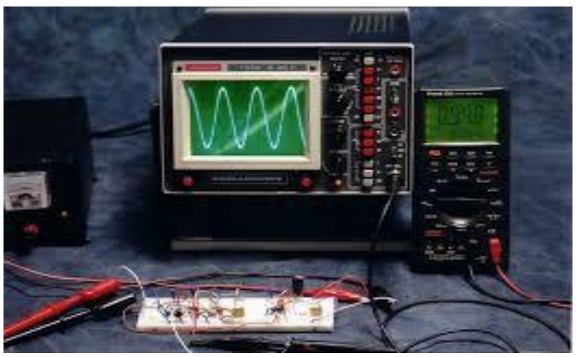
LINEAR IC'S &PULSES CIRCUITS LABORATORY MANUAL (EEE-318)
(III/IV EEE Ist Semester)



DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING
ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES
(Affiliated to AU, Approved by AICTE & Accredited by NBA)
SANGIVALASA-531 162, Bheemunipatnam Mandal, Visakhapatnam District
Phone: 08933-225083/84/87 Fax : 226395

Linear IC's and Pulse Circuit Laboratory
III/IV EEE Ist semester
(CODE - EEE 318)

DEPARTMENT OF
ELECTRONICS & COMMUNICATION ENGINEERING

LABORATORY MANUAL

Prepared By:

Mrs.P.S.M.Veena
Assistant Professor,
Department of ECE.

Approved By:

Dr.V.Rajya Lakshmi,
Professor & HOD,
Department of ECE,ANITS

ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES
(Affiliated to Andhra University)
Sangivalasa-531162, Bheemunipatnam Mandal, Visakhapatnam.
Phone:089335084,226395

INTRODUCTION

In pulse circuits lab students will be able to analyze and design different linear and non-linear waveforms with different time constants and different types of inputs, with and without reference voltages using linear and non-linear wave shaping circuits. Design, analysis and voltage regulators circuits will be done.

In Integrated circuits lab Design and analysis of linear and non-linear circuits using operational amplifiers, 1st & 3rd order active filters, voltage regulators, multivibrators using timers, Schmitt trigger circuits will be done. Identification, verification and applications of ICs like LM741, 555 timer and three terminal regulators (7805, 7808 etc) will be taught. With this knowledge students will be able to do the mini-projects with the help of integrated circuits.

LINEAR INTEGRATED CIRCUITS & PULSE AND DIGITAL CIRCUITS LABORATORY	
EEE 318	**Credits : 2**
Instruction : 3 Periods / Week	**Sessional Marks : 50**
End Exam : 3 Hours	**End Exam Marks : 50**

Prerequisites:

1. Pulse and Digital Circuits (EEE 313)
2. Linear IC's and Applications (EEE 314)

Course Objectives:

At the end of the course students should understand:

- To understand the linear and non-linear applications of operational amplifiers(741)
- To familiarize with theory and applications of 555 timers.
- To design and construct waveform generation circuits using Op-Amp
- Understand the response of linear circuits for different signals.
- Determine the voltage transfer characteristics of non linear circuits and also learn about comparators
  - ~~Understand different energy resources~~
  - ~~Electrical energy generation by using different types of plants.~~
  - ~~Utilization of generated electrical energy for various purposes.~~
  - ~~Electric heating, welding, illumination are known~~

Course Outcomes:

At the end of the course student should be able to:	
1.	Design the circuits using op-amps for various applications like adder, subtractor, integrator, differentiator and Schmitt trigger
2.	Design active filters for the given specifications and obtain their frequency response characteristics.
3.	Design and analyze multi vibrator circuits using op-amp, Transistor and 555Timer
4.	Design application based on linear and nonlinear circuits
5.	Understand the operation & application of Bootstrap circuit

Mapping of course outcomes with program outcomes:

		PO												PSO	
		1	2	3	4	5	6	7	8	9	10	11	12	1	2
CO	1	1	1	3	3	1	1	0	0	2	1	0	0	1	0
	2	1	1	3	1	1	1	0	0	2	1	0	0	1	0
	3	1	1	3	2	1	1	0	0	2	1	0	0	1	0
	4	1	1	3	3	1	1	0	0	2	1	0	0	1	0
	5	1	1	3	1	1	1	0	0	2	1	0	0	1	0

Formatted: Not Highlight

Formatted: Indent: Hanging: 0.55"

Formatted: Font: Not Bold, Not Highlight

Formatted: Font: Not Bold, Not Highlight

Formatted: List Paragraph, Indent: Left: 0.94", Hanging: 0.19", Don't add space between paragraphs of the same style, Line spacing: single, Outline numbered + Level: 4 + Numbering Style: 1, 2, 3, … + Start at: 1 + Alignment: Left + Aligned at: 2.5" + Tab after: 3" + Indent at: 3", Tab stops: Not at 3"

Formatted: Font: Not Bold, Not Highlight

Formatted: Not Highlight

Formatted: Indent: Hanging: 0.49"

Formatted: Indent: Left: 0.31", Hanging: 0.19"

Formatted: Not Highlight

Formatted: Font: Bold, Not Highlight

Formatted: Not Highlight

Formatted: Not Highlight

~~TEN EXPERIMENTS BASED ON EEE 313 & EEE 314 SYLLABI~~

Formatted: Normal, No bullets or numbering

Linear IC's and Pulse Circuit Laboratory

III/IV EEE Ist semester

(CODE - EEE 318)

List of Experiments

EXPERIMENT NO: 1
APPLICATIONS OF OPERATIONAL AMPLIFIER

AIM : To realize Summing Amplifier, Subtracting Amplifier, Integrator and Differentiator by using 741 Op-Amp.

APPARATUS:

1. Op-Amp LM 741
2. Resistors – 1KΩ (4), 100KΩ (1), 10KΩ (1)
3. Capacitors – 0.01µf(1), 330pf(1)
4. Function Generator
5. TRPS
6. CRO & CRO Probes
7. Bread Board
8. Connecting Wires.

THE IDEAL OP AMP:

An ideal op amp would exhibit the following electrical characteristics.

1. Infinite voltage gain A.
2. Infinite input resistance R_i so that almost any signal source can drive it and there is no loading of the preceding stage.
3. Zero output resistance R_0 so that output can drive an infinite number of other devices.
4. Zero output voltage when input voltage is zero.
5. Infinite bandwidth so that any frequency signal from 0 to ∞ Hz can be amplified without attenuation.
6. Infinite common mode rejection ratio so that output common – mode noise voltage is zero.
7. Infinite slew rate so that output voltage changes occur simultaneously with input voltage changes.

APPLICATIONS OF OP AMP:

1. Summing Amplifier:

Op amp may be used to design a circuit whose output is the sum of several input signals. Such a circuit is called a summing amplifier or a summer. If V1, V_2 are two input signals given to the inverting terminal, then

$$\boxed{V_o = -\frac{R_F}{R}(V_1 + V_2)}$$

2. **Subtracting Amplifier**:

The function of a subtractor is to provide an output, which is equal to the difference of two input signals (or) proportional to the difference of two input signals. If V_1 and V_2 are the input voltages at inverting and non – inverting terminals, then

$$\boldsymbol{V_o = \frac{-R_F}{R}(V_1 - V_2)}$$

3. Integrator:

A circuit in which the output voltage waveform is the integral of the input voltage waveform is the integrator or the integration amplifier. Such a circuit is obtained by using a basic inverting amplifier configuration with the feedback resistor R_F replaced by a capacitor C_F. The output voltage is given by

$$\boldsymbol{V_o = -\frac{1}{RC}\int V_1\,dt}$$

Integrator is used in signal wave shaping circuits and in analog computers. If the input is a sine wave, the output is a cosine wave. If the input is a square wave, the output will be a triangular wave. In the practical integrator, R_F is connected across feedback capacitors C_F. This R_F limits the low frequency gain and minimizes the variation in the output voltage. The input signal will be integrated properly if the time constant

$T = R_1 C_F$ is larger than the time period T of the input signal

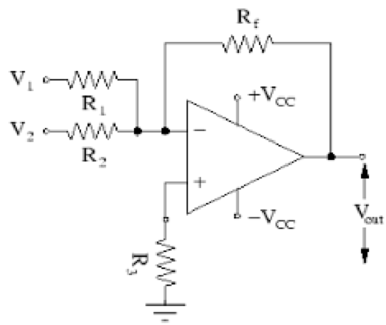
4. Differentiator:

The function of a differentiator is to give an output voltage, which is proportional to the rate of change of input voltage. The differentiator may be constructed from a basic inverting amplifier if an input resistor is replaced by capacitor C_1. The output voltage is given by

$$V_o = -RC\,(dV_i / dt)$$

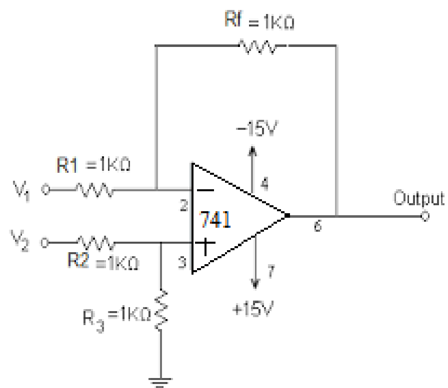
The condition for differentiator is τ << T where τ = $R_1 C_F$ for sine wave and square wave inputs, the resulting differentiated outputs are cosine wave and spike outputs respectively. Differentiator is used to detect high frequency components in an input signal.

