SVR ENGINEERING COLLEGE

Approved by AICTE & Permanently Affiliated to JNTUA

Ayyalurmetta, Nandyal – 518503. Website: www.svrec.ac.in

Department of Electronics and Communication Engineering

(15A04507P) INTEGRATED CIRCUITS AND APPLICATIONS LAB

III B. Tech (ECE) I Semester 2021-22

STUDENT NAME	
ROLL NUMBER	
SECTION	

SVR ENGINEERING COLLEGE

Approved by AICTE & Permanently Affiliated to JNTUA

Ayyalurmetta, Nandyal – 518503. Website: www.svrec.ac.in

DEPARTMENT OF

ELECTRONICS AND COMMUNICATION ENGINEERING

CERTIFICATE

ACADEMIC YEAR: 2021-22

This is to certify that the bonafide record work done by Mr./Ms.____ bearing H.T.NO. _____________________ of III B. Tech I Semester in the

Integrated circuits and applications laboratory.

Faculty In-Charge **Head of the Department**

ECE DEPT VISION & MISSION PEOs and PSOs

Vision

To produce highly skilled, creative and competitive Electronics and Communication Engineers to meet the emergingneeds of the society.

Mission

- Impart core knowledge and necessary skills in Electronics and Communication Engineering throughinnovative teaching and learning.
- Inculcate critical thinking, ethics, lifelong learning and creativity needed for industry and society
- Cultivate the students with all-round competencies, for career, higher education and self-employability

I. PROGRAMME EDUCATIONAL OBJECTIVES (PEOS)

PEO1: Graduates apply their knowledge of mathematics and science to identify, analyze and solve problems in the field of Electronics and develop sophisticated communication systems.

PEO2: Graduates embody a commitment to professional ethics, diversity and social awareness in theirprofessional career.

PEO3: Graduates exhibit a desire for life-long learning through technical training and Professional activities.

II. PROGRAM SPECIFIC OUTCOMES (PSOS)

PSO1: Apply the fundamental concepts of electronics and communication engineering to design a variety of components and systems for applications including signal processing, image processing, communication, networking, embedded systems, VLSI and control system

PSO2: Select and apply cutting-edge engineering hardware and software tools to solve complex Electronics and Communication Engineering problems.

III. PROGRAMME OUTCOMES (PO'S)

1. Engineering knowledge: Apply the knowledge of mathematics, science, engineering fundamentals, and anengineering specialization to the solution of complex engineering problems.

2. Problem analysis: Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.

3. Design/development of solutions: Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety,and the cultural, societal, and environmental considerations.

4. Conduct investigations of complex problems: Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.

5. Modern tool usage: Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.

6. The engineer and society: Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.

7. Environment and sustainability: Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.

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

9. Individual and team work: Function effectively as an individual, and as a member or leader in diverse teams, andin multidisciplinary settings.

10. Communication: Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.

11. Project management and finance: Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects andin multidisciplinary environments.

12. Life-long learning: Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.

IV. COURSE OBJECTIVES

- To familiarize different Analog ICs.
- To implement linear and nonlinear application circuits by Op amp.
- To realize active filters using Op amp.
- To design of various multi-vibrator circuits using 555 timer application
- To design and Understand the working of mixed signal circuits like Analog to Digital Convertors, Digital to analog Convertors and Phase Locked Loop.
- To understand the working of a few application specific analog ICs and to design circuits based on these ICs.

V. COURSE OUTCOMES

After the completion of the course students will be able to

Course Outcomes	Course Outcome statements	BTL
CO1	Understand the working of Op amp ICs & Application specific analog ICs.	L1
CO2	Analyze operational amplifier based circuits for linear and non-linear applications.	L2
CO3	Design Operational amplifiers for linear and nonlinear application, Multivibrator circuits using 555 & application specific ICs.	L3
CO4	Simulate all linear and nonlinear application based Op amp Circuits and circuits based on application specific ICs.	L4
CO5	Compare theoretical, practical & simulated results in integrated circuits.	L5

VI. COURSE MAPPING WITH PO'S AND PEO'S

CourseTitle	PO 1	PO 2	PO 3	PO 4	PO 5	PO 6	PO 7	PO 8	PO 9	PO 10	PO 11	PO 12	PSO 1	PSO 2
Integrated Circuits and Applications Lab	3.0	2.4	2.0	3.0	2.0	2.7	2.2	1.5	2.0	2.0	2.0	2.0	2.4	1.6

VII. MAPPING OF COURSE OUTCOMES WITH PEO'S AND PO'S

Course Title	PO 1	PO 2	PO 3	PO 4	PO 5	PO 6	PO 7	PO 8	PO 9	PO 10	PO 11	PO 12	PSO 1	PSO 2
CO1	3	2	2	3	2	3	3	1	2	2	3	1	3	2
CO2	3	2	2	3	2	3	3	1	1	3	1	3	3	2
CO3	3	3	2	3	2	2	3	2	3	2	2	2	3	1
CO4	3	3	2	1	2	1	1	2	2	1	1	1	2	2
CO5	3	2	2	2	2	2	1	2	1	1	1	1	1	1

LABORATORY INSTRUCTIONS

1. While entering the Laboratory, the students should follow the dress code. (Wear shoes and White apron,Female Students should tie their hair back).

2. The students should bring their observation book, record, calculator, necessary stationery items And graphsheets if any for the lab classes without which the students will not be allowed for doing The experiment.

3. All the Equipment and components should be handled with utmost care. Any breakage or damage Will becharged.

4. If any damage or breakage is noticed, it should be reported to the concerned in charge immediately.

5. The theoretical calculations and the updated register values should be noted down in the Observation bookand should be corrected by the lab in-charge on the same day of the laboratory Session.

6. Each experiment should be written in the record note book only after getting signature from the lab In-charge in the observation notebook.

7. Record book must be submitted in the successive lab session after completion of experiment.

8. 100% attendance should be maintained for the laboratory classes.

Precautions.

1. Check the connections before giving the supply.
2. Observations should be done carefully.

Lab syllabus copy

B. Tech III-I Sem. (ECE)

15A04507 IC APPLICATIONS LABORATORY

All experiments are based upon 741 / TL 082/ASLK Kits.

1. Study the characteristics of negative feedback amplifier
Aim: Design the following amplifiers:
a) A unity gain amplifier
b) A non-inverting amplifier with a gain of "A"
c) An inverting amplifier with a gain of „A"
Apply a square wave of fixed amplitude and study the effect of slew rate on the three type of amplifiers.

2. Design of an instrumentation amplifier
Aim: Design an instrumentation amplifier of a differential mode gain of „A" using three Amplifiers

3. Study the characteristics of regenerative feedback system with extension to designAn astable multivibrator
Aim: Design and test an astable multivibrator for a given frequency.

4. Study the characteristics of integrator circuit
Aim: Design and test the integrator for a given time constant.

5. Design of Analog filters – I
Aim: Design a second order butter worth band-pass filter for the given higher and lower Cut-off frequencies.

6. Design of Analog filters – II
Aim: Design and test a notch filter to eliminate the 50Hz power line frequency.

7. Design of a self-tuned Filter
Aim: Design and test a high-Q Band pass self-tuned filter for a given center frequency.

8. Design of a function generator
Aim: Design and test a function generator that can generate square wave and Triangular wave output for a given frequency.

9. Design of a Voltage Controlled Oscillator
Aim: Design and test voltage controlled oscillator for a given specification (voltage Range and frequency range).

10. Design of a Phase Locked Loop (PLL)
Aim: Design and test a PLL to get locked to a given frequency „f". Measure the locking Range of the system and also measure the change in phase of the output signal as input Frequency is varied within the lock range.

11. Automatic Gain Control (AGC) Automatic Volume Control (AVC)
Aim: Design and test an AGC system for a given peak amplitude of sine-wave Output.

12. Design of a low drop out regulator
Aim: Design and test a Low Dropout regulator using op-amps for a given voltage Regulation characteristic and compare the characteristics with TPS7250 IC

13. DC-DC Converter
Aim: Design of a switched mode power supply that can provide a regulated output Voltage for a given input range using the TPS40200 IC

INDEX

BEYOND THE SYLLABUS

Experiment No: 1 **Date:**

STUDY THE CHARACTERISTICS OF NEGATIVE FEED BACK AMPLIFIER
(Inverting, Non-Inverting and Unity Gain Amplifier)

AIM:

To Design the following Negative Feedback amplifiers:

a) Unit gain amplifier (Inverting or Non-inverting).
b) A Non-inverting amplifier with gain'(A_{CL})'.
c) An inverting amplifier with gain'(A_{CL})'.
d) Apply a square wave of fixed amplitude and study the effect of slew rate on the three types of amplifiers.

APPARATUSREQUIRED:

S. No.	Name of the Apparatus	Range	Quantity
1.	Function Generator	3MHz	1
2.	CRO	30MHz	1
3.	Dual RPS	0-30V	1
4.	ALSK PRO Kit	TL082	1
5.	CRO probes	--------	3
6.	Connecting connectors	--------	As required

THEORY:

An inverting-amplifier circuit is built by grounding the positive input of the operational amplifier and connecting resistors R1 and R2, called the feedback networks, between the inverting input and the signal source and amplifier output node, respectively. With assumption that reverse-transfer parameter is negligibly small, open-circuit voltage gain Av, input resistance Zin and output resistance Zo can be calculated.

The operational amplifier can also be used to construct a non-inverting amplifier with the circuit indicated below. The input signal is applied to the positive or non-inverting input terminal of the operational amplifier, and a portion of the output signal is fed back to the negative input terminal. Analysis of the circuit is performed by relating the voltage at V_2 to both the input voltage V_{in} and the output voltage V_o.
The output is applied back to the inverting (-) input through the feedback circuit (closed loop) formed by the input resistor R_1 and the feedback resistor R_2. This creates ve feedback as follows. Resistors R_1 and R_2 form a voltage-divider circuit, which reduces V_o and connects the reduced voltage V_2 to the inverting input.

A voltage follower is also known as a unity gain amplifier, a voltage buffer, or an isolation amplifier. In a voltage follower circuit, the output voltage is equal to the input voltage; thus, it has a gain of one (unity) and does not amplify the incoming signal......Op amps also have a very low output impedance

CIRCUIT DIAGRAMS:

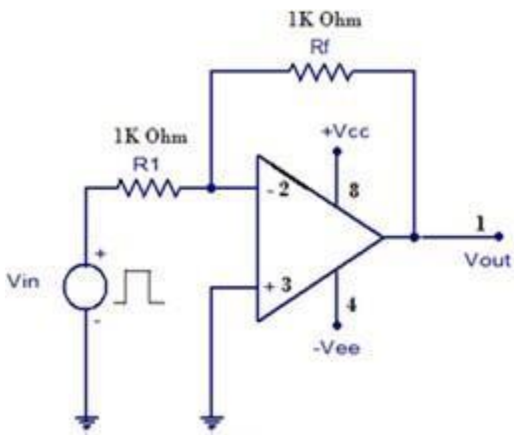


FIG INVERTING AMPLIFIER

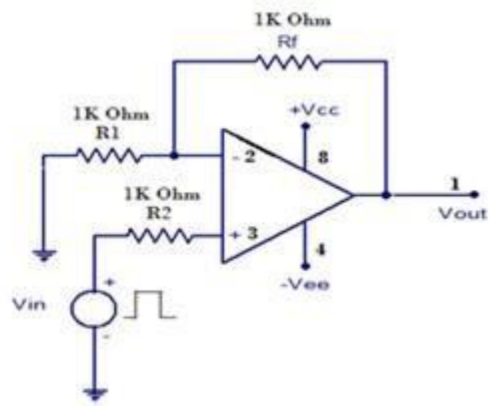


FIG NON- INVERTING AMPLIFIER

PROCEDURE:

For Inverting Amplifier

1) Connect the circuit as per the given circuit diagram.
2) Apply biasing voltage at pins 4 and8 as-10Vand+10Vrespectively.
3) Apply a square wave input $2V_{p-p}$ 1-KHtotheinvertingterminalatpinNo.2usingfunctiongenerator.
4) Observetheoutputatthepinno.1in CRO by changing feedback resistance (R_f) for desired gain.
5) Also observe the slew-rate i.e., response time of TL082forchangeof input in the CRO.
6) Compare the theoretical and practical
7) values.

For Non-Inverting Amplifier

1) Connect the circuit as per the given circuit diagram.
2) Apply biasing voltage at pins 4 and8 as-10Vand+10Vrespectively.
3) Apply a square waveinput2V_{p-p} and $1KH_Z$ to the inverting terminal at pin No.2 using function generator.
4) Observe the output at the pin no.1 in CRO by changing feedback resistance (R_f) for desired gain.
5) Also observe the slew-rate i.e., response time of TL082forchangeof input in the CRO.
6) Compare the theoretical and practical values.

EXPECTEDWAVE FORMS:

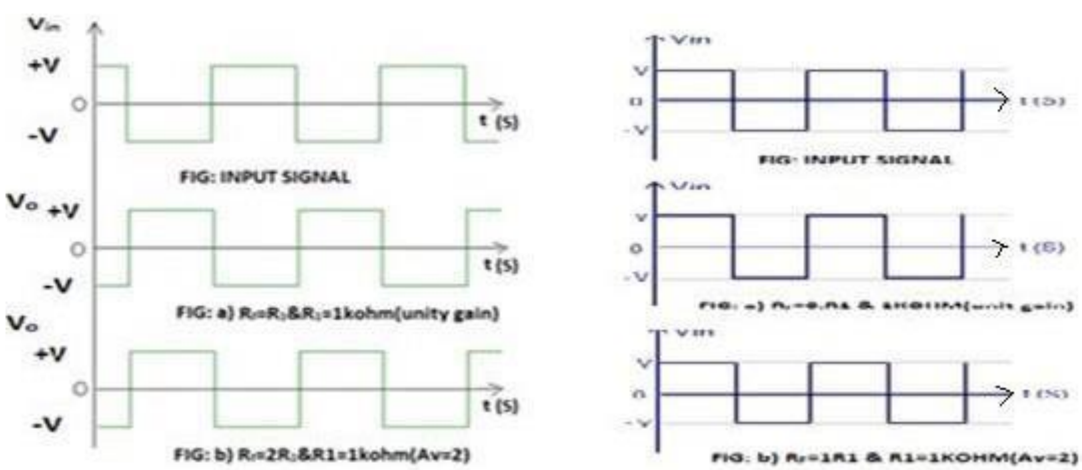


FIG: INVERTING AMPLIFIER FIG: NON-INVERTING AMPLIFIER

1. Peak-to-peak Amplitude(Vo) =

OBSERVATIONS:

1. Observe the output waveform from CRO. An inverted and amplified waveform will be observed.
2. Measure the input and output voltage from the input and output waveform in the CRO.
3. calculate

$$V0 = -\frac{R_2}{R_1} V_{IN}$$

4. Compare the theoretical voltage gain from the above equation with the experimental value obtained by dividing output voltage by input voltages observed.
5. Observe outputs of the inverting amplifier circuit using different input waveforms.

TABULARCOLUMN:

For Inverting Amplifier

S. No.	Gain (ACL)	$R1$	R_f	V_{in}	$V_{out-theo}$	$V_{out-practi}$	Practical Gain

For Non-Inverting Amplifier

S. No.	Gain (ACL)	$R1$	R_f	V_{in}	$V_{out-theo}$	$V_{out-practi}$	Practical Gain

RESULT:

We have studied and designed the Design the following Negative Feedback amplifiers:

a) Unit gain amplifier (Inverting or Non-inverting).
b) A Non-inverting amplifier with gain‘(ACL)’.
c) An inverting amplifier with gain‘(ACL)’.
d) Apply a square wave of fixed amplitude and study the effect of slew rate on the three types of amplifiers.

VIVA QUESTIONS:

1. What is an op-amp?
2. What is called a differential amplifier?
3. What is an inverting and non-inverting operational amplifier?
4. Non-inverting operational amplifier?
5. What are the applications of inverting amplifier, Non-inverting amplifier, Voltage follower and Summing amplifier?

Experiment No: 2 **Date:**

STUDY THE CHARACTERISTICS OF REGENERATIVE SYSTEM WITH EXTENSION TO DESIGN AN ASTABLE MULTIVIBRATOR

AIM:

To study the characteristics of regenerative feedback system with extension to design an Astable multi-vibrator using ASLK Pro Kit.

APPARATUS REQUIRED:

S. No.	Name of the Apparatus	Range	Quantity
1.	CRO	30MHz	1
2.	Dual RPS	0-30V	1
3.	ALSKPRO Kit	TL082	1
4.	CRO probes	--------	2
5.	Connecting connectors	--------	As required

THEORY:

A_multivibrator is a one type of electronic circuit that is used to implement a two state system like flip-flops, timers and oscillators. Multivibrators are categorized by two amplifying devices like electron tubes, transistors and other devices like capacitors and cross coupled by resistors. Multivibrators are classified into three types based on the circuit operation, namely Astable multivibrators, Bistable multivibrators and Monostable multivibrators. The astable multi vibrator is not stable and it repeatedly switches from one state to the other. In monostable multivibrator, one state is stable and remaining state is unstable. A trigger pulse is the root to the circuit to enter the unstable state. When the circuit enters into the unstable state, then it will return to the normal state after a fixed time. A bistable mutivibrator circuit is stable that can be changed from one stable to other stable by an external trigger pulse. This multi vibrator circuit is also called as flip-flop which can be used to store one bit of data.

CIRCUITDIAGRAM:

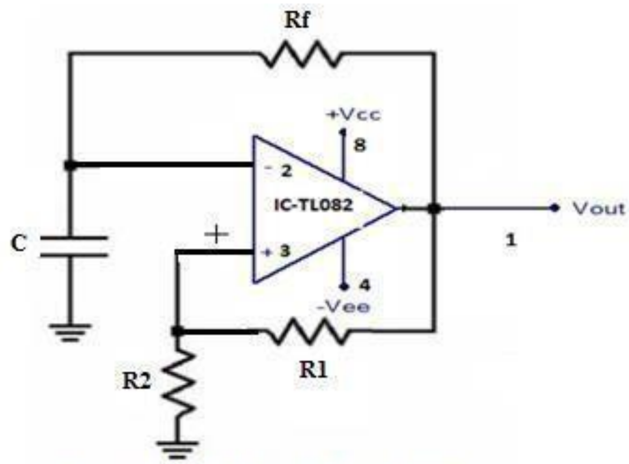
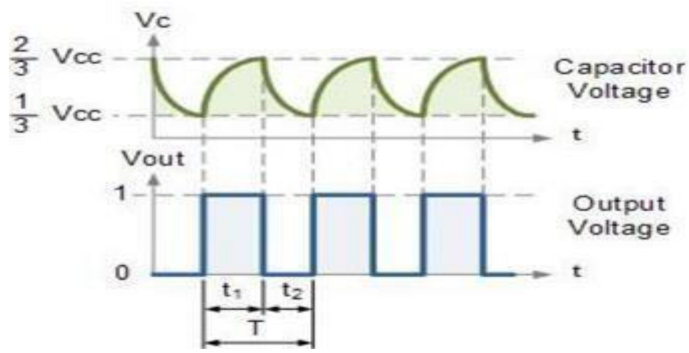


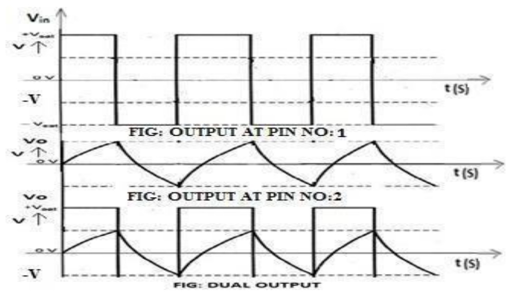
FIG: ASTABLE MULTIVIBRATOR

PROCEDURE:

1) Connect the circuit as per the circuit diagram.
2) In RPS the bias voltages are applied to both 4 and 8as -10V and + 10V respectively.
3) To obtain the $1KH_Z$ frequency we have to use R1=R2=1kΩ and C=0.1μF.
4) Observe the output of the capacitor by placing the CRO probe at pin no.2 .Here we can observe the charging and as well as discharging point.
5) Observe the square wave output at pin no.1.
6) Plot the output waveforms.

EXPECTED WAVE FORMS:





TABULAR COLUMNS:

S. No	Capacitor(µF)	Resistor(Rf)	Theoretical Time Period (ms)	TheoreticalFrequency	Practical Time - period(ms)	Practical Frequency (KHZ)

RESULT:

We studied the characteristics of regenerative feedback system with extension to designed an Astable multi-vibrator using ASLK Pro Kit and drawn the output wave forms on the graph sheet.

VIVA QUESTIONS:

1. What is astable multivibrator?
2. Why astable multivibrator is called astable?
3. What are applications of astable multivibrator?

Experiment No: 3 **Date:**

STUDY THE CHARACTERISTICS OF INTEGRATOR CIRCUIT

AIM:

To Design and test an integrator for a given time constant using ASLK Pro Kit.

APPARATUS REQUIRED:

S. No.	Name of the Apparatus	Range	Quantity
1	Function Generator	3MHz	1
2	CRO	30MHz	1
3	Dual RPS	0-30V	1
4	Resistors	1KΩ	1
		2.2KΩ	1
5	Capacitor	0.1µF	1
6	ALSKPRO Kit	TL082	1
7	CRO probes	--------	2
8	Connecting connectors	--------	As required

THEORY:

The circuit in fig 1 is an integrator, which is also a low-pass filter with a time constant=R_1C. When a voltage, V_{in} is firstly applied to the input of an integrating amplifier, the uncharged capacitor C has very little resistance and acts a bit like a short circuit (voltage follower circuit) giving an overall gain of less than 1, thus resulting in zero output. As the feedback capacitor C begins to charge up, its reactance X_c decreases and the ratio of Z_f/R_1 increases producing an output voltage that continues to increase until the capacitor is fully charged. At this point the ratio of feedback capacitor to input resistor (Z_f/R_1) is infinite resulting in infinite gain and the output of the amplifier goes into saturation. (Saturation is when the output voltage of the amplifier swings heavily to one voltage supply rail or the other with no control in between). The circuit design generate triangular wave providing square wave as input to the integrator. Hence, the integrator circuit generates integral output with respect to the input waveform.

CIRCUITDIAGRAM:

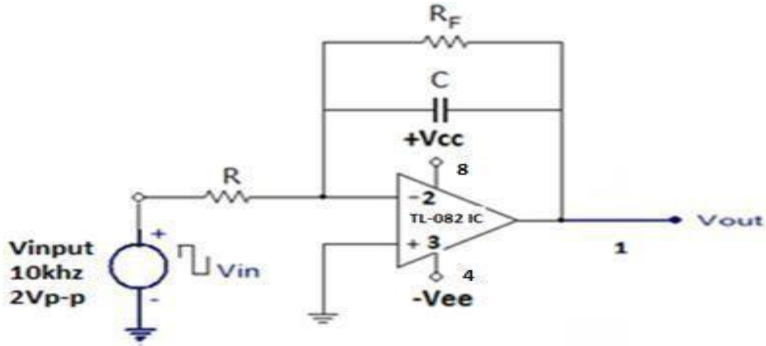


FIG: INTEGRATOR CIRCUIT

PROCEDURE:

1. Connect the circuit as shown in the circuit diagram.
2. Give the input signal as specified.
3. Switch on the power supply.
4. Note down the outputs from the CRO
5. Draw the necessary waveforms on the graph sheet.

OBSERVATIONS:

1. Observe the output waveform from CRO. A square wave will generate a triangular wave and sine wave will generate a cosine wave.
2. Measure the frequency and the voltage of the output waveform in the CRO.
3. Calculate

$$V_o = -\frac{1}{R_1 C}\int V_{in\,dt}$$

4. Compare the calculated output voltage with the experimentally observed voltage from the output waveform.
5. Observe outputs of the integrator circuit using different input waveforms.

