LENDI INSTITUTE OF ENGINEERING AND TECHNOLOGY

(Approved by A.I.C.T.E & Affiliated to JNTU, Kakinada)
Jonnada (Village), Denkada (Mandal), Vizianagaram Dist – 535005
Phone No. 08922-241111, 241112
E-Mail: lendi_2008@yahoo.com **Website:** www.lendi.org

DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

PULSE AND DIGITAL CIRCUITS LABORATORY OBSERVATION

Name:

Regd No: **Year&Sem:**

Academic Year:

LIST OF EXPERIMENTS

Minimum Twelve experiments to be conducted:

1. Linear wave shaping.
2. Non Linear wave shaping – Clippers.
3. Non Linear wave shaping – Clampers.
4. Transistor as a switch.
5. Study of Logic Gates & Some applications.
6. Study of Flip-Flops & some applications.
7. Sampling Gates.
8. Astable Multivibrator.
9. Monostable Multivibrator.
10. Bistable Multivibrator.
11. Schmitt Trigger.
12. UJT Relaxation Oscillator.
13. Bootstrap sweep circuit.

INTRODUCTION TO PDC LAB:

This Lab is organized into 12 experiments and the outline is as follows:

When non-sinusoidal signals are transmitted through a linear network, the shape of the waveform undergoes a change. This process is called linear wave shaping and is our 1st experiment.

In communication systems it is required to remove a part of the waveform above or below some reference level, this process is called clipping. In many pulse systems, quite often a dc level is required to be added to a waveform to fix the top or bottom of waveform at some reference level, this process is called clamping. Clipping and Clamping together is called non- Linear wave shaping and is our 2nd and 3rd experiments.

The switching characteristics of transistors are designed in the 4th experiment.

Logic gates are the fundamental building blocks of any digital system. Realization of logic gates using diodes and transistors are done in the 5th experiment.

The different types of flip- flops are designed in the 8th experiments which are used for storing digital data.

When signals are to be transmitted only for specified intervals of time and are to be blocked during other intervals of time, we require sampling gates. Various types of sampling gates are designed in our 7th experiment.

Memory is the basic requirement of all computers. The basic memory element is flip- flop i.e. the bistable multivibrator. The monostable multivibrator is the basic gating circuit. The astable multivibrator is used as a master oscillator and the Schmitt trigger circuit as a basic voltage comparator. The various types of multivibrators designing is done in the 8,9,10 and 11 experiments.

Time- base generators are essential for display of signals on the screen. Voltage and current time- base generators are designed in 12th and 13th experiments.

EXPERIMENT NO:1 Date:

LINEAR WAVE SHAPING

AIM : *a) To study the response of RC Low pass circuit and to determine rise time for a square wave input for different time constants.*
i) RC>>T ii) RC = T iii) RC<<T
b) To study the response of RC High pass circuit and to determine percentage tilt for a square input for different time constants.
i) RC>>T ii) RC = T iii) RC<<T

COMPONENTS REQUIRED :

1. Resistors - 10kΩ, 100 kΩ, 1MΩ
2. Capacitor - 0.01μF

APPARATUS REQUIRED :

1. Bread Board.
2. CRO
3. Function Generator.
4. Connecting Wires.

THEORY:

LINEAR WAVE SHAPING

The process of where by the form of a non-sinusoidal signal is altered by transmission through a linear network is called "*LINEAR WAVE SHAPING".*

a) RC Low Pass Circuit :

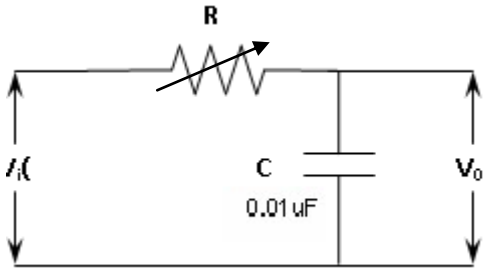


Figure1.1: RC Low Pass Circuit.

The circuit passes low frequencies readily but attenuates high frequencies because the reactance of the capacitor decreases with increasing frequency. At very high frequencies the capacitor acts as a virtual short circuit and the output falls to zero. This circuit also works as integrating circuit. A circuit in which the output voltage is proportional to the integral of the input voltage is known as integrating circuit. The condition for integrating circuit is RC value must be much greater than the time period of the input wave (RC>>T)

Let V_i = alternating input voltage, i = resulting current
Applying Kirchoff's Voltage Law to RC low pass circuit (fig.1).

$$V_i = iR + \frac{1}{C}\int_o^T i.dt$$

Multiplying throughout by C, we get

$$CV_i = iRC + \int_o^T i.dt$$

As $RC >> T$, the term $\int_o^T i.dt$ may be neglected

$$\therefore \quad CV_i = iRC$$

Integrating with respect to T on both sides, we get

$$\int_0^T CV_i.dt = RC\int_0^T i.dt$$

$$\frac{1}{C}\int_0^T i.dt = \frac{1}{RC}\int_0^t V_i.dt$$

$$V_0 = \frac{1}{C}\int_0^T i.dt$$

$$\therefore \quad V_0 = \frac{1}{RC}\int_0^t V_i.dt$$

The output voltage is proportional to the integral of the input voltage.

EXPECTED GRAPH:

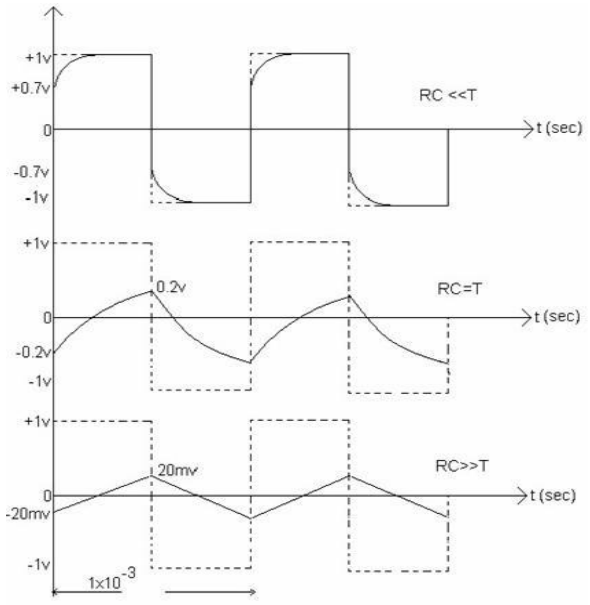


Fig1.2:Output Of Low Pass Filter

b) RC High Pass Circuit.

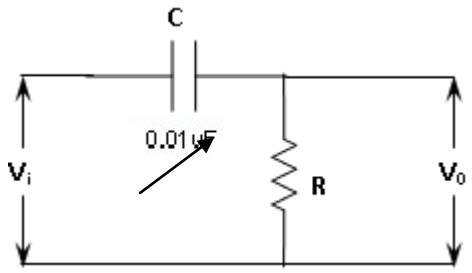


Figure: 1.3. RC High Pass Circuit.

The higher frequency components in the input signal appears at the output with less attenuation than the lower frequency components because the reactance of the capacitor decreases with increase in frequency. This circuit works as a differential circuit. A circuit in which the output voltage is proportional to the derivative of the input voltage is known as differential circuit. The condition for differential circuit is RC value must be much smaller then the time period of the input wave *(RC<<T)*.

Applying Kirchoff's Voltage Law to RC high pass circuit (fig.2)

$$V_i = \frac{1}{C}\int_o^T i.dt + iR\,.$$

Divide throughout by R

$$\frac{V_i}{R} = \frac{1}{RC}\int_o^T i.dt + i\,.$$

As *RC << T*, the above equation is modified as

$$\frac{V_i}{R} = \frac{1}{RC}\int_o^T i.dt$$

Differentiating above equation with respect to *T*.

$$\frac{1}{R}\frac{d}{dt}V_i = \frac{1}{RC}.i$$

$$RC\frac{d}{dt}V_i = iR$$

$$V_0 = iR$$

Therefore $V_0 = RC\frac{d}{dt}V_i\,.$

$$V_0 \propto \frac{d}{dt}V_i$$

EXPECTED GRAPHS:

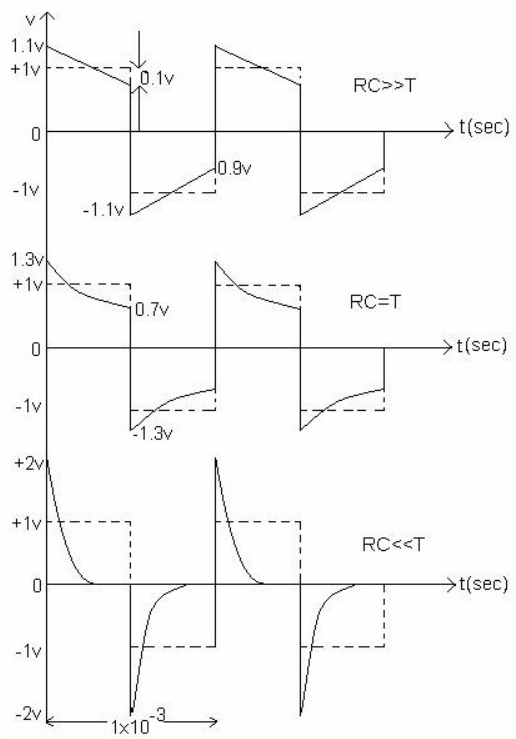


Fig 1.4:Output Of Low Pass Filter

DESIGN:

1. Choose T = 1msec.
2. Select C = 0.01 μF.
3. For RC = T; select R.
4. For RC >> T; select R.
5. For RC << T; select R.
6. If RC << T, the High pass circuit works as a differentiator.
7. If RC >> T, the Low pass circuit works as an integrator.

PROCEDURE:

1. Connect the circuit as shown in the figure1 &2.
2. Connect the function generator at the input terminals and CRO at the output terminals of the circuit.
3. Apply a square wave signal of frequency 1KHz at the input. (T = 1 msec.)
4. Observe the output waveform of the circuit for different time constants.
5. Calculate the rise time for low pass filter and tilt for high pass filter and compare with the theoretical values.
6. For low pass filter select rise time (t_r) = 2.2 RC (theoretical). The rise time is defined as the time taken by the output voltage to rise from 0.1 to 0.9 of its final value.
7. % tilt = $(T/2RC) \times 100$ (theoretical)
   % tilt = $[(V_1 – V_1') / (V / 2)] \times 100$ (practical)

RESULT:

1. Rise time for lowpass filter when RC <<T
   Theoretical = Practical =

2. % tilt for highpass filter when RC = T.
   Theoretical = Practical =

Response of RC Low pass circuit is observed and rise time calculated.
Response of RC High pass circuit is observed and percentage tilt is calculated.

VIVA QUESTIONS:

1. What is high pass circuit under what condition it acts as a differentiator?
2. What is low pass circuit under what condition it acts as a integrator?
3. Show theoretically how you get a triangular wave when a square wave is given to an integrator?
4. What happens when a sine wave is applied to a differentiator or integrator circuit?
5. What are different applications of a differentiator?
6. What are different applications of a integrator?
7. What is the ideal value of phase shift offered by an RC circuit?

APPLICATIONS:

1. Linear wave shaping networks as a low pass filter and high pass filter used to control the transmission with respect to frequency.
2. The output of High pass network for less time constant can be used as a trgger for monostable multivibrator.
3. The low pass network can be used to generate the triangular wave for high time constant.

DESIGNING PROBLEM:

1. A 1khz symmetrical square wave of $\pm$10V is applied to a RC circuit having 1msec Time constant .calculate and plot the scale for the configurations :
   a) High Pass Circuit
   b) Low Pass Circuit

GRAPH

GRAPH

EXPERIMENT NO:2 Date:

NON LINEAR WAVE SHAPING - CLIPPERS

AIM : *To study the clipping circuits for the following reference voltages and to verify the responses.*

Components Required:

1. Resistors - 1KΩ
2. IN4007 Diode – 2No.

Apparatus Required :

1. Bread board.
2. Function generator
3. CRO
4. Power supply (0-30V)
5. Connecting wires

THEORY:

The non-linear semiconductor diode in combination with resistor can function as clipper circuit. Energy storage circuit components are not required in the basic process of clipping.

These circuits will select part of an arbitrary waveform which lies above or below some particular reference voltage level and that selected part of the waveform is used for transmission. So they are referred as voltage limiters, current limiters, amplitude selectors or slicers.

There are three different types of clipping circuits.

1) Positive Clipping circuit.
2) Negative Clipping.
3) Positive and Negative Clipping (slicer).

In positive clipping circuit positive cycle of Sinusoidal signal is clipped and negative portion of sinusoidal signal is obtained in the output of reference voltage is added, instead of complete positive cycle that portion of the positive cycle which is above the reference voltage value is clipped.

In negative clipping circuit instead of positive portion of sinusoidal signal, negative portion is clipped.

In slicer both positive and negative portions of the sinusoidal signal are clipped.

I. <u>Positive Clipping</u>

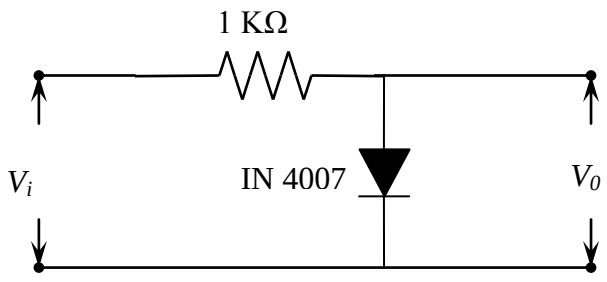


Figure:2.1.positive clipper

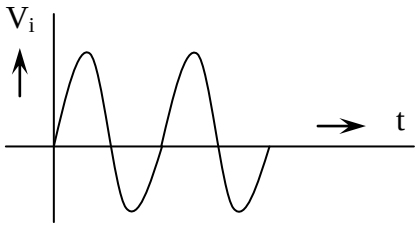


Figure: 2.2. Input waveform

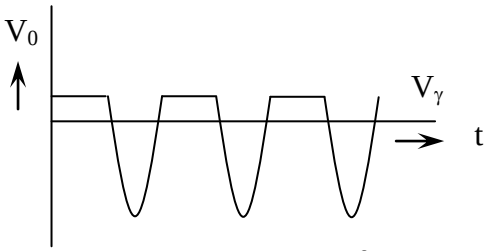


Figure: 2.3.Output waveform.

V_i is a input sinusoidal signal as shown in the figure 2(a) . For positive portion of the sinusoidal the diode IN4007 gets forward biased. The output voltages in the voltage across the diode under forward biased which is cut-in-voltage of the diode. Therefore the positive portion above the cut-in-voltage is clipped or not observed in the output *(V_0)* as shown in figure 2(b).

II. Positive Clipping with Positive Reference Voltage

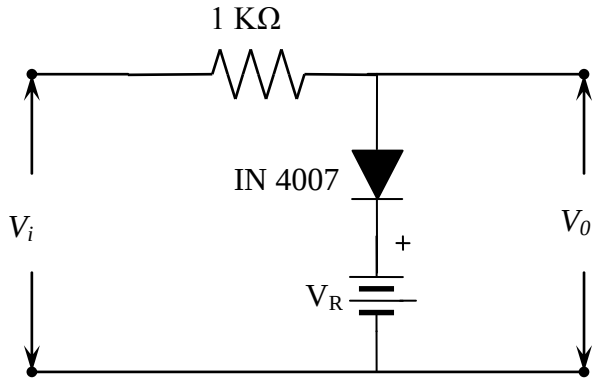


Figure:2.4. Positive Clipping with Positive Reference Voltage

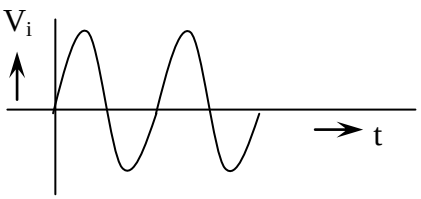


Figure:2.5.Input waveform

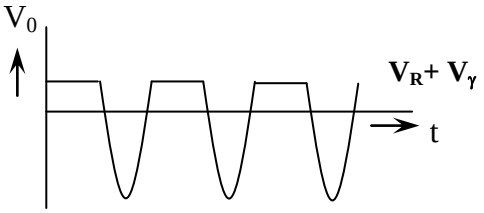


Figure:2.6.Output waveform.

The input sinusoidal signal (V_i) in figure 4(a) can make the diode to conduct when its instantaneous value is greater than V_R. Up to that voltage (V_R) the diode is open circuited and the output voltage is same as the input voltage. After that voltage (V_R) the output voltage is V_R plus the cut-in-voltage (V_γ) of the diode as shown in figure 4(b).

III. Positive Clipping with Negative Reference Voltage

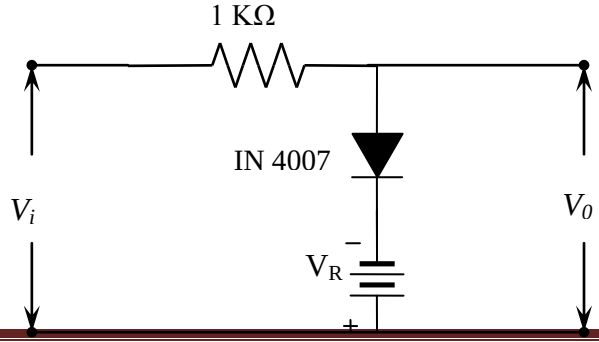
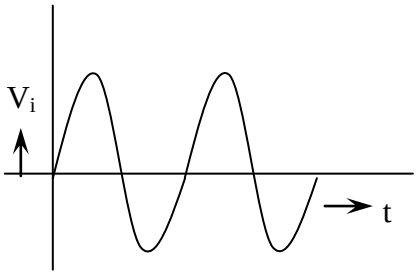


Figure: 2.7.Positive Clipping with Negative Reference Voltage



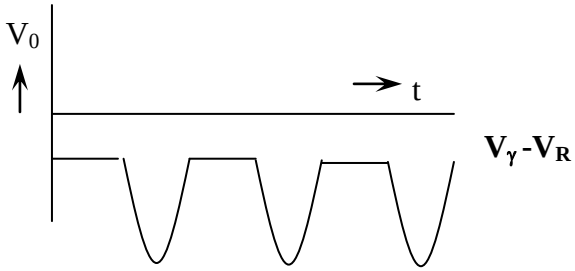


Figure:2.8.Input waveform Figure:2.9.Output waveform.

In this circuit the diode conducts the output voltage is same as input voltage. The diode conducts at a voltage less by V_R from cut-in-voltage called as V_γ. For voltage less than V_γ, the diode is open circuited and output is same as input voltage.

IV Negative Clipping Circuit

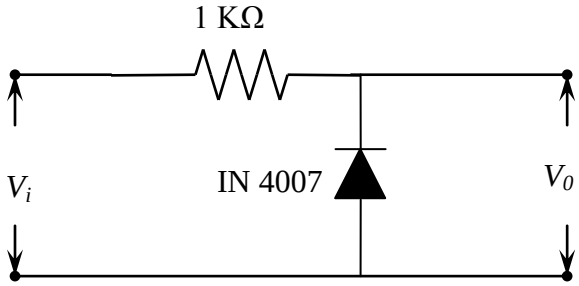
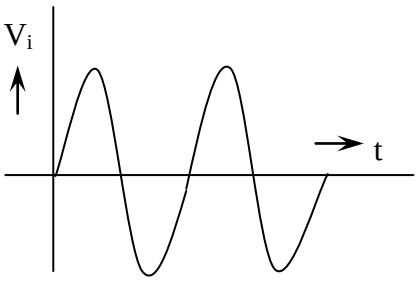


Figure:2.10.Negative Clipping Circuit



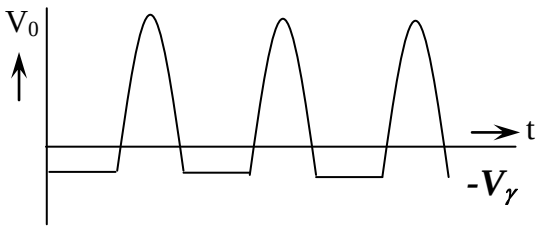


Figure: 2.11. Input waveform Figure: 2.12.Output waveform.

For this portion of the input sinusoidal signal (Vi), the diode gets reverse biased and it is open. Then the output voltage is same as input voltage. For the negative portion of the signal the diode gets forward biased and the output voltage is the cut-in-voltage $(-V_\gamma)$ of the diode. Then the input sinusoidal variation is not seen in the output. Therefore the negative portion of the input sinusoidal signal (Vi) is clipped in the output signal (V_0).

V. Negative Clipping with Negative Reference Voltage

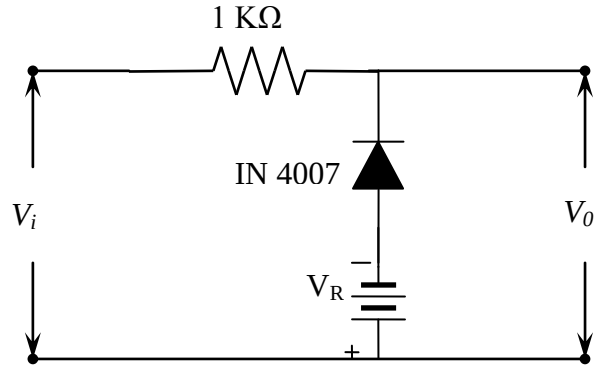


Figure:2.13.Negative Clipping with Negative Reference Voltage

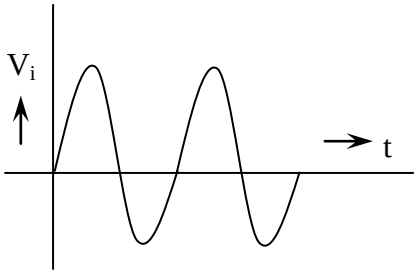


Figure:2.14. Input waveform.

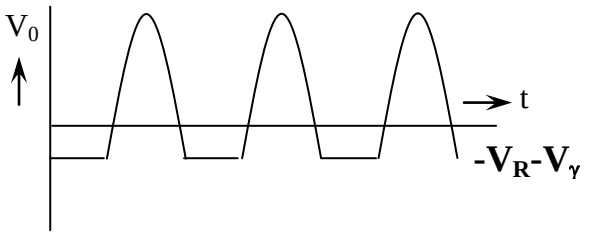


Figure:2.15. Output waveform.

In this circuit, the diode gets forward biased for the input sinusoidal voltage is less than ($-V_R$). For input voltage greater than ($-V_R$), the diode is non-conducting and it is open. Then the output voltage is same as input voltage.

VI. Negative Clipping with Positive Reference Voltage

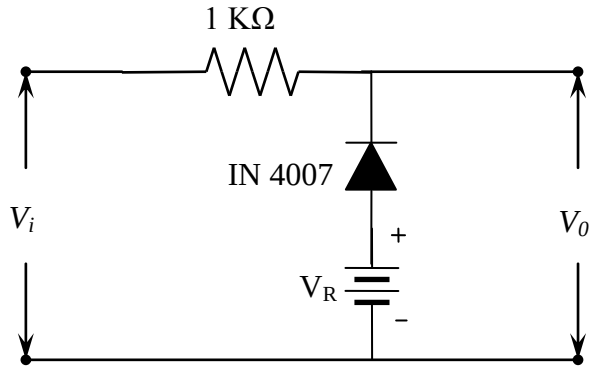


Figure:2.16.Negative Clipping with Positive Reference Voltage

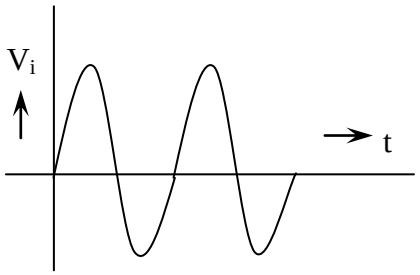


Figure: 2.17. Input waveform

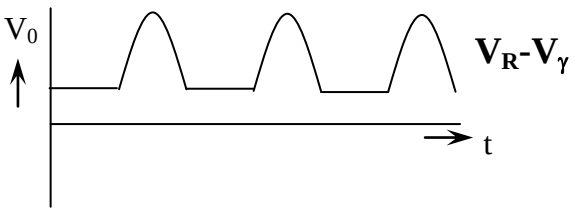


Figure: 2.18. Output waveform.

For input sinusoidal signal voltage less than V_R, the diode is shorted and the output voltage is fixed ar V_R. For input sinusoidal voltage greater than V_R the diode is reverse biased and open circuited. Then the output voltage is same as input voltage.

VII. Slicer

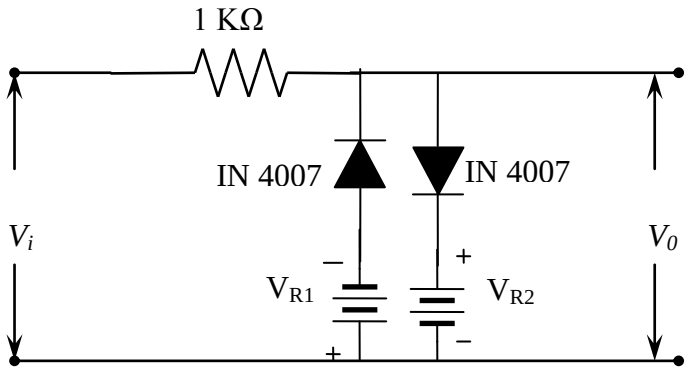


Figure: 2.19.Slicer

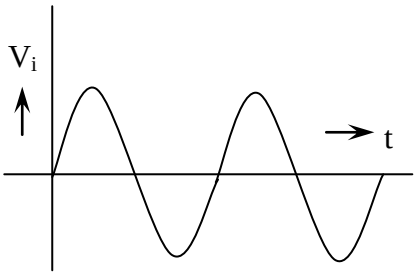


Figure:2.20. Input waveform

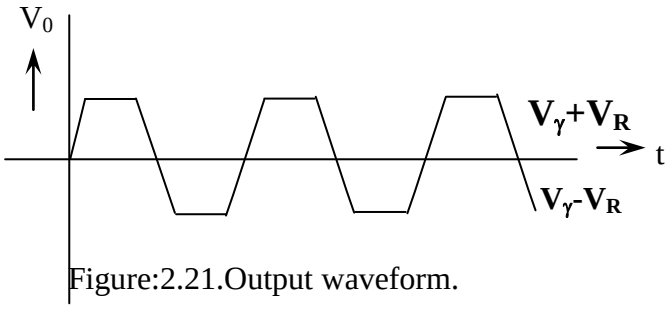


Figure:2.21.Output waveform.

DESIGN:

1. For positive clipping at 'V' volts reference select V_R = V.
2. For negative clipping at 'V' volts reference select V_R = V.
3. For clipping at two independent levels at V_1&V_2 reference voltages select $V_{R1} = V_1$, $V_{R2} = V_2$ and $V_{R2} > V_{R1}$.

PROCEDURE:

1. Connect the circuit as shown in the figure 1.
2. Connect the function generator at the input terminals and CRO at the output terminals of the circuit.
3. Apply a sine wave signal of frequency 1KHz at the input and observe the output waveforms of the circuits.
4. Repeat the procedure for figure 3, 5, 7, 9, 11 and 13.

RESULT:

Vγ =

Clipping circuits for different reference voltages are studied.

VIVA QUESTIONS:

1. Define clipping? Describe (i) Positive clipper (ii) Biased clipper (iii) Combination clipper.
2. Define clamping?
3. Define peak inverse voltage of diode?
4. What are the other names for the clamper?
5. What are the applications of clampers?

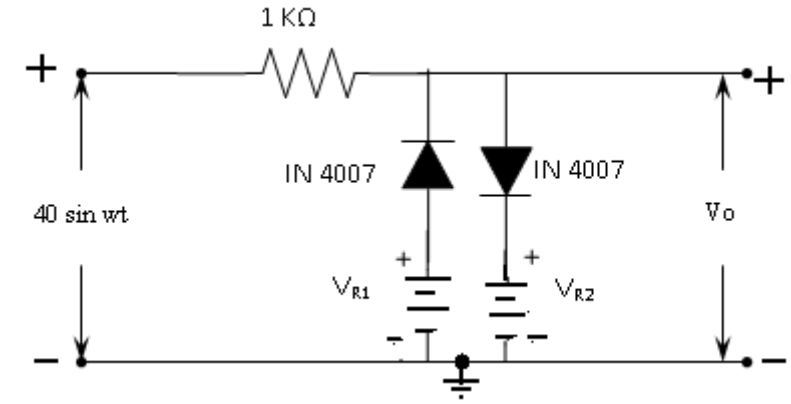
6. Explain the clipping process?
7. Discuss the differences between shunt and series clipper.
8. Draw the o/p waveforms for
   i) Series diode +ve clipper
   ii) Series diode -ve clipper
   iii) Shunt diode +ve clipper
   iv) Shunt diode -ve clipper
   v) Two level clipper
9. Draw the o/p wave forms for
   i). +ve clamper
   ii). –ve clamper

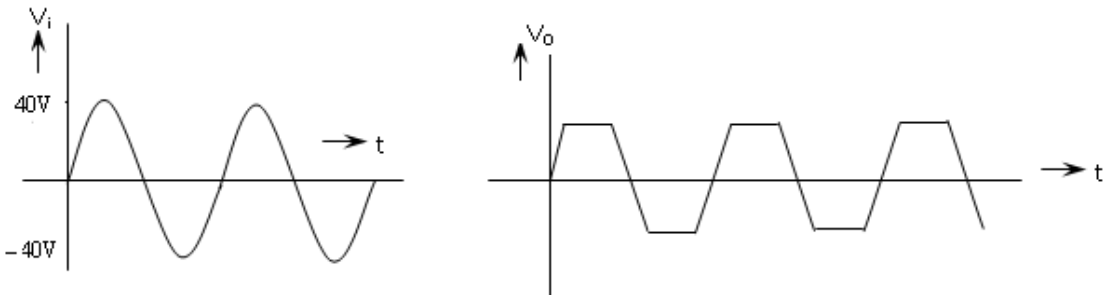
APPLICATIONS:

1. Used in radars, digital computers and other electronic systems for removing unwanted portions of the input signal voltages above or below a specified level.
2. used in radio-receivers for communication circuits where noise pulses that rise well above the signal amplitude are clipped down to the desired level.

DESIGNING PROBLEM:

1. The circuit shown in Figure below is used to "square" a 1Khz input sine wave whose peak is 40V. It is desired that the output voltage waveform be flat for 90% of time. Find the values if V_{R1} and V_{R2}. Assume ideal diodes. At what value of input will the waveform be clipped?





Input waveform Output waveform.

GRAPH

GRAPH

EXPERIMENT NO:3 Date:

NON LINEAR WAVE SHAPING – CLAMPERS

AIM : *To get positive and negative clamping for sinusoidal and Square wave inputs.*

COMPONENTS REQUIRED:

1. Resistors - 1kΩ
2. IN4007 Diode
3. Capacitor -10μF

APPARATUS REQUIRED:

1. Bread board
2. Function generator
3. CRO
4. Power supply 0-30V
5. Connecting Wires.

THEORY:

Clamping Circuit

"A clamping circuit is one that takes an input waveform and provides an output that is a faithful replica of its shape but has one edge tightly clamped to the zero voltage reference point".

There are various types of Clamping circuits, which are mentioned below:

1. Positive Clamping Circuit.
2. Negative Clamping Circuit.
3. Positive Clamping with positive reference voltage.
4. Negative Clamping with positive reference voltage.
5. Positive Clamping with negative reference voltage.
6. Negative Clamping with negative reference voltage.

Negative Clamping Circuit

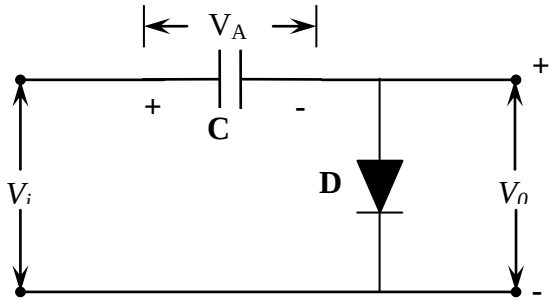


Figure3.1. Negative Clamping Circuit

The input signal is a sinusoidal which begins at t=0. The capacitor C is charged at $t = 0$. The waveform across the diode at various instant is studied.

During the first quarter cycle the input signal rises from zero to the maximum value V_m. The diode being ideal, no forward voltage may appear across it. During this first quarter cycle the capacitor voltage $V_A = V_i$. The voltage across C rises sinusoidally, the capacitor is charged through the series combination of the signal source and the diode. Throughout this first quarter cycle the

output V_0 has remained zero. At the end of this quarter cycle there exists across the capacitor a voltage $V_A = V_m$.

After the first quarter cycle, the peak has been passed and the input signal begins to fall, the voltage V_A across the capacitor is no longer able to follow the input voltage. For in order to do so, it would be required that the capacitor discharge, and because of the diode, such a discharge is not possible. The capacitor remains charged to the voltage $V_A = V_m$, and, after the first quarter cycle the output is $V_0 = V_i - V_m$. During succeeding cycles the positive excursion of the signal just barely reaches zero. The diode need never again conduct, and the positive extremity of the signal has been clamped to zero. The average value of the signal is $-V_m$.

Positive Clamping Circuit:

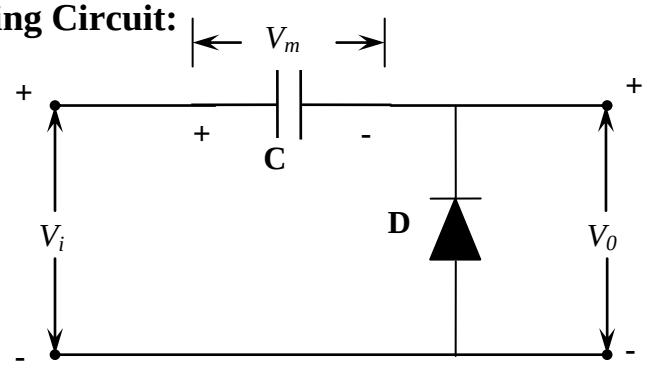


Figure3.2. Positive Clamping Circuit

It is also called as negative peak clamper, because this circuit clamps at the negative peaks of a signal.

Let the input signal be $V_i = V_m \sin\omega t$. When V_i goes negative, diode gets forward biased and conducts. The capacitor charges to voltage V_m, with polarity as shown. Under steady state condition, the positive clamping circuit is given as,

$$V_0 = V_i - (-V_m)$$

$$\boxed{V_0 = V_i + V_m} \qquad \text{Eq.1}$$

During the negative half cycle of V_i, the diode conducts and C charges to $-V_m$ volts, i.e., the negative peak value. The capacitor cannot discharge since the diode cannot conduct in the reverse direction. Thus the capacitor acts as a battery of $-V_m$ volts and the output voltage is given by equation.1 above. It is seen for figure 2, that the negative peaks of the input signal are clamped to zero level. Peak-to-peak amplitude of output voltage $2V_m$, which is the same as that of the input signal.

Negative Clamping with Positive Reference Voltage

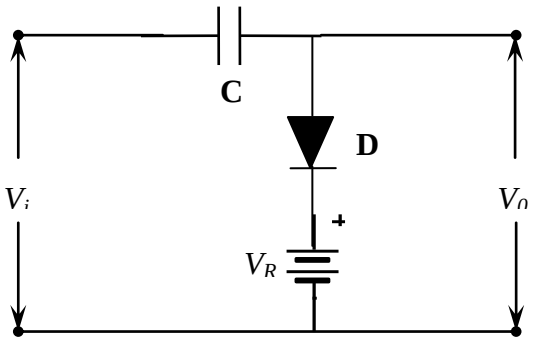


Figure:3.3. Negative Clamping with Positive Reference Voltage

Since V_R is in series with the output of negative clamping circuit, now the average value of the output becomes $(-V_m + V_R)$.
Similarly, the average of

i) Negative clamping with negative reference voltage is $(-V_m + V_R)$.
ii) Positive clamping is $+V_m$.
iii) Positive clamping with positive reference voltage is $V_m + V_R$.
iv) Positive clamping with negative reference voltage is $V_m - V_R$.

Clamping Circuit Theorem:

It states that for any input waveform the ratio of the areas under the output voltage curve in forward direction to that in the reverse direction is equal to the ratio (R_f / R).

$$\frac{A_f}{A_r} = \frac{R_f}{R}.$$

Where A_f = area of the output wave in forward direction.
A_r = area of the output wave in reverse direction.
Rf and *R* are forward and reverse resistances of the diode.

I. <u>Negative Clamping</u>

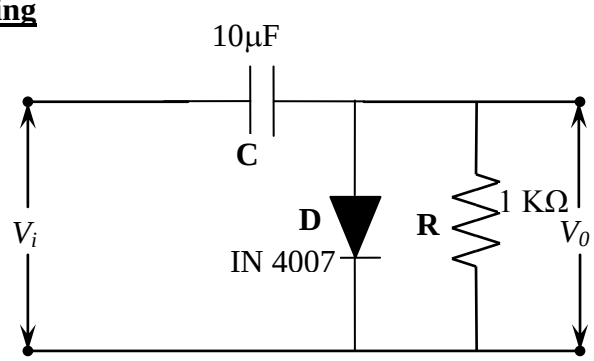


Figure:3.4. Negative Clamping

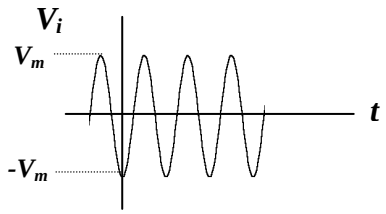


Figure:3.5.Input waveform

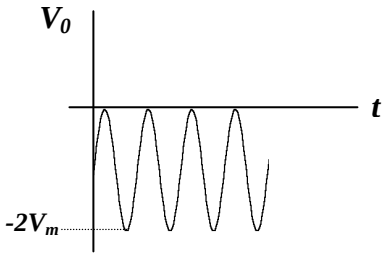


Figure:3.6. Output waveform.

II. Negative Clamping with Positive Reference Voltage.

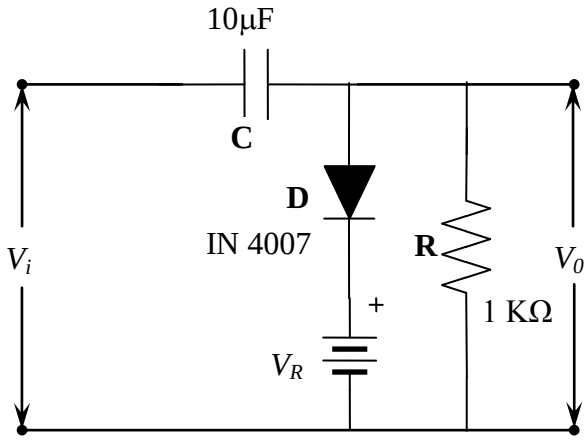


Figure:3.7. Negative Clamping with Positive Reference Voltage

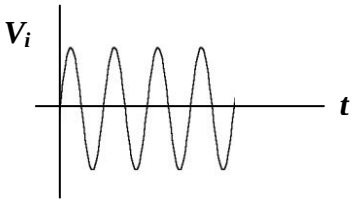


Figure:3.8.Input waveform

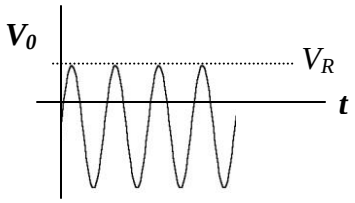


Figure:3.9.Output waveform.

III. Negative Clamping with Negative Reference Voltage.

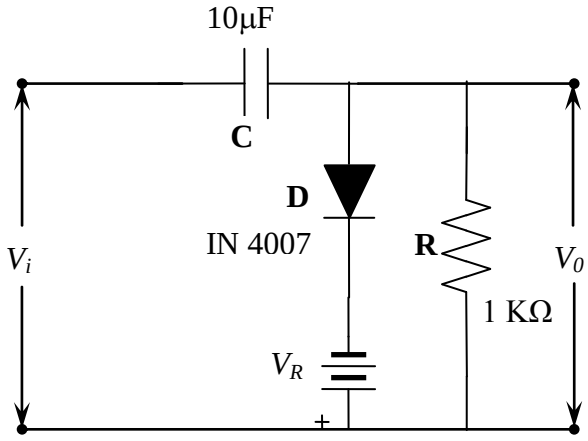


Figure:3.10. Negative Clamping with Negative Reference Voltage

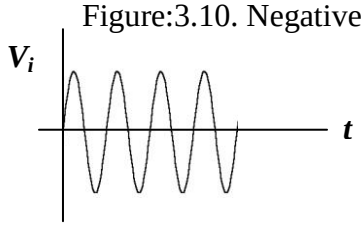


Figure:3.11.Input waveform

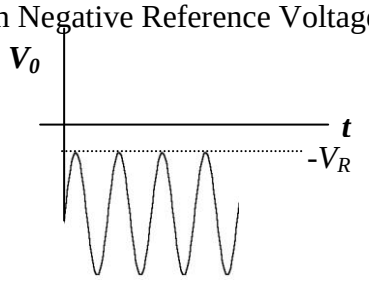


Figure:3.12 Output waveform.

IV. <u>Positive Clamping.</u>

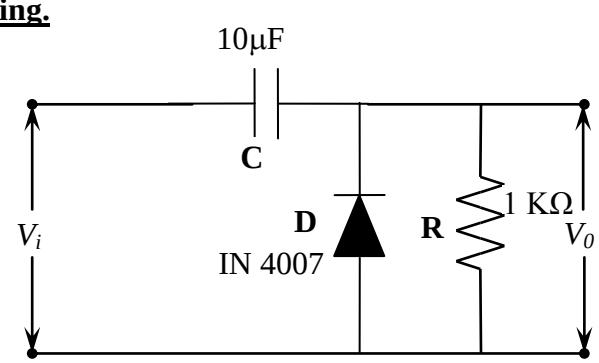


figure:3.13.Positive Clamping

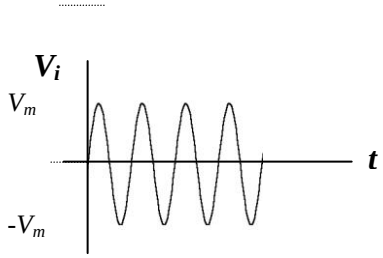


Figure:3.14.Input waveform

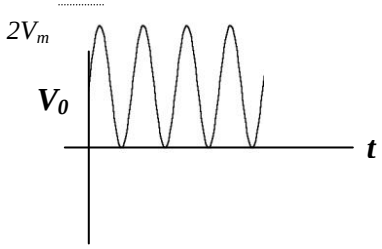


Figure:3.15. Output waveform.

V. <u>Positive Clamping with Negative Reference Voltage</u>.

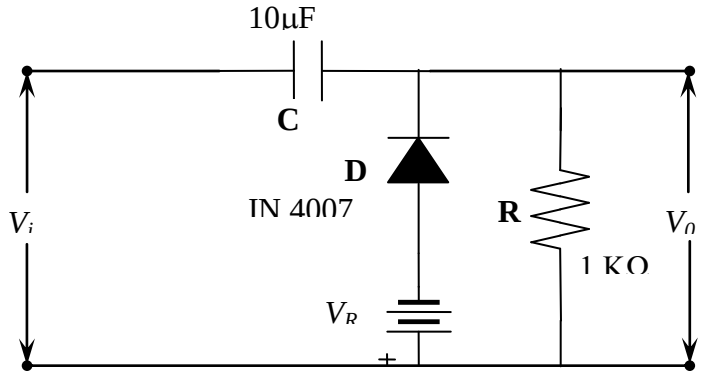


Figure: 3.16. Positive Clamping with Negative Reference Voltage

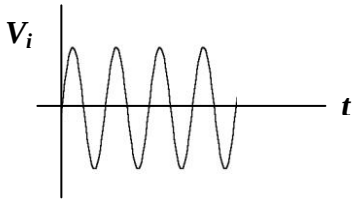


Figure:3.17.Input waveform

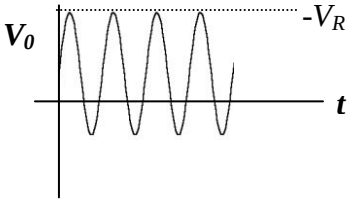


Figure:3.18. Output waveform.

VI. <u>Positive Clamping with Positive reference Voltage.</u>

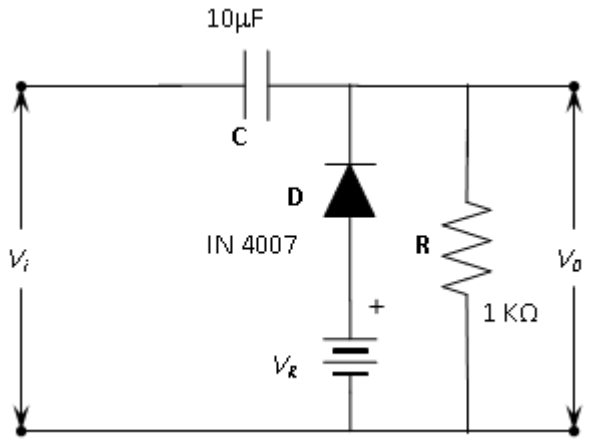


Figure: 3.19. Positive Clamping with Positive reference Voltage

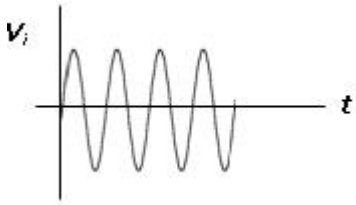


Figure:3.20.Input waveform

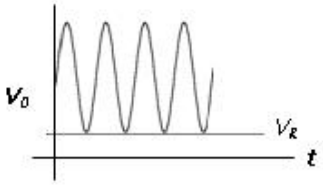


Figure:3.21. Output waveform.

PROCEDURE:

1. Connect the circuit as shown in the figure 3.
2. Connect the function generator at the input terminals and CRO at the output terminals of the circuit.
3. Apply a sine wave and square wave signal of frequency 1kHz at the input and observe the output waveforms of the circuits in CRO.
4. Repeat the above procedure for the different circuit diagram as shown inf figure 5, 7, 9, 11 and 13.

RESULT: The clamping voltages for positive and negative clamping circuits are noted.

QUESTIONS:

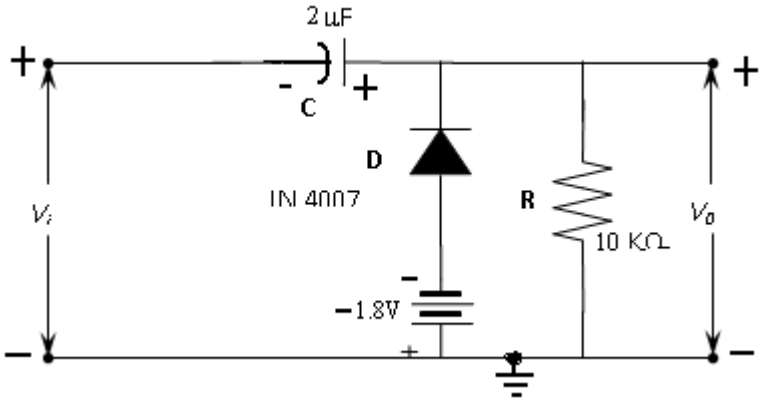
a. Explain the operation of a clamping circuit for a square wave input.
b. Differentiate the clippers with clampers.
c. Give the applications of clampers.

APPLICATIONS:

1. Used in T.V receivers as dc restorers.
2. Used in test equipment, radar systems, electronic countermeasure systems, and sonar systems.

DESIGNING PROBLEM:

1. Design a diode clamper to restore a dc level of +5V to an input signal of peak-to-peak valve 15V. Assume the drop across the diode as 0.7V.


2 uF
C
D
IN 4007
R
10 KΩ
V_i
V_o
-1.8V

GRAPH

GRAPH

GRAPH

EXPERIMENT NO:4 Date:

TRANSISTOR AS A SWITCH

AIM : *To design and observe the performance of a transistor as a switch.*

COMPONENTS REQUIRED :

1. Resistors
2. LED
3. 2N2369 Transistor

APPARATUS REQUIRED:

1. Bread board
2. Function generator 1Hz- 1MHz
3. CRO 1Hz -20MHz
4. Power supply 0-30V

THEORY:

Transistors are widely used in digital logic circuits and switching applications.in these applications the voltage levels periodically alternate between a "LOW" and a "HIGH" voltage,such as 0v and +5v.in switching circuits ,a transistor is operated at cutoff for the off condition ,ans the saturation for the ON condition.the active linear region is passed through abruptly from cutoff to saturation or vice versa.in cut off region,both the transistor junctions between Emitter and Base and the junction between base and collector are reverse biased and only the reverse current which is very small and practically neglected,flows in the transistor.in saturation region both junctions are in forward bais and the values of V_{ce} (sat) and $V_{be(sat)}$ are small.

CIRCUIT DIAGRAM:

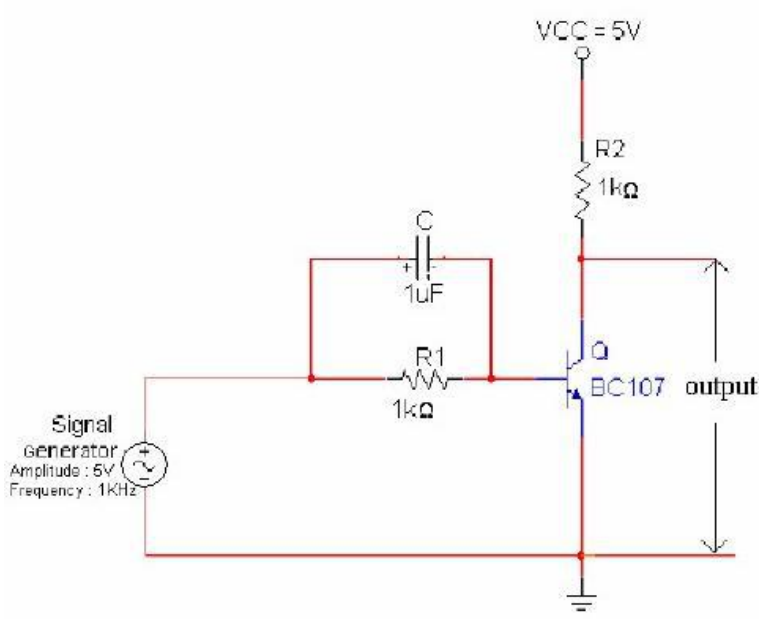


Figure 4.1.circuit diagram of transistor working as a switch

PROCEDURE:

1.Connect the circuit as per circuit diagram.
2.Obtain a constant amplitude square wave from function generator of 5V p-p and give the signal as input to the circuit.
4.Draw the input and output waveforms
3.Observe the output waveform and note down its voltage amplitude levels.

MODEL WAVEFORM:

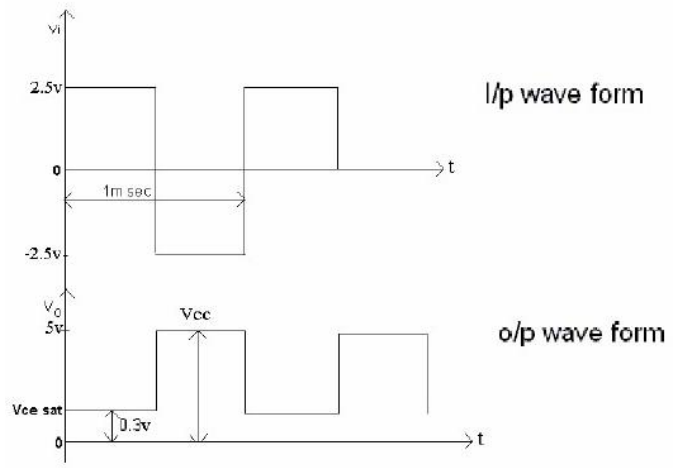


Figure4.2.model waveforms

OBSERVATIONS:

	Vbe	Vce	Vcb
when the trasistor is ON			
when the transistor is OFF			

THEORETICAL CALCULATIONS:

When Vi=+2.5v,the transistor goes into saturation region .
So Vout=Vce (sat)=0.3V.
When Vi=-2.5v,the transistor goes into cutoff region .
So Vout=Vcc =5v

PRECAUTIONS:

1. Connections should be made carefully.
2. Verify the circuit before giving supply voltage.
3. Take reading without any parallax error.

RESULT:

Switchinng charactristics of a transistor are observed.

1. Discuss the advantages of an electronic switch over a mechanical or electro mechanical switch.
2. What is a switching circuit?
3. Explain the terms collector leakage current and saturation collector current.

VIVA QUESTIONS:

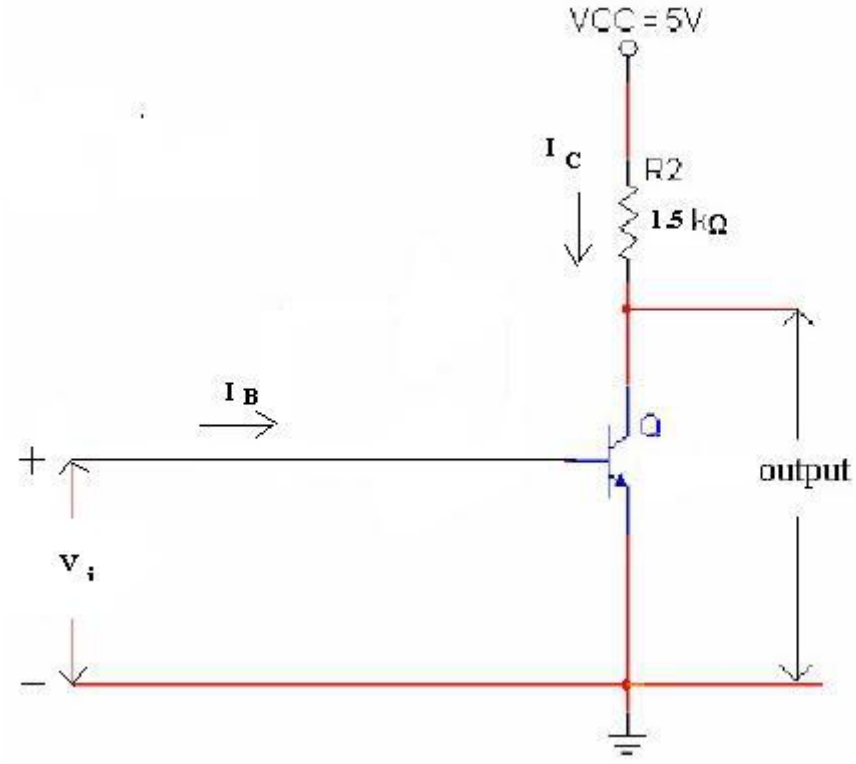
1. What are the three regions of operation of a transistor.
2. What do you mean by time delay of a transistor.
3. When does the transistor acts as (a) a closed switch (b) an open switch.
4. How are the junctions of a transistor biased for a active region operation.
5. What do you mean by turn ON time of a transistor.

APPLICATIONS:

Used for switching purpose in analog and digital communication networks**.**

DESIGNING PROBLEM:

1. For a CE transistor circuit with Vcc = 15V, Rc = 1.5KΩ. Calculate the transistor power dissipation (a) at cutoff and (b) at saturation. The figure is shown below.



GRAPH

EXPERIMENT NO: 5 **Date:**

STUDY OF LOGIC GATES

AIM : *To study the various logic gates by using discrete components.*

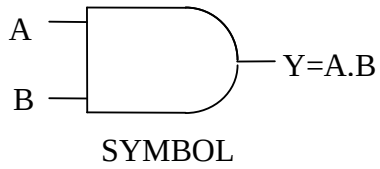
COMPONENTS REQUIRED :

1. Resistors - 1kΩ -1, 10 kΩ -2
2. IN4007 Diode – 2 no
3. Transistor 2N2369

APPARATUS REQUIRED:

1. Power supply 0-30V
2. Bread board
3. Connecting wires

THEORY:



SYMBOL

TRUTH TABLE

A	B	Y
0	0	0
0	1	0
1	0	0
1	1	1

AND GATE:

The AND gate as a high output when all the inputs are high the figure 1 shows one way to build the AND gate by using diodes.

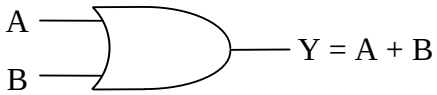
Case 1: When both A and B are low then the diodes are in the saturation region then the supply from V_{CC} will flow to the diodes then the output is low.

Case 2: When A is low and B is high then diode D_1 will be in the saturation region and D_2 will be in the Cut-off region, then the supply from V_{CC} will flow through diode D_1 then the output will be low.

Case 3: When A is high, B is low the diode D_1 will be in the Cut-off region and diode D_2 will be in saturation region then the supply from V_{CC} will flow through the diode D_2, therefore the output will be low.

Case 4: When both the A and B are high then the two diodes will be in Cut-off region therefore the supply from V_{CC} will flow through V_{out} then V_{out} is high.

OR GATE:



SYMBOL

TRUTH TABLE

A	B	Y
0	0	0
0	1	1
1	0	1
1	1	1

An OR gate has two or more inputs but only one output signal. It is called OR gate because the output voltage is high if any or all the inputs are high.

The figure 2 shows one way to build OR gate (two inputs) by using diodes.

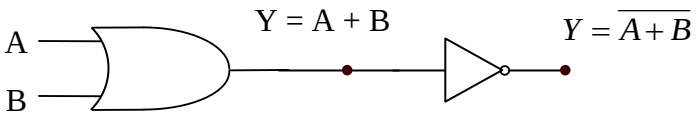
Case 1: When A and B are low then the two diodes D_1 and D_2 are in Cut-off region. Then the V_{out} is low.

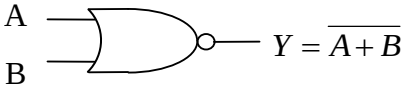
Case 2: When A is low and B is high then the diode D_1 is in Cut-off region and diode D_2 is in saturation region, then the V_{out} is high.

Case 3: When A is high and B is low then the diode D_2 is in saturation region and diode D_1 is in Cut-off region, then the V_{out} is high.

Case 4: When both A and B are high the diodes D_1 and D_2 are in saturation region then the output V_{out} is high.

NOR GATE:





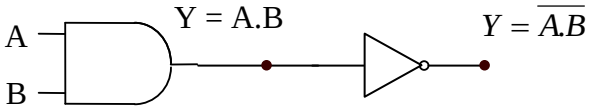
Symbol

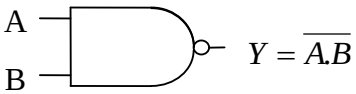
TRUTH TABLE

A	B	Y
0	0	1
0	1	0
1	0	0
1	1	0

NOR gate is referred to a NOT OR gate because the output is $Y = \overline{A+B}$. Read this as Y = NOT A OR B or Y = compliment of the A OR B. the circuit is in an OR gate followed by a NO gate OR inverter. The only to get high output is to have both inputs low.

NAND GATE:





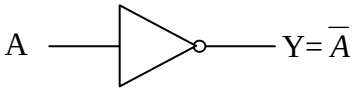
Symbol

TRUTH TABLE

A	B	Y
0	0	1
0	1	1
1	0	1
1	1	0

NAND gate is referred to as NOT AND GATE because the output is $Y = \overline{A.B}$ read this as Y = NOT A AND B or Y = Compliment of A AND B. By this gate the output is low when all the inputs are high.

NOT GATE:



Symbol

TRUTH TABLE

A	Y
0	1
1	0

The Inverter or NOT gate is with only one input and only one output. It is called inverter because the output is always opposite to the input.

The figure5 shows the one way to build inverter circuit by using transistor (CE mode) when the Vin is low then the transistor will be in the Cut-off region. Then the supply from VCC will flow to Vout. Then the Vout is high. When Vin is high then the transistor is in the saturation region then the supply from VCC will flow through the transistor to the ground, then the Vout is low.

CIRCUIT DIAGRAM :

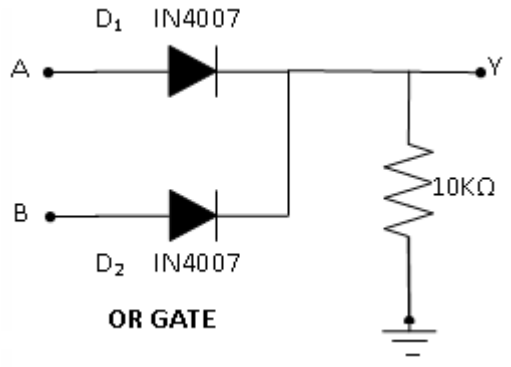


Figure 5.1.OR gate

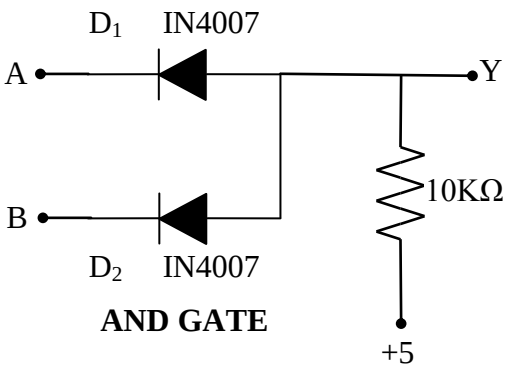


Figure 5.2.AND Gate

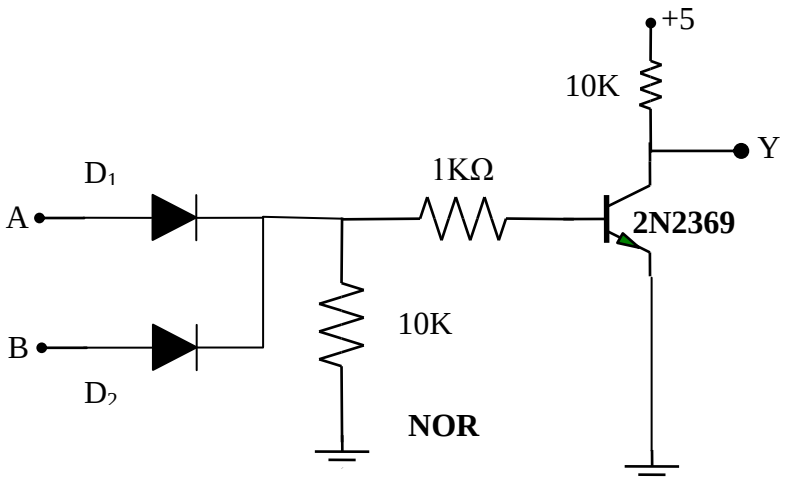


Figure 5.3.NOR Gate

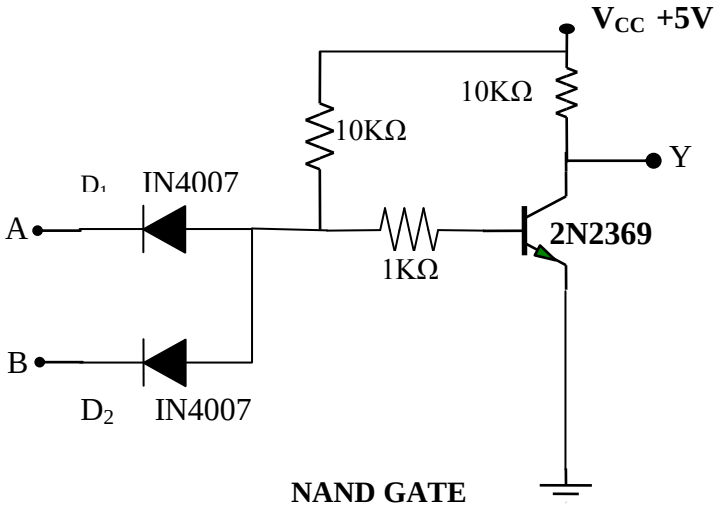


Figure 5.4.NAND Gate

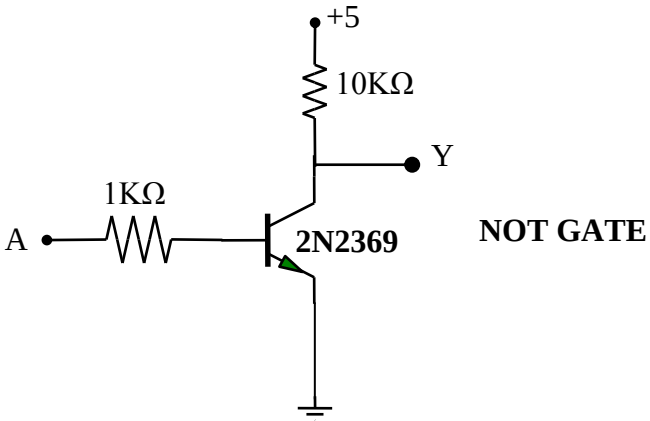


Figure 5.5.NOT Gate

PROCEDURE:

1. Connect the circuit as shown in figures above.
2. Verify the truth tables of various gates for different conditions of inputs.
3. Repeat the steps 1&2 for figures 2, 3, 4 & 5.

TRUTH TABLES:

AND GATE

A	B	Y
0	0	
0	1	
1	0	
1	1	

OR GATE

A	B	Y
0	0	
0	1	
1	0	
1	1	

NAND GATE

A	B	Y
0	0	
0	1	
1	0	
1	1	

NOR GATE

A	B	Y
0	0	
0	1	
1	0	
1	1	

NOT GATE

A	Y
0	1
1	0

Where 5V is represented by logic 1.

RESULT: Different logic gates are studied and their truth tables are verified .

QUESTIONS:

1. Realize AND, OR, NOT gates using NAND & NOR gates
2. Why NAND & NOR gates are called universal gates.

APPLICATIONS:

1. Arithmetic adders, multiplexers, and some kinds of memories.
2. Computers,arger components like Internet routers, digital television sets, your cell phone, tablet, Blue Ray player, or iPad.

GRAPH

EXPERIMENT NO:6 **Date:**

STUDY OF FLIP-FLOPS

AIM: *to verify the truth table of SR latch ,JK flipflop ,JK master slave flipflop,T-flipflop and D-flipflop.*

COMPONENTS REQUIRED:

1. IC 74LS00
2. IC 74LS10
3. IC 74LS04.
4. Digital Trainer Kit.

THEORY:

RS FLOP-FLOP:

Consider the logic Symbol for RS flip-flop shown in Fig. Notice that the RS flip-flop has two inputs, labeled S and R. The two outputs are labeled Q and Q . Note that the outputs are always opposite, or complementary in the flip-flops.

Fig 6.1 shows the timing diagram for an RS flip-flop. Notice that the output Q goes high whenever R goes low; and the output Q goes LOW whenever S goes LOW. The logic levels (0,1) are on the left side of the wave forms. The output Q 'holds' whenever the both inputs are high.

Figure 6.1: Logic symbol for RS Flipflop

CLOCKED RS FLIP-FLOP:

We know that the flip-flops are synchronous bistable devices. The term synchronous indicates that the output changes its state only at a specified point on a triggering input called the 'clock' i.e. changes in the output occur in synchronization with the clock.; By adding gates to the inputs of the basic circuit, the flip-flop can be made to respond to input levels during the occurrence of a clock pulse. Note that the clock signal is a square wave signal and the signal prevents the flip-flops from changing the states until the right time occurs. The clocked RS flip-flop using NAND gates are shown in Fig.6.3

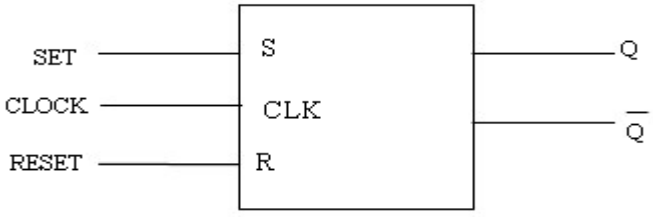
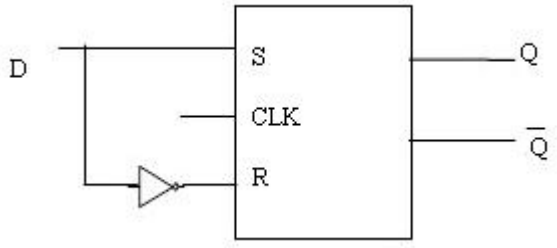


Figure 6.2: Logic symbol for Clocked RS Flipflop

INPUTS			OUTPUTS		Mode of Operation
CLK	R	S	Q	$\overline{Q}$	
0 0 0 0 1	0 0 1 1 0	0 1 0 1 0	NO CHANGE		HOLD
1 1	0 1	1 0	1 0	0 1	SET RESET
1	1	1	Do not Use		prohibited

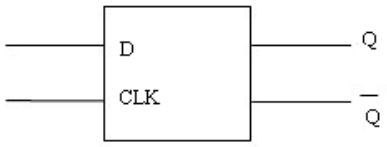
Table 6.1. Truth Table For Clocked RS Flipflop



D (DELAY) FLIP-FLOP:

The D (Delay) flip-flop is used for storing the information. It is basically an RS flip-flop with an inverter in the R input. Fig. 6.3 shows a clocked D flip-flop. NAND gates 1 and 2 from a basic RS flip-flop and gates 3 and 4 modify it into a clocked RS flip-flop. The D input is to the S input and its complement through gate 5 is applied to the R input. The D flip-flop is often called a 'delay flip-flop' The word '.delay' describes what happens to the data or information at input D. In other words, the data, i.e. 0 or 1 at the input D is delayed by one clock pulse from getting the output Q.

Figure 6.3: Modification of clocked RS Flip-Flop into D Flip-Flop



Inputs		Outputs	
CLK	D	Q	$\overline{Q}$
0 0	0 1	No Change	
1 1	0 1	0 1	1 0

Figure 6.4: Logic symbol and Truth Table for D Flip-flop

JK- FLIPFLOP:

The JK flip-flop has the features of all other flip-flops, and hence it can also be considered as 'Universal' flip-flop. This JK flip-flop is a refinement of the RS flip-flop.
The indeterminate state (when R=S=1) of the RS type is defined in the JK type. In that condition the state of the output is changed; i.e. the complement of the previous state is available. In other words, if the previous state of the output Q is 0; it becomes 1; and vice versa. This can be written as

Q_{n+1} = complement $\{Q_n\}$.

The logic diagram of a clocked JK flip-flop is shown in Fig.6.5. Inputs J and K behave like inputs S and R to set and reset the flip-flop. Note that in a JK flip-flop, the letter j is for set and the letter K is for reset.

Fig.6.5 shows the logic symbol for JK flip-flop and its truth table is shown in the Table 6.2. Note that the RS flip-flop is converted into JK flip-flop by making S= □J**.**Q' and

R =□ K'**.**Q

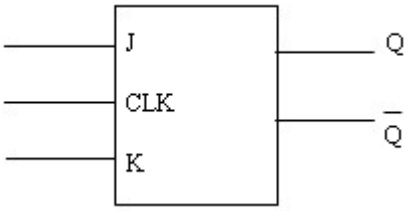


Figure 6.5. Logic Symbol for JK Fliop flop

INPUTS			OUTPUTS	
CLK	K	J	Q	$\overline{Q}$
0 0 0 0 1	0 0 1 1 0	0 1 0 1 0	NO CHANGE	
1 1	0 1	1 0	1 0	0 1
1	1	1	Toggle	

Table 6.2. Truth Table For JK-Flipflop

JK Master Slave Flip flop:

In the operation of a JK flip-flop, it is very difficult to satisfy the requirements, which should be fulfilled to avoid the 'racing condition' when both the data inputs are at High. The, practical approach to overcome this problem is to use 'Master-slave JK flip-flop' The Logic diagram of a master-slave JK flip-flop is shown in Fig.6.6 which contains two clocked flop-flops.

One flip-flop serves as a MASTER and the other as a SLAVE; and overall circuit is known as 'Master-Slave flip-flop' Whenever the clock is HIGH, the master is active. According to the state of the data inputs, The output of the master is set or reset. At this stage, the slave is inactive and its output remains in the previous state. Whenever the input clock is LOW, then the master is inactive and the slave is active. Note that final output of the master-slave flip-flop is the output of the slave flip-flop. Hence the final output of the master-slave flip-flop is available at the end of a clock pulse.

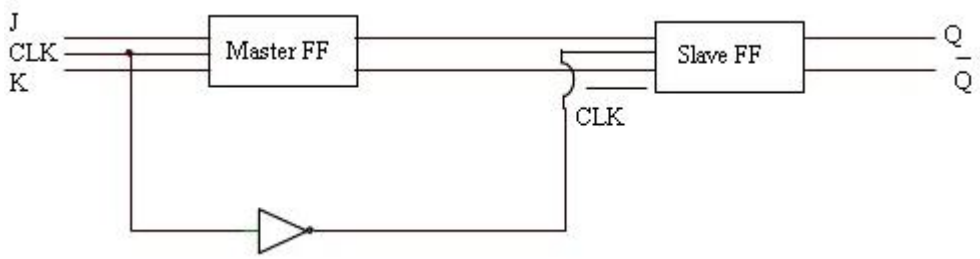
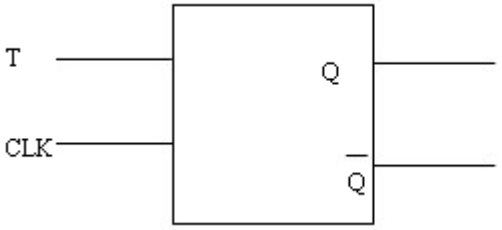


Figure 6.6: Logic Diagram Of Master-Slave J-K Flip-Flop

T(TOGGLE) FLIP FLOP:

