astable multivibrators, the charging and discharging time of capacitor is made unequal to get asymmetrical square wave with different T_{on} and T_{off}.

Procedure:

1. Check the components.
2. Setup the symmetric astable multivibrator circuit on the breadboard and check the connections.
3. Switch on the power supply.
4. Observe output and capacitor voltage on two channels of the oscilloscope simultaneously.
5. Draw the waveforms on the graph.
6. Measure the frequency of oscillation and duty cycle .
7. Repeat the procedures for asymmetric astable multivibrator.

a) SYMMETRICAL ASTABLE MULTIVIBRATOR

Circuit Diagram:



SYMMETRICAL ASTABLE MULTIVIBRATOR

Design:

Given $f = 1$ KHz

So $T = 1/f = 1ms$

And $\beta = R_1 / (R_1+R_2)$

Let $R_1 = 10K\Omega$, and $R_2 = 10K\Omega$

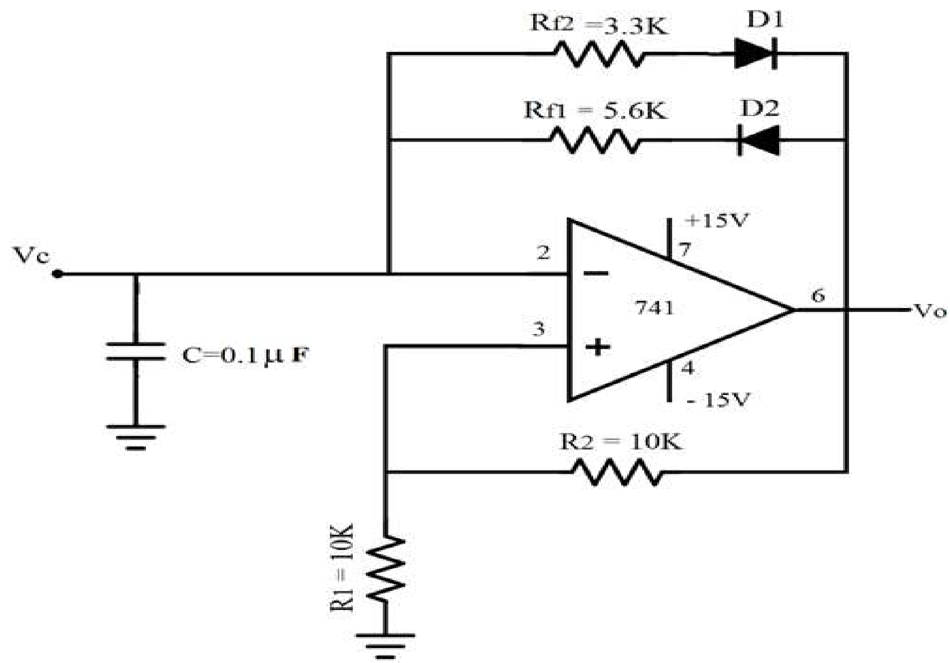
Then $\beta = 0.5$

Therefore $T = 2.2RC$

Let $C = 0.1\mu F$ Then $R =$
$4.7K\Omega$

b) ASYMMETRCAL ASTABLE MULTIVIBRATOR

Circuit Diagram:



ASYMMETRICAL ASTABLE MULTIVIBRATOR

Design:

Given $f = 1$ KHz

So $T = T_{on} + T_{off} = 1/f = 1ms$

Also Duty cycle $= T_{on}/(T_{on}+T_{off}) = 0.66$ or 66% Solving above two equations, $T_{on} = 0.66ms$ $T_{off} = 0.33ms$

For β=0.5,

$T_{on}= 1.1R_{f1}C = 0.66ms$

Let $C = 0.1\mu F$

Then $R_{f1} = 6.2K\Omega = 5.6K\Omega$ (Std)

Similarly $T_{off} = 1.1R_{f2}C = 0.33ms$

Then $R_{f2} = 3K\Omega = 3.3 K\Omega$ (Std)

bservation:

a) **Symmetrical astable multivibrators**

V_o(p-p)= ?

f = ?

