15EC204J- LINEAR INTEGRATED CIRCUITS LAB

RECORD

ACADEMIC YEAR: 20---20—

NAME :

REG.NO. :

DEPARTMENT OF ELECTRICAL & ELECTRONICS ENGINEERING

FACULTY OF ENGINEERING & TECHNOLOGY

SRM INSTITUE OF SCIENCE AND TECHNOLOGY

(Under section 3 of UGC Act, 1956)

S.R.M. NAGAR, KATTANKULATHUR – 603 203

KANCHEEPURAM DISTRICT

SRM INSTITUTE OF SCIENCE AND TECHNOLOGY
(under Section 3 of UGC Act, 1956)
S.R.M. NAGAR, KATTANKULATHUR -603 203
KANCHEEPURAM DISTRICT

BONAFIDE CERTIFICATE

Register No_____

Certified to be the bonafide record of work done by _____ *of EEE , B.Tech Degree course in the Practical 15EC204J- LINEAR INTEGRATED CIRCUITS LAB, in* ***SRM INSTITUTE OF SCIENCE AND TECHNOLOGY, Kattankulathur*** *during the academic year 20__-20__ .*

Lab Incharge

Date: **Year Co-ordinator** **HOD**

Submitted for University Examination held in Lab, ***SRM UNIVERSITY, Kattankulathur.***

Date: **Examiner-1** **Examiner-2**

15EC204J –LINEAR INTEGRATED CIRCUITS

LIST OF EXPERIMENTS

1. Basic op-amp circuits.
2. Integrators and Differentiators.
3. Rectifiers.
4. Comparators.
5. Wave shaping Circuits.
6. Waveform generators: using op-amp.
7. Waveform generators: using 555 Timer.
8. Design of LPF, HPF, BPF and Band Reject Filters.
9. IC Voltage Regulators.
10. R -2R ladder DAC.
11. Flash Type ADC.
12. Simulation experiments using EDA Tools.

DEPT. OF ELECTRICAL & ELECTRONICS ENGINEERING
SRM UNIVERSITY, Kattankulathur – 603203.

INDEX

SI. NO	DATE OF EXPERIMENT	TITLE OF EXPERIMENT	EVALUATION MARKS (40)	QUIZ / VIVA (05)	TOTAL MARKS (45)	SIGN OF STAFF

DEPT. OF ELECTRICAL & ELECTRONICS ENGINEERING
SRM UNIVERSITY, Kattankulathur – 603203.

INDEX

SI. NO	DATE OF EXPERIMENT	TITLE OF EXPERIMENT	EVALUATION MARKS (40)	QUIZ / VIVA (05)	TOTAL MARKS (45)	SIGN OF STAFF

DEPT. OF ELECTRICAL & ELECTRONICS ENGINEERING
SRM UNIVERSITY, Kattankulathur – 603203.

Title of Experiment :	
Name of the candidate :	
Register Number :	
Date of Experiment :	
Date of submission :	

S.No:	Marks split up	Maximum Marks	Marks Obtained
1	Pre viva questions	05	
2	Preparation of circuit diagrams	10	
3	Execution of experiment	15	
4	Calculations and Graph	10	
	Total	40	

Staff Signature

1. Basic Op-amp circuits

Pre Lab Questions:

1. What does operational amplifier refers to?

2. State the ideal characteristics of Op-amp.

3. What causes slew rate?

4 . State some linear applications of Op-amp.

5. State some non-linear applications of Op-amp.

1. BASIC OP-AMP CIRUITS

AIM:

1. To measure the following parameters of op-amp.
   i)Input bias current ii) Input offset voltage

2. To operate and obtain the output of i) Summing amplifier ii) Subtractor.

APPARATUS REQUIRED:

S.No.	APPARATUS	TYPE	RANGE	QUANTITY
1)	Op-Amp	µA741		1
2)	Resistors		100 / 10K / 1M	2 / 4/ 2
3)	Capacitors		0.01µF	1
4)	Signal Generator			1
5)	CRO			1
6)	Dual power supply			1
7)	Bread Board			1
8)	Connecting wires			

THEORY:

Input bias current: The inverting and noninverting terminals of an op-amp are actually two base terminals of transistors of a differential amplifier. In an ideal op-amp it is supported that no current flows through these terminals. However, practically a small amount of current flows through these terminals which is on the order of nA (typical and maximum values are 80 and 1500nA) in bipolar op-amps and pA for FET op-amps. Input bias current is defined as the average of the currents entering into the inverting and noninverting terminals of an op-amp. To compensate for bias currents a compensating resistor R_{comp} is used. Value of R_{comp} is parallel combination of the resistors connected to the inverting terminal. Input bias current $I_B = (I_{B1} + I_{B2})/2$, where I_{B1} and I_{B2} are the base bias currents of the op-amp.

Input offset voltage: Even if the input voltage is zero, output voltage may not be zero. This is because of the circuit imbalances inside the op-amp. In order to compensate this, a small voltage should be applied between the input terminals. Input offset voltage is defined as the voltage that must be applied between the input terminals of an op-amp to nullify the output voltage. Typical and maximum values of input offset voltage are 2mV and 6mV.

Summing Amplifier: Op-amp may be used to perform summing operation of several input signals in inverting in inverting and non-inverting mode. The input signals to be summed up are given to inverting terminal or non-inverting terminal through the input resistance to perform inverting and non-inverting summing operations respectively.

Subtractor: The basic difference amplifier can be used as a subtractor. The signals to be subtracted are connected to opposite polarity inputs i.e. in inverting or non-inverting terminals of the op-amp.

PROCEDURE:

i) Input Bias Current:

1. Connect the circuit as shown.
2. Measure the output voltage from which the inverting input bias current can be calculated as $I_B^- = Vo / R$
3. Connect the circuit as shown .
4. Measure the output voltage from which the non-inverting input bias current can be calculated as $I_B^+ = Vo / R_f$.
5. Average of magnitude of both I_B^- and I_B^+ gives the input bias current.

ii) Input Offset Voltage:

1. Connect the circuit as shown.
2. Measure the output voltage using multimeter.
3. Calculate offset voltage as $V_{ios} = V_o / (1+ R_f / R_1)$.

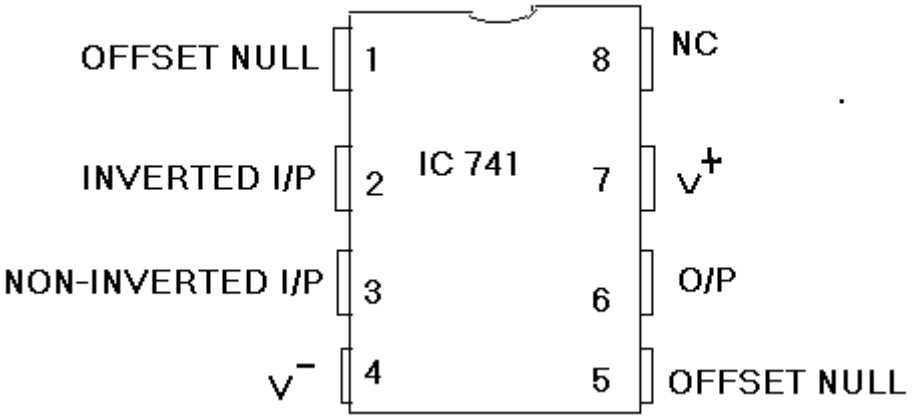
iii) Inverting Summing Amplifier:

1. Connect the circuit as shown in figure.
2. Connect batteries for voltage V_1, V_2.
3. Measure and note the output voltage and compare it with theoretical value , $Vo = -(R_f / R_i) (V_1+V_2)$.

iv) Subtractor:

1. Connect the circuit as shown in figure.
2. Measure and note the output voltage and compare it with theoretical value, $Vo = (V_1- V_2)$.

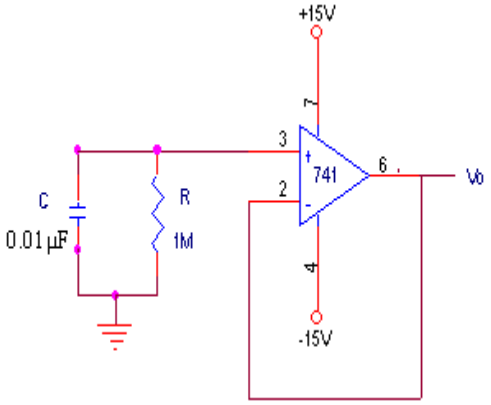
PIN DIAGRAM:

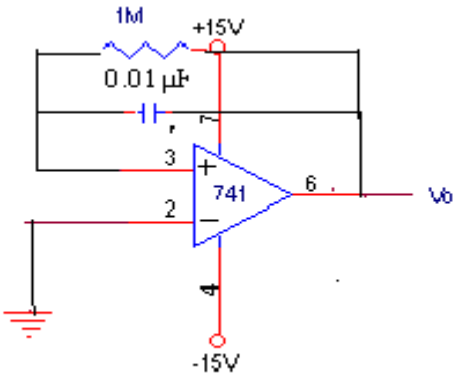


CIRCUIT DIAGRAMS:

1) To measure the following parameters:

i)To measure input bias current:

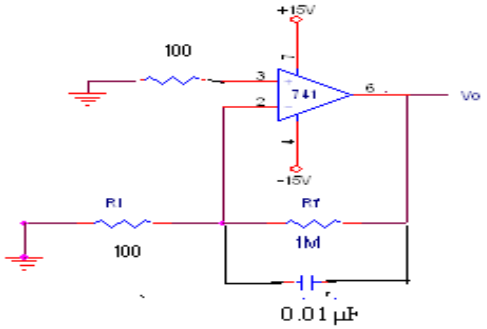




TABULAR COLUMN:

SI.NO	$I_B^- = Vo/R$	$I_B^+ = Vo/R_f$
	Input bias current , $I_B = \frac{I_B^+ + I_B^-}{2} =$	

ii)To measure input offset voltage :



TABULAR COLUMN:

SI.NO	$V_{i_{os}} = V_o / (1+ R_f / R_1)$..
	$V_o =$

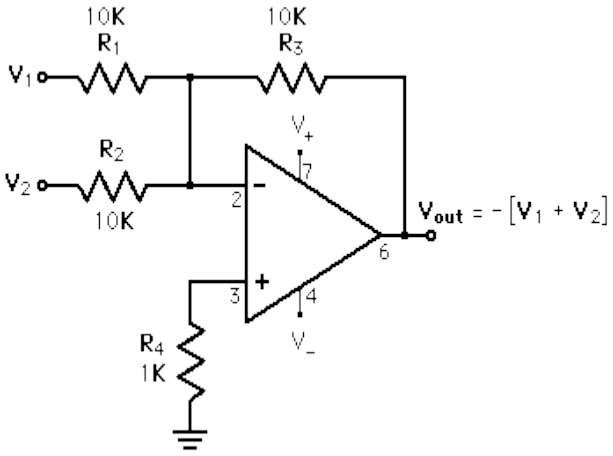
TYPICAL VALUES OF ELECTRICAL CHARACTERISTICS OF µA741:

Input bias current = 80-500nA

Input offset voltage = 1-5mV

2. To know the applications of op-amp:

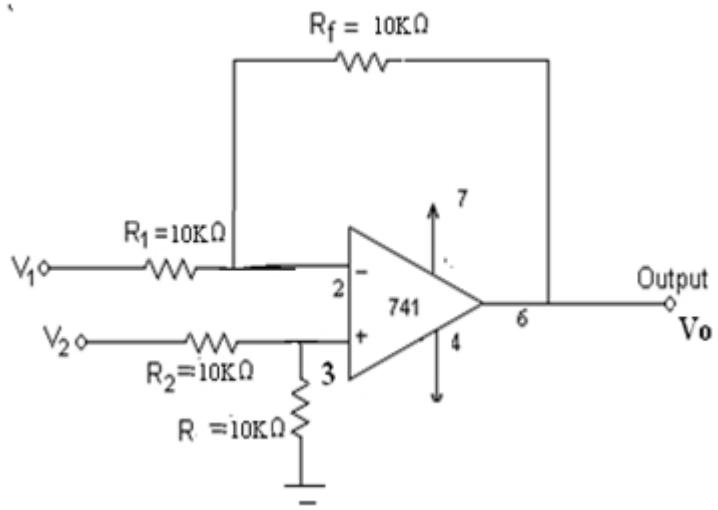
i) Summing Amplifier:



TABULAR COLUMN:

S.No.	V_1	V_2	Theoretical $V0 = -(V_1 + V_2)$	Practical V0

ii) Subtractor:



TABULAR COLUMN:

S.No.	V1	V2	Theoretical V0=V1-V2	Practical V0

RESULT:

1. The input bias current and input offset voltage of the op-amp were determined.

   Input bias current = ...…...A
   Input offset voltage = ……..mV

2. The applications of op-amps like summing amplifier and subtractor were understood.

Post Lab Questions

1. Why differential amplifier is used as an input stage of IC op-amp?

2. Why do we use R_{comp} resistor?

3. **What is thermal drift?**

4. **Why is IC741 op-amp not used for high frequency applications?**

5. What is unity gain circuit?

DEPT. OF ELECTRICAL & ELECTRONICS ENGINEERING
SRM UNIVERSITY, Kattankulathur – 603203.

Title of Experiment :	
Name of the candidate :	
Register Number :	
Date of Experiment :	
Date of submission :	

S.No:	Marks split up	Maximum Marks	Marks Obtained
1	Pre viva questions	05	
2	Preparation of circuit diagrams	10	
3	Execution of experiment	15	
4	Calculations and Graph	10	
	Total	40	

Staff Signature

2. Integrators and Differentiators

Pre Lab Questions:

1. **What are integrators and differentiators?**

2. **What are the applications of integrators?**

3. **Op-amp is used mostly as an integrator than a differentiator.Why?**

-

4. **Mention some applications of differentiators.**

5. **Draw the frequency response curves for integrator and differentiator.**

2. INTEGRATORS AND DIFFERENTIATORS

AIM:

To demonstrate the use of op-amp as Integrator and Differentiator .

APPARATUS REQUIRED:

S.No.	APPARATUS	TYPE	RANGE	QUANTITY
1)	Op-Amp	µA741-		1
2)	Resistors		1.5K / 10K/ 15K / 100K	2/2/ /1 / 1
3)	Capacitors		0.1 µ F / 0.01 µ F	1 / 1
4)	Signal Generator			1
5)	CRO			1
6)	Dual power supply			1
7)	Bread Board			1
8)	Connecting wires			

THEORY:

Integrator: Integrator is used to integrate the input waveform. i.e; $V_O = \int V_{in}\, dt$. Here in the inverting amplifier configuration, the feedback resistor R_f is replaced by capacitor C_f. Integrators are commonly used in wave shaping news, signal generators etc. For proper wave integration, T >> RC. Gain and linearity of the o/p are two advantages of op-amp integrators. Linearity is due to linear charging of capacitor. Its limitation is for Vin=0 and for low frequencies, $X_{Cf} = \infty$ or the capacitor C_f acts as an open circuit. Therefore the op-amp integrator works as an open loop amplifier and the gain becomes infinity or very high.

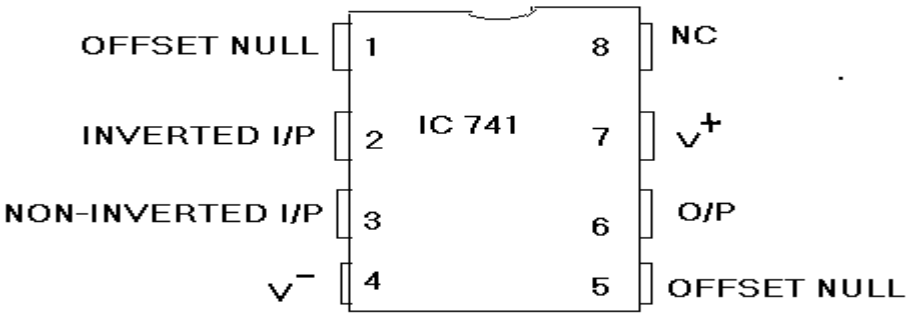
Differentiator: Here the output waveform is the derivative of the i/p waveform. In a basic inverting amplifier, if R_1 is replaced by C_1, we get the differentiator. But at high frequencies, the gain of the circuit (Rf/XC1) increases with increase in frequency at the rate of 20dB/decade. This makes the circuit unstable. Also X_{C1} decreases when frequency increases.

PROCEDURE:

Integrator & Differentiator:

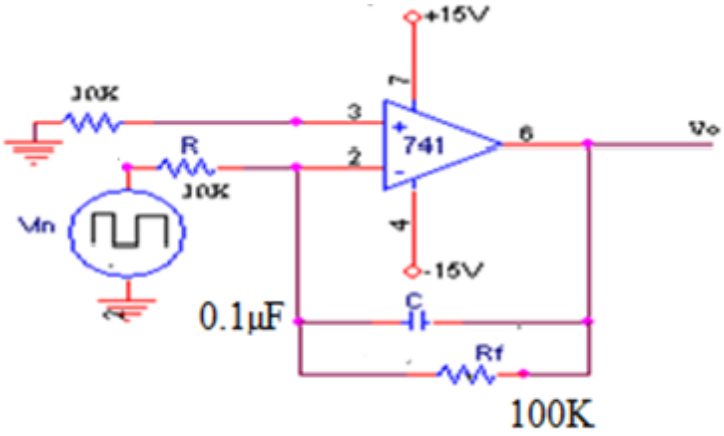
1. Connections are made as per the diagram.
2. Apply an i/p voltage of 1-2Vpp with 1kHz frequency and check the waveform on the CRO.
3. Measure the value of V_O by varying the frequency of the i/p signal.
4. Calculate gain using the formulae 20 log (V_O / V_{IN}).

PIN DIAGRAM:

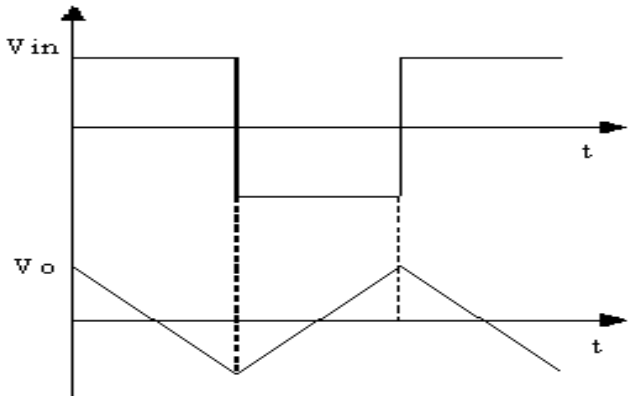


CIRCUIT DIAGRAM:

Integrator:



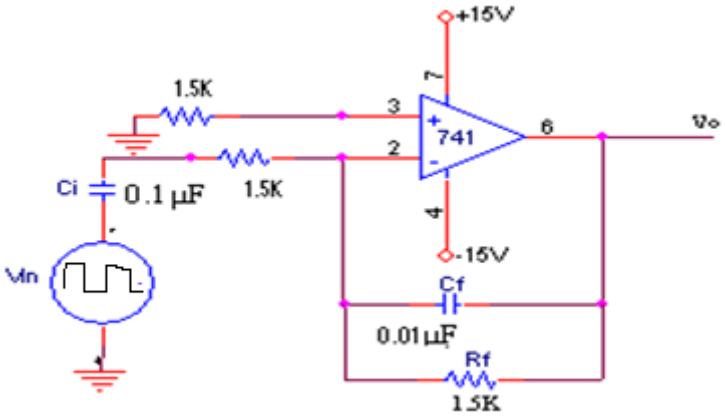
Model Graph:



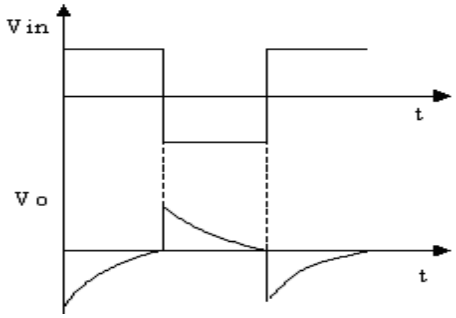
TABULAR COLUMN:

S.No:	INPUT		OUTPUT	
	Vin (V)	Time (msec)	Vout(V)	Time (msec)

Differentiator:



Model Graph:



TABULAR COLUMN:

S.No:	INPUT		OUTPUT	
	Vin (V)	Time (msec)	Vout(V)	Time (msec)

RESULT:

Thus the output waveforms of integrator and differentiator were obtained and the graphs were drawn.

Post viva :

1. **What is the requirement of a high pass filter to act as differentiator?**

2. **What is time constant of an RC circuit?**

3. **What is the requirement of a low pass filter to act as integrator?**

4. **What are the modes in which op-amp is operated with finite gain and infinite gain?**

5. **Differences between active and passive differentiators.**

DEPT. OF ELECTRICAL & ELECTRONICS ENGINEERING
SRM UNIVERSITY, Kattankulathur – 603203.

Title of Experiment	:
Name of the candidate	:
Register Number	:
Date of Experiment	:
Date of submission	:

S.No:	Marks split up	Maximum Marks	Marks Obtained
1	Pre viva questions	05	
2	Preparation of circuit diagrams	10	
3	Execution of experiment	15	
4	Calculations and Graph	10	
	Total	40	

Staff Signature

3. Comparators

Pre lab Questions :

1. What is a comparator?

2. What is the difference between a basic comparator and ZCD?

3. Depending on the value of input and reference voltage a comparator can be named as ……..

4.How can we obtain high rate of accuracy in comparator?

5. Zero crossing detectors are also called as ……..

3. COMPARATORS

AIM:

To obtain the output of voltage comparator and zero crossing detector.

APPARATUS REQUIRED:

S.No.	APPARATUS	TYPE	RANGE	QUANTITY
1)	Op-Amp	µA741		1
2)	Resistors		1K	2
4)	Signal Generator			1
5)	CRO			1
6)	Dual power supply			1
7)	Bread Board			1
8)	Connecting wires			

THEORY:

Voltage Comparator: A comparator is a circuit which compares a signal voltage applied at one input of an op-amp with output $\pm V_{sat} = (V_{cc})$. If the signal is applied to the inverting terminal of the op-amp it is called inverting comparator and if the signal is applied to non-inverting terminal of the op-amp it is called non-inverting comparator. In an inverting comparator if input signal is less than reference voltage, output will be $+V_{sat}$. When input signal voltage is greater than reference voltage output will be $-V_{sat}$. The vice-versa takes place in non-inverting comparator.

Zero Crossing Detector: Zero crossing comparator (ZCD) is an application of voltage comparator. It converts any time varying signal to square of same time period with amplitude $\pm V_{sat}$. The reference voltage is set as zero volts. When the polarity of the input signal changes, output square wave changes polarity.

PROCEDURE:

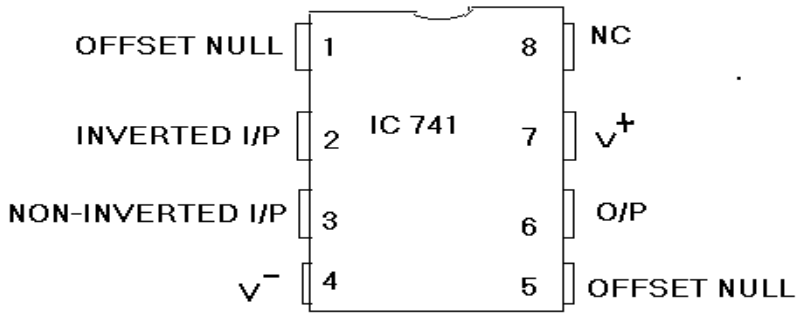
1) **Voltage comparator:**

1. Connect the circuit as shown in the figure.
2. Connect an alternating waveform to the non-inverting input of the op-amp.
3. Connect a reference voltage source to inverting input of the op-amp.
4. Plot the input and output waveforms.

2) Zero crossing detector:

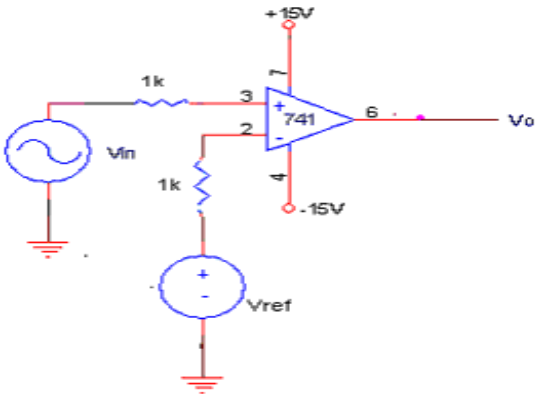
1. Connect the circuit as shown in figure.
2. Connect the input to a signal generator generating a sin wave with one volt peak to peak at 1kHz.
3. Connect the input and output to dual channel CRO and compare the input and output.
4. Plot the input and output waveforms on a graph.

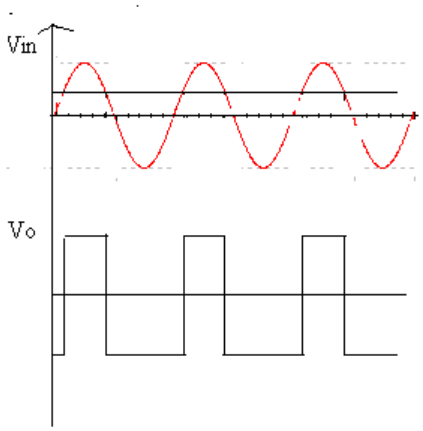
PIN DIAGRAM:



CIRCUIT DIAGRAM:

Voltage Comparator:

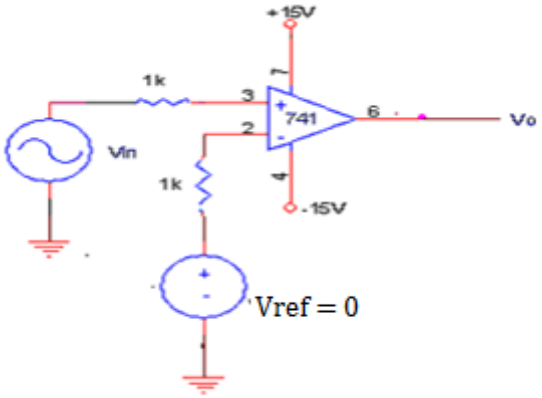




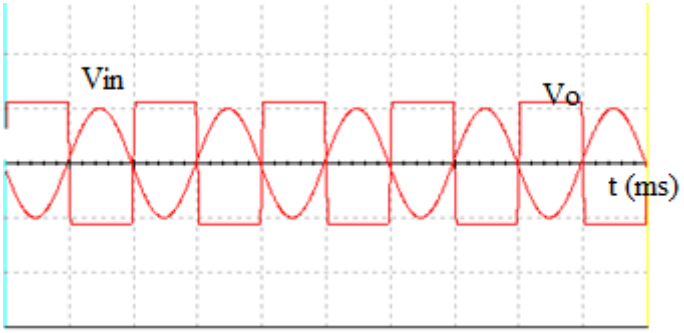
TABULAR COLUMN:

SI. NO	INPUT		OUTPUT	
	VOLTAGE (V)	TIME (ms)	VOLTAGE (V)	TIME (ms)

Zero Crossing Detector:



MODELGRAPH:



TABULAR COLUMN:

SI. NO	INPUT		OUTPUT	
	VOLTAGE (V)	TIME (ms)	VOLTAGE (V)	TIME (ms)

RESULT:

Thus the output of voltage comparator and zero crossing detector were obtained and the graphs have been drawn.

Post Lab Questions:

1. **Give some applications of comparator.**

2. **What is a window detector?**

3. **Draw the characteristics of ideal and practical comparator.**

4. **What is the basic difference between a basic comparator and a Schmitt trigger?**

5. **Define hysteresis .**

DEPT. OF ELECTRICAL & ELECTRONICS ENGINEERING
SRM UNIVERSITY, Kattankulathur – 603203.

Title of Experiment :	
Name of the candidate :	
Register Number :	
Date of Experiment :	
Date of submission :	

S.No:	Marks split up	Maximum Marks	Marks Obtained
1	Pre viva questions	05	
2	Preparation of circuit diagrams	10	
3	Execution of experiment	15	
4	Calculations and Graph	10	
	Total	40	

Staff Signature

4. Wave shaping circuits

Prelab Questions:

1. State Barkhausen criterion.

2. Effect of reference voltage when clipping a circuit.

3. What are the other names of clippers?

4. How clippers are applicable in noise reduction in audio applications?

5. Why clamper is called a DC restorer?

4. WAVESHAPING CIRCUITS

AIM:

To know the operation of clippers and clampers.

APPARATUS REQUIRED:

S.No.	APPARATUS	TYPE	RANGE	QUANTITY
1)	Op-Amp	µA741		1
2)	Resistors		4.7K Ω/ 1KΩ / 10 KΩ / 10KΩ POT	1 / 2 / 2 / 1
3)	Capacitors		0.1 µ F	1
4)	Signal Generator			1
5)	Diode	IN 4001		1
6)	CRO			1
7)	Dual power supply			1
8)	Bread Board			1
9)	Connecting wires			

THEORY:

1) Clipper :

Clipper is a circuit that removes positive or negative level of the input signal and can be designed using op amp with rectifier diodes. The op amp is basically used as a voltage follower with the feedback path ,the reference voltage determines the level of voltage to be clipped both either positive or negative . A negative clipper is obtained by just reversing the diode.

2) Clamper :

Clamper is a circuit used to add D.C voltage to the input signal .It is also called a D.C inverter or/ restorer.

PROCEDURE:

1) Clippers:

1. Connect the trainer to mains and switch on the power supply
2. Measure the output voltage of regulated power supply i.e. +12V and -12V using digital multimeter
3.Observe the output of the on board signal generator with the help of Oscilloscope Signal should be sine wave of 1KHZ frequency with 10Vpp amplitude or connect external signal generator

Positive clippers

4. Connect the circuit as shown.
5. Observe the input and output waveforms with the help of dual trace Oscilloscope and compare them with the expected waveform
6. Repeat the same at different voltage source by varying 10 K pot
7. Now connect a negative V ref fig. and observe the wave forms

Negative clippers

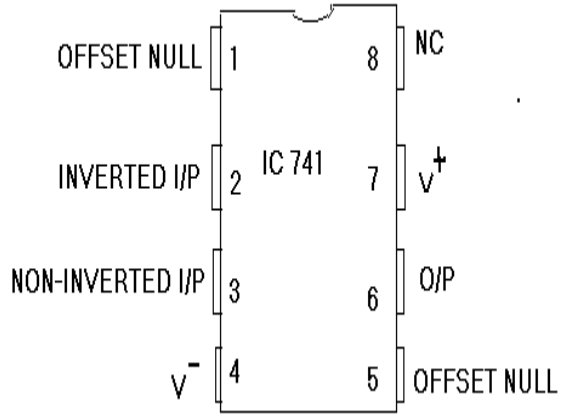
8. Connect the circuit as shown.
9. Observe the input and output waveforms with the help of dual trace Oscilloscope and compare them with the expected waveform.
10. Repeat the same at different voltage source and positive Vref ..

2) Clampers:

Positive clampers

1. Connect the circuit as shown in fig.
2. Observe the input and output waveforms with the help of dual trace Oscilloscope and compare them with the expected waveform.
3. Repeat the same at different voltage source and changing the Vref

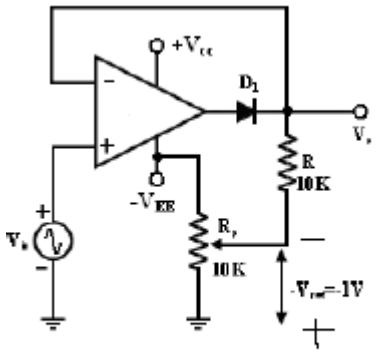
PIN DIAGRAM:

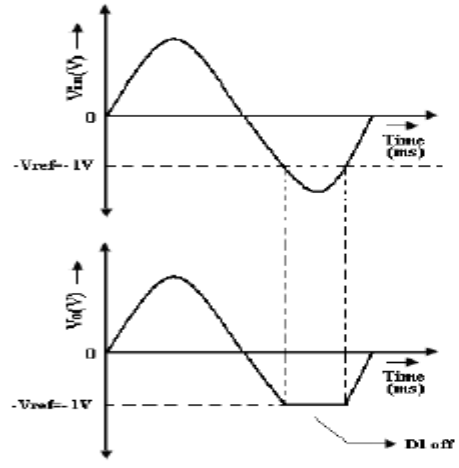


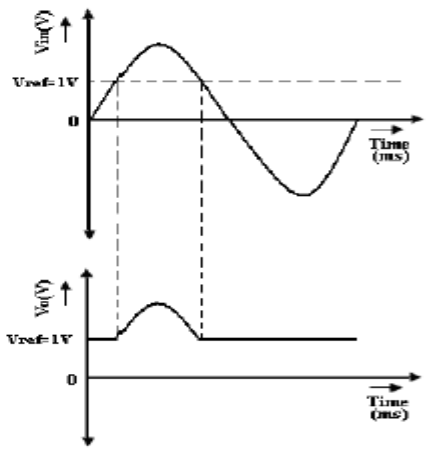
CIRCUIT DIAGRAM:

1) **Clippers:**

NEGATIVE CLIPPER:



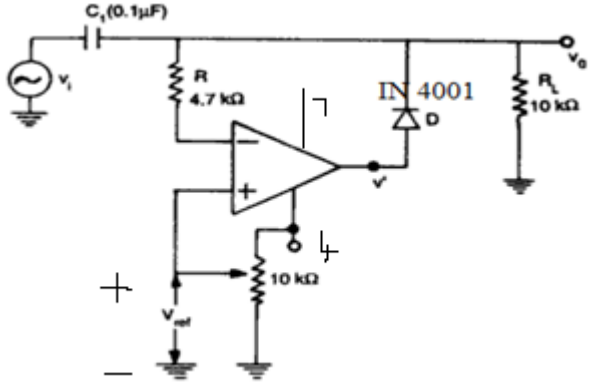


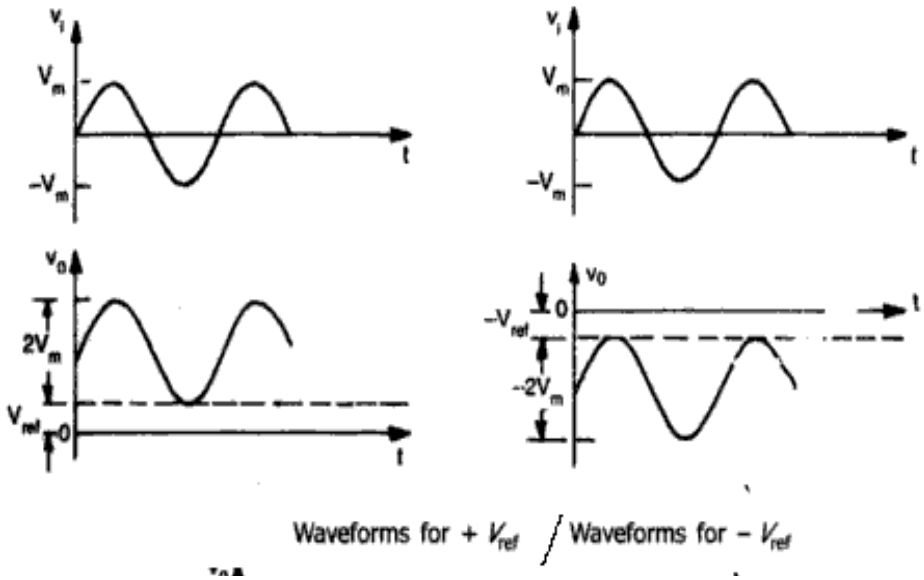


TABULAR COLUMN:

S.No:	INPUT		OUTPUT	
	Vin (V)	Time (msec)	Vout(V)	Time (msec)

2) Clamper:





Waveforms for + V_{ref} / Waveforms for – V_{ref}

TABULAR COLUMN:

S.No:	INPUT		OUTPUT	
	Vin (V)	Time (msec)	Vout(V)	Time (msec)

RESULT:

The operation of clipper and clamper have been studied and its output waveforms have been drawn.

Postlab Questions:

1. What is the difference between slicing and clipping of a voltage waveform?

2. How can you obtain triangular wave using schmitt trigger?

3. What is the factor which determines clipping level in a clipper?

4. What is the use of resistor in clamper circuits?

5. What are the applications of clippers and clampers?

DEPT. OF ELECTRICAL & ELECTRONICS ENGINEERING
SRM UNIVERSITY, Kattankulathur – 603203.

Title of Experiment :	
Name of the candidate :	
Register Number :	
Date of Experiment :	
Date of submission :	

S.No:	Marks split up	Maximum Marks	Marks Obtained
1	Pre viva questions	05	
2	Preparation of circuit diagrams	10	
3	Execution of experiment	15	
4	Calculations and Graph	10	
	Total	40	

Staff Signature

5. Waveform generators using op-amp

Prelab Questions:

1. **What is the basic classification of oscillator?**

2. **What is a regenerative comparator?**

3. **Wien bridge oscillator uses positive and negative feedback. Why?**

4. **What is the function of lead-lag network in Wien bridge oscillator?**

5. **What is the difference between an amplifier and an oscillator?**

5.WAVEFORM GENERATING CIRCUITS.

AIM:

To design a Schmitt trigger and Wien bridge oscillator and to study their operation.

APPARATUS REQUIRED:

S.No.	APPARATUS	TYPE	RANGE	QUANTITY
1)	Op-Amp	µA741		1
2)	Resistors		29KΩ/10KΩ POT/ 16KΩ, 1.6KΩ	1/1 / 1/ 2
3)	Capacitors		0.1µF	2
4)	Signal Generator			1
5)	CRO			1
6)	Dual power supply			1
7)	Bread Board			1
8)	Connecting wires			

THEORY:

1) Schmitt Trigger

Schmitt trigger is otherwise called regenerative comparator. In this comparator circuit a positive feedback is added. The input voltage Vi triggers the output Vo very time it exceeds certain voltage levels. These voltages are known as upper threshold voltage (V_{UT}) and lower threshold voltage V_{LT}. The difference between the two threshold voltages ($V_{UT} - V_{LT}$) gives the hysteresis width.

$$V_{UT} = V_{ref} + (R_2 / (R_1 + R_2)) * (V_{sat} - V_{ref})$$

V_{ref} - applied reference voltage
V_{sat} - saturation voltage of OP-AMP
R_1, R_2 - Voltage divider resistances

$$V_{LT} = V_{ref} - (R_2 / (R_1 + R_2)) * (V_{sat} - V_{ref})$$

When input voltage is greater than V_{UT}, output goes to negative saturation and when input voltage is less than V_{LT}, output goes to positive saturation.

2) **Wien bridge Oscillator:**

Wien bridge circuit is connected between the amplifier input terminals and output terminals. 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, resistors R_1 and R_f are connected. The phase angle criterion for oscillation is that the total phase shift around the circuit must be 0^o .This conditions occurs only when the bridge is balanced, that is, at resonance. The frequency of oscillation is exactly the resonant frequency of the balanced Wien bridge.

Design of Schmitt trigger

Given $V_{UT} = 0.5V$ $V_{LT} = -0.5V$

$$V_{UT} = \frac{R_2}{R_1 + R_2} V_{sat}$$

$$V_{LT} = \frac{R_2}{R_1 + R_2} (-V_{sat})$$

Taking $\pm V_{sat} = \pm 15V$,

$$0.5 = \frac{R_2}{R_1 + R_2} (15V)$$

$$\frac{R_2}{R_1 + R_2} = 30$$

$$\frac{R_2}{R_2} = 29$$

$= R_1 = 29R_2$

Taking $R_2 = 1K\Omega$

$R_1 = 29K\Omega$ (set using 100K POT).

Design of Wien bridge oscillator:

f = $1 / 2 \Pi RC = 0.159 / RC$

Assuming that the resisters are equal in value, and capacitors are equal in value in the reactive leg of the wien bridge. At this frequency the gain required for substained oscillation is given by

Ao -> open loop gain the op-amp

β -> feed back factor

$$Ao = \frac{1}{3}$$

$$\frac{1 + Rf}{R} = 3$$

$$\frac{R_1 + R_f}{R_1} = 3$$

$R_1 + R_f = 3R_1$, $R_f = 2R_1$

PROCEDURE:

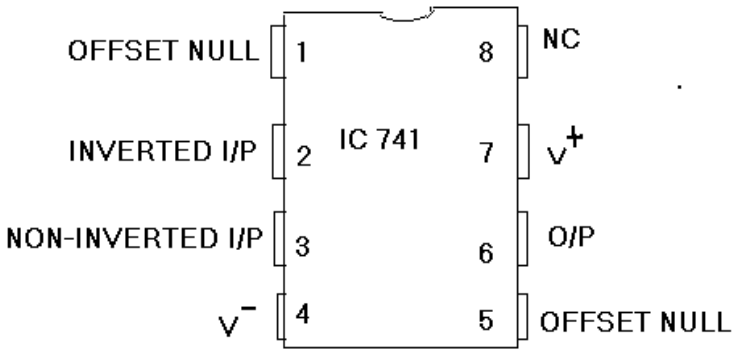
1) Schmitt trigger:

1. Connect the circuit as shown in diagram
2. See the input sine wave and output from pin.6 in a dual channel, CRO
3. Plot the observed waveforms in a linear graph.
4. Calculate the lower threshold voltage and upper threshold voltage from the plotted graph.
5. Calculate the lower threshold voltage and upper threshold voltage theoretically using the formula.

2) Wien bridge Oscillator:

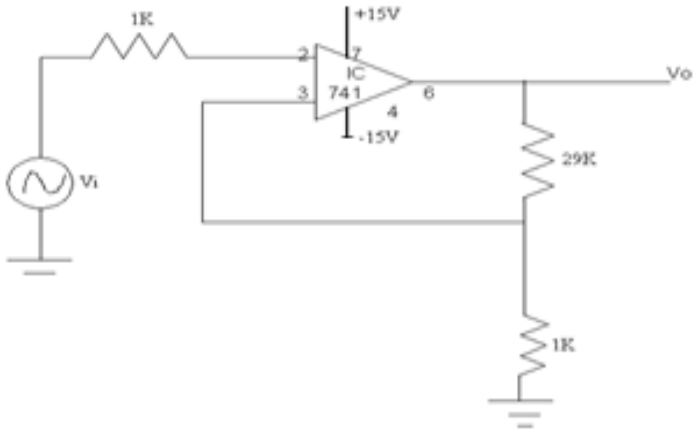
1. Design the oscillator for desired frequency using equations.
2. Connect the circuit as per the circuit diagram in figure.
3. Connect the output to the CRO.
4. Adjust the POT Rf in feedback loop so that the output is a sine wave.
5. Plot the output in a graph.

PIN DIAGRAM:

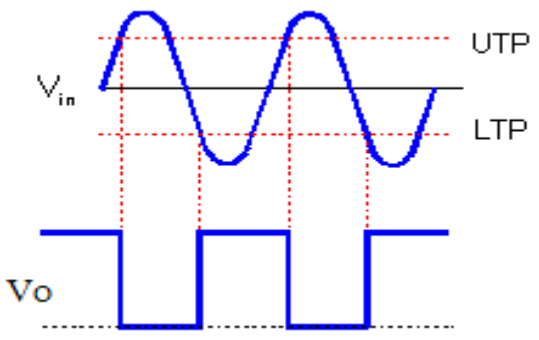


CIRCUIT DIAGRAM:

1) **Schmitt Trigger:**



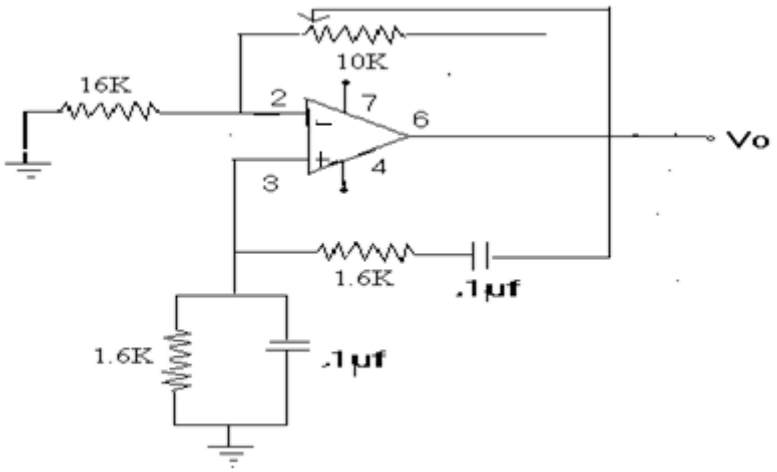
MODEL GRAPH:

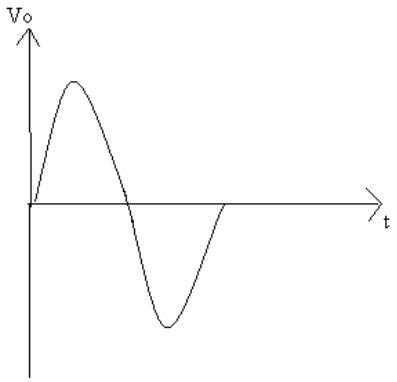


TABULAR COLUMN:

S.No:	INPUT		OUTPUT	
	Vin (V)	Time (msec)	Vout(V)	Time (msec)

2) Wein Bridge Oscillator:





TABULAR COLUMN:

S.No:	OUTPUT		Theoretical Frequency(Hz)	Practical Frequency(Hz)
	Vin (V)	Time (msec)		

RESULT:

Thus the Schmitt trigger and Wein bridge oscillator hsve been designed and its output has been obtained.

Postlab Questions:

1. **Differentiate RC and LC oscillators.**

2. **What is relaxation oscillator?**

3. **Give some practical applications of oscillators.**

4. **List out some advantages of Wien bridge oscillator.**

5. **List out some disadvantages of Wien bridge oscillator.**

DEPT. OF ELECTRICAL & ELECTRONICS ENGINEERING
SRM UNIVERSITY, Kattankulathur – 603203.

Title of Experiment :	
Name of the candidate :	
Register Number :	
Date of Experiment :	
Date of submission :	

S.No:	Marks split up	Maximum Marks	Marks Obtained
1	Pre viva questions	05	
2	Preparation of circuit diagrams	10	
3	Execution of experiment	15	
4	Calculations and Graph	10	
	Total	40	

Staff Signature

6.Waveform Generators using 555 Timer

Prelab Questions:

1. **Give the pin diagram of 555 timer IC.**

2. **What is multivibrator?**

3. **What is quasi stable state?**

4. **What are the various modes of operation of multivibrator? Explain.**

5. **What is one-shot multivibrator?**

6. WAVEFORM GENERATOR USING 555 TIMER

AIM:

To study the application of IC555 as an astable multivibrator.

APPARATUS REQUIRED *:*

S.NO	APPARATUS	RANGE	QUANTITY
1)	IC	NE555	1
2)	Resistor	1KΩ, 2.2KΩ	1
3)	Capacitor	0.1μF, 0.01μF	1
4)	CRO	-	1
5)	RPS	DUAL(0-30) V	1
6)	Connecting Wires		

THEORY:

The IC555 timer is a 8 pin IC that can be connected to external components for astable operation. The simplified block diagram is drawn. The OP-AMP has threshold and control inputs. Whenever the threshold voltage exceeds the control voltage, the high output from the OP –AMP will set the flip-flop. The collector of discharge transistor goes to pin 7. When this pin is connected to an external trimming capacitor, a high Q output from the flip flop will saturate the transistor and discharge the capacitor.

When Q is low the transistor opens and the capacitor charges. The complementary signal out of the flip-flop goes to pin 3 and output. When external reset pin is grounded it inhibits the device. The on – off feature is useful in many application. The lower OP- AMP inverting terminal input is called the trigger because of the voltage divider. The non-inverting input has a voltage of +Vcc/3, the OP-Amp output goes high and resets the flip flop.

The output frequency is,

$$f = 1.44/(R_A + R_B)C$$

The duty cycle is,

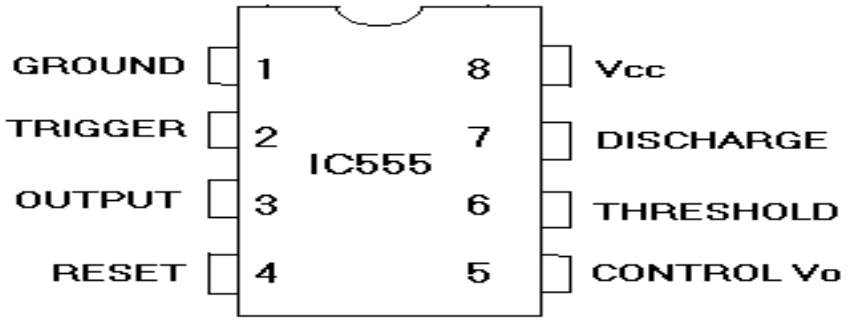
$$D = R_A + R_B / (R_A + 2R_B) * 100\%$$

The duty cycle is between 50 to 100% depending on R_A and R_B.

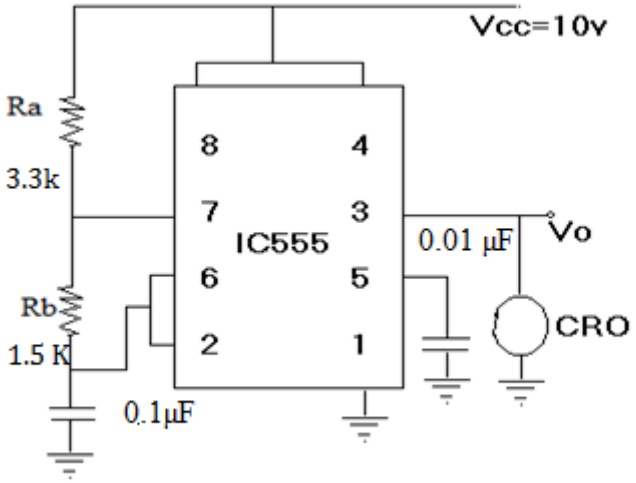
PROCEDURE:

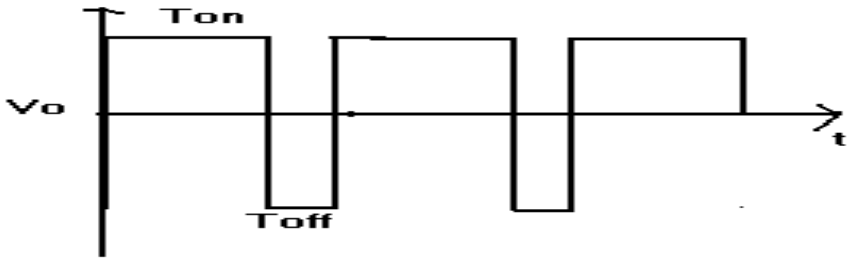
1. The connections are made as per the circuit diagram and the values of R and C are calculated assuming anyone term.
2. The output waveform is noted down and graph is drawn and also the theoretical and practical time period is verified.

PIN DIAGRAM:



CIRCUIT DIAGRAM:





TABULAR COLUMN:

S.No:	OUTPUT		Theoretical Frequency(Hz)	Practical Frequency(Hz)
	Vin (V)	Time (msec)		

RESULT:

Thus the astable multivibrator circuit using IC555 is constructed and verified its theoretical and practical time period.

Post Lab Questions:

1. **List the basic blocks of IC 555 timer?**

2. **Give the applications of 555-timer Astable multivibrator**.

3. **What is the advantage of 555 IC over op amp?**

4. **List the applications of monostable mode of 555 timer.**

5. **Name other circuits that employ 555 TimerIC.**

DEPT. OF ELECTRICAL & ELECTRONICS ENGINEERING
SRM UNIVERSITY, Kattankulathur – 603203.

Title of Experiment :	
Name of the candidate :	
Register Number :	
Date of Experiment :	
Date of submission :	

S.No:	Marks split up	Maximum Marks	Marks Obtained
1	Pre viva questions	05	
2	Preparation of circuit diagrams	10	
3	Execution of experiment	15	
4	Calculations and Graph	10	
	Total	40	

Staff Signature

7. Rectifiers

Prelab Questions:

1. **What is a rectifier?**

2. **Give some applications of rectifier.**

3. **What are the types of rectifiers and their configurations?**

4. **What do you mean by the terms ripple factor and peak inverse voltage?**

5. **Why half wave rectifiers are generally not used in rectifiers?**

7. RECTIFIERS

AIM:

To obtain the output of half wave and full wave rectifier.

APPARATUS REQUIRED:

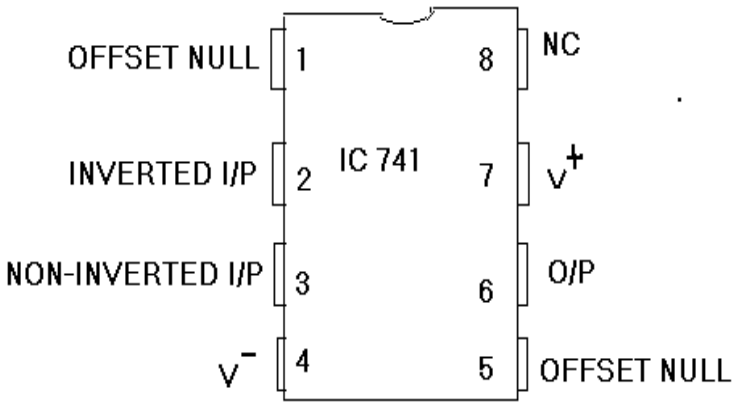
S.No.	APPARATUS	TYPE	RANGE	QUANTITY
1)	Op-Amp	µA741		2
2)	Resistors		10KΩ/ 1K	5 / 2
3)	Diodes	IN 4001		2
4)	Signal Generator			1
5)	CRO			1
6)	Dual power supply			1
7)	Bread Board			1
8)	Connecting wires			

THEORY:

HALF WAVE RECTIFIER: When the input signal is positive, *i.e.*, $V_i > 0$, D_1 is forward biased and D_1 conducts. D_2 is reverse biased and D_2 does not conduct. Therefore, no current flows through R_f and output voltage $V_o = 0$. When the input signal is negative, *i.e.*, $V_i < 0$, D_1 is reverse biased and D_1 does not conduct. Therefore, D_2 conducts and the circuit behaves like an inverter causing the output voltage to become positive.

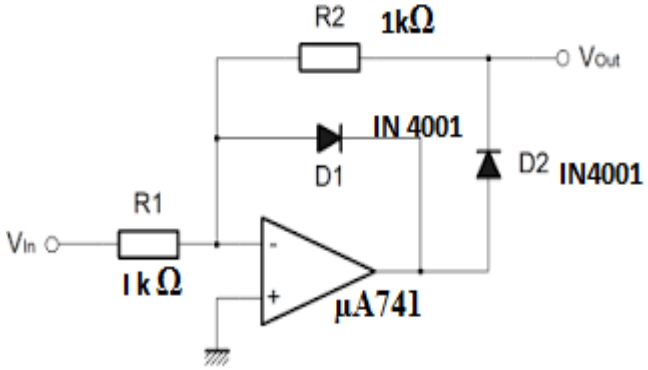
FULL WAVE RECTIFIER: The First Operational amplifier (OP AMP) (shown in circuit) act as inverting amplifier while the second op amp act as Non Inverting amplifier. The circuit mainly works in two modes, depending upon the polarity of input voltage. One is inverting and the other is in non inverting mode. When Vin is Positive diode D1 conducts at that time diode D2 doesn't conduct. The limitation of this circuit is that it doesn't have high input impedance. Op amp rectifier is also called as precision rectifier, because it is able to rectify lower amplitude signals. In ordinary diode rectifiers it need a minimum input voltage, in the order of cut in voltage of diode.

PIN DIAGRAM:

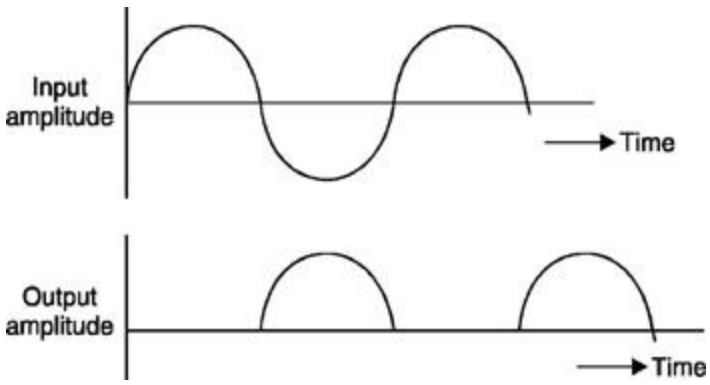


CIRCUIT DIAGRAM:

HALF WAVE RECTIFIER:



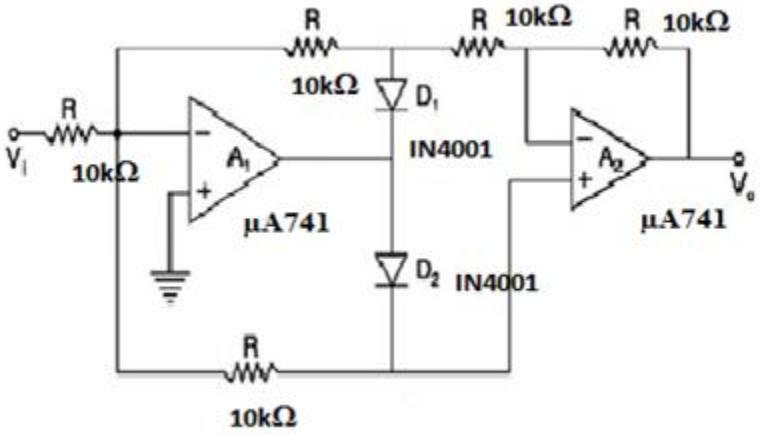
MODEL GRAPH:



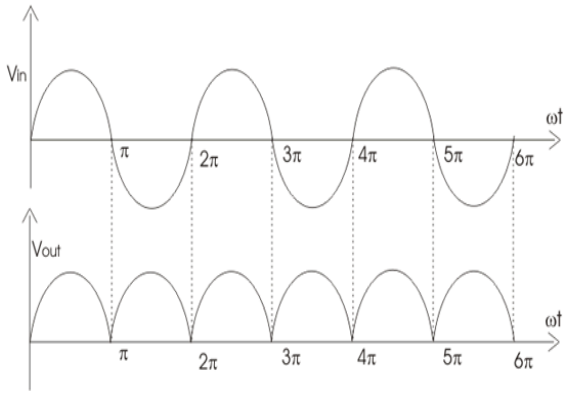
TABULAR COLUMN:

S.No:	INPUT		OUTPUT	
	Vin (V)	Time (msec)	Vout(V)	Time (msec)

FULL WAVE RECTIIFIER:



MODEL GRAPH:



TABULAR COLUMN:

S.No:	INPUT		OUTPUT	
	Vin (V)	Time (msec)	Vout(V)	Time (msec)

RESULT:

Thus the rectifier circuits were operated and their corresponding waveforms have been drawn.

Postlab Questions:

1. **What is transformer utilization factor?**

2. **Whay series inductor filters are not used in half wave rectifiers?**

3. **Give the ripple factor for half wave and full wave rectifiers.**

4. **Compare half wave and full wave rectifier.**

5. **List out some disadvantages of full wave rectifier.**

DEPT. OF ELECTRICAL & ELECTRONICS ENGINEERING
SRM UNIVERSITY, Kattankulathur – 603203.

Title of Experiment	:
Name of the candidate	:
Register Number	:
Date of Experiment	:
Date of submission	:

S.No:	Marks split up	Maximum Marks	Marks Obtained
1	Pre viva questions	05	
2	Preparation of circuit diagrams	10	
3	Execution of experiment	15	
4	Calculations and Graph	10	
	Total	40	

Staff Signature

8.Design of LPF, HPF, BPF and Band Reject Filters.

Pre Lab Questions:

1. **What is a filter?**

2. **State the advantage of active filters over passive?**

3. **Define order of a filter?**

4. **What are the types of active filters?**

5. **What is frequency response?**

8.DESIGN OF LPF, HPF, BPF AND BAND REJECT FILTERS

AIM:

To demonstrate the use of op-amp as low pass and high pass filter.

APPARATUS REQUIRED:

S.No.	APPARATUS	TYPE	RANGE	QUANTITY
1)	Op-Amp	µA741		1
2)	Resistors		10K / 5.86K/ 1.6K/ 3.4K / 6.8K/ 39.8K/7.9K	4 / 1/ 2 / 1 / 2/ 1 / 1
3)	Capacitors		0.1µF / 0.01 µF / 0.47 µF/ 0.94 µF	2/2/2/ 1
4)	Signal Generator			1
5)	CRO			1
6)	Dual power supply			1
7)	Bread Board			1
8)	Connecting wires			

THEORY:

A frequency selective circuit that passes electric signals of desired band of frequencies and attenuates the other band of frequencies outside the band is called an electric filter. At radio frequencies, inductors become problematic. Hence RC filters are used. As op-amp is used in non-inverting mode, it offers high input impedance and low output impedance. This improves the load driving capacity. The most commonly used filters are low pass filter (LPF), high pass filter (HPF), band pas filter (BPF), band stop filter (BSF). Low pass filter passes signals whose frequency is less than cut-off frequency. High pass filter passes signals whose frequency is greater than cut-off frequency. Depending upon the number of RC networks used in the circuit, filters are classified as first order filter if the network contains one RC network and as second order if it contains two RC networks.

DESIGN:

$f_c = (1/2\Pi RC)$

f_c - cut-off frequency

1. BAND PASS FILTER:

Given specifications: Wide BPF (Q <10),

$f_l = 400\ Hz$, and f_h = 2 KHz , A_o = 2

$A_0 = 1 + \frac{R_f}{R_i} = 2$

Let $R_f = R_i = 10\ K\Omega$ for both HPF and LPF sections

LPF:

f_h = 2 KHz = $\frac{1}{2\pi R_1 C_1}$, Let C_1 = 0.01 µ F

Therefore, $R_1 = 7.9\ K\Omega \approx$---------- KΩ

HPF:

f_l = 400 Hz = $\frac{1}{2\pi R_2 C_2}$, Let C_2 = 0.01 µ F

Therefore, $R_2 = 39.8\ K\Omega \approx$---------- KΩ

$f_o = \sqrt{f_h f_l}$ =$\sqrt{2000\ x400}$ = 894.4 Hz

Q = $\frac{f_o}{BW}$ =$\frac{f_o}{f_h - f_l}$ = $\frac{894.4}{2000-400}$ =0.56 , Q = 0.56<10, Wide BPF

2. BAND STOP FILTER:

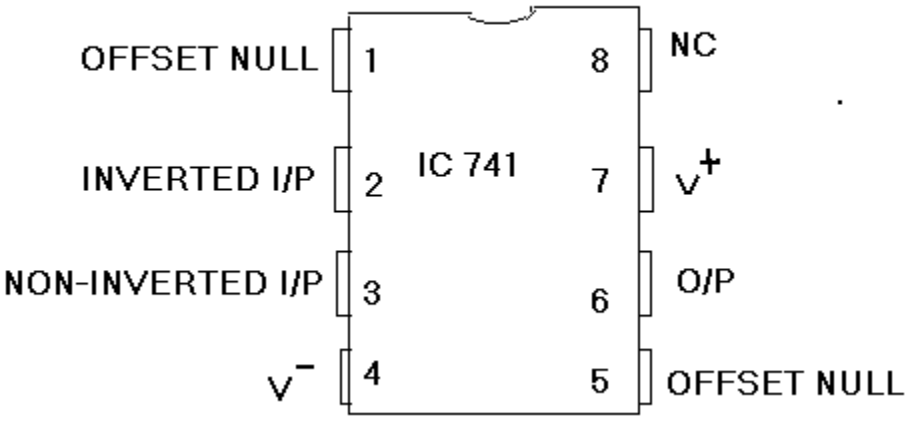
Given specifications, Narrow BRF (Q >10), $C = 0.47\ \mu F$

R=$\frac{1}{2*\pi*50*0.47*10^{-6}}$ = 6.772kΩ.

PROCEDURE:

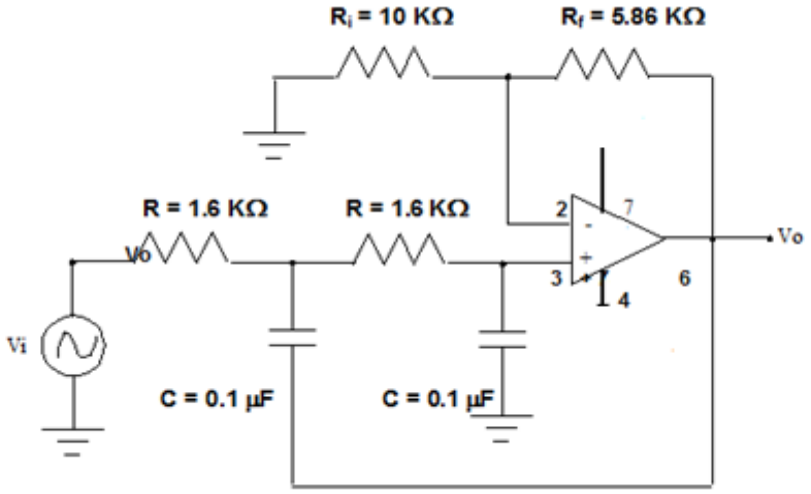
1. Connect the circuit as per the circuit diagram.
2. Give the sine input through the function generator at 1kHz.
3. Observe the output in the CRO for various frequencies.
4. Calculate voltage gain and plot the graph between voltage gain and frequency.
5. Draw 3dB line to determine the cut-off frequency and verify it with the design frequency.

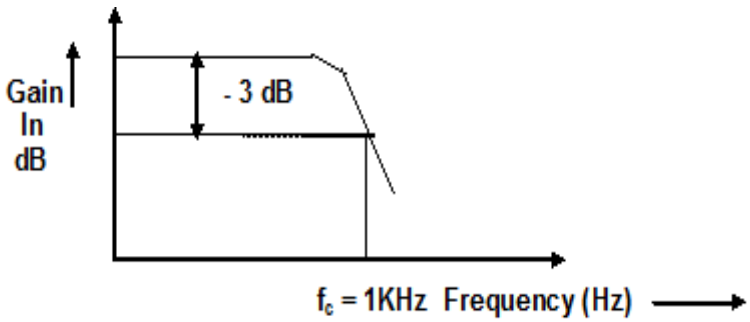
PIN DIAGRAM:



CIRCUIT DIAGRAM:

1) Second Order LPF:

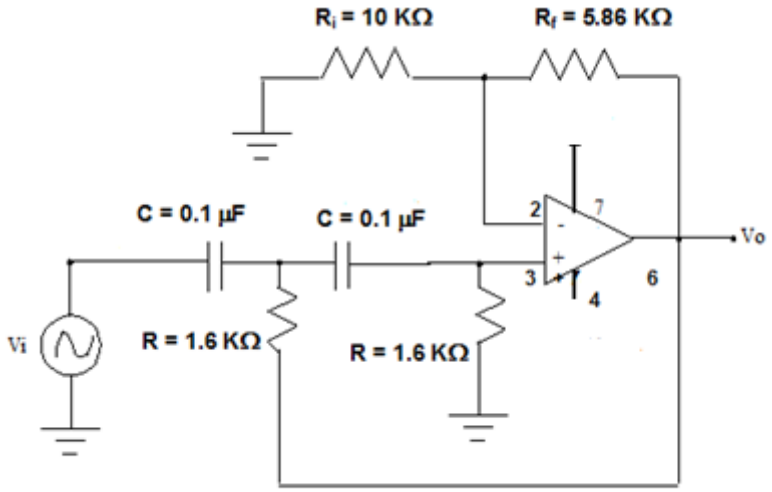




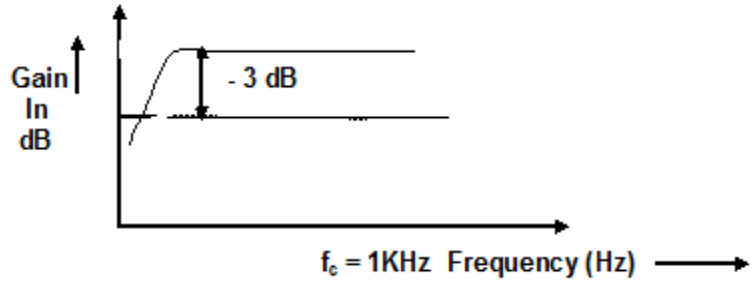
TABULAR COLUMN:

SI NO	FREQUENCY	INPUT VOLTAGE (V)	OUTPUT VOLTAGE (V)	GAIN = V_O/V_{IN}	GAIN= 20 log (V_O/V_{IN}) db

2) Second Order HPF:



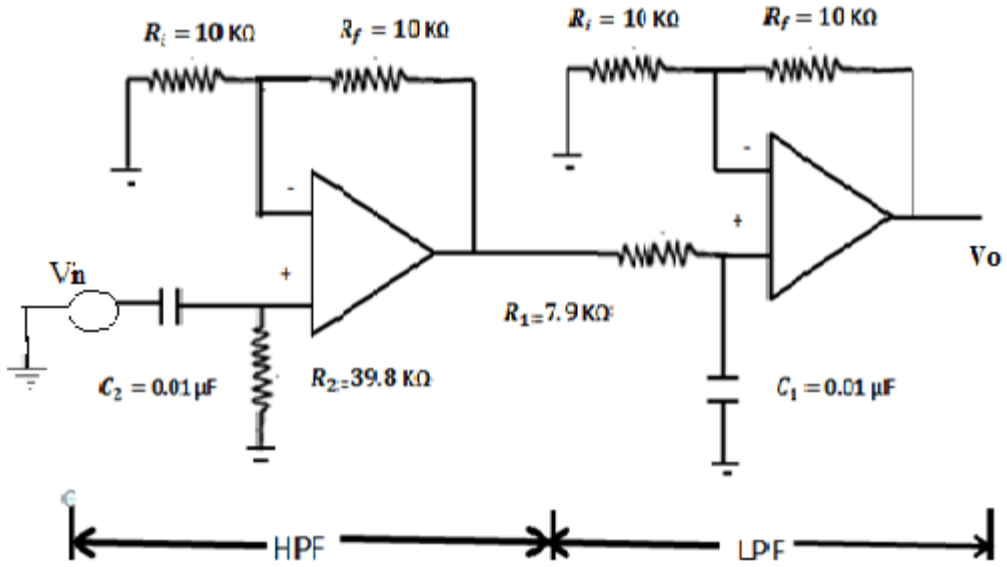
MODEL GRAPH:



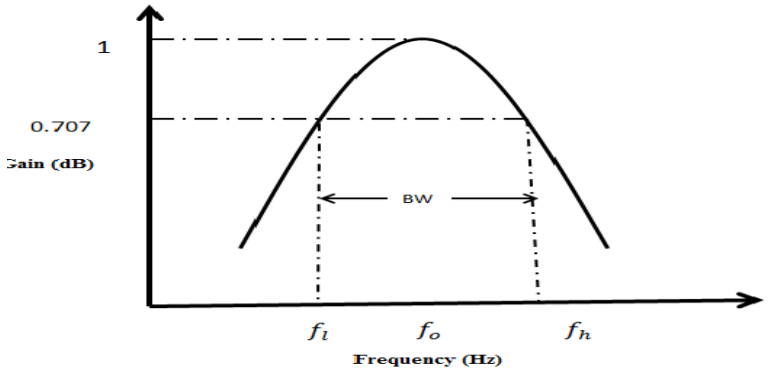
TABULAR COLUMN:

SI NO	FREQUENCY	INPUT VOLTAGE (V)	OUTPUT VOLTAGE (V)	GAIN = V_O/V_{IN}	GAIN= 20 log (V_O/V_{IN}) db

3) Band Pass Filter:



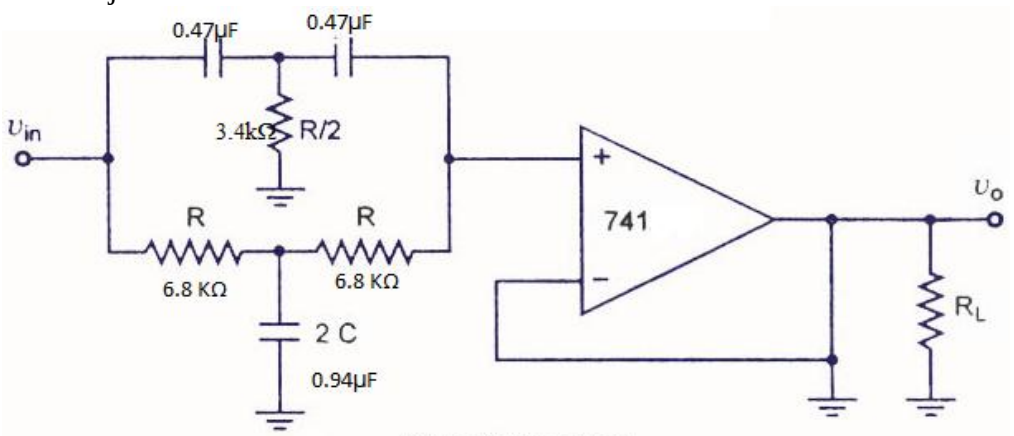
MODEL GRAPH:



TABULAR COLUMN:

SI NO	FREQUENCY	INPUT VOLTAGE (V)	OUTPUT VOLTAGE (V)	GAIN = V_O/V_{IN}	GAIN= 20 log (V_O/V_{IN}) db

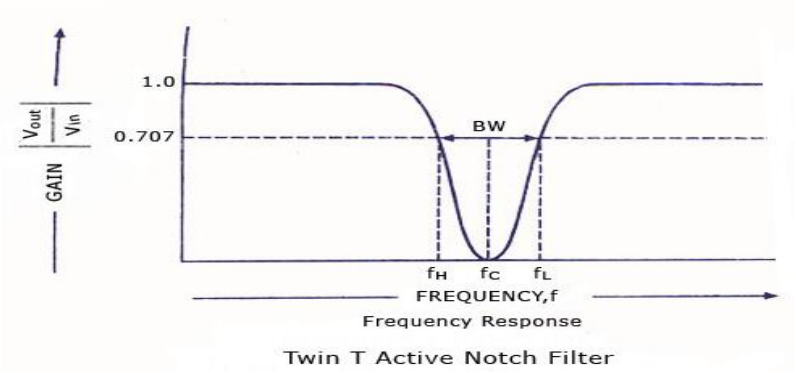
4) Band Reject Filter:



Circuit Diagram

Twin T active notch filter

MODEL GRAPH:



Frequency Response

Twin T Active Notch Filter

TABULAR COLUMN:

SI NO	FREQUENCY	INPUT VOLTAGE (V)	OUTPUT VOLTAGE (V)	GAIN = V_O/V_{IN}	GAIN= 20 log (V_O/V_{IN}) db

RESULT:

Thus the frequency response curve of second order low and high pass filter, band pass and band reject filters were plotted.

Post Lab Questions:

1. **Determine the type of filter and the corresponding cut-off rate.**

2. **What is the significance of 3 db line in frequency response?**

3. **What are the applications of filters?**

4. **What is state variable filter?**

5. **How do you classify active filters based on damping ratio?**

DEPT. OF ELECTRICAL & ELECTRONICS ENGINEERING
SRM UNIVERSITY, Kattankulathur – 603203.

Title of Experiment :	
Name of the candidate :	
Register Number :	
Date of Experiment :	
Date of submission :	

S.No:	Marks split up	Maximum Marks	Marks Obtained
1	Pre viva questions	05	
2	Preparation of circuit diagrams	10	
3	Execution of experiment	15	
4	Calculations and Graph	10	
	Total	40	

Staff Signature

9. IC Voltage Regulator

Pre Lab Questions:

1. **What is a voltage regulator?**

2. **What are the different types of voltage regulators?**

3. **What are switching regulators?**

4. **Compare linear and switching regulators.**

5. **What is thermal shut down of IC Regulator.**

9. VOLTAGE REGULATOR USING OP-AMP

AIM:

To design a high current, low voltage and high voltage linear variable dc regulated power supply and test its line and load regulation.

APPARATUS REQUIRED:

S.NO	APPARATUS	SPECIFICATION	QUANTITY
1.	Transistors	TIP122,2N3055	1 each
2.	Integrated Circuit	LM723	1
3.	Digital Ammeter	(0 – 10) A	1
4.	Digital Voltmeter	(0 – 20) V	1
5.	Variable Power Supply	(0 – 30) V-2A	1
6.	Resistors	300Ω ,430Ω ,1KΩ ,678KΩ ,67 8Ω 1Ω	1 each 2
7.	Capacitors	0.1µ F,100pF	1 each
8.	Rheostat	(0 – 350) Ω	1

THEORY:

A voltage regulator is an electrical regulator designed to automatically maintain a constant voltage level. A voltage regulator may be a simple "feed-forward" design or may include negative feedback control loops. It may use an electromechanical mechanism, or electronic components. Depending on the design, it may be used to regulate one or more AC or DC voltages. Load regulation is the change in output voltage for a given change in load current. Line regulation or input regulation is the degree to which output voltage changes with input (supply) voltage changes - as a ratio of output to input change. Active regulators employ at least one active (amplifying) component such as a transistor or operational amplifier. linear regulator is a voltage regulator based on an active device (such as a bipolar junction transistor, field effect transistor or vacuum tube) operating in its "linear region" .

PROCEDURE:

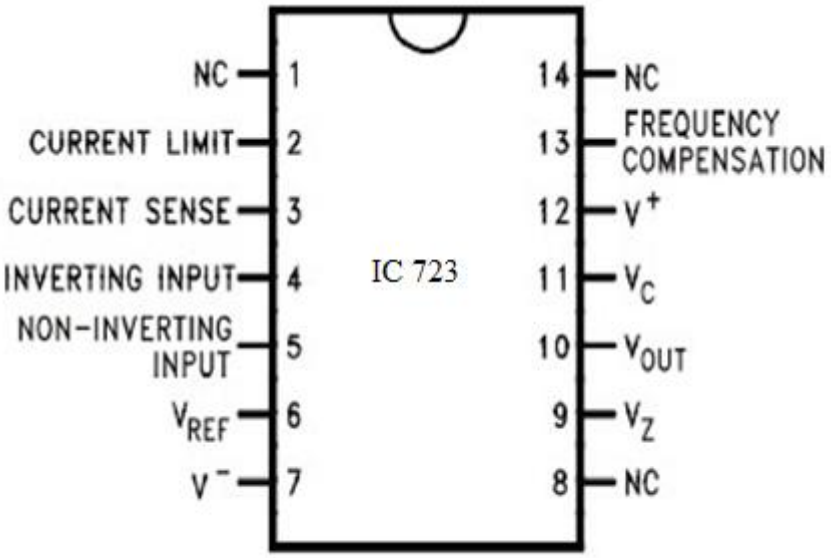
Line Regulation:

1. Give the circuit connection as per the circuit diagram
2. Set the load Resistance to give load current of 0.25A
3. Vary the input voltage from 7V to 18V and note down the corresponding output voltages
4. Similarly set the load current (I_L) to 0.5A & 0.9A and make two more sets of measurements.

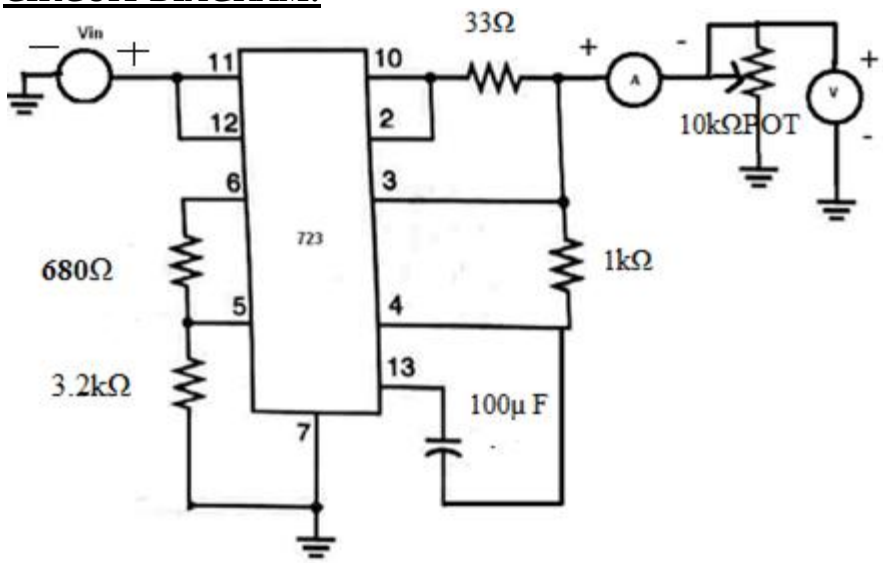
Load Regulation:

1. Set the input voltage to 10V.
2. Vary the load resistance in equal steps from 350Ω to 5Ω and note down the corresponding output voltage and load current.
3. Similarly set the input voltage (Vin) to 14V & 18V and make two more sets of measurements.

PIN DETAIL OF IC 723:



CIRCUIT DIAGRAM:



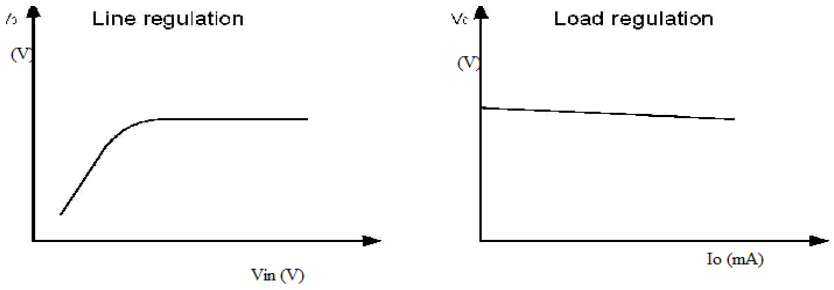
Line Regulation:

	I o =	
S.No	**I/P Voltage (V)**	**O/P Voltage (V)**

Load Regulation:

S.No:	I/P Voltage =	
	O/P Current (mA)	O/P Voltage (V)

MODEL GRAPH:



RESULT:

Thus the line and load regulation of low voltage linear variable dc regulated power supply was tested.

Post Lab Questions:

1. **What are the main advantages of voltage regulators?**

2. **Define line regulation or source regulation.**

3. **Define Load regulation.**

4. **What are the limitations of 723 regulators?**

5. **What is current limiting ability?**

DEPT. OF ELECTRICAL & ELECTRONICS ENGINEERING
SRM UNIVERSITY, Kattankulathur – 603203.

Title of Experiment :	
Name of the candidate :	
Register Number :	
Date of Experiment :	
Date of submission :	

S.No:	Marks split up	Maximum Marks	Marks Obtained
1	Pre viva questions	05	
2	Preparation of circuit diagrams	10	
3	Execution of experiment	15	
4	Calculations and Graph	10	
	Total	40	

Staff Signature

10. R -2R ladder DAC

Prelab Questions:

1. **Which error is the most common in DACs?**

2. **What are the different types of DAC?**

3. **What are the different types of ADC?**

4. **What is the resolution of a DAC?**

5. **A 4-bit R/2R digital-to-analog (DAC) converter has a reference of 5 volts. What is the analog output for the input code 0101?**

10.R-2R LADDER DAC

AIM:

To study the operation of R-2R ladder DAC.

APPARATUS REQUIRED:

S.NO	APPARATUS	RANGE	QUANTITY
1)	IC	µA741	1
2)	Resistor	1KΩ, 2KΩ	1
3)	Multimeter	-	1
4)	RPS	DUAL(0-30) V	1
5)	Connecting Wires		

THEORY:

The R-2R ladder type DAC uses resistor of only two values R and2R.The inputs to resistor network may be applied through digitally connected switches or from output pins of a counter. The analogue output will be maximum, when all inputs are of logic high.

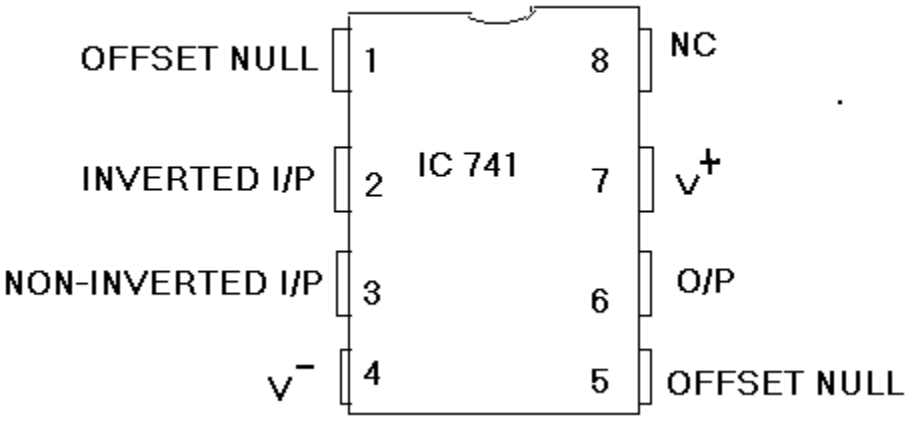
$$V=-R_f/R\ (1/2\ D_3+1/4D_2+1/8D_1+1/16D_0)$$

In a 3 input ADC, if the analog signal exceeds the reference signal, comparator turns on. If all comparators are off, analog input will be between 0 and V/4.If C1 is high and C2 is low input will be between V/4 andV/2.If C1 andC2 are high and C3 is low input will be between 3V/4 and V.

PROCEDURE:

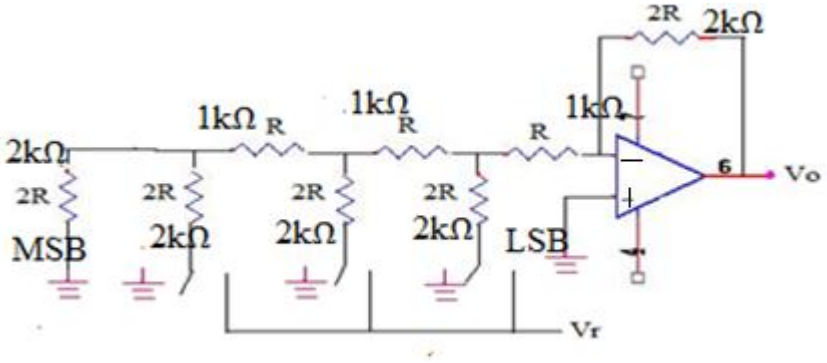
1. Connect the circuit as shown in circuit diagram.
2. For various inputs, measure the outputs using multimeter.

PIN DIAGRAM:



CIRCUIT DIAGRAM:

R-2R Ladder DAC:



TABULAR COLUMN:

S.No.	D2	D1	D0	Vth (V)	Vprac (V)
1)	0	0	0		
2)	0	0	1		
3)	0	1	0		
4)	0	1	1		
5)	1	0	0		
6)	1	0	1		
7)	1	1	0		
8)	1	1	1		

RESULT:

The operation of R-2R ladder was studied and the output voltages were verified.

Post Lab Questions:

1. **The basic step of 9 bit DAC is 10.3mV. If 000000000 represents 0V. What output is produced if the input is 101101111?**

2. **State the applications of DAC and ADC .**

3. **The difference between analog voltage represented by two adjacent digital codes, or the analog step size, is ……..**

4. **Mention the merits of R -2R ladder DAC .**

5. **For R-2R ladder 4 bit type DAC find the output voltage if digital input is 1111. Assume VR = 10V, R = Rf = 10K.**

DEPT. OF ELECTRICAL & ELECTRONICS ENGINEERING
SRM UNIVERSITY, Kattankulathur – 603203.

Title of Experiment	:
Name of the candidate	:
Register Number	:
Date of Experiment	:
Date of submission	:

S.No:	Marks split up	Maximum Marks	Marks Obtained
1	Pre viva questions	05	
2	Preparation of circuit diagrams	10	
3	Execution of experiment	15	
4	Calculations and Graph	10	
	Total	40	

Staff Signature

12.Simulation of op-amp circuits using PSPICE

Prelab Questions:

1. **List out some commonly used EDA tools.**

2. **Give some linear applications of op-amp.**

3. **Give some nonlinear applications of op-amp.**

4. **What is window detector?**

5. **Draw a sample and hold circuit.**

12. SIMULATION OF OP-AMP CIRCUITS USING PSPICE

AIM :

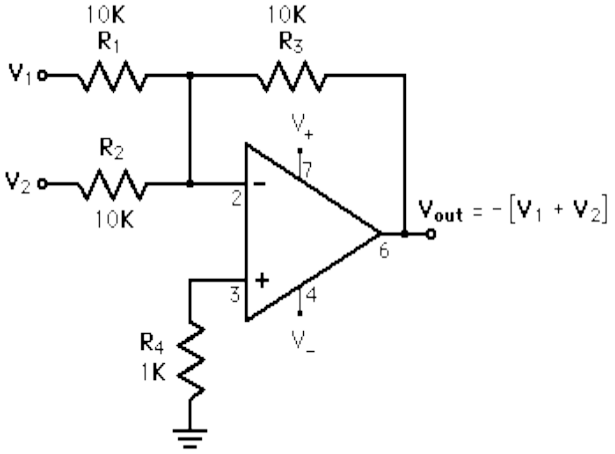
To simulate op-amp circuits like summing amplifier, subtractor and zero crossing detector, using PSPICE.

SOFTWARE REQUIRED:

PSPICE

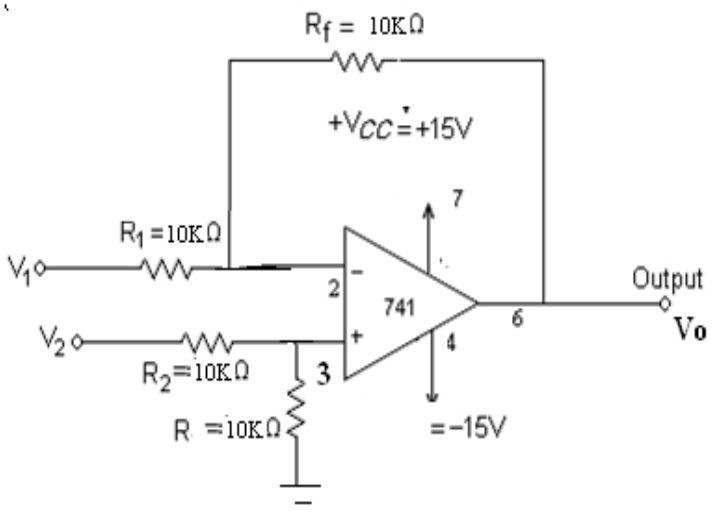
CIRCUIT DIAGRAM:

1) Summing Amplifier:



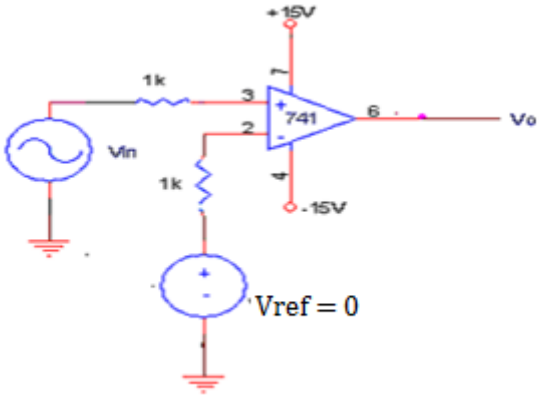
S.No.	V1	V2	Theoretical V0= - (V1+V2)	Practical V0

2) Subtractor:

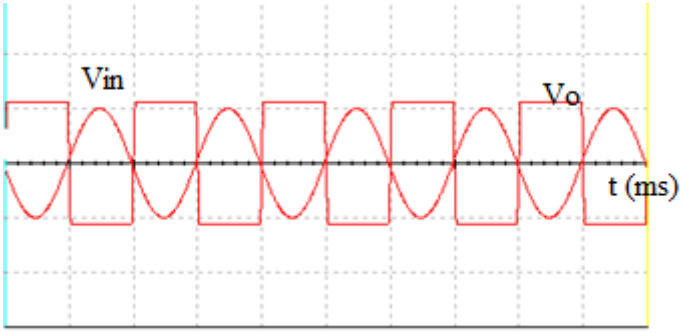


S.No.	V1	V2	Theoretical V0=V1-V2	Practical V0

1) Zero Crossing Detector:



MODEL GRAPH:



TABULAR COLUMN:

SI. NO	INPUT		OUTPUT	
	VOLTAGE (V)	TIME (ms)	VOLTAGE (V)	TIME (ms)

RESULT:

Thus the summing amplifier, subtractor and zero crossing detector circuits were simulated using PSPICE and results verified with hardware setup.

Postlab Questions:

1. **What is a voltage follower?**

2. **What do you mean by inverting precision rectifier?**

3. **Draw the circuit for time marker generator.**

4. **List out some merits of PSPICE and MATLAB simulation tools.**

5. **What is a Schmitt trigger?**