CIRCUIT DIAGRAM: (i) Summing Amplifier



R1=R2=R3=Rf =1KΩ

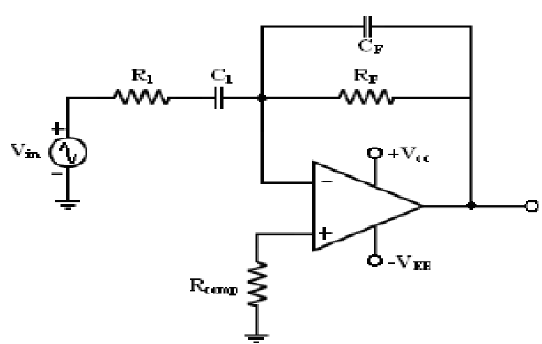
(ii) Subtracting Amplifier



R1=R2=R3=R4=1KΩ

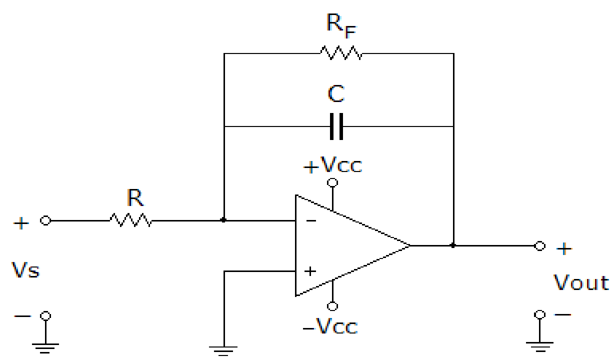
(iii) Differentiator:

R1=1KΩ, R2=100KΩ ; C1=0.01µf, Cf=330pf ;Vin = 0.4v peak to peak at 1kHz



(iv)Integrator:

R=10KΩ, Rf=100KΩ, C=0.01µf, Vs = 4v peak to peak at 10kHz



PROCEDURE:

I. Summing Amplifier:

1. Connections are made as per the circuit diagram.
2. Input voltages V_1 and V_2 are given and the corresponding output voltage V_o is measured from CRO.
3. Output varies as $V_o = -(V_1 + V_2)$, since $R_F = R$.

II. Subtracting Amplifier:

1. Connections are made as per the circuit diagram.
2. Input voltage V_1 and V_2 are given to the inverting and non – inverting terminals respectively and corresponding output voltage is measured from CRO.
3. Output varies as $V_o = V_2 – V_1$.

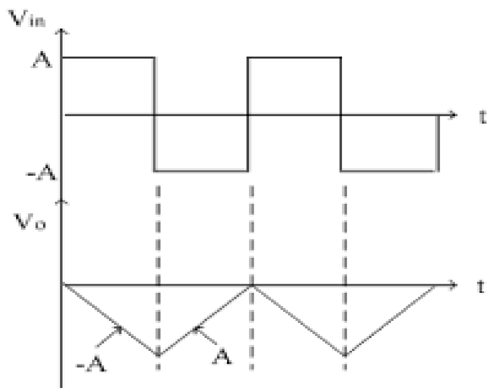
III Differentiator:

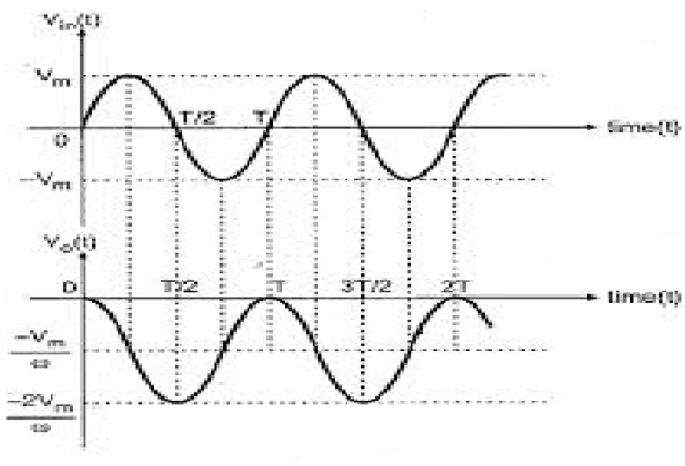
1. Connections are made as per the circuit diagram.
2. A square wave input of 4V (p-p) and frequency of 1KHZ is applied from function generator.
3. Output waveform is observed. Corresponding amplitude and time period is observed and frequency is calculated.
4. With the above data plot the output graphs with time on X-axis and voltage on Y-axis.

IV. Integrator:

1. Connections are made as per the circuit diagram.
2. By using a function generator, a square wave input 4Vp-p is given.
3. The frequency applied is 10 KHz.
4. A perfect triangular wave is obtained. The peak-to-peak voltage and the time period of input and output waves are measured from CRO.
5. The waveforms are plotted.

MODEL GRAPHS:
INTEGRATOR:





DIFFERENTIATOR:

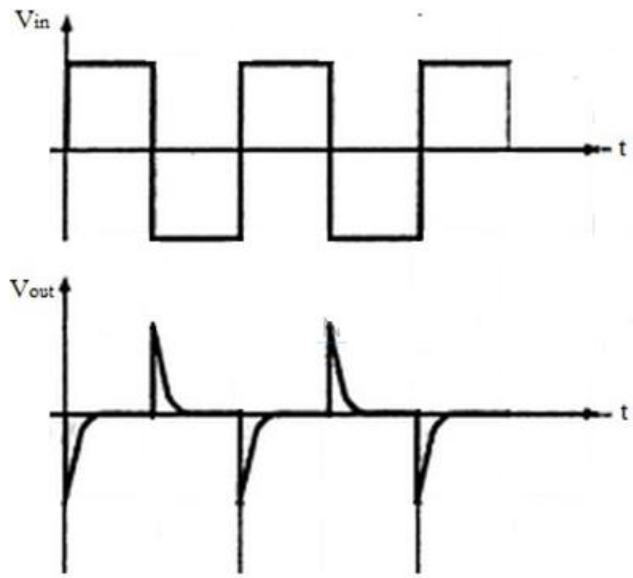
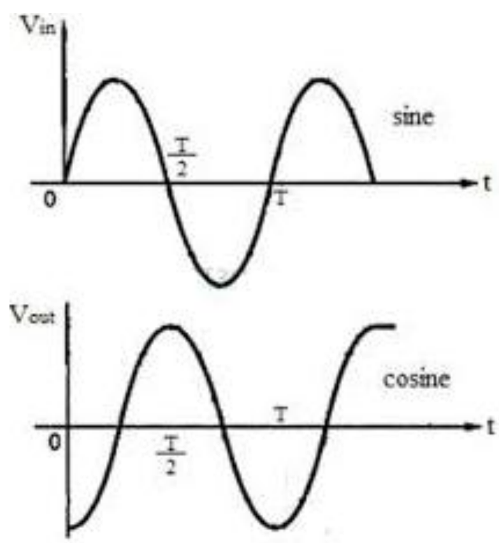


Fig: Input and Output waveforms for Square wave



PRECAUTIONS:

1. Loose and wrong connections are to be avoided.
2. Waveforms should be obtained without any distortion.

Conclusion:

RESULT:

Viva questions

1. What are the ideal characteristics of an OP-AMP?
2. Define OP-AMP.
3. What do you mean by CMRR?
4. Define slew rate.
5. What are the applications of differentiator?
6. What are the applications of integrator?
7. What is a difference between inverting and non-inverting amplifier?

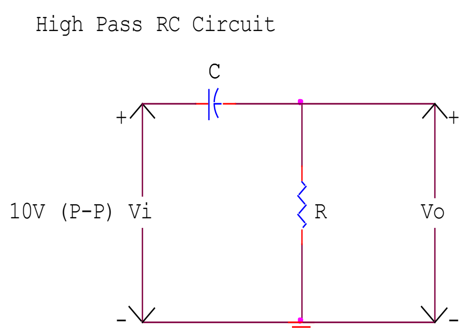
EXPERIMENT NO: 2
LINEAR WAVE SHAPPING

AIM: To observe the process of linear wave shaping for square wave input for high pass RC circuit and low pass RC circuit.

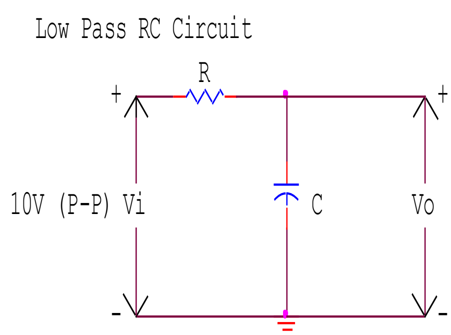
APPARATUS:

1. Resistors – 1KΩ (1) & 10KΩ (1)
2. Capacitors – 1µf(1) & 0.01µf(1)
3. Function Generator
4. TRPS
5. CRO & CRO probes
6. Bread Board and connecting wires.

CIRCUIT DIAGRAM:



R=10KΩ, C=1µf



R=1KΩ, C=0.01µf

High pass RC Circuit:

The reactance of a capacitor decreases with increasing frequency; the higher frequency components in input signal appear at the output with less attenuation than do the lower frequency components. At very high frequency the capacitor acts almost as a short circuit and virtually all the input appears at the output. This is behavior accounts for the designation 'High Pass Filter '.

Square wave responses of a high pass RC circuit.

The dashed curve represents the output if RC >> T.

$V_1^1 = V1 \exp(-T_1/ RC)$ $\qquad V_1^1 - V_2 = V$

$V_2^1 = V2 \exp(-T_2/ RC)$ $\qquad V_1 - V_2^1 = V$

A symmetrical square wave is one for which $T_1 = T_2 = T/2$. Because of symmetry $V_1 = -V_2$ and $V_1^1 = -V_2^1$

$$V_1 = V / (1+\exp^{-T/2RC})$$
$$V_1^1 = V / (1+\exp^{T/2RC})$$

Peaking of square wave resulting from a time constant small compared with T.

The high pass RC circuit acts as a differentiator if time constant is very small in comparisons with the time required for the input signal to make an appreciable change.

Low Pass R-C Circuit:

The low pass RC circuit passes low frequencies readily but attenuates high frequencies because the reactance of capacitor decreases with increasing frequency. At high frequencies, the capacitor acts as a virtual short circuit and output falls to zero.

Square wave input (b-d) , output of low pass RC circuit. The time constant is smallest for (b) and largest for (d).

Equation of rising portion

$$V_{01} = V^1 + (V_1 - V^1) \exp^{(-t/RC)}$$

V_1 = initial value of output voltage .

Equation of falling portion

$$V_{02} = V^{11} + (V_2 - V^{11}) \exp^{-(t-T1)/RC}$$

The low pass RC circuit acts as an integrator if time constant is very large in comparison with time required for the input signal to make an appreciable change.