Duty cycle = ?

b) **Asymmetrical astable multivibrators**

V_o(p-p)= ?

f = ?

Duty cycle = ?

Graph:

a) **Symmetrical astable multivibrators**

b) **Asymmetrical astable multivibrators**

Result:

MONOSTABLE MULTIVIBRATOR USING OP AMP

Aim: To design and setup a monostable multivibrator using Op-amp 741 and

(i) Plot the waveforms (ii) Measure the time delay

Objectives: After completion of this experiment the students are able to design and set up the monostable multivibrator circuit and delay circuits.

Equipments/Components

Sl. No.	Name and spification		Quantity
1	Dual Power Supply	+/-15V	1
2	Resistors		4
3	Capacitor	0.1μF;0.01μF	1 each
4	IC μA 741		1
5	Function generator	(0-1)MHz	1
6.	Oscilloscope		1
7.	Diode	1N 4001	2
8	Bread board		1
9	Connecting wires and probes		As required

Theory:

The monostable multivibrator is also called as one shot multivibrator. The circuit produces a single pulse of specified duration in response to each external trigger pulse. It always has one stable state (+Vsat). When an external trigger is applied, the output state changes and the new state is called quasi stable state (-Vsat). The circuit remains in this state for a fixed interval of time and then it returns to the original state after this interval. This time interval is determined discharging of the capacitor from 0.7V to -βVsat.The time period of quasi stable state or the delay is given by

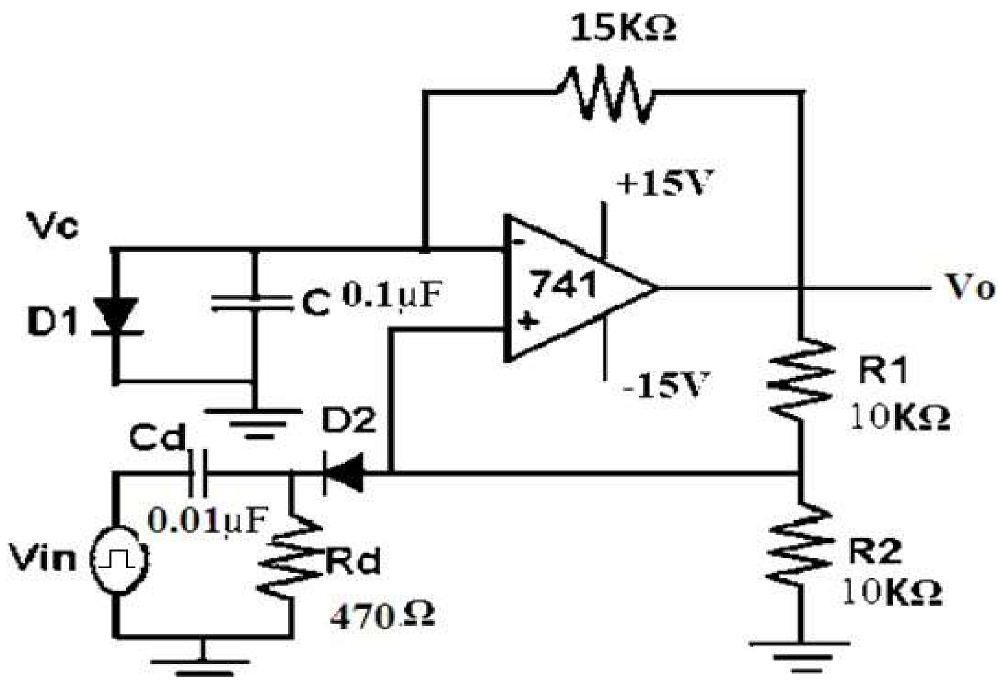
$$T = 0.69RC$$

Procedure:

1. Check the components.
2. Setup the monostable multivibrator circuit on the breadboard and check the connections.
3. Switch on the power supply.