PROCEDURE:

1. Connect the circuit as per the circuit diagram.
2. In RPS bias voltages are applied at 4 and 8 as-10V and +10V respectively.
3. Keep the capacitor 0.1µF at 2nd terminal to1st terminal.
4. The channels of the CRO are used as input and output.
5. A square wave 2Vp-p 10 KHz is supplied as input through function generator
6. The output is taken at the pin number no.1
7. Note down the outputs from the CRO
8. Draw the necessary waveforms on the graph sheet.

CALCULATIONS:

If input V_{in} = 2.09 sin (2*50*t)
Output of the integrator will be equal to

$$V_o = -\frac{1}{R_1 C}\int V_{in\,dt}$$

Thus,

$$V_o = -\frac{1}{1.369\times10^{3}\times10^{-6}}\int 2.09\sin(100\pi t)\,dt$$

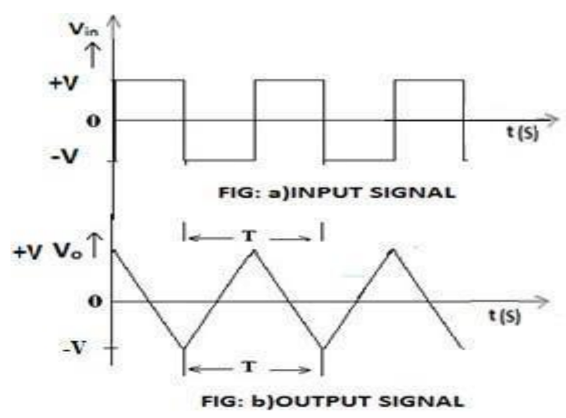
$$V_o = -\frac{2.09}{1.369\times10^{-3}\times100\pi}[-\cos(100\pi t)]$$

$$V_o = 4.72\cos(100\pi t)$$

TABULARCOLUMN:

S. No.	Observation	Peak Voltage(Vp-p)	Time Period(µS)

EXPECTED WAVE FORMS:



RESULT:

We studied and designed the integrator for a given time constant using ASLK Pro Kit and output wave forms have been drawn.

VIVA QUESTIONS:

1. What is integrator used for?
2. What is the output voltage of the integrator?
3. Why capacitor is used in integrator?
4. Which configuration is used in integrator?

Experiment No: 4 **Date:**

DESIGN OFA FUNCTION GENERATOR

AIM:

To Design and test a function generator that can generate square wave and triangular wave output for a given frequency using ALSK Pro Kit.

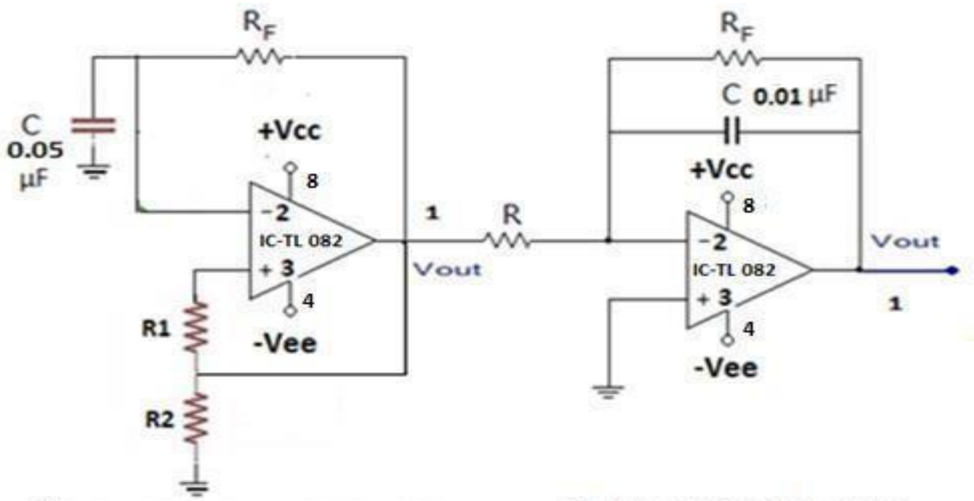
APPARATUSREQUIRED:

S. No.	Name of the Apparatus	Range	Quantity
1.	CRO	30MHz	1
2.	Dual RPS	0-30V	1
3.	ALSKPRO Kit	TL082	1
4	Resistors	1KΩ,	2
		2.2KΩ,	1
		4.7KΩ	1
5	Capacitors	0.1µF	2
6.	CRO probes	--------	2
7.	Connecting connectors	--------	As required

THEORY:

Signal generators which provide a variety of waveforms are referred to as unction generator. These are commonly available as test instruments. The variety of wave shapes provided generally determines the complexity and cost of the generator system. It facilitates such as voltage control of frequency and ability to provide a signal wave or group of the wave. Function generator system can be readily synthesized using operational amplifiers on an approach which uses full when the need for a special purpose generator arises or when a function generator is inconvenient or prohibited by cost consideration. The basic wave shapes produced by most function generators are square wave & triangular wave. These can be shaped by nonlinear amplifiers of other wave forms, including sinusoidal waveforms.

CIRCUITDIAGRAM:



A)FIG: ASTABLE MULTIVIBRATOR B)FIG: INTEGRATOR CIRCUIT

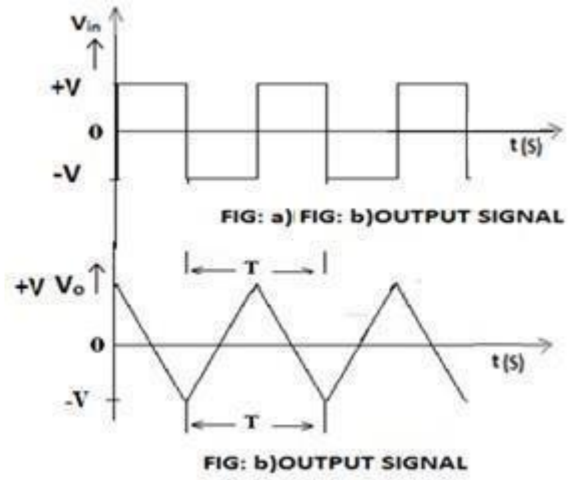
PROCEDURE:

1. Connect the circuit as per given the circuit diagram.
2. In RPS the bias voltage of 10V to both 4 and 8 as-10V and+10Vrespectively.
3. To obtain1khz frequencywehavetouseR1=R2=10kΩ and Rf=4.7kΩ ,2.2kΩ and C=0.1µF
4. Observe the square waveoutputatpinno.1withtheCRO.
5. Plot the output waveforms.
6. The output is connected to R1andconnect feedback resisitance Rf=to2ndand1stterminal.
7. Connect the channel of CRO at input and output.
8. Plot the wave forms are drawn on graph sheet

TABULAR COLUMN:

S. No.	Theoretical values	Practical values	
		Square wave	Triangular wave

EXPECTED WAVE FORMS:



RESULT:

We studied and Designed and test a function generator that can generate square wave and triangular wave output for a given frequency using ALSK Pro Kit and output wave forms are drawn.

VIVA QUESTIONS:

1. Who invented function generator?
2. Who invented function generator?
3. What are the types of function generator?

Experiment No: 5 **Date:**

DESIGN OF ANALOG FILTER-II

AIM:

Design and test a Notch filter to eliminate, 50Hz power line frequency.

APPARATUSREQUIRED:

S. No.	Name of the Apparatus	Range	Quantity
1.	Function Generator	3MHz	1
2.	CRO	30MHz	1
3.	Dual RPS	0-30V	1
4.	ALSK PRO Kit	TL082	1
5.	CRO probes	--------	2

COMPONENTSREQUIRED:

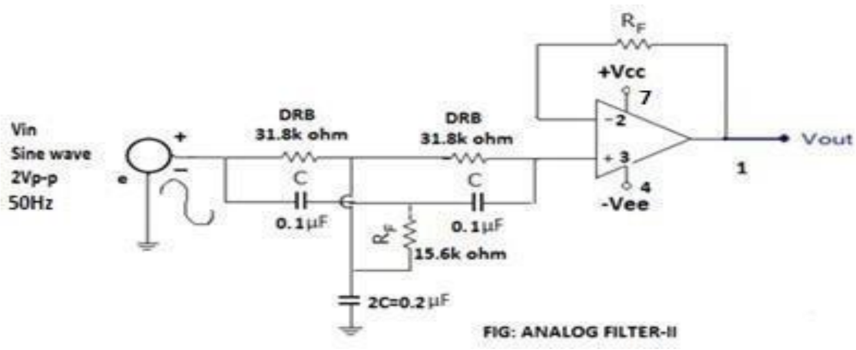
S. No.	Name of the Apparatus	Range	Quantity
1.	Resistors	31.8kΩ	2
2.	Decade Resistance Box	---------	2
3.	Capacitors	0.1μf	4
4.	IC-741	--------	1
5.	Breadboard	---------	1
6.	Connecting wires	--------	As required

THEORY:

Analogue filters are a basic building block of signal processing much used in electronics. Amongst their many applications are the separation of an audio signal before application to bass, mid-range, and tweeter loudspeakers; the combining and later separation of multiple telephone conversations onto a single channel; the selection of a chosen radio station in a radio receiver and rejection of others.

Passive linear electronic analogue filters are those filters which can be described with linear differential equations (linear); they are composed of capacitors, inductors and, sometimes, resistors (passive) and are designed to operate on continuously varying (analogue) signals. There are many linear filters which are not analogue in implementation (digital filter), and there are many electronic filters which may not have a passive topology – both of which may have the same transfer function of the filters described in this article. Analogue filters are most often used in wave filtering applications, that is, where it is required to pass particular frequency components and to reject others from analogue (continuous-time) signals

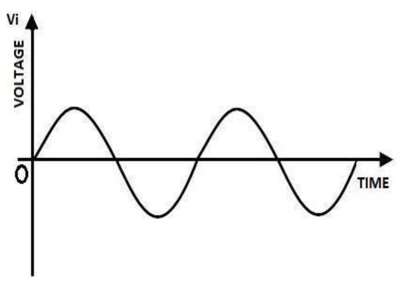
CIRCUITDIAGRAM:

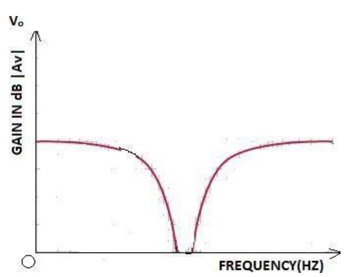


PROCEDURE:

1. Connect the circuit as per given circuit diagram.
2. InRPSbiasvoltagesof10Vtoboth4and7as-10Vand+10Vrespectively.
3. Set2Vp-pofsinewaveinCROwith50Hzinfunctiongenerator.
4. Check the output voltages for different input frequencies as 10Hz, 20Hz, 30Hz, 40Hz, 50Hz, 60Hz, 70Hz, 80Hz,90Hz, and 100Hz.
5. Herewecanobservethattheoutputvoltagedecreasesupto50Hzandthenincreases.

EXPECTEDWAVEFORMS:





TABULARCOLUMN:

S. No.	Input Voltage	Input frequency	Output Voltage	A_v=20log(V_0/V_{in})

RESULT:

We studied and Designed and test a Notch filter to eliminate, 50Hz power line frequency.and output wave forms are drawn.

VIVA QUESTIONS:

1. What is the use of notch filter?
2. Which filter is called as notch filter?
3. What is the purpose of notch filter Mcq?
4. Applications of notch filter?

Experiment No: 6 **Date:**

DESIGN OF ANALOG FILTER-I

AIM:

To design a second order Butterworth band pass filter for the specified cut-off frequencies ALSK Pro Kit.

APPARATUS REQUIRED:

S. No.	Name of the Apparatus	Range	Quantity
1.	Function Generator	3MHz	1
2.	CRO	30MHz	1
3.	Dual RPS	0-30V	2
4.	ALSK PRO Kit	TL082	2
5.	Decade Resistance Boxes	--------	2
6.	CRO probes	--------	2

COMPONENTSREQUIRED:

S. No.	Name of the Apparatus	Range	Quantity
1.	Resistors	1kΩ	4
2.	Resistors	560Ω	2
3.	IC-741	--------	2
4.	Capacitors	0.1µF	4
5.	Breadboard	--------	1
6.	Connecting wires	--------	As required

THEORY:

Analogue filters are a basic building block of signal processing much used in electronics. Amongst their many applications are the separations of an audio signal before application to bass, mid-range, and tweeter loudspeakers; the combining and later separation of multiple telephone conversations onto a single channel; the selection of a chosen radio station in a radio receiver and rejection of others.

Passive linear electronic analogue filters are those filters which can be described with linear differential equations (linear); they are composed of capacitors, inductors and, sometimes, resistors (passive) and are designed to operate on continuously varying (analogue) signals. There are many linear filters which are not analogue in implementation (digital filter), and there are many electronic filters which may not have a passive topology – both of which may have the same transfer function of the filters described in this article. Analogue filters are most often used in wave filtering applications, that is, where it is required to pass particular frequency components and to reject others from analogue (continuous-time) signals.

CIRCUITDIAGRAM:

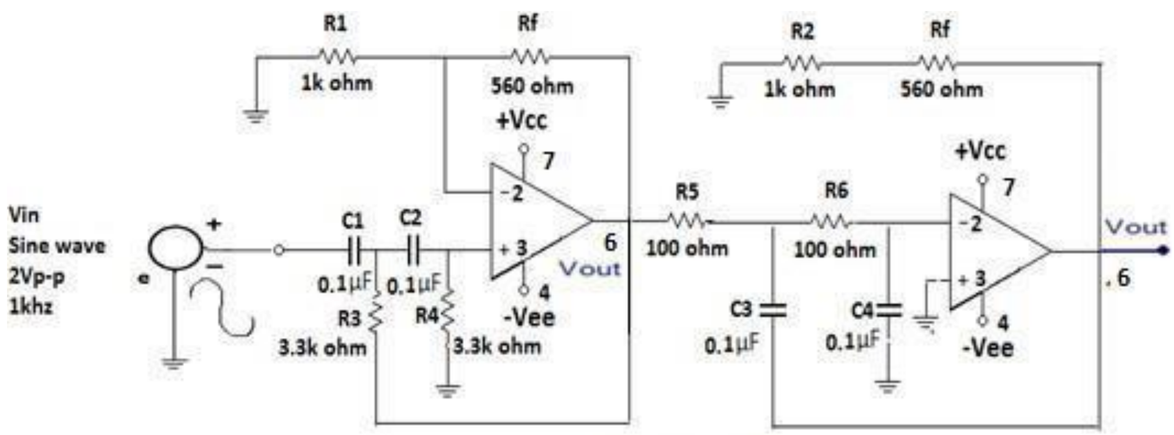
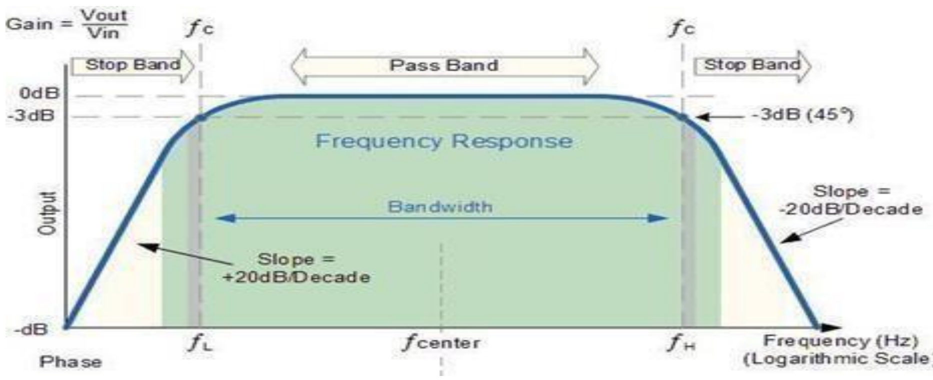


FIG: ANALOG FILTER-I

PROCEDURE:

1. Connect the circuit as per the given circuit diagram. In RPS bias voltages of 10V to both 4^{th} and 7^{th} pin nos. Set $2V_{p-p}$ of sine wave inCROwith1kHz in function generator.
2. Check the output voltages for different frequencies as10Hz, 100Hz,200Hz,300Hz,400Hz, 500Hz, 1kHz, 2kHz, and 10kHz.
3. Here we can observe that the output voltage increases upto1 kHz and then decreases.
4. Note down the output voltages V_{0p-p} in tabular column.
5. Calculate gain by using the formula $|A_v| = 20\log[V_0/V_{in}]$.
6. Plot the graph by taking frequency on x-axis and gain in dB on y-axis.

EXPECTEDGRAPH:



TABULARCOLUMN:

S. No.	Frequency	Output Voltage (Vo)Vp-p	Av=20log(V0/Vin)

RESULT:

We studied and designed the second order Butterworth band pass filter for the specified cut-off frequencies using ALSK Pro Kit and output wave forms are drawn.

VIVA QUESTIONS:

1 What is analog filter design?

2. What are analog filters used for?

3. Which analog filter has two stop bands?

4. Which analogue filter has to stop bands?

5. What is the application of band pass filter?

.

Experiment No: 7 **Date:**

DESIGN OF SELF TUNED FILTER

AIM:

To Design and test a high quality band pass self-tuned filter for a given Centre frequency using ALSK Pro Kit.

APPARATUSREQUIRED:

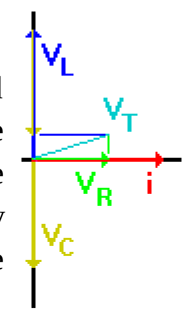
S. No.	Name of the Apparatus	Range	Quantity
1.	Function Generator	3MHz	1
2.	CRO	30MHz	1
3.	Dual RPS	0-30V	2
4.	ALSK PRO Kit	TL082	2
5.	Decade Resistance Box	--------	1
6.	CRO probes	--------	2

COMPONENTSREQUIRED:

S. No.	Name of the Apparatus	Range	Quantity
1.	Resistors	1kΩ	4
2.	Resistors	100Ω	2
3.	IC-741	--------	2
4.	Capacitors	0.1µF	2
5.	Breadboard	--------	1
6.	Connecting wires	--------	As required

THEORY:

We must take into account the different phase angles between voltage and current for each of the three components in the circuit. The vector diagram to the right illustrates this concept.



Since this is a series circuit, the same current passes through all the components and therefore our reference is at a phase angle of 0°. This is shown in red in the diagram. The voltage across the resistor, V_R, is in phase with the current and is shown in green. The blue vector shows V_L at +90°, while the gold vector represents V_C, at -90°. Since they oppose each other diametrically, the total reactive voltage is V_L - V_C. It is this difference vector that is combined with V_R to find V_T (shown in cyan in the diagram).

We know that V_T = 10 V RMS. Now we can see that V_T is also the vector sum of (V_L - V_C) and V_R. In addition, because of the presence of R, the phase angle between V_T and "i" will be arc tan((V_L-V_C)/V_R), and can vary from -90° to +90°.

CIRCUIT DIAGRAM:

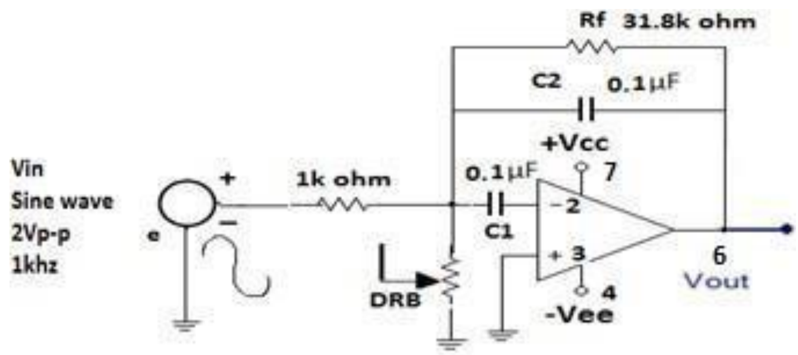
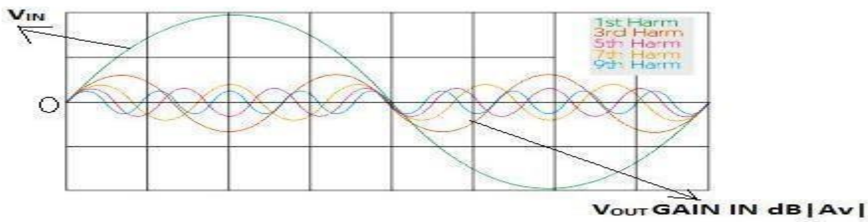


FIG: NARROW BANDPASS FILTER

PROCEDURE:

1. Connect the circuit as per the given circuit diagram.
2. In RPS bias voltages of 10Vp-p to both 4thand 7thpin numbers.
3. Set 2Vp-p of sine wave in CROwith1 kHz in function generator.
4. Check the output voltages for different input frequenciesas100Hz, 500Hz, 1kHz, 2 kHz, and5kHz10kHz.
5. Here we can observe that V0ismaximumfor1 kHz.
6. Check the output voltages for different input frequencies.
7. We can observe that V0 is maximum for 2 kHz.
8. Note down the output voltages VP-P and calculate gain in AV= 20log [V0\VIN].
9. Plot the graph by taking frequency on x-axis and gain in dB on y-axis.

EXPECTEDGRAPHS:



TABULARCOLUMN:

Case: I F_C=1 KHz and Case: Fc=2 KHz

S. No.	Frequency	Output Voltage V0 p-p	Gain in dbAv=20log(V 0/Vin)

Case: II F_C=1 KHz and Case: Fc=2 KHz

S. No.	Frequency	Output Voltage V0 p-p	Gain in dbAv=20log(V0/Vin)

RESULT:

We studied and designed the test a high quality band pass self-tuned filter for a given Centre frequency using ALSK Pro Kit.

VIVA QUESTIONS:

1. What is a self tuned filter?
2. What is the quality factor of a filter?
3. What is the type of filter shown in the figure?
4. What are the application self tuned filter?

Experiment No: 8 **Date:**

DESIGN OFAN INSTRUMENTATION AMPLIFIER

AIM:

To Design an instrumentation amplifier of a differential mode gain of ‘A’ using three amplifier using ALSK Pro Kit.