PROCEDURE:

1. Connections are made as per the circuit diagram.
2. To the high pass circuit a square wave input of amplitude 10V (p-p) is given.
3. The time period of waveform is adjusted such that RC << T, RC = T and RC >> T to get spikes and tilted output respectively. The time period and amplitude are noted.
4. Now to the low pass circuit a square wave input of amplitude 10V (p-p) is given.
5. The time period of input signal is adjusted with the help of a function generator such that RC << T and RC >> T to get the corresponding waveforms. The time period and amplitude are noted.
6. Graphs are plotted for both input and output waveforms of both the circuits when RC << T and RC >> T.

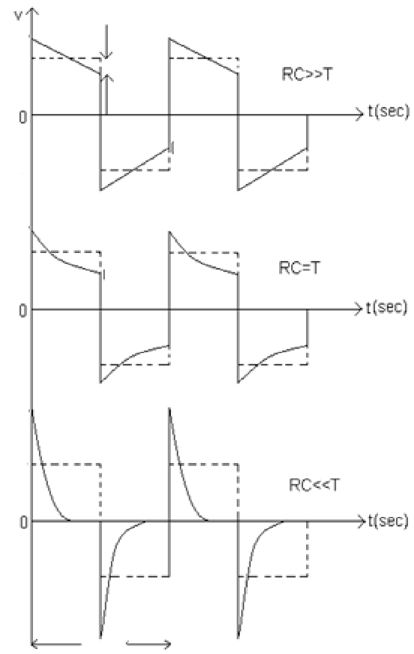
Observation Table:

	High Pass RC Circuit									
	$RC<<T$		$RC==T$				$RC>>T$			
	V_1	V_2	V_1	V_2	V'_1	V'_2	V_1	V_2	V'_1	V'_2
Theoretical Calculations										
Practical Calculations										

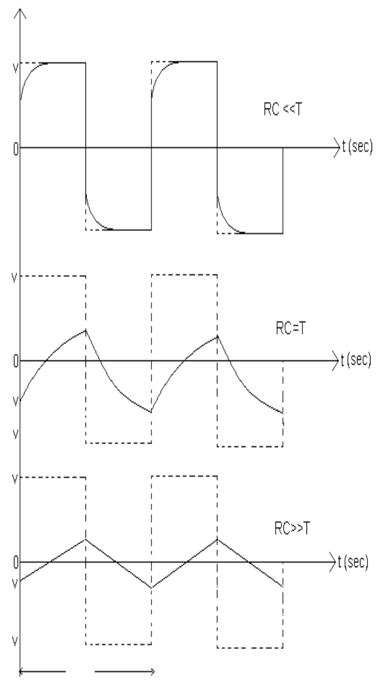
	Low Pass RC Circuit					
	$RC<<T$		$RC==T$		$RC>>T$	
	V_1	V_2	V_1	V_2	V_1	V_2
Theoretical Calculations						
Practical Calculations						

MODEL GRAPHS:

High pass circuit



Low Pass Circuit



PRECAUTIONS:

1. Loose and wrong connections are to be avoided.
2. The output waveforms should be obtained without and distortion
3. Parallax error should be avoided.

Conclusion:

RESULT:

Viva questions

1. What is linear wave shaping?
2. How low pass RC circuit works as an integrator?
3. How low pass RC circuit works as differentiator?
4. Define time constant.
5. Define tilt.
6. Explain the output wave forms of high pass and low pass circuit for different conditions.

<u>**EXPERIMENT NO:3**</u>

CLIPPER CIRCUITS

AIM:

To observe the waveforms of clipper circuits using

a. Positive clipper
b. Negative clipper
c. Two level clipper or slicer circuit.

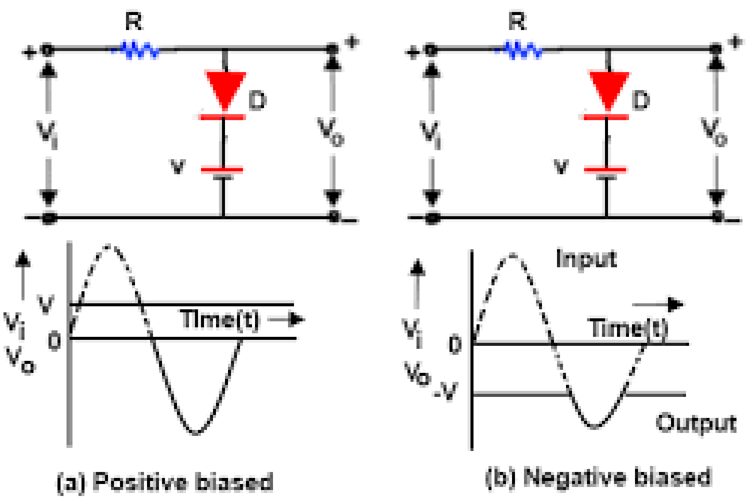
APPARATUS:

1. 1N 4007 diodes (2)
2. Resistor-10K (1)
3. TRPS
4. Function Generator
5. Bread board and connecting wires
6. CRO with CRO probes.

CIRCUIT DIAGRAM:

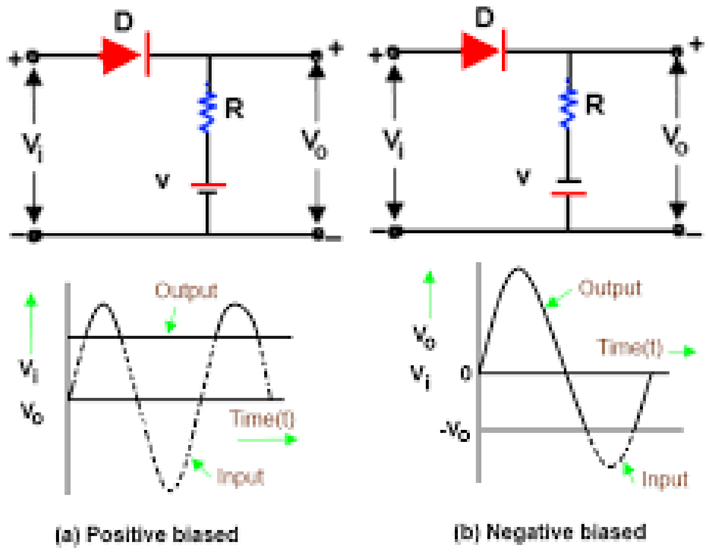
Shunt Clippers Negative clipper:

R =10KΩ



Biased Shunt Positive Clipper

Series Clippers : R1=10KΩ



Biased series negative clipper

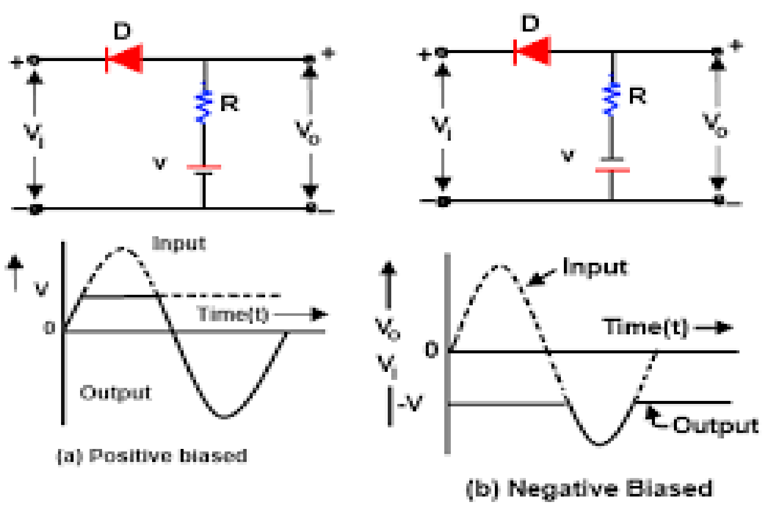
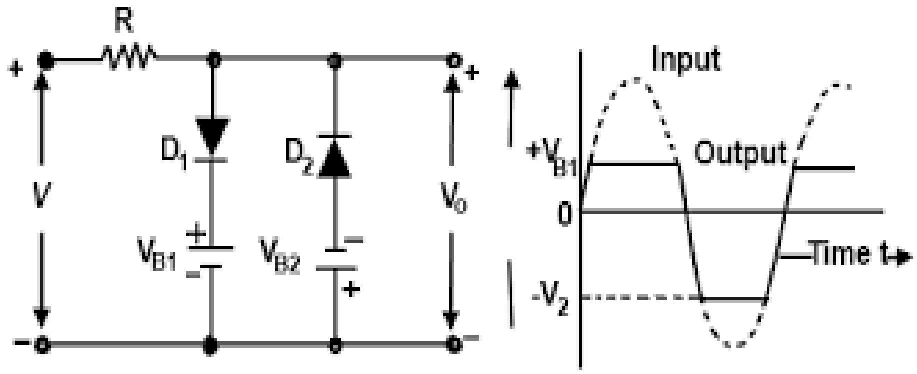


Figure 2: Series positive clipper with bias

Two level clipper: R1=10KΩ



Dual (Combination) Diode Clipper

CIRCUIT OPERATION:

Clippers are used to select a part of signal waveform above or below a reference voltage for transmission.

Negative Clipper:

For Vi < V_R +V_r , The diode D is OFF ,since it is reverse biased and hence does not contact. Since no current flows, there is no voltage drop across R.

$V_O = V_i$ for $V_i < V_R + V_r$

Where V_r is Cut-in voltage of the diode.

For $V_i > V_R + V_r$, the diode D is ON, Since it is forward biased and the potential barrier is overcome

$$V_o = V_R + V_r$$

Transfer characteristic Equation:

$V_o= V_i$ for $V_i < V_R + V_r$

$V_o= V_R + V_r$ for $V_i > V_R + V_r$

Positive Clipper:

When $V_i > V_R + V_r$ the diode is forward biased and hence it conducts since it is ON it is short circuited .It is obvious that $V_o= V_R + V_r$ Whatever the comment.

When $V_i < V_R + V_r$ the diode is reverse biased and hence it is OFF. It acts as an open Circuit. $V_o = V_i$

Transfer Characteristic Equation:

$$V_o = V_i \text{ for } V_i < V_R + V_r$$
$$V_o = V_R + V_r \text{ for } V_i > V_R + V_r$$

Procedure:

1. Connections are made as per the circuit diagram
2. For the positive clipper the diode is connected along with reference voltage as shown by applying the input and the output is observed on the C.R.O.
3. For the negative clipper the directions of diode and the reference voltage are reversed and by giving the input, the output is observed on the C.R.O.
4. For the Slicer Circuit has two Diodes along with reference voltages are connected as shown and output is observed on the C.R.O.
5. A sinusoidal input 10V (p-p) 1KHZ is given to positive clipper, negative clipper and slicer circuit and corresponding output is observed.