4. Put the function generator output to square wave mode. Adjust the amplitude to 5V.
5. Observe trigger input , output and capacitor voltage on different channels of the oscilloscope simultaneously.
6. Draw the waveforms on the graph.
7. Measure the time delay .

Circuit Diagram:



Design:

Time Period T= 0.69RC

Let T = 1ms; and C = 0.1µF.

Then R = 15KΩFeedback factor

$\beta = R_2/(R_1+R_2)$ Let $\beta = 0.5$ or 1/2

$R_2/(R_1+R_2) = 1/2$

If R_1=10KΩ ; R_2 =10KΩ

For triggering circuit

$R_d C_d = 0.0016t$

Let $t = 3ms$ and $C_d = 0.01\ \mu F$; then Rd = 470Ω D_1 & D_2 are diodes 1N 4001

Observations:

Measured time period or delay =?

Graph:

Result:

Experiment No.6

SCHMITT TRIGGER

Aim: To design and setup a Schmitt trigger, plot the input output waveforms and measure V_{UT} and V_{LT}.

Objectives: After completion of this experiment, student will be able to design and setup a Schmitt trigger circuit using OP AMP.

Equipments/Components:

Sl .No	Name and Specification	Quantity required
1	Dual power supply +/- 15V	1
2	Function generator(0- 1MHz)	1
3	Oscilloscope	1
4	Bread board	1
5	IC 741C	1
6	Resistor	3
7	Probes and connecting wires	As required.

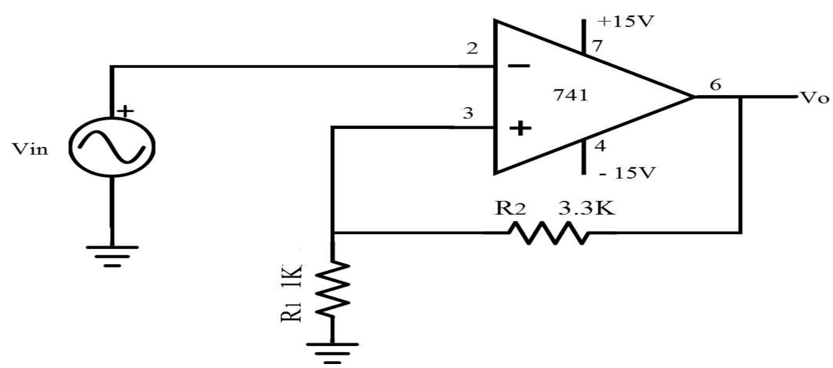
Theory:

It is a regenerative comparator or it is a comparator with hysteresis. This circuit uses positive feedback and the op-amp is operated in saturation. The output can take two values
+Vsat and –Vsat. When output = +Vsat, the voltage appearing at the non-inverting terminal is V_{UT} or UTP = +Vsat(R_1/R_1+R_2) called the upper threshold point. Similarly When output = - Vsat, the voltage appearing at the non-inverting terminal is V_{LT} or LTP = -Vsat(R_1/R_1+R_2) called the lower threshold point. When Vin is greater than UTP, the output will switch from +Vsat to –Vsat. Similarly When Vin is less than LTP; the output will switch from -Vsat to +Vsat which is shown in the graph. The difference between UTP-LTP is called hysteresis. Hysteresis avoids false triggering of the circuit by noise. Hysteresis curve is the plot of Vo versus Vin . Schmitt trigger circuit is used to convert any irregular wave into square wave.

Procedure:

1. Check the components.
2. Setup the circuit on the breadboard and check the connections.
3. Switch on the power supply.
4. Give V_i= 10 Vpp / 1KHz sine wave.
5. Observe input and output on two channels of oscilloscope simultaneously.
6. Note down and draw the input and output waveforms on the graph.

Circuit Diagram



Design:

UTP = +Vsat(R_1/R_1+R_2)

Let UTP = +3V and LTP = -3V,

Vsat=+13V

UTP, +3 = +13(

R_1/R_1+R_2) Let R_1 = 1 KΩ

Then R_2 = 3.3KΩ

Observations:

UTP=

LTP =

Graph:

Result:

Experiment no. 7

AIM: To design, construct and plot the frequency response of second order low pass having the fc of 1 kHz.

APPARATUS REQUIRED:

S.No.	Name of the Apparatus	Range/Value	Qty
1.	Bread Board	-	1
2.	IC Power Supply	±15 V	1
3.	Resistor	10 k Ω, 5.86 k Ω 1.6 k Ω	1 2
4.	IC 741 Op-Amp	-	1
5.	CRO	20 MHz.	1
6.	Function Generator	0-3MHz.	1
7.	Capacitor	0.1µF	2
8.	Connecting Wires	-	Few

THEORY:

An improved filter response can be obtained by using a second order active filter. A second order filter consist of two RC pairs has a roll-off rate of –40db/decade. The transfer function of a Low pass filter is $H(s)$. For n=2, the damping factor α = 1.414, the pass band gain A_0 = 3 – α = 1.586. Cutoff frequency of the filter = 1/ 2π RC= □h. HPF is the complement of the Low pass filter and can be obtained simply by interchanging R and C in the low pass configuration

DESIGN:

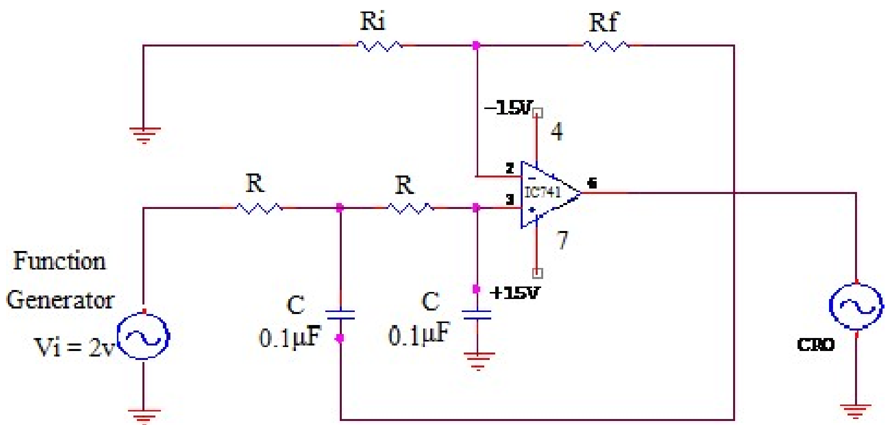
f_c = 1KHz, Assume C = 0.1µF, R= 1/2πfcC=

The gain for the second order filter is known as 1.5816.

Let R_i = 10KΩ, Gain =Ao = 1.5816 => 1+ R_f / R_i = 1.586 => R_f = 0.586

CIRCUIT DIAGRAM:

Low Pass Filter



PROCEDURE:

1. Connect the Low pass filter circuit as shown in the circuit diagram.
2. Give an input signal Vi of 2V(p-p) and measure the output voltage for different frequency from theCRO.
3. Plot the frequency response 20 log V_o/V_i versus input frequency and find 3db frequency.
4. Determine the cut-off frequency from the plot.

TABULATION:

Low Pass Filter **INPUT VOLTAGE: V_i =** volts

Frequency Hz	Output voltage Vo volts	Gain in db 20 logV_o/V_i

Result -

Experiment no. 8

AIM: To design, construct and plot the frequency response of second order High pass filter having the f_c of 1 kHz.

APPARATUS REQUIRED:

S.No.	Name of the Apparatus	Range/Value	Qty
1.	Bread Board	-	1
2.	IC Power Supply	±15 V	1
3.	Resistor	10 k Ω, 5.86 k Ω 1.6 k Ω	1 2
4.	IC 741 Op-Amp	-	1
5.	CRO	20 MHz.	1
6.	Function Generator	0-3MHz.	1
7.	Capacitor	0.1μF	2
8.	Connecting Wires	-	Few

THEORY:

An improved filter response can be obtained by using a second order active filter. A second order filter consist of two RC pairs has a roll-off rate of –40db/decade. The transfer function of a Low pass filter is $H(s)$. For n=2, the damping factor α = 1.414, the pass band gain A_0 = 3 – α = 1.586. Cutoff frequency of the filter = 1/ 2π RC= ω_h. HPF is the complement of the Low pass filter and can be obtained simply by interchanging R and C in the low pass configuration

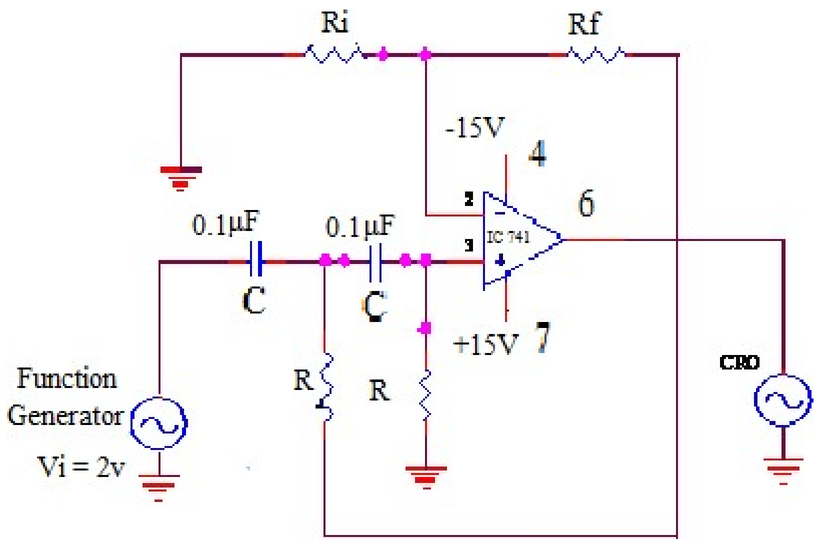
DESIGN:

f_c = 1KHz, Assume C = 0.1μF, R= 1/2πfcC=

The gain for the second order filter is known as 1.5816.

Let R_i = 10KΩ, Gain =Ao = 1.5816 => 1+ R_f / R_i = 1.586 => R_f = 0.586 R_i =

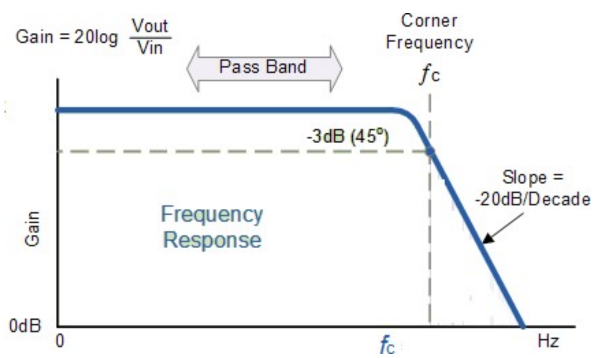
Second order High Pass Filter:



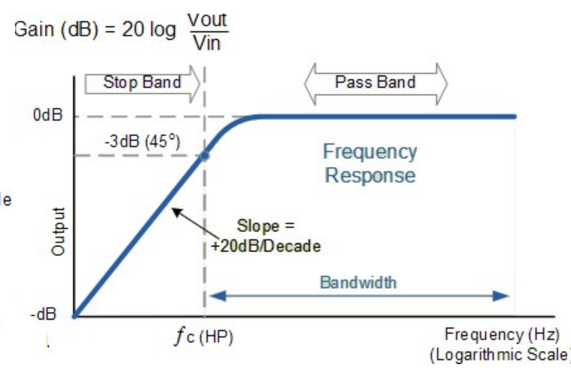
High Pass Filter: **INPUT VOLTAGE: Vi** = volts

Frequency Hz	Output voltage Vo volts	Gain in db 20 logVo/Vi

MODEL GRAPH:



Low Pass Filter:



High Pass Filter:

RESULT:

Thus the Second order low pass filter and High pass filter was designed and frequency response plot wasdrawn.

LPF: i. Theoretical = ii. Practical =

HPF: i. Theoretical = ii. Practical =

EXPERIMENT NO. 9

BANDPASS FILTER

AIM: To design, construct, test and to plot the frequency response of wide band pass filter.

APPARATUS REQUIRED:

S.No.	Name of the Apparatus	Range/Value	Qty
1.	Bread Board	-	1
2.	IC Power Supply	±15 V	1
3.	Resistor	10 k Ω, 39.8 k Ω, 7.9 k Ω	4, 1
4.	IC 741 Op-Amp	-	1
5.	CRO	20 MHz.	1
6.	Signal Generator	0-3 MHz.	1
7.	Capacitor	0.01μF	2
8.	Connecting Wires	-	Few

THEORY:

A wide band pass filter can be formed by cascading a HPF and LPF section. If the HPF and LPF are of the first order, then the band pass filter (BPF) will have a roll off rate of -20 dB/decade. A wide band pass filter formed by cascading I order HPF and I order LPF is shown in the circuitdiagram.

DESIGN:

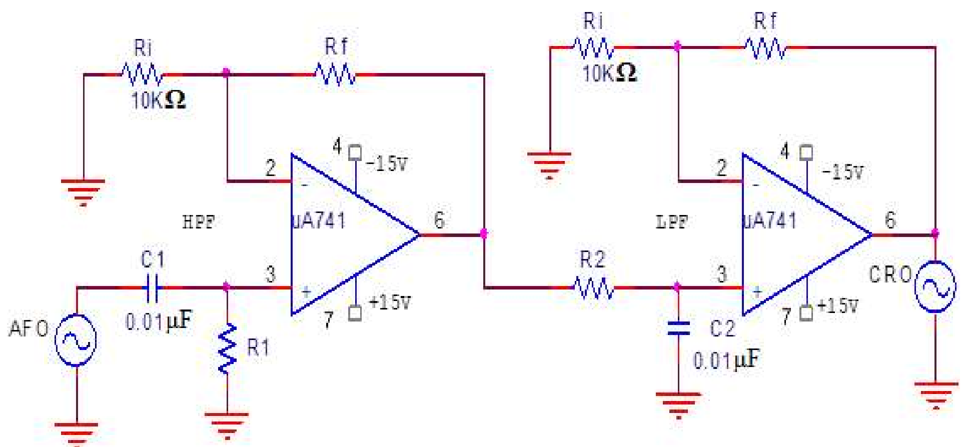
f_h = 2KHz; f_l = 400Hz; pass band gain A_0= 4.

LPF and HPF sections may be designed to have a gain of 2.

As the opamp is used in non-inverting configuration $A_o = 1+ (R_f/R_i) = 2 \Rightarrow R_f/R_i =$
$1 \Rightarrow R_f = R_i$.Let R_i =10 kΩ, R_f = .

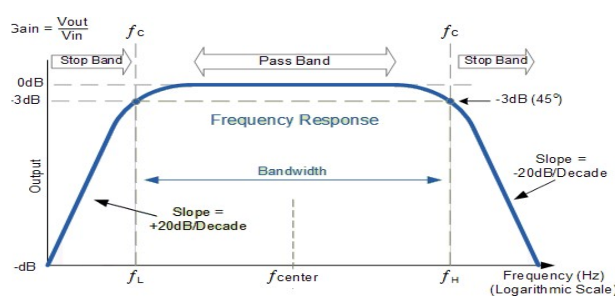
$f_h = 1/(2\pi R_2 C_2)$ = 2KHz. Let C_2= 0.01μF, R_2 =
$1/(2\pi X2X10^3X0.01X10^{-6})$ =$f_l = 1/(2\pi R_1 C_1)$ = 400Hz. Let C_1=
0.01μF, $R_1 = 1/(2\pi X400X0.01X10^{-6})$

CIRCUIT DIAGRAM



PROCEDURE:

1. Connect the Band pass filter circuit as shown in the circuit diagram.
2. Give an input signal V_i of 1V (p-p) and measure the output voltage for different frequency.
3. Plot the frequency response 20 log V_o/V_i versus input frequency and find 3db frequency.
4. Determine the cut-off frequency f_h and f_l .



TABULATION: **INPUT VOLTAGE: V_i** = volts

Frequency (Hz)	Output voltage V_o (volts)	Gain in db 20 logV_o/V_i

RESULT:

Thus the Second order Band pass filter was designed and frequency response plot was drawn

Lower cutoff frequency: i. Theoretical = ii. Practical =

Upper cutoff frequency: i. Theoretical = ii. Practical

Experiment No. 10

RC PHASE SHIFT OSCILLATOR USING OP AMP

Aim: To Design and setup a RC phase shift oscillator using Op-Amp 741 and (i) Plot the output waveform (ii) Measure the frequency of oscillation

Objectives: After completion of this experiment the students are able to design and set up the RC phase shift oscillator for desired frequency.

Equipments/Components

Sl. No.	Name and specification	Quantity
1	Dual Power Supply +/-15V	1
2	Resistors	5
3	Capacitor 0.01µF	3
4	IC µA 741	1
5	Oscilloscope	1
6	Bread board	1
7	Connecting wires and probes	As required

Theory:

RC phase shift oscillator uses op-amp, in inverting amplifier mode and the circuit generates its own output signal. It consists of an op-amp as an amplifier and 3 RC cascaded network as the feedback circuit. Since the op-amp is used in the inverting mode, any signal that appears at the inverting terminal is shifted by 180^0 at the output. An additional 180^0 phase shift required for oscillation is provided by the cascaded RC network. Thus the total phase shift around the circuit is 360^0 or 0^0 . At some specific frequency, the phase shift of the cascaded RC network is exactly 180^0 and feedback factor is 1/29. If the gain of the amplifier is 29, the total loop gain of the circuit becomes 1. The circuit will oscillate at this specific frequency and is given by

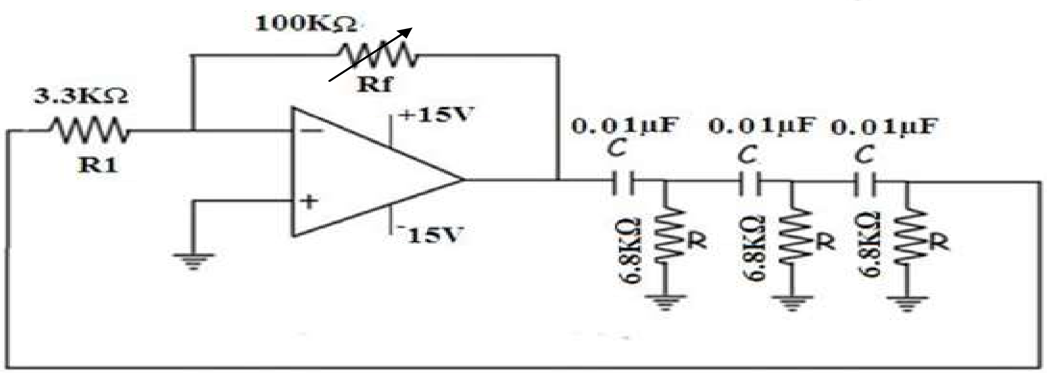
$$f_{\text{oscillation}} = \frac{1}{2\pi RC\sqrt{6}}$$

Procedure:

1. Check the components.
2. Setup the RC phase shift oscillator circuit on the breadboard .
3. Switch on the power supply.
4. Observe output voltage on oscilloscope.
5. Draw the waveforms on the graph.
6. Measure the frequency of oscillation .

Result:

Circuit Diagram:



$$f_{\text{oscillation}} = \frac{1}{2\pi RC\sqrt{6}}$$

Design:

Let f = 1 KHz, and C= 0.01µF

R = 6.8KΩ

Gain = 29

$R_f/R_1 = 29$

If R1 = 3.3KΩ ; Rf = 95.7KΩ Use 100KΩ pot

Result

EXPERIMENT NO.11

WIEN BRIDGE OSCILLATOR USING OP AMP

Aim: To design and construct a Wien bridge oscillator using Op-Amp 741 and

(i) Plot the output waveform (ii) Measure the frequency of oscillation

Objectives: After completion of this experiment the students are able to design and set up the Wien oscillator for desired frequency

Equipments/Components

Sl. No.	Name and specification	Quantity
1	Dual Power Supply +/-15V	1
2	Resistors	4
3	Capacitor 0.1µF	2
4	IC µA 741	1
5	Oscilloscope	1
6	Bread board	1
7	Connecting wires and probes	As required

Theory:

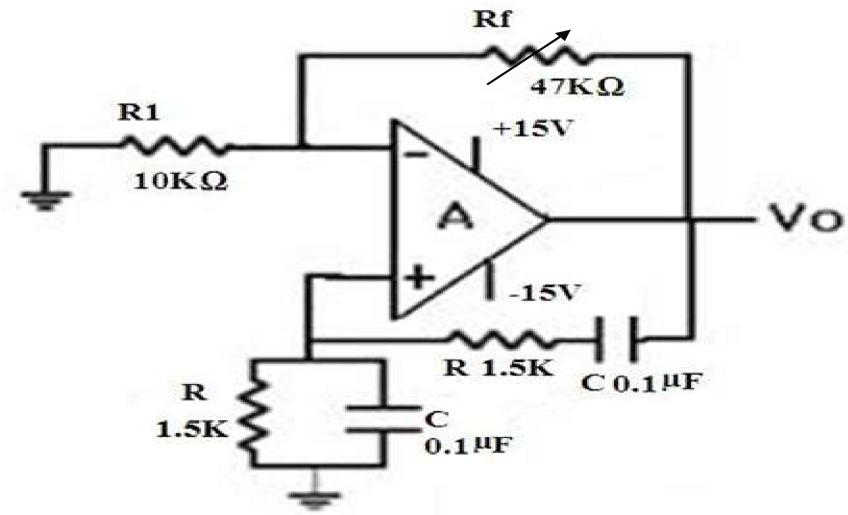
It is the commonly used audio frequency oscillator which employs both positive and negative feedback. The feedback signal is connected in the non-inverting input terminal so that the amplifier is working in non-inverting mode. The Wien bridge circuit is connected between amplifier input terminal and output terminal. The bridge has a series RC network in one arm and a parallel RC network in the adjoining arm. In the remaining two arms of the bridge, resistor R1 and Rf are connected. The phase angle criterion for oscillation is that the total phase shift around the circuit must be zero. This condition occurs when bridge is balanced. At resonance, the frequency of oscillation is exactly the resonance frequency of balanced Wien bridge and is given by $f_0 = 1/(2\pi RC)$. At this frequency, the gain required for sustained oscillation is 3

Procedure:

1. Check the components.
2. Setup the Wien bridge oscillator circuit on the breadboard and check the connections.
3. Switch on the power supply.

4. Observe output voltage on oscilloscope.
5. Draw the waveforms on the graph.
6. Measure the frequency of oscillation .

Circuit Diagram:



Design:

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

Let f = 1KHz , and C= 0.1μF

R=1.5KΩ

Gain=3

$1+ (R_f/R_1) =3$

If R_1 = 10KΩ ,R_f = 20KΩ Use 47KΩ pot

Observations:

Measured frequency of oscillation is =?

RESULT