APPARATUS REQUIRED:

S. No.	Name of the Apparatus	Range	Quantity
1.	Function Generator	3MHz	1
2.	Digital multimeter	1	1
3.	Dual RPS	0-30V	2
4.	Decade Resistance Box	--------	1
5.	CRO probes	--------	As required

COMPONENTS REQUIRED:

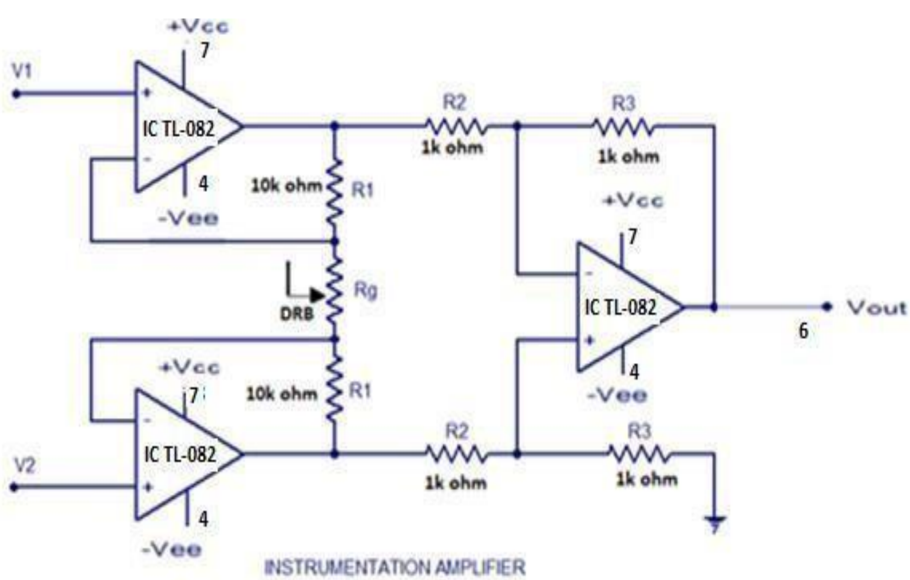
S. No.	Name of the Apparatus	Range	Quantity
1.	Resistors	10kΩ	2
2.	Resistors	1kΩ	4
3.	IC-741	--------	3
4.	Breadboard	--------	1
5.	Connecting connectors	--------	As required

THEORY:

Select an Op Amp. Selecting an appropriate op amp is an important part in designing an instrumentation amplifier. The instrumentation amplifier will require three op amps. Almost any op amp will work for this design. However, with circuit size in mind, this design will use the LM324. The LM324 IC has four op amps on the chip, which will reduce size and amount of wires needed to build the circuit. The LM324 can be seen below in figure 4, illustrating how the four op amps are laid out within the chip. The maximum gain of this amplifier will depend on the input. The LM324 op amp will begin to saturate at VCC – 1.5 V. If the input multiplied by the gain is above this value, the op amp will saturate and give an incorrect output

A typical instrumentation amplifier will have seven resistors. These resistors are oriented between the op amps as shown in figure 5. Instrumentation amplifiers are unique in the fact that resistor values can be selected so that only one resistor will dictate the overall gain. This can be done if all other resistor values are chosen properly. The transfer function of a typical instrumentation amplifier is: Vout = (V1 – V2)(R2/R1)(1+2R5/RG) This is the case if R5 = R6, R2 = R4 and R1 = R3. However, this transfer function can be simplified further if R2 = R4 = R1 = R3 = R5 = R6 = R. The transfer function would then become: Vout = (V1 – V2)(1+2R/RG) This transfer function allows the gain to be decided by one resistor, RG. Finally, in order to allow large gain, select R to be between 25 KOhm and 1 MOhm. For this application note R will be selected to be 25 KOhm.

CIRCUIT DIAGRAM:



PROCEDURE:

1) Connect the circuit as per the given circuit diagram.
2) In RPS bias voltages of $10V_{p-p}$ to both 4^{th}&7^{th}pins for three ICs
3) The voltage V1 & V2 can be given by using RPS by taking corresponding V1 & V2 values which are mentioned in the table.
4) Now, the output voltage measured at pinno.1withthehelpofmultimeter.
5) Note down the output voltage in the tabular column.
6) Now replace R_G by 10kΩ and noted down the corresponding output voltage.
7) Note down the values in table for both10kΩ and 20kΩ.

INFERENCE:

1. Voltage gain of the instrumentation amplifier(A)=
2. Measured output signal voltage amplitude peak –to–peak(V_O)=
3. Frequency(f)=

TABULARCOLUMN:

S. NO.	V_1	V_2	A_V=2 V_O	A_V=3 V_O	A_V=1 V_O

RESULT:

We studied and designed the instrumentation amplifier of a differential mode gain of ‘A’ using three amplifier using ALSK Pro Kit

VIVA QUESTIONS:

1. What is the working principle of instrumentation amplifier?
2. What instrument is used to amplify?
3. What are instrumentation amplifiers used for?
4. What are the applications of Instrumentation Amplifier?

Experiment No: 9 **Date:**

DESIGN OF AUTOMATIC GAIN CONTROL (AGC)

AIM:

To Design and test an AGC system for a given peak amplitude of sine wave output using ALSK Pro Kit.

APPARATUSREQUIRED:

S. No.	Name of the Apparatus	Range	Quantity
1.	Function Generator	3MHz	1
2.	CRO	30MHz	1
3.	Dual RPS	0-30V	2
4.	ALSK PRO Kit	TL082	2
5.	Bread board	--------	1
6.	Digital multimeter	--------	1
7.	Connecting connectors and CRO probes	--------	As required

THEORY:

Automatic gain control (AGC) is a closed-loop feedback regulating circuit in an amplifier or chain of amplifiers, the purpose of which is to maintain suitable signal amplitude at its output, despite variation of the signal amplitude at the input. The average or peak output signal level is used to dynamically adjust the gain of the amplifiers, enabling the circuit to work satisfactorily with a greater range of input signal levels. It is used in most radio receivers to equalize the average volume (loudness) of different radio stations due to differences in received signal strength, as well as variations in a single station's radio signal due to fading. Without AGC the sound emitted from an AM radio receiver would vary to an extreme extent from a weak to a strong signal; the AGC effectively reduces the volume if the signal is strong and raises it when it is weaker. In a typical receiver the AGC feedback control signal is usually taken from the detector stage and applied to control the gain of the IF or RF amplifier stages.

CIRCUITDIAGRAM:

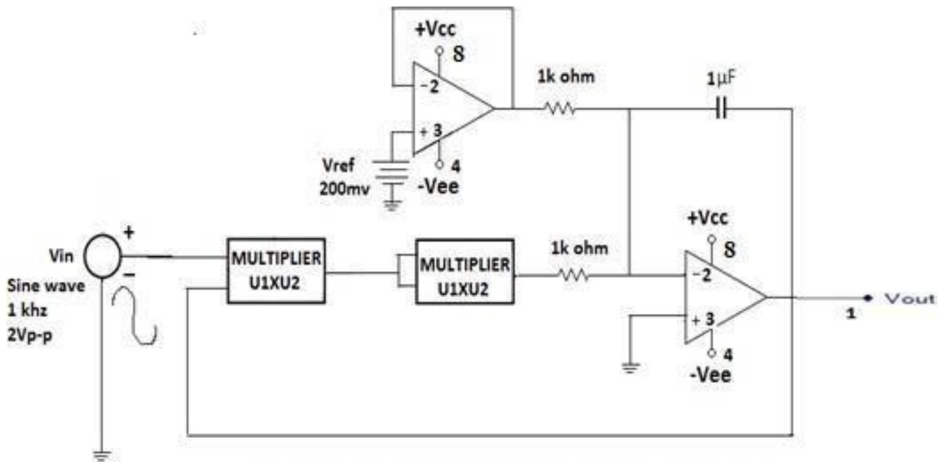


FIG: AUTOMATIC GAIN CONTROL/ AUTOMATIC VOLTAGE CONTROL

PROCEDURE:

1) Connect the circuit as per the given circuit diagram with help of patch chords
2) In RPS bias voltages of10Vp-ptoboth4th&8thpins.
3) Set the input wave with1khz in function generator and set 2Vp-pin CRO.
4) Vary the Vrefas200mv in multimeter and check the output for different voltages like 5Vp-p,10Vp-p.
5) Take the readings of Vout in tabular column.
6) We can observe that output voltage is controlled as input voltage increases.

TABULARCOLUMN:

S.NO	Vin	Vout

RESULT:

We studied and designed the AGC system for a given peak amplitude of sine wave output using ALSK Pro Kit and output values is observed.

VIVA QUESTIONS:

1. Why is AGC used?
2. What is the principle of automatic gain control?
3. Why is the automatic gain control necessary in radio receiver Mcq?
4. what are the application of AGC?

Experiment No: 10 **Date:**

VOLTAGE CONTROLLED OSCILLATOR

AIM:

To Design and test a voltage controlled oscillator for the given specification (Voltage range& frequency range) using ALSK Pro Kit.

APPARATUSREQUIRED:

S. No.	Name of the Apparatus	Range	Quantity
1.	Function Generator	3MHz	1
2.	CRO	30MHz	1
3.	Dual RPS	0-30V	2
4.	ALSKPRO Kit	TL082	2
5.	Breadboard	--------	1
6.	Digital multimeter	--------	1
7.	Connecting connectors and CRO probes	--------	As required

THEORY:

A **voltage-controlled oscillator** (**VCO**) is an electronic oscillator whose oscillation frequency is controlled by a voltage input. The applied input voltage determines the instantaneous oscillation frequency. Consequently, a VCO can be used for frequency modulation (FM) or phase modulation (PM) by applying a modulating signal to the control input. A VCO is also an integral part of a phase-locked loop. VCOs are used in synthesizers to generate a waveform whose pitch can be adjusted by a voltage determined by a musical keyboard or other input.

Voltage controlled oscillator (VCO), from the name itself it is clear that the output instantaneous frequency of the oscillator is controlled by the input voltage. It is a kind of oscillator which can produce output signal frequency over a large range (few Hertz-hundreds of Giga Hertz) depending on the input DC voltage given to it

CIRCUIT DIAGRAM:

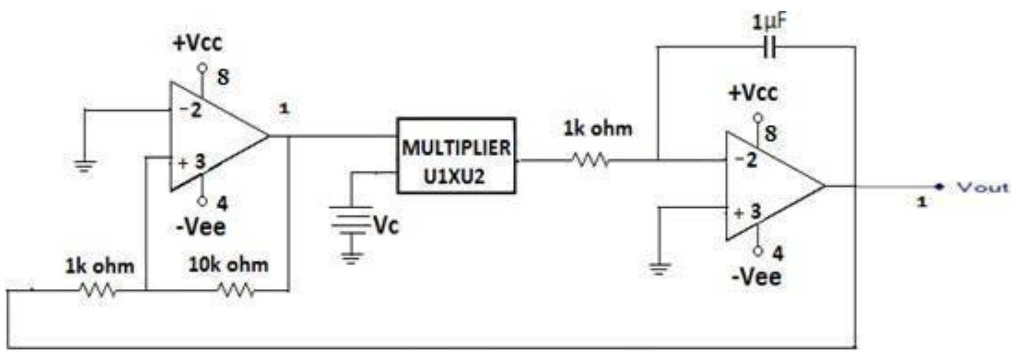
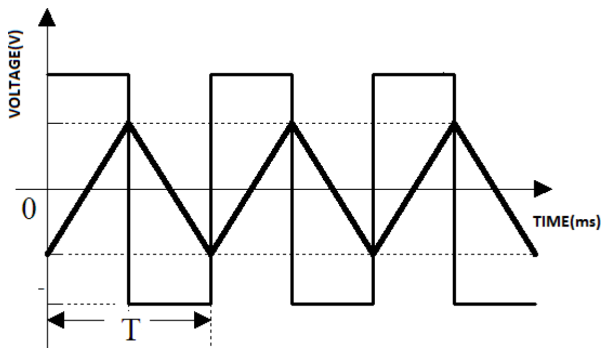


FIG: VOLTAGE CONTROL OSCILLATOR

PROCEDURE:

1) Connect the circuit as per the given circuit diagram with help of patch chords.
2) In RPS bias voltages of10Vp-p to both 4th&8thpins.
3) By varying the control voltage Vc and check output at voltages like 1V, 2V &3V.
4) Note the frequency for the respective voltages.
5) Take the readings in the tabular column.

EXPECTEDWAVEFORMS:



TABULAR COLUMN:

S. No.	Control voltage	Freq. of oscillations (fo)	Time period (sq.ms)

RESULT:

We studied and designed the voltage controlled oscillator for the given specification (Voltage range& frequency range) using ALSK Pro Kit and output wave forms is observed.

VIVA QUESTIONS:

1. What is the basic principle of VCO?
2. What is the principle of automatic gain control?
3. What is the purpose of the voltage controlled oscillator Mcq?
4. what are the applications of VCO?

Experiment No: 11 **Date:**

DC-DC CONVERTER

AIM:

To design of a switch mode power supply that can provide a regulated output voltage for a given Input range using (TPS40200IC) ASLK Pro Kit.

APPARATUS REQUIRED:

S. No.	Name of the Apparatus	Range	Quantity
1.	Function Generator	3MHz	1
2.	CRO	30MHz	1
3.	Dual RPS	0-30V	2
4.	ALSKPRO Kit	TL082	2
5.	Breadboard	--------	1
6.	Digital multi-meter	--------	1
7.	Connecting connectors	--------	As required

THEORY:

As its name implies, a DC-DC converter convert's one DC voltage to another.The operating voltage of different electronic devices such as ICs can vary over a wide range, making it necessary to provide a voltage for each device. A Buck Converter outputs a lower voltage than the original voltage, while a Boost Converter supplies a higher voltage

DC, which stands for Direct Current, is characterized by current that does not change in polarity over time. A boost converter step-up converter is a DC-to-DC power converter that steps up voltage while stepping down current) from its input supply to its output load. It is a class of switched-mode power supply SMPS containing at least two semiconductors a diode and a transistor and at least one energy storage element: a capacitor, inductor, or the two in combination. To reduce voltage ripple, filters made of capacitors sometimes in combination with inductors are normally added to such a converter's output load-side filter and input supply-side filter. Power for the boost converter can come from any suitable DC source, such as batteries, solar panels, rectifiers, and DC generators. A process that changes one DC voltage to a different DC voltage is called DC to DC conversion. A boost converter is a DC to DC converter with an output voltage greater than the source voltage. A boost converter is sometimes called a step-up converter since it "steps up" the source voltage. Since power must be conserved, the output current is lower than the source current

CIRCUITDIAGRAM:

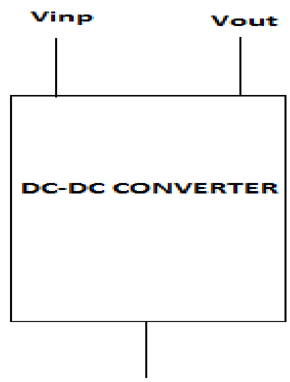


FIG: DC-DC CONVERTER

PROCEDURE:

1) Connect the circuit as per the given circuit diagram by using ANALOGSYSTEM PROG. Kit.
2) In RPS bias voltages of10V$_{p-p}$ to both 4th&8thpins.
3) Apply the input voltage of DC-DC converter as to specified pin i.e., Vin.
4) When the Vout set voltage is 5v and 3.3v by changing the Vin by using the trimmer circuit.
5) The output voltage is observed in the multimeter and it is observed to be constant.

TABULARCOLUMN:

S. No.	VIN	VOUT=5V	VOUT=3.3V

RESULT:

We studied and designed the DC-DC Converter of a switch mode power supply that can provide a regulated output voltage for a given Input range using (TPS40200IC) ASLK Pro Kit and output values is observed.

VIVA QUESTIONS:

1. What is DC-to-DC converter called?
2. What is the function of DC-DC converter?
3. What is the principle of dc/dc conversion?
4. What are the three basic DC to DC converters?
5. What are the applications of DC-DC converter?

Experiment No: 12 **Date:**

DESIGN OF A PHASE LOCKED LOOP

AIM:

Design and test a phase locked loop to get locked to a given frequency. Measure the lock-in-range of the system and also measure the change in

Phase of output signal as input frequency is varied within the lock range.

APPARATUS REQUIRED:

S. No.	Name of the Apparatus	Range	Quantity
1.	Function Generator	3MHz	1
2.	CRO	30MHz	1
3.	Dual RPS	0-30V	2
4.	ALSK PRO Kit	TL082	2
5.	Bread board	--------	1
6.	Digital multimeter	--------	1
7.	Connecting connectors and CRO probes	--------	As required

THEORY:

The terms **hold-in range**, **pull-in range** (acquisition range), and **lock-in range** are widely used by engineers for the concepts of frequency deviation ranges within which phase-locked loop-based circuits can achieve lock under various additional conditions. In the classic books on phase-locked loops, published in 1966, such concepts as hold-in, pull-in, lock-in, and other frequency ranges for which PLL can achieve lock, were introduced. They are widely used nowadays (see, e.g. contemporary engineering literature and other publications). Usually in engineering literature only non-strict definitions are given for these concepts. Many years of using definitions based on the above concepts has led to the advice given in a handbook on synchronization and communications, namely to check the definitions carefully before using them .Later some rigorous mathematical definitions were given in

CIRCUIT DIAGRAM:

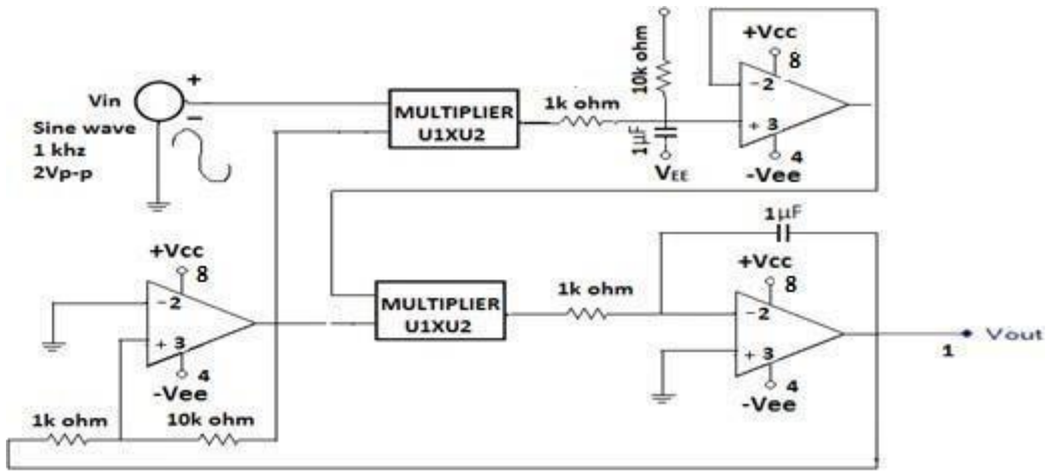


FIG: DESIGN OF A PHASE LOCKED LOOP

PROCEDURE:

1) Connect the circuit as shown in figure using analog system lab kit.
2) In RPS bias voltages of 10Vp-p to both 4th&8th pins.
3) Make the function generator to ground and keep the CRO probe at output of integrator.
4) Measure the free running frequency of the integrator at CRO.
5) Again connect the function generator at input of multiplier.
6) Apply an input of sine wave from function generator.
7) Vary the frequencies at function generator upto the free running frequency, there should not be any change in output for the input.
8) Phase is also same for the wave forms until it exceeds the free running frequency.

OBSERVATIONS:

S. No.	INPUT FREQUENCY	OUTPUT FREQUENCY

RESULT:

We studied and designed the phase locked loop to get locked to a given frequency and output values is observed.

VIVA QUESTIONS:

1. What is a phase-locked loop used for?
2. What is the function of low pass filter in phase-locked loop?
3. At which stage the phase-locked loop?
4. Which oscillator is used in phase-locked loop?
5. what are the applications of PLL?

Experiment No: 13 **Date**:

LOW DROP OUT (LDO) REGULATOR

AIM:

To verify the action of low drop out using IC-723
To find the values of line regulation and load regulation.

APPARATUS REQUIRED:

Regulated DC Power Supply	-----------------------------	1

COMPONENTS REQUIRED:

Resistor10kΩ	---------------------------	1
Resistor100Ω	----------------------------	1
Resistor560Ω	----------------------------	1
Capacitor(Disc)100pf	----------------------------	1
Decade Resistance Box	----------------------------	1
Digital Multimeter	----------------------------	1
IC-723	----------------------------	1
Bread Board	----------------------------	1
Connecting Wires	----------------------------	A fewnos.

THEORY:

In the classic books on phase-locked loops, published in 1966, such concepts as hold-in, pull-in, lock-in, and other frequency ranges for which PLL can achieve lock, were introduced. They are widely used nowadays (see, e.g. contemporary engineering literature and other publications). Usually in engineering literature only non-strict definitions are given for these concepts. Many years of using definitions based on the above concepts has led to the advice given in a handbook on synchronization and communications, namely to check the definitions carefully before using them. Later some rigorous mathematical definitions were given in

PIN DIAGRAM:

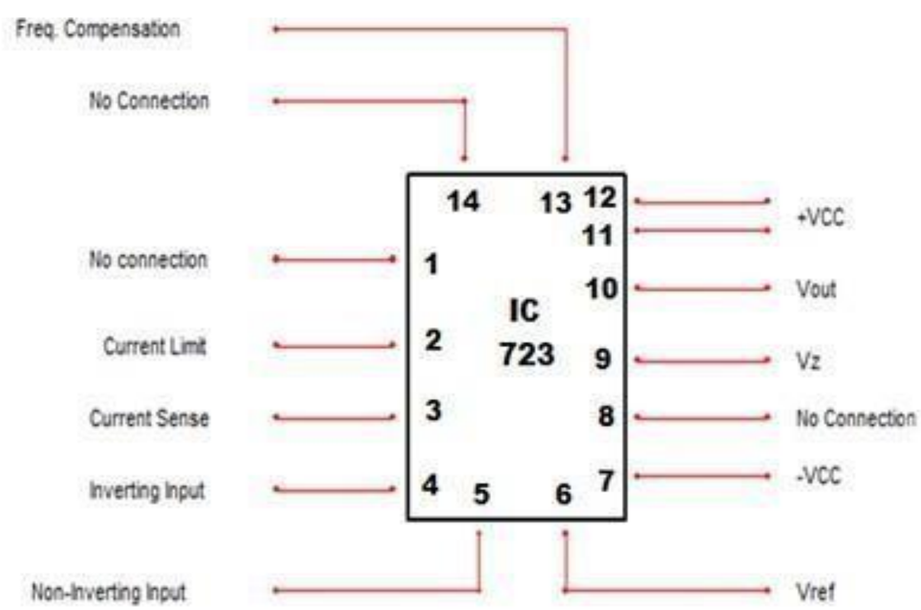


FIG: PIN DIAGRAM OF LOW DROP OUT(LDO) REGULATOR USING 723 IC

CIRCUIT DIAGRAM:

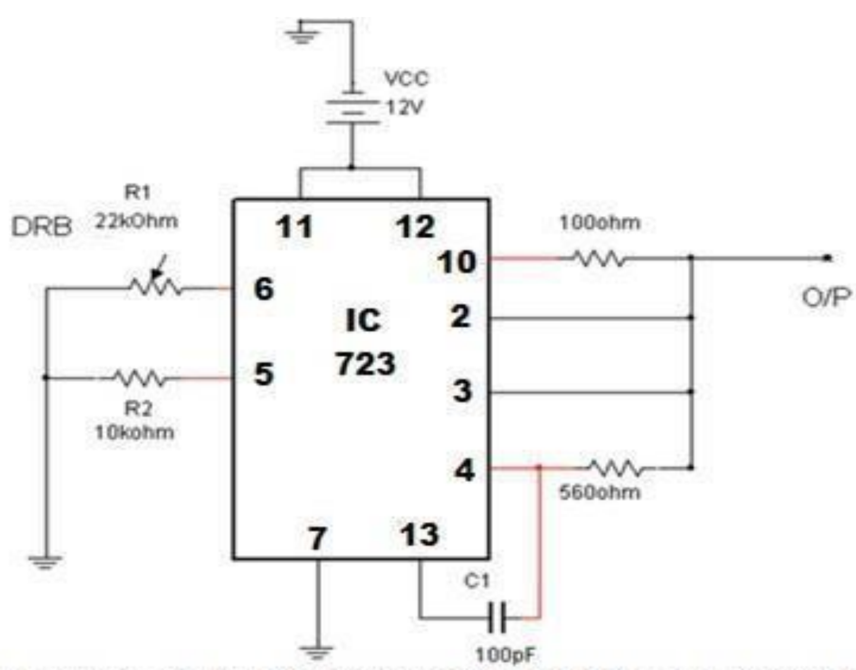


FIG: LOW DROP OUT (LDO) REGULATOR USING 723 IC

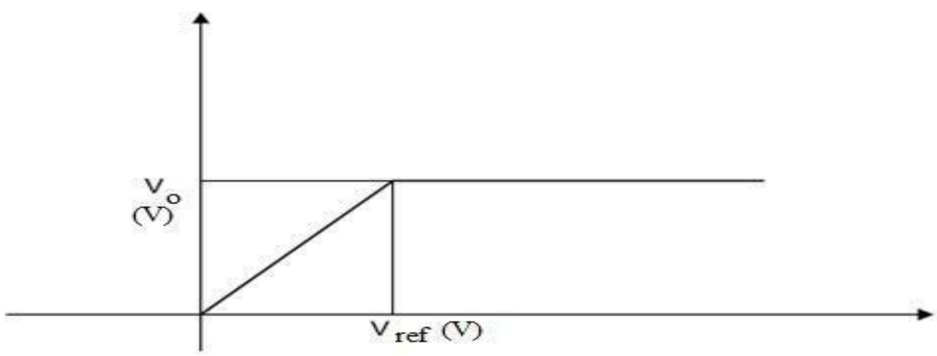
PROCEDURE:

1) Connect the circuit as per given the circuit diagram.
2) Measure the reference voltage at pinno.6 Itshouldbegreaterthan7V.
3) By varying the resistance (Decade Resistance Box) measures the output voltage with digital multimeter.
4) Calculate the theoretical values and compare with the practical values.
5) Plot the graph between X-axis on Resistance and Y-axis on Voltage.

OBSERVATIONS:

SL.NO.	Load Resistance(RL)in Ω	Pin No.6VREF(V)	$V_{OUT} = \left(V_{REF} \times \frac{R2}{R1 + R2}\right)$	Practical V_o

EXPECTED GRAPH:



RESULT:

We studied and designed the action of low drop out using IC-723 and we find the values of line regulation and load regulation.values is observed.

VIVA QUESTIONS:

1. Which is the output pin for 723 voltage regulator?
2. What is the function of IC 723?
3. What are the limitations of 723 regulators?
4. Which type of regulator is IC 723?
5. what are the applications of IC 723?

BEYOND THE SYLLABUS HARDWARE EXPERIMENTS

Experiment No: 1 **Date:**

4-BIT D-A CONVERTER

AIM:

1) To obtain analog output voltages for the digital input data using 4-bit R-2R type Digital to Analog converter.
2) Calculate output voltages corresponding to different input theoretical and compare them with practical values.

APPARATUS REQUIRED:

Regulated Power Supply	--------------------------------	3

COMPONENTS REQUIRED:

Resistors 10kΩ	--------------------------------	4
Resistors 22kΩ	--------------------------------	6
I.C. 741	---------------------------------	1
Digital Multimeter	---------------------------------	1
Connecting Wires	--------------------------------	As required

THEORY:

In the weighted resistor type D/A Converter, each digital voltage is converted into an equivalent analog voltage or current. In a 4-bit D/A converter, from 0000 to 1111, there are **15 discrete levels** and hence it is convenient to divide the output analog signal into 15 discrete levels.

A D/A Converter is used when the binary output from a digital system is to be converted into its equivalent analog voltage or current.The binary output will be a sequence of 1's and 0's. Thus they may be difficult to follow. But, a D/A converter help the user to interpret easily.Basically, a D/A converter have an op-amp. It can be classified into 2 types. They are A D/A converter using binary-weighted resistors is shown in the figure below. In the circuit, the op-amp is connected in the inverting mode. The op-amp can also be connected in the non-inverting mode. The circuit diagram represents a 4-digit converter. Thus, the number of binary inputs is four.

CIRCUIT DIAGRAM:

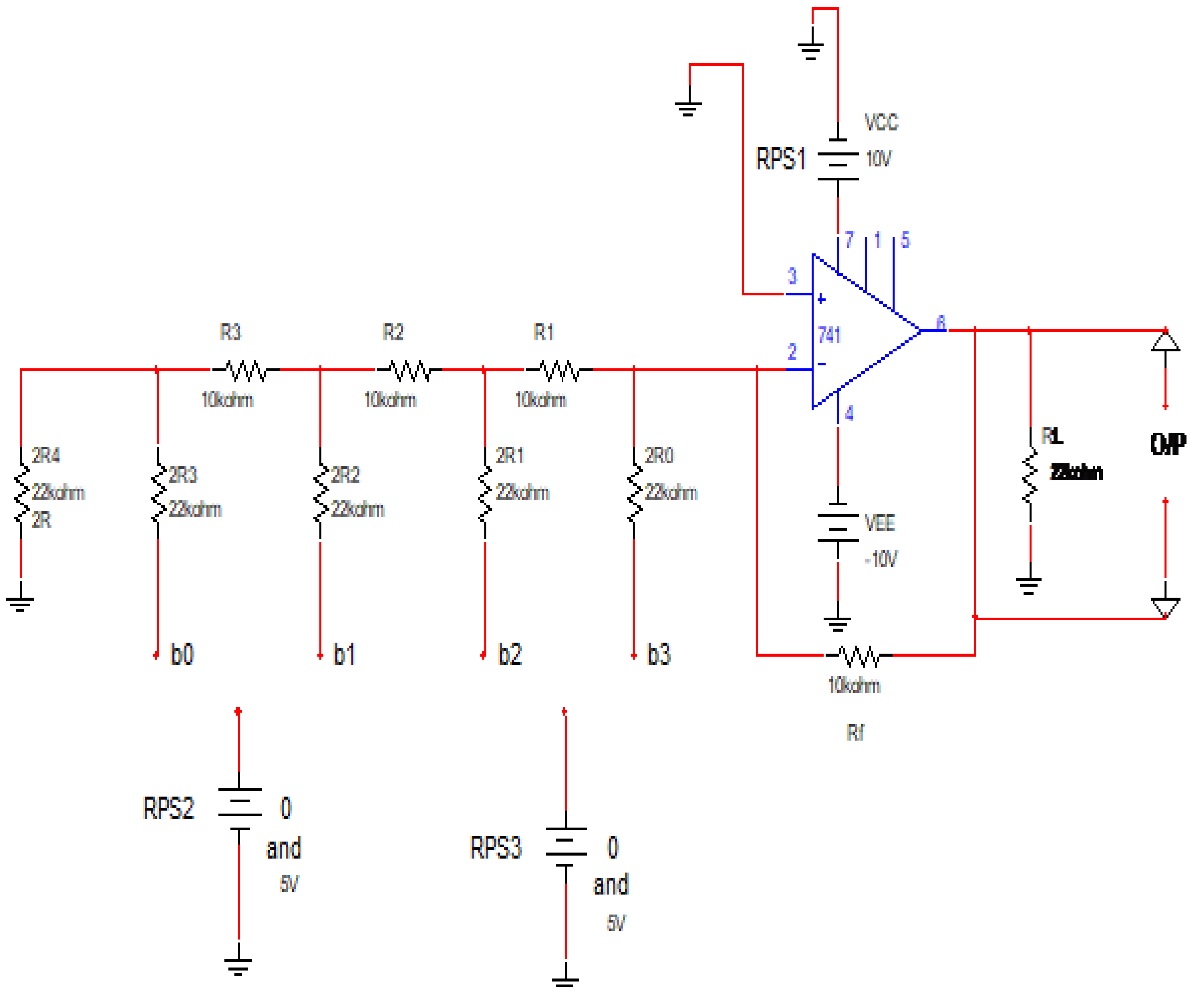


FIG: 4-BIT D TO A CONVERTER

PROCEDURE:

1) Connect the circuit as per given the circuit diagram.
2) The bias voltages are applied to the circuit i.e. 4th and 7th pins numbers.
3) While taking reading in the D.C. Power Supply bit 1 means take reading as a 5V.
4) By taking the output at pin no.6 measure with the digital multimeter.
5) The input starting from 0000 to 1111 are connected and corresponding output are noted.

$$V_0 = -R_F \left(\frac{b_3}{2R} + \frac{b_2}{4R} + \frac{b_1}{8R} + \frac{b_0}{16R} \right).$$

6) Calculate the theoretical values and compare with the practical values.
7) Graph is drawn between output voltage verses digital input.

For Example:

When b_3 is high and other inputs are low then the output voltage is

When $R_F = -10k\Omega$

$$V_0 = -R_F \left(\frac{5}{2R} + \frac{0}{4R} + \frac{0}{8R} + \frac{0}{16R} \right) = -10K \left(\frac{5}{2x10K} \right) = -2.5V$$

TABULAR COLUMN:

Input				Output, V_{OUT}
B_0	B_1	B_2	B_3	$V_{OUT} = \frac{V_{ref}\ R_f}{2^n\ R} B_{in} = \frac{5\ R}{2^4\ R} B_{in} = \frac{5}{16} B_{in}$
0	0	0	0	
0	0	0	1	
0	0	1	0	
0	0	1	1	
0	1	0	0	
0	1	0	1	
0	1	1	0	
0	1	1	1	
1	0	0	0	
1	0	0	1	
1	0	1	0	
1	0	1	1	
1	1	0	0	
1	1	0	1	
1	1	1	0	
1	1	1	1	

EXPECTED WAVE FORMS:

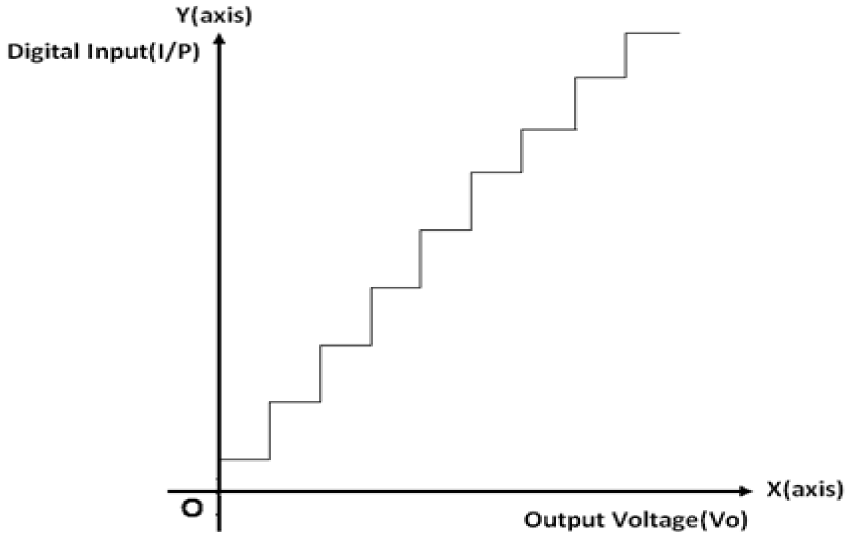


FIG: 4-BIT D/A CONVERTER

RESULT:

we studied and designed the 4-bit R-2R type Digital to Analog converter using 741 I.C. output values is observed.

VIVA QUESTIONS:

1. What is the main function of DAC converter?
2. Why d'A converter is useful?
3. What is full scale output voltage for DAC?
4. Which type of regulator is IC 723?

Experiment No: 2 **Date:**

ACTIVE FILTER APPLICATIONS – LPF, HPF (FIRST ORDER)

AIM:

1) To design low pass and high pass filter of first order.
2) To study the frequency response of first order low pass and high pass filters using IC-741.

APPARATUS REQUIRED:

Cathode Ray Oscilloscope	---------------------------------	1
Function Generator	---------------------------------	1
Regulated DC Power Supply	--------------------------------	1
CRO Probes	---------------------------------	2

COMPONENTS REQUIRED:

Resistors 10 kΩ	---------------------------------	2
Potentiometer (pot) 10 kΩ	----------------------------------	1
Capacitor (Disc) 0.01μf	-----------------------------------	1
I.C. 741	-----------------------------------	1
Bread Board	-----------------------------------	1
ConnectingWires	-----------------------------------	As required

THEORY:

By combining a basic RC Low Pass Filter circuit with an operational amplifier we can create an Active Low Pass Filter circuit complete with amplification

In the RC Passive Filter tutorials, we saw how a basic first-order filter circuits, such as the low pass and the high pass filters can be made using just a single resistor in series with a non-polarized capacitor connected across a sinusoidal input signal.

We also noticed that the main disadvantage of passive filters is that the amplitude of the output signal is less than that of the input signal, ie, the gain is never greater than unity and that the load impedance affects the filters characteristics.

With passive filter circuits containing multiple stages, this loss in signal amplitude called "Attenuation" can become quiet severe. One way of restoring or controlling this loss of signal is by using amplification through the use of **Active Filters**.

An Active High Pass Filter can be created by combining a passive RC filter network with an operational amplifier to produce a high pass filter with amplification

Technically, there is no such thing as an **active high pass filter**. Unlike Passive High Pass Filters which have an "infinite" frequency response, the maximum pass band frequency response of an active high pass filter is limited by the open-loop characteristics or bandwidth of the operational amplifier being used, making them appear as if they are band pass filters with a high frequency cut-off determined by the selection of op-amp and gain.
In the Operational Amplifier tutorial we saw that the maximum frequency response of an op-amp is limited to the Gain/Bandwidth product or open loop voltage gain (A_V) of the operational amplifier being used giving it a bandwidth limitation, where the closed loop response of the op amp intersects the open loop response

CIRCUIT DIAGRAM:

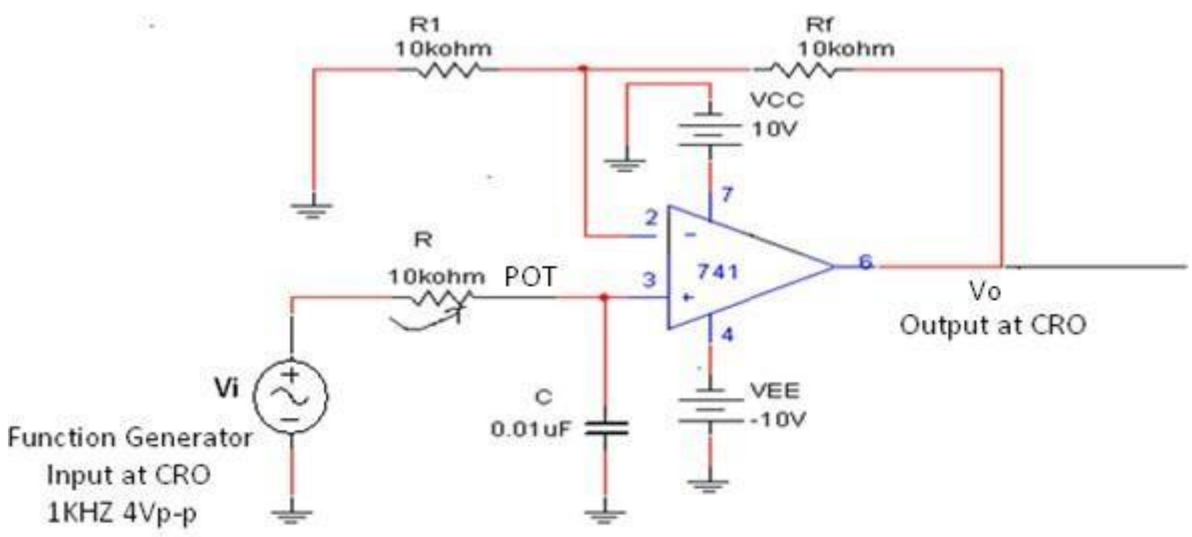


FIG: LOW PASS FILTER USING 741 I.C.

PROCEDURE:

For low pass filter

1) Connect the circuit as per the circuit diagram.
2) The bias voltages are applied to the circuit i.e. 4^{th} and 7^{th} pin numbers.
3) Using function generator, adjust the amplifier of input sinusoidal voltage to $4V_{P-P.}$
4) Vary the frequency from 30 HZ to 30 KHZ note down output voltage by observing wave forms on C.R.O.
5) Calculate gain i.e., $20\log\left(\frac{V_o}{V_{in}}\right)$
6) Plot the graph between frequency and voltage gain.

7) Compare theoretical and practical cut-off frequencies.

DESIGN:

1) Low pass filter: F_H= higher cut-off frequency = $\frac{1}{2\Pi RC}$

Let F_H = 1 KHZ

Pass band A_F = 2

Select C = 0.01μf

$R = \frac{1}{2\Pi RC} = 15.9k\Omega$

$A_F = 1 + \frac{R_F}{R_1} = 2$

$R_F = 1R_1$

Choose 10KΩ

=10KΩX1

R_F =10kΩ

2) High pass filter: F_L = lower cut-off frequency = $\frac{1}{2\Pi RC}$

$A_F = 1 + \frac{R_F}{R_1}$

F_L = 1 KHZ

C = 0.01μf A_F=2

R = 15.9kΩ

TABULAR COLUMN: 1) For Low Pass Filter

Let $V_{IN} = 4\ V_{P-P}$

SL. No.	Input Frequency	V_O	V_O/V_{IN}	Gain in db $20\log(\frac{V_o}{V_{in}})$

CIRCUIT DIAGRAM:

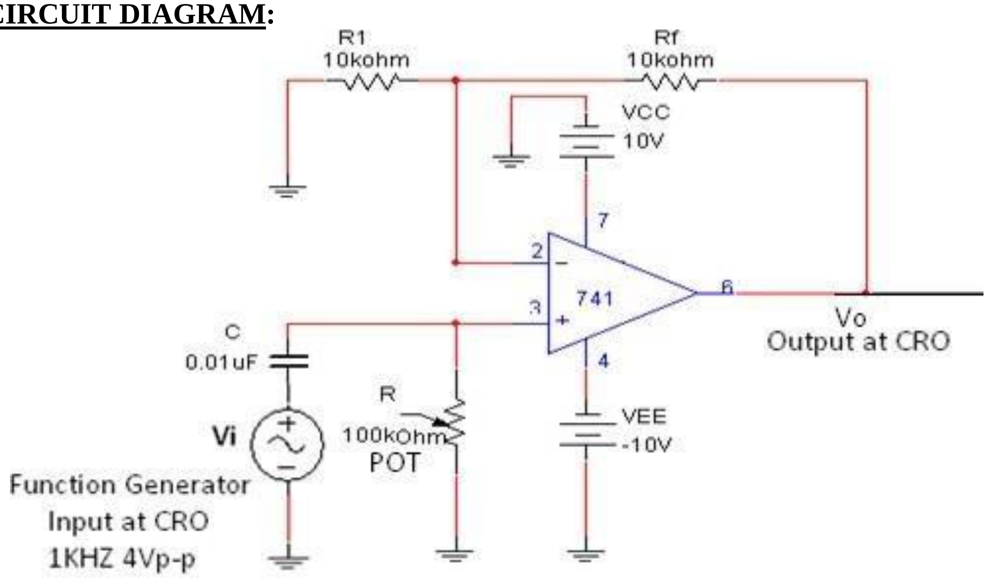


FIG: HIGH PASS FILTER USING 741 I.C.

PROCEDURE:

For high pass filter

1) Connect the circuit as per given the circuit diagram.
2) The bias voltages are applied to the circuit i.e. 4^{th} and 7^{th} pin numbers.
3) Using function generator, adjust the amplifier of input sinusoidal voltage to 4 $V_{P-P.}$
4) Vary the frequency from 30 HZ to 30 KHZ and note down output voltage by observing wave forms on C.R.O.
5) Calculate gain i.e., 20log ($\frac{V_o}{V_{in}}$).
6) Plot the graph between frequency verses voltage gain.
7) Compare theoretical and practical cut-off frequencies

TABULAR COLUMN:

Let $V_{IN} = 4V_{P-P}$

SL. No.	Input Frequency	V_O	V_O/V_{IN}	Gain in db 20log ($\frac{V_o}{V_{in}}$).

EXPECTED WAVE FORMS:

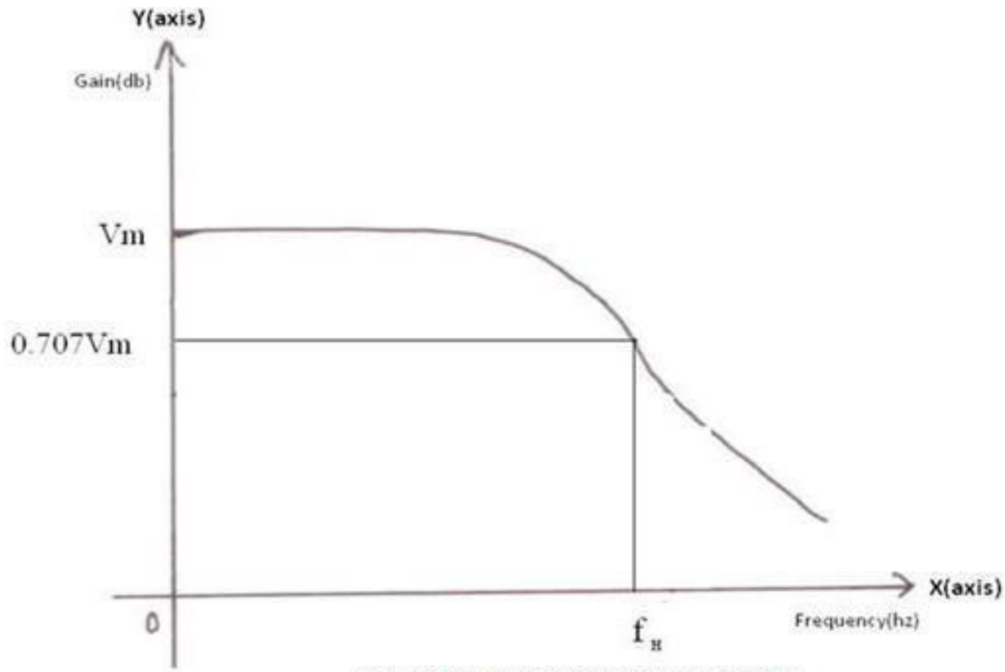


FIG: LOW PASS FILTER USIING 723 I.C.

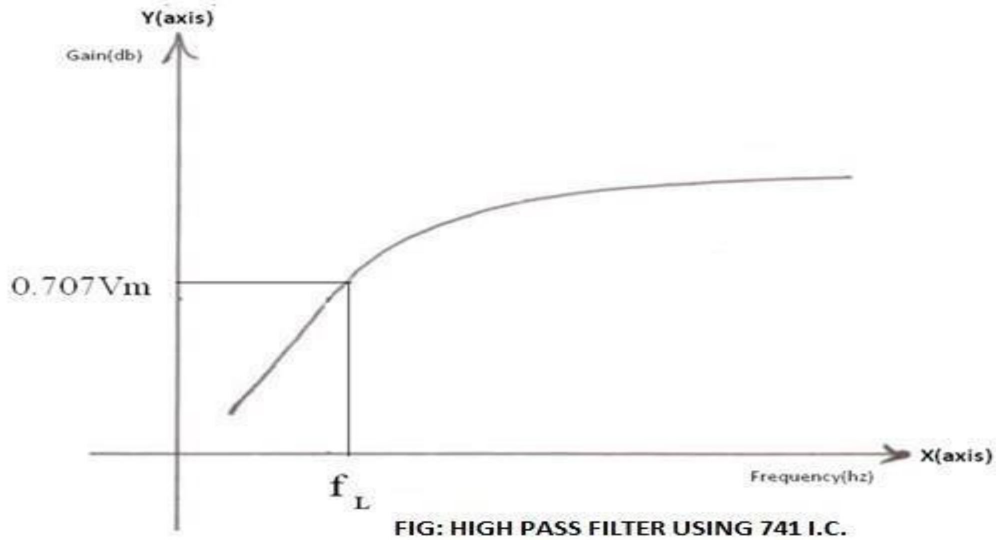


FIG: HIGH PASS FILTER USING 741 I.C.

RESULT:

We studied and designed the low pass and high pass filter of first order using IC-741 and graph is drawn and output voltages is observed.

VIVA QUESTIONS:

1. What is a low-pass filter?
2. What is the difference between LPF and HPF?
3. What is the purpose of a high pass filter?
4. What are the applications of LPF&HPF?