OBSERVATIONS:

<table>
<tr><td>Name Of the Clipper</td><td colspan="2">Negative Clipper O/P</td><td>Output waveform</td></tr>
<tr><td>Wave Form</td><td>Positive peak</td><td>Negative peak</td><td rowspan="3"></td></tr>
<tr><td>Amplitude (p-p)</td><td></td><td></td></tr>
<tr><td>Time Period</td><td></td><td></td></tr>
<tr><td>Name Of the Clipper</td><td colspan="2">Positive Clipper O/P</td><td>Output waveform</td></tr>
<tr><td>Wave Form</td><td>Positive peak</td><td>Negative peak</td><td rowspan="3"></td></tr>
<tr><td>Amplitude (p-p)</td><td></td><td></td></tr>
<tr><td>Time Period</td><td></td><td></td></tr>
</table>

Name Of the Clipper	2-Level Clipper O/P		Output waveform
Wave Form	Positive peak	Negative peak	
Amplitude (p-p)			
Time Period			

PRECAUTIONS:

1. Loose and wrong connections are to be avoided.
2. The output waveforms should be obtained without distortion.
3. Parallax error should be avoided

Conclusion:

RESULT:

Viva questions.

1. What is meant by non linear wave shaping?
2. What is clipper? What are the different types of clippers?
3. What are the different applications of clipper?
4. What is two level clipper
5. Explain the operation of positive and negative clipper?

EXPERIMENT NO:4

CLAMPER CIRCUITS

AIM:

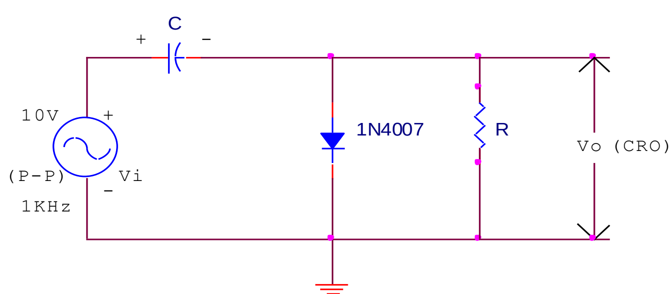
To observe the waveforms of the Positive and Negative clamping circuits.

APPARATUS:

1) Capacitor 4.7µf - 1
2) IN4007 diode - 1
3) Resistor 1M Ω - 1
4) Function Generator
5) Bread board
6) CRO and CRO probes
7) Connecting wires.

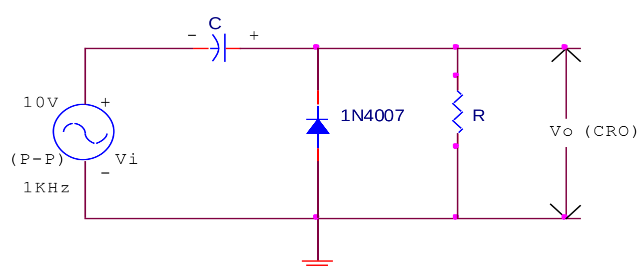
Circuit Diagrams:

Negative Clamper



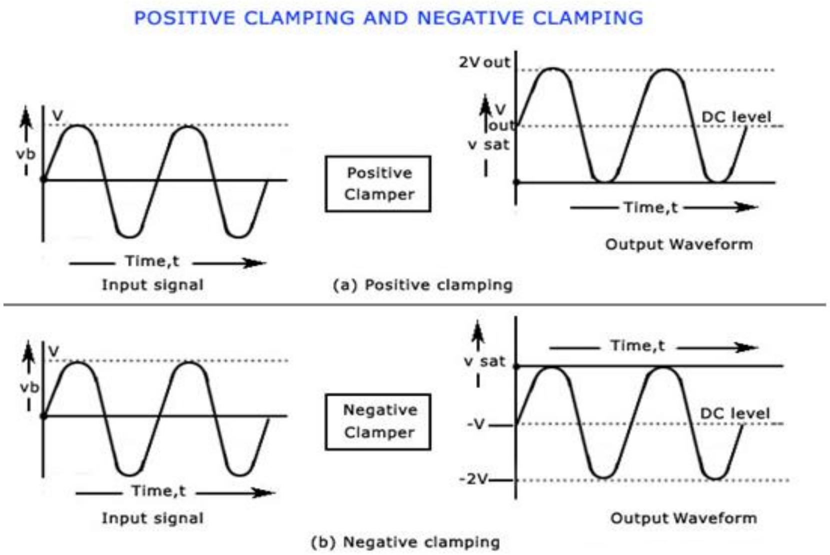
C=4.7µf , R=1M Ω

Positive Clamper



C=4.7µf, R=1M Ω

Model Graph:



PROCEDURE:

1. The circuits are connected as per the circuit diagram.
2. The input signal Vi of (10V p-p) frequency (1KHz) is applied to each of the circuits.
3. The corresponding output waveforms are noted from the C.R.O.
4. The input and output waveform are plotted on the graph sheets.

OBSERVATIONS:

Name Of the Clamper	Negative Clamper O/P		Output waveform
Wave Form	Positive peak	Negative peak	
Amplitude (p-p)			
Time Period			

Name Of the Clamper	Positive Clamper O/P		Output waveform
Wave Form	Positive peak	Negative peak	
Amplitude (p-p)			
Time Period			

PRECAUTIONS:

1. Loose and wrong connections are to be avoided.
2. The output waveforms should be obtained without and distortion.
3. Parallax error should be avoided

Conclusion:

RESULT:

Viva questions

1. What do mean by clamper?
2. What are the different types of clamping circuits?
3. What are the different applications of clampers?
4. Why clamper is called DC inserter?
5. Explain the operation of positive clamper and negative clamper?

EXPERIMENT NO: 5
SCHMITT TRIGGER

AIM:

To observe the output waveform of a Schmitt trigger circuit and to note down the hysteresis voltage V_{HY} with the reference of V_{UT} and V_{LT} .

Apparatus: 1) IC 741 OP-AMP -1NO
2) Resistors 2.2KΩ -2, 10kΩ -1
3) Function Generator
4) TRPS
5) CRO, Bread Board and connecting wires.

Theory:

The circuit shown is known as the Schmitt trigger or Squaring Circuit. It shows an working comparator with positive feedback. This circuit converts an irregular shaped waveform to a square wave hence it is called as a square wave generator.

If positive feedback is added to a basic comparator circuit, Gain can be increased greatly.

The input voltage V_{in} triggers the output V_o every time it exceeds certain voltage levels called upper threshold voltage V_{UT} and lower threshold voltage V_{LT}

The threshold voltages are obtained by using the voltage divider R_1 - R2 where the voltage across R_1 is fed back to the (+) input. The voltage across R_1 is variable reference threshold voltage that depends on the value the polarity of the output voltage V_0 . When V_0 = + V_{SAT} the voltage across R_1 is called the upper threshold Voltage V_{UT}

The input voltage V_{IN} must be slightly more positive than V_{UT} in order to cause the output V_o to switch from $+V_{SAT}$ to $-V_{SAT}$ as long as $V_{IN} < V_{UT}$, V_O is at $+V_{SAT}$

$$V_{UT} = \frac{R_1}{R_1+R_2}(+V_{SAT})$$

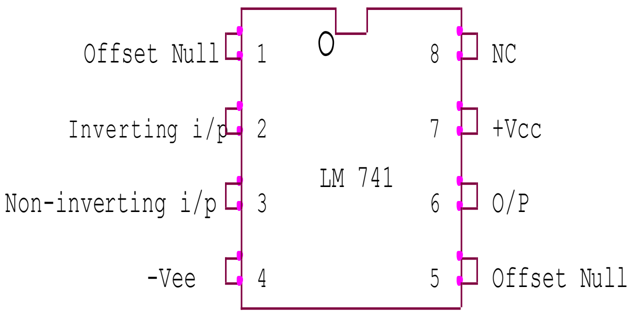
$$V_{LT} = \frac{R_1}{R_1+R_2}(-V_{SAT})$$

The hysteresis voltage is equal to difference between V_{UT} and V_{LT}

$$V_{HY} = V_{UT} - V_{LT}$$

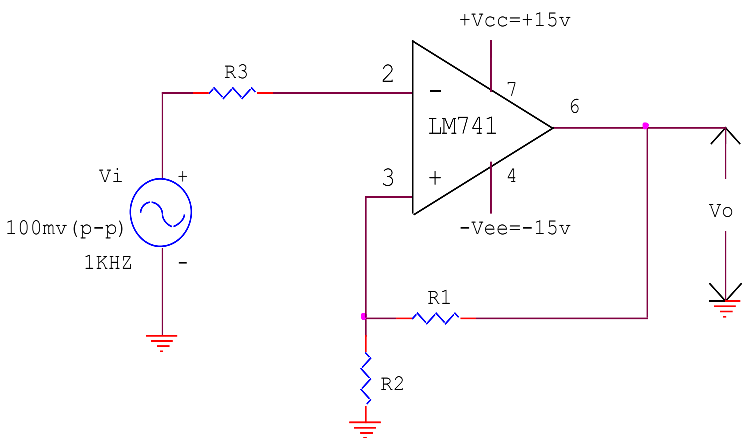
$$V_{HY} = \frac{R_1}{R_1+R_2}(+V_{SAT}) - \frac{R_1}{R_1+R_2}(-V_{SAT})$$

Pin Diagram:



Circuit Diagram:

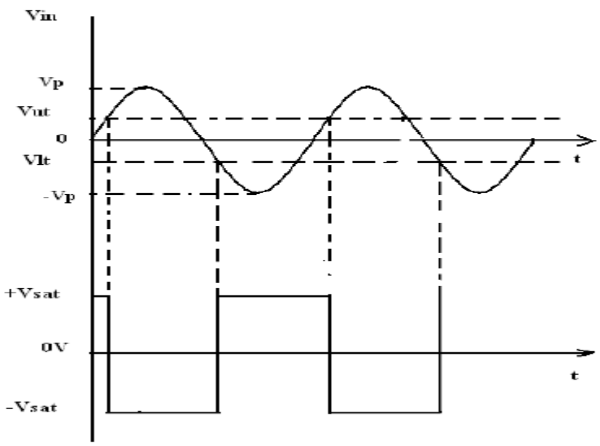
Schmitt Trigger



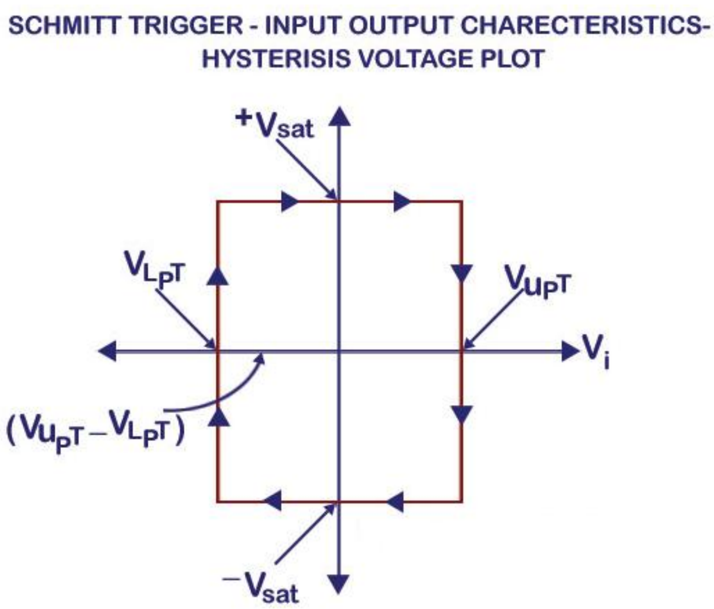
R1=10KΩ,R2=R3=2.2KΩ

MODEL GRAPHS:

Input and output waveform of Schmitt trigger:



V_0 versus V_{in} plot of hysteresis voltage:



PROCEDURE:

1. The circuit for Schmitt trigger is connected as per the given circuit diagram.
2. A sinusoidal input of 1 KHz is applied with the help of function generator.
3. A square wave output is obtained for the corresponding input for which the positive peak voltage (+Vsat) and negative peak voltage (-Vsat) are noted.
4. The upper threshold voltage (V_{UT}) and lower threshold voltage (V_{LT}) are calculated for the corresponding output.
5. The shift angle (θ) is calculated using the formula

$$V_{UT} = V_p \sin\theta$$

$$\sin\theta = V_{UT}/V_p$$

$$\theta = \sin^{-1}(V_{UT}/V_p)$$

6. The hysteresis voltage (V_H) is calculated using the formula

$$V_{HY} = V_{UT} - V_{LT}$$

Observations:

Input applied: V_i (p-p mV) =

F=

T=

Output obtained: +Vsat =

-Vsat =

T=

F=

Calculations:

Upper threshold voltage:

$$V_{UT} = \frac{R_1}{R_1+R_2}(+V_{SAT})$$

Lower threshold voltage:

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

Hysteresis voltage $V_{HY} = V_{UT} - V_{LT}$

Shift angle $\theta = \sin^{-1}(V_{UT}/V_p)$

Tabular Form:

		Amplitude	**Time period**
Input applied	**Vi (p-p) =**		
output applied	**+Vsat=**		
	-Vsat =		

PRECAUTIONS:

1. Loose and wrong connections are to be avoided.
2. The output waveforms should be obtained without and distortion.
3. Parallax error should be avoided.

Conclusion:

RESULT:

Viva questions

1. What do mean by Schmitt trigger?
2. What are the different applications Schmitt triggers?
3. What is meant by Hysteresis voltage?
4. What is meant by threshold voltage?

EXPERIMENT: 6
IC VOLTAGE REGULATOR

Aim:

To obtain the voltage regulation of a 3-terminal fixed IC voltage regulator.

Apparatus:

1) IC 7808
2) Capacitor 1µF, 0.1 µF
3) Voltmeter (0-10) V
4) Ammeter (0-100) mA
5) Decade resistance box (DRB)
6) TRPS
7) Bread board
8) Connecting wires.

Theory:

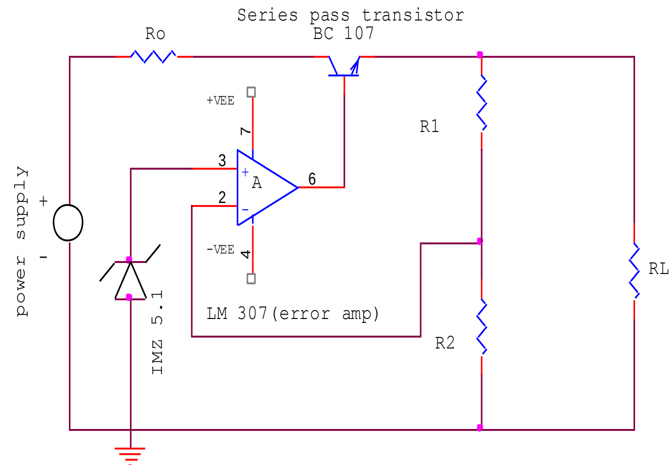
A voltage regulator is an electronic device that provides a stable dc voltage independent of load current, temperature and a.c voltage variations. Figure shows a regulated power supply using discrete components. The circuit consists of following parts.

1. Reference voltage circuit
2. Error amplifier
3. Series pass transistor
4. Feedback network.

It can be seen from the figure that the power transistor Q1 is in series with the un-regulated dc voltage V_{in} and the regulated output voltage Vo so it must absorb the difference between these two voltages whenever any fluctuation in output voltage Vo occurs

The transistor Q1 is also connected as an emitter follower and therefore provides sufficient current gain to drive the load. The output voltage is sampled by R1-R2 divider and feedback to the negative input terminal of op-amp error amplifier sample the output voltage. This sampled voltage is compared with the reference voltage V_{ref}. The output voltage Vo' of the error amplifier drives the transistor Q1.

Internal Diagram:



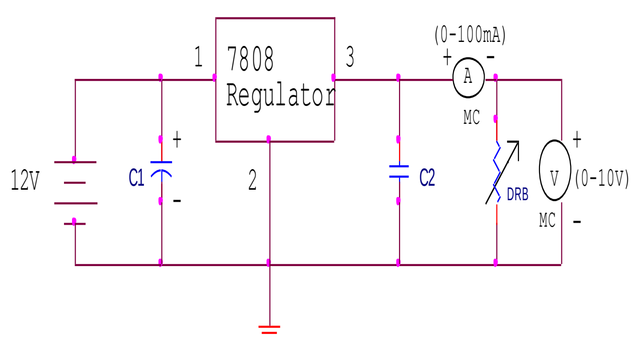
78XX series are three terminal positive fixed voltage regulators. There are seven voltage options available such as 5, 6, 8,12,15,18 and 24V. In 78XX series the last two numbers indicate the output voltage. For example 7808 indicates 8V regulator.

79 series are also 3-terminal IC regulator with fixed output negative voltage regulator.

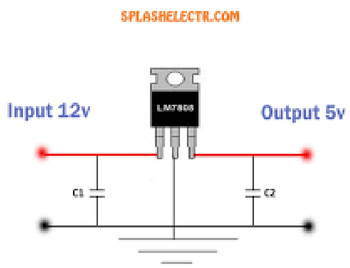
In the standard representation of monolithic voltage regulator a capacitor 'C' is usually connected between input terminal and ground to cancel the inductive effect due to long distribution leads.

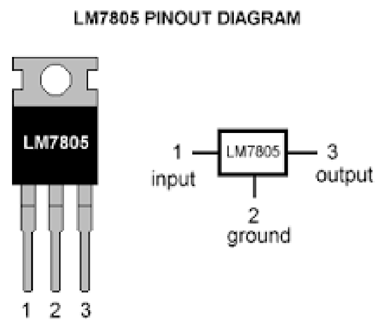
CIRCUIT DIAGRAM:

3-Terminal Fixed Voltage Regulator



C1 = 1µF , C2 = 0.1 µF





Procedure:

1. Connections are made as per the circuit diagram.
2. By adjusting the Voltage across RPS to 12V, the load terminals open circuited, the voltmeter reading is noted. This gives the no load voltage.
3. The load is varied from 10KΩ to 50Ω with the help of decade resistance box the corresponding voltmeter and ammeter reading are noted.
4. A graph is drawn between % voltage regulation on y-axis and load resistance on x-axis.

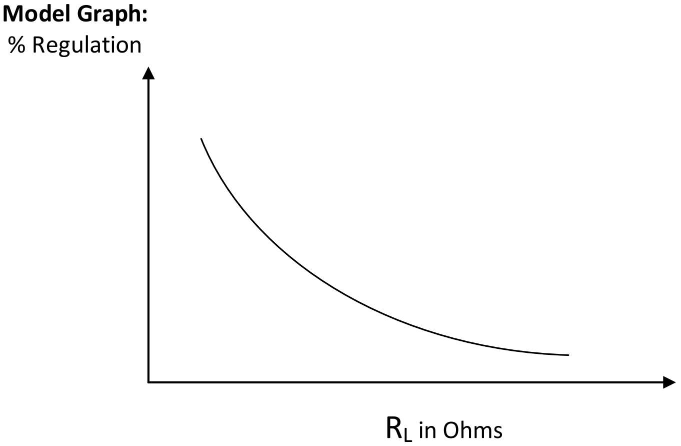
% voltage Regulation =

$$\frac{VNL - Vl}{VL} \times 100$$

Tabular form V_{NL} =

$R_{L\,(\Omega)}$	I (mA)	V (Volts)	% Regulation
10KΩ to 100Ω			

Model Graph:



PRECAUTIONS:

1. Loose and wrong connections are to be avoided.
2. The output waveforms should be obtained without and distortion.
3. Parallax error should be avoided

Conclusion:

RESULT:

Viva questions

1. What do mean by voltage regulator?
2. What is error amplitude?
3. What is meant by error amplitude?
4. What is meant by threshold voltage?

EXPERIMENT: 7
UJT AS A RELAXATION OSCILLATOR

AIM:

To obtain a saw tooth waveform using UJT and test its performance as an oscillator

APPARATUS:

1) UJT-2N 2646 - 1
2) Resistors- 100 Ω -2
   15 K Ω -1
3) Capacitors- 0.01 µf -1
4) TRPS -1
5) CRO -1
6) Bread board CRO probes and connecting wires

ANALYSIS OF UJT:

The voltage V_{BB} is applied between B_1 and B_2. If I_E=0, then voltage drop across R_{B1} is given by,

$$V_1 = \frac{RB1}{RB1 + RB2} \text{ x } VBB$$

The ratio $\frac{RB1}{RB1+RB2}$ is termed as Intrinsic stand off ratio and is denoted by η.

$V_1 = \eta V_{BB}$

The value of emitter voltage, which makes the diode conduct, is termed as peak voltage and is given by

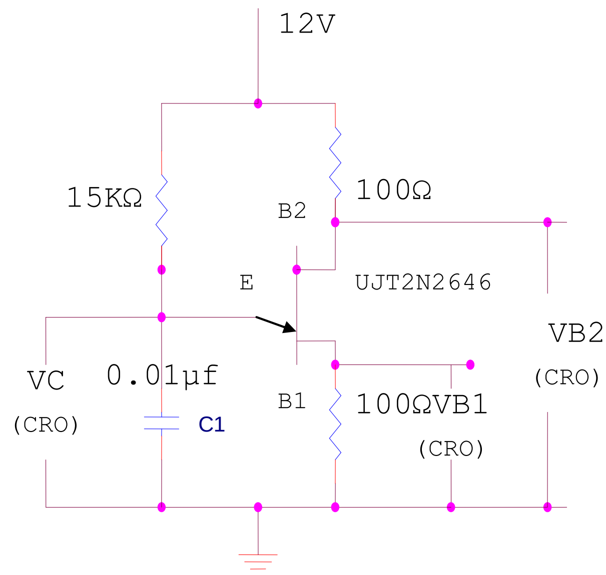
$V_P = V_D + V_1$

$V_P = V_D + \eta V_{BB}$

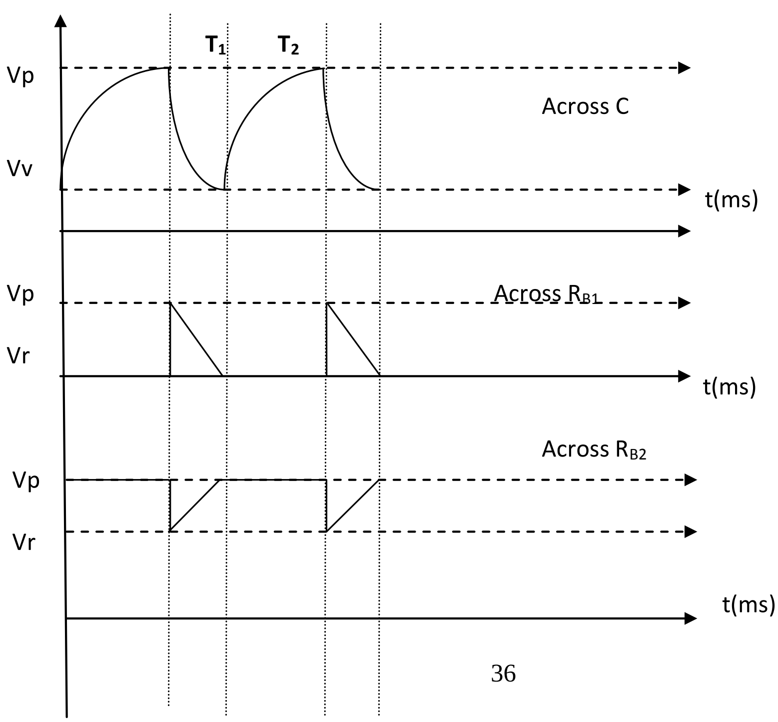
Expression for frequency of oscillation:

F =1/T
=1/RC ln (1/1-η)

CIRCUIT DIAGRAM



MODEL GRAPHS:



Observations:

	Vv	Vp	T_1	T_2
Across C				
Across R_{B1}				
Across R_{B2}				

T1----charging time period
T2 ----discharging time period
F -----frequency of oscillations
F=1/T
F=1/[RC ln [(1/(1- η))]
η =RB1/(RB1+RB2)

PROCEDURE:

1. The circuit is connected as per the circuit diagram.
2. A supply of V_{BB} =12v is applied to the circuit with the help of TRPS.
3. The output waveforms across capacitor, resistor R_{B1} and resistor R_{B2} are obtained from the CRO.
4. The frequency of the corresponding signals is noted and the waveforms are plotted on the graph sheet.

PRECAUTIONS:

1) Loose and wrong connections must be avoided.
2) Parallax error should be avoided while taking the readings.

Conclusion:

RESULT:

Viva questions

1. What do mean by Intrinsic standoff ratio?
2. Why the wave form of RB2 is getting negative spikes?
3. What is meant by base bias resistor?

EXPERIMENT: 8
ASTABLE MULTIVIBRATOR USING 555 IC

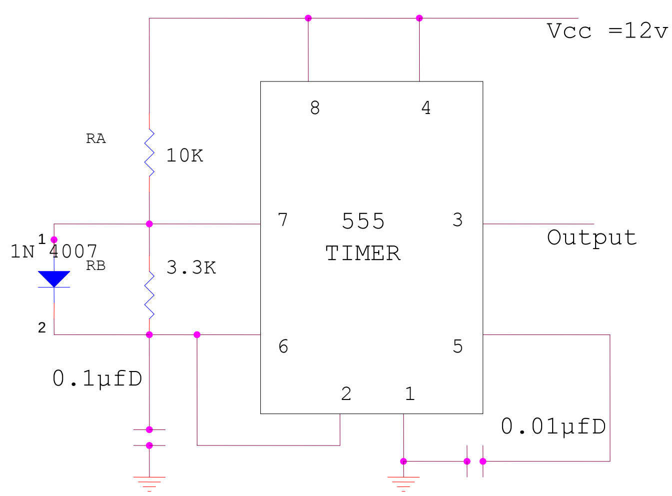
AIM:

To obtain a symmetric square wave output by maintaining certain duty cycle by using 555 IC.

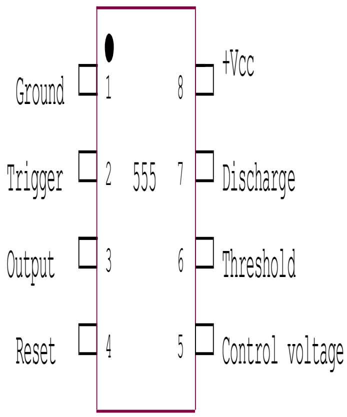
APPARATUS:

1) 555 IC
2) Resistors 3.3 KΩ
3) Capacitors 0.1µF, 0.01µF
4) 10KΩ Potentiometer
5) TRPS
6) Diode 1N4007 1
7)CRO and CRO Probes

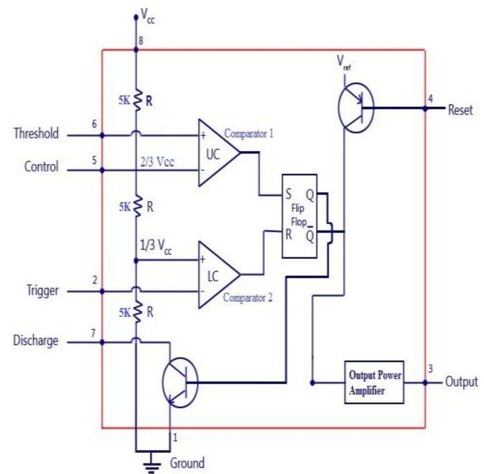
CIRCUIT DIAGRAM:



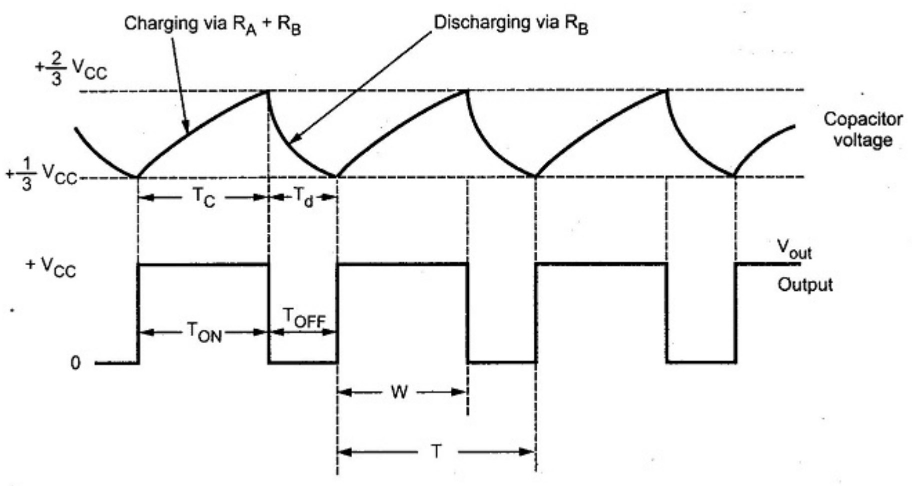
PIN DIAGRAM:



555 IC Timer Block Diagram



MODEL GRAPHS:



Waveforms of astable operation

Duty cycle: The capacitor voltage for a low pass RC circuit subjected to a step input of Vcc volts is given by

$$V_c = V_{cc} (1- \exp(-t/ RC))$$

The time t_1 taken by the circuit to charge from 0 to 2/3 Vcc is,

$$2/3\ V_{cc} = V_{cc} (1- \exp(-t_1/ RC))$$

$$t_1 = 1.09\ RC$$

The time t_2 to charge from 0 to 1/3 Vcc is,

$$1/3\ V_{cc} = V_{cc} (1- \exp(-t_2/ RC))$$

$$t_2 = 0.405\ RC$$

So the time to charge from 1/3 Vcc to 2/3 Vcc is

$$t_{HIGH} = t_1 - t_2 = 1.09\ RC - 0.405\ RC = 0.69\ RC$$

So, for the given circuit,

$$t_{HIGH} = 0.69 (R_A + R_B) C$$

The output is low while the capacitor discharges from 2/3 Vcc to 1/3 Vcc and the voltage across the capacitor is given by

$$1/3\ V_{cc} = 2/3\ V_{cc} (\exp(-t/RC))$$

$$t_{LOW} = 0.69\ RC$$

For the given circuit, $t_{LOW} = 0.69\ R_B C$

Total time period, $T = t_{HIGH} + t_{LOW} = 0.69 (R_A+2R_B) C$

Duty cycle = $t_{HIGH} /T = (R_A+ R_B)/ (R_A+ 2R_B)$

For the modified circuit Duty cycle = $R_A / (R_A+ R_B)$

PROCEDURE:

1. The connections are made as per the circuit diagram.
2. Now the potentiometer is adjusted till the 50% duty cycle is achieved. Output waveform is observed on the CRO.
3. Time periods of the output waveform are noted and output waveform is plotted to the scale.
4. The corresponding waveforms for other duty cycles are also obtained and plotted to scale.

Tabular Form:

Duty cycle	$R_A(\Omega)$	T_{high} µ Sec	T_{low} µ Sec	Across pinNo:6		Across pin No:3	Frequency Theoretical	Frequency Practical
				V_1 (v)	V_2 (v)			

PRECAUTIONS:

1. Loose and wrong connections should be avoided.
2. Parallax error should be avoided.

Conclusion:

RESULT:

Viva questions

1. What do mean by duty cycle?
2. What is RS flip flop?
3. What is comparator?
4. What are the applications of astable multivibrator?
5. How many stable states we have in astable multivibrator?
6. What is quasi stable state?

EXPERIMENT: 9

BISTABLE MULTIVIBRATOR

AIM: To design and study the Fixed Bistable multivibrator using transistors.

APPARATUS:

1) Transistors BC 107 –2
2) Resistors 100KΩ –2
   2.2KΩ -2
   15kΩ -2
3) Light emitting diodes -2
4) Regulated power supply-1
5) Bread board and connecting wires
6) Digital multi meter (0-20V) -1

DESIGN OF A BISTABLE MULTIVIBRATOR:

For the given Vcc, V_{BB}, $h_{fe\,(min)}$, $I_{c\,(sat)}$ it is possible to compute the values of R_{c1}, R1 and R_2. The following assumptions are made in order to design the bistable fixed bias multivibrator.

1. If Q_1 and Q_2 are identical silicon transistors, the junction voltages are assumed as $V_{CE\,(sat)} = 0.3$ V and $V_{BE\,(sat)} = 0.7$ V.
2. The base current of the ON transistor is taken as 1.5 times of the minimum value of base current.

   $I_B = 1.5\, I_{B\,(min)}$

   Where $I_{B\,(min)} = I_{c\,(sat)} / h_{fe\,(min)}$
3. The current through R_2 of the ON transistor is taken as one tenth of $I_{C.}$

   If Q_2 ON, $I_4 = I_{C2}/10$.

4 The current through R_1 is ignored since it is quite small in comparison with the collector current of ON transistor.

To find R_C:

$R_C = V_{CC} - V_D / I_2 = V_{CC} - V_{CE\,(sat)} / I_{C\,(sat)}$

$R_{C1} = R_{C2}$

To find R_2:

The current though R_2 is I_4, where $I_4 = I_{C2}/10 = I_{C(sat)}/10$.

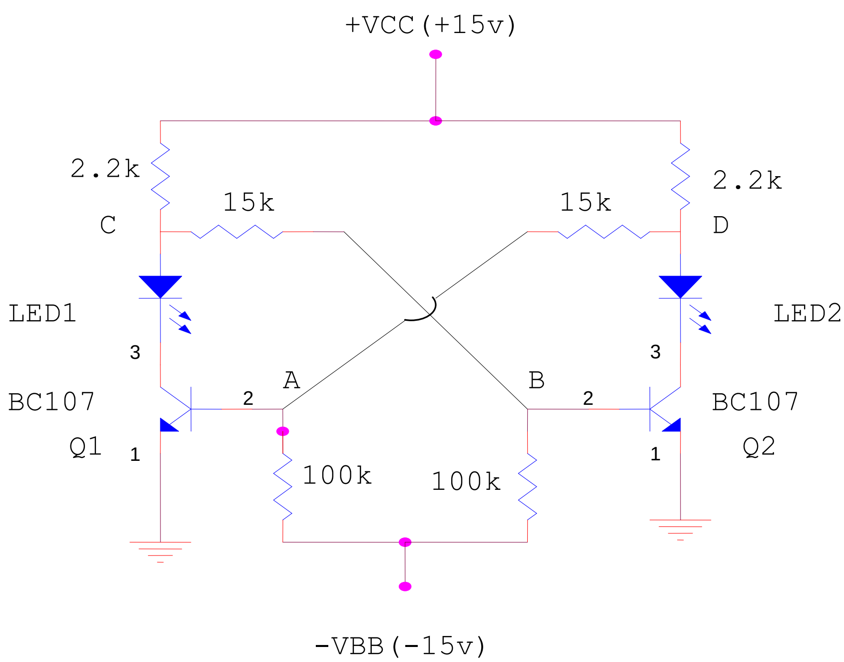
$R_2 = [V_B - (-V_{BB})]/I_4$

To find R_1:

The current through R_1 is I'

$$I' = I_{B2} + I_4$$
$$I_{B2} = 1.5\ I_{B(min)}$$
$$\text{Where } I_{B(min)} = I_{C(sat)}/h_{fe(min)}.$$
$$I_1 = I'$$
$$I_1 = V_{CC} - V_B/R_{C1} + R_1$$
$$R_1 = [V_{CC} - V_{BE(SAT)}/I_1] - R_{C1}$$

<u>CIRCUIT DIAGRAM</u>:



PROCEDURE:

1. The connections are made as per the circuit diagram.
2. The supply is switched on and it is observed that one LED is ON whereas the other is OFF.
3. Now the base voltages of both the transistors V_{B1} and V_{B2} and collector voltages V_{C1} and V_{C2} are noted.
4. A negative trigger is given at the base of the ON transistor to change the states of the transistors.
5. In this steady state the base voltages of both the transistors V_{B1} and V_{B2} and also the collector voltages V_{C1} and V_{C2} are noted.

PRECAUTIONS:

1. Loose and wrong connections should be avoided.
2. Parallax error should be avoided.

Conclusion:

RESULT:

Viva questions

1. What is stable state?
2. Name the types of multivibrators?
3. What is quasi stable state?
4. How many stable states are there in binary?
5. What is the need of triggering
6. What are the types of triggering are there in multivibrator?

EXPERIMENT: 10

FREQUENCY RESPONSE OF ACTIVE FILTER

AIM:

To obtain the response of active filters by varying the frequency.

APPARATUS:

1) OP-AMP LM 741C –2
2) Resistors 10kΩ –4
   16 KΩ -3
3) Capacitors –0.01μf –3
4) Function generator
5) TRPS
6) CRO & CRO Probes
7) Connecting wires & bread board

THEORY:

Filters are frequency selective networks, which can allow desired range of frequencies and attenuates other frequencies. Filters are classified:

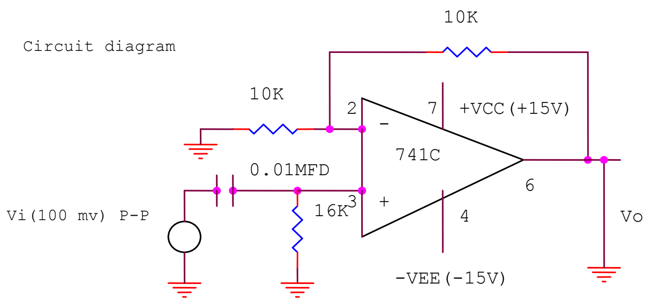
1. Passive and Active filters
2. Analog and Digital Filters

Depending on the type of the elements used as resistor, capacitor, and inductor such a type of filter is called as passive filters. By using op-amp and transistor on addition to passive elements, they are called as active filters.

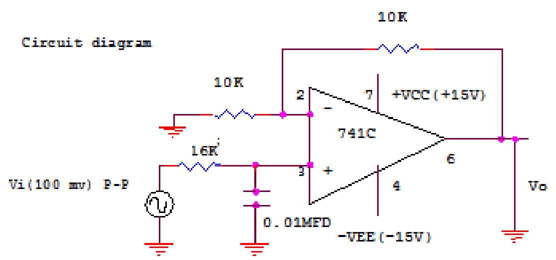
Depending on the range of frequencies the active filters can be classified as low pass, band pass, high pass, all pass, band reject filters.

CIRCUIT DIAGRAM:

HIGH PASS BUTTERWORTH FILTER

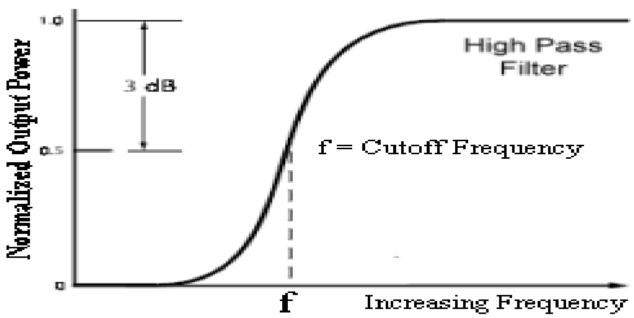


LOW PASS BUTTERWORTH FILTER

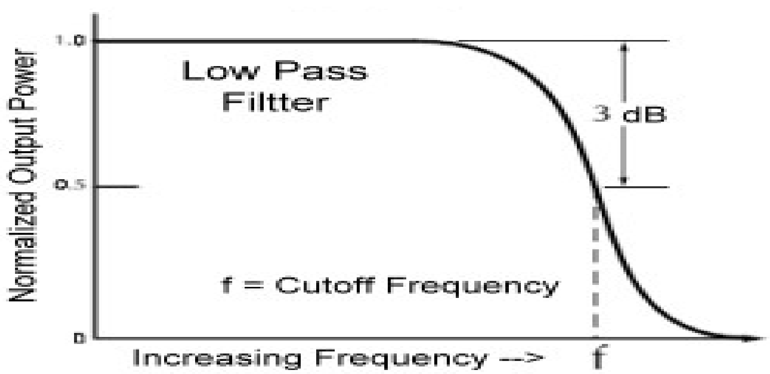


MODEL GRAPHS:

HGIH PASS BUTTERWORTH FILTER



LOW PASS BUTTERWORTH FILTER



DESIGN:

Design of I order Butter worth filter:

Given the cut off frequency F_L, A_O,

$F_L = 1/2\pi RC$

Assume C and then substituting the value in the above formula

Find R, using A_O and assuming R1 find RF

PROCEDURE:

1. The circuit is connected as per the circuit diagram
2. The Frequency of the input signal is varied and the Corresponding out put voltage is noted. The magnitude of the input Signal is kept constant through out the experiment.
3. The gain for each frequency is calculated using the formula

   Gain in dB=20 log (V_O/V_I).
4. A graph for gain v/s frequency is plotted which is known as Frequency response.

TABULAR FORM:

INPUT VOLTAGE:100 mv (p-p)

HGIH PASS BUTTERWORTH FILTER

Frequency (Hz)	Out Put Voltage(V)	Gain=20log(v_0/v_i) (dB)
100Hz to 1M Hz		

LOW PASS BUTTERWORTH FILTER

Frequency (Hz)	Out Put Voltage(V)	Gain=20log(v_0/v_i) (dB)
100Hz to 1M Hz		

PRECAUTIONS:

1. Loose and wrong connections should be avoided.
2. Parallax error should be avoided.

Conclusion:

RESULT:

Viva questions

1. What is filter?
2. What is an active filter?
3. What is high pass filters & low pass filters?
4. Name the types of filtes?
5. What is butter worth filter?

EXPERIMENT: 11

IC OP-AMP BOOT STRAP RAMP GENERATOR

AIM: To generate a ramp wave forms by maintaining constant current conditions by using a boot strap ramp generator with an op-amp 741IC as voltage follower.

APPARATUS:

1. Diode 1N4007 - 1
2. Op-amp: LM 741C -1
3. Capacitors --33kpf -1
   0.02µf -1
   2.2 µf -1
4. Resistors - 47kΩ, 470kΩ, 10kΩ
5. Transistors-BC107
6. Function Generator.
7. TRPS
8. Bread Board & Connecting Wires
9. CRO & CRO Probes

DESIGN:

For a given
Maximum diode current is I_R =3µA
Allowing 1% non linearity due to I_R
$I_1=100\ I_R$
$C_1=I_1$ (Ramp time/V_P)
Assume V_P =8V
$V_{R1}=V_{CC}-V_{D1}-V_{CE\ (sat)}$
$R_1=V_{R1}/I_1$

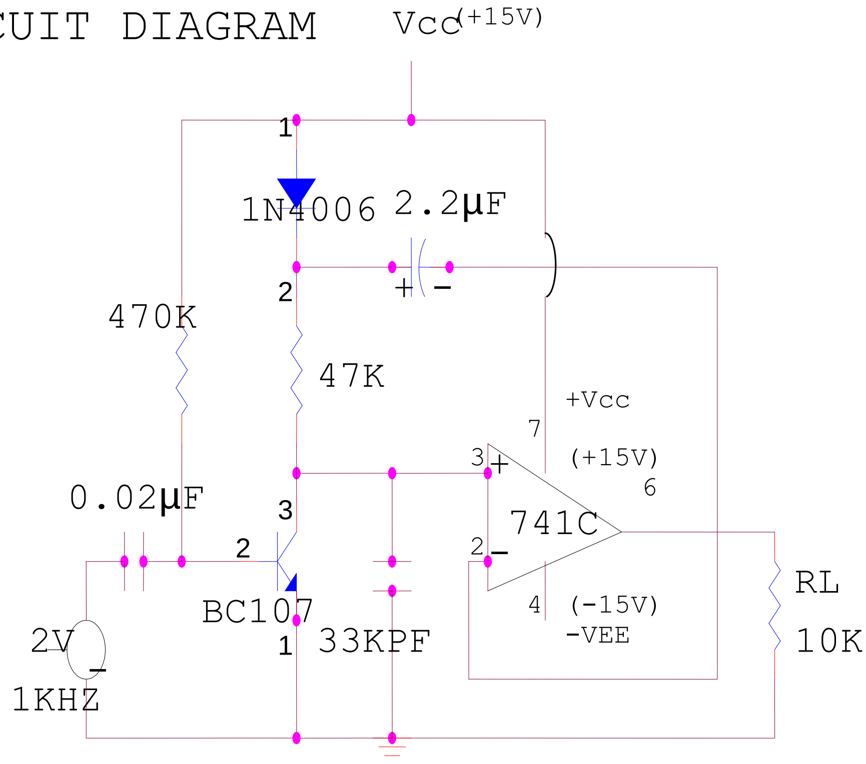
For 1% non-linearity due to C_3 discharge:

$V_{C3}=V_{CC}$
ΔV_{C3}=1% of Vcc
C_3 discharges current I_1
$C_3=I_1\ (t/\Delta V_{CC})$

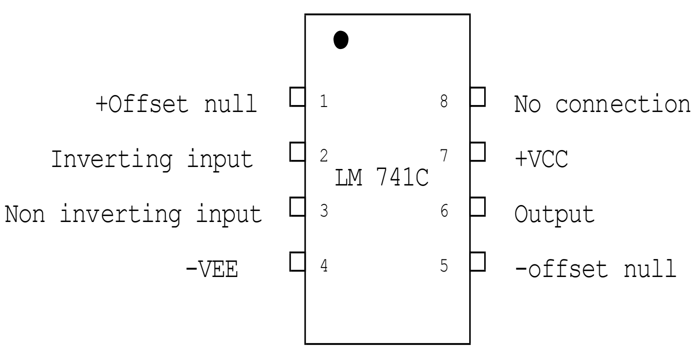
Calculation of C_2:

Minimum Ic of Q_1= 10 I_1
$I_B=I_C/h_{fe}$
$I=V_{CC}-V_1/R_B$ where $R_B=V_{CC}-V_{BE}/I_B$
$C_2=I\ (t/\Delta V)$

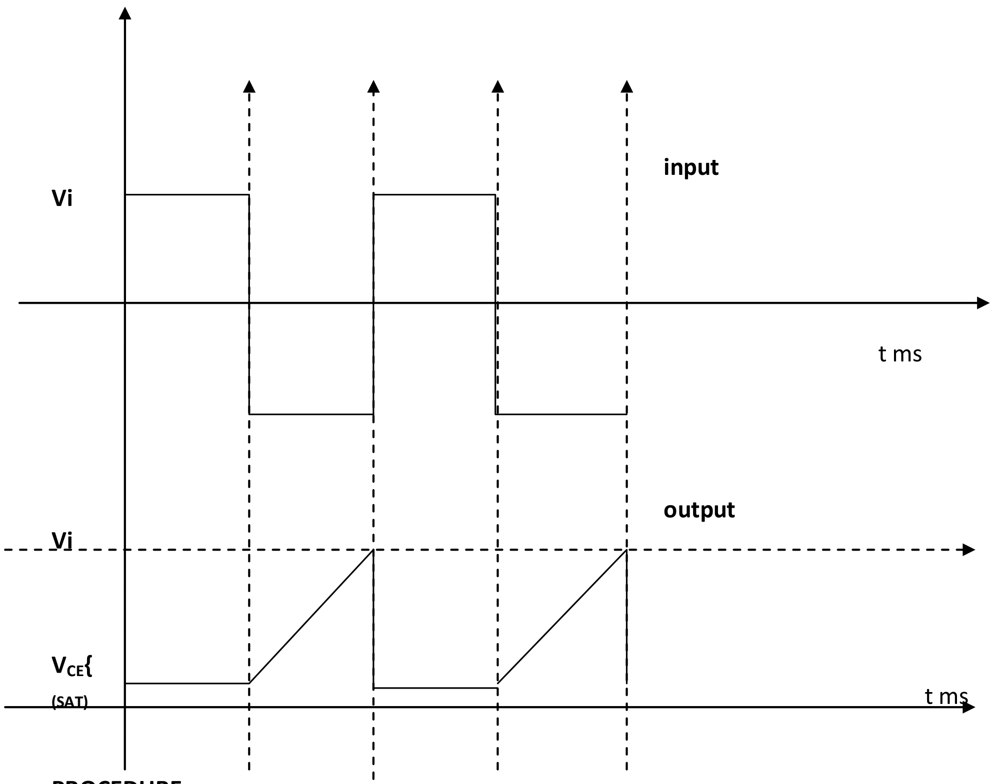
CIRCUIT DIAGRAM



PIN DIAGRAM



MODEL GRAPHS:



PROCEDURE:

1. The circuit is connected as per the circuit diagram.
2. A square wave of 2V (p-p), 1 KHz is applied with the help of function generator to the base of a transistor.
3. The corresponding input and output waveforms are noted from the CRO
4. The graphs are plotted for the input and output waveforms.

OBSERVATION

V_{CE} =

Output Voltage Vo =

Sweep time Ts =

PRECAUTIONS:

1. Loose and wrong connections should be avoided.
2. Parallax error should be avoided.

RESULT:

Viva questions

1. What is sweep circuit?
2. How the Op-amp acts as a emitter follower?
3. How the current constant is maintained in Boot strap?
4. Name different types of ramp generator?

MAJOR EQUIPMENT IN
LINEAR IC'S & PULSES CIRCUITS LABORATORY

S.NO	DESCRIPTION	MAKE	QUANTITY
1.	20 MHz DUAL TRACE OSCILLOSCOPE	AP LAB /SCIENTIFIC	21
	20 MHz DIGITAL STORAGE OSCILLOSCOPE	FALCON	
2.	1 MHz FUNCTION GENERATOR WITH DIGITAL DISPLAY	AP LAB/ PACIFIC	18
3.	TRPS 0-30V, 2A DUAL CHANNEL	ITL/PACIFIC /FALCON	20
4.	DC MICRO & MILLI AMMETERS	MECO/HI-Q/AQUILA	46
5.	DC MICRO VOLTMETER	MECO/HI-Q/AQUILA	18

6.	**BENCH TOP DIGITAL MULTIMETER**	**METRAVI/ MECO**	**15**
7.	**5KVA SERVO CINTROLLED STABILIZER**	**ITL**	**01**

TOTAL EXPENDITURE OF LABORATORY: Rs: 6,30,679.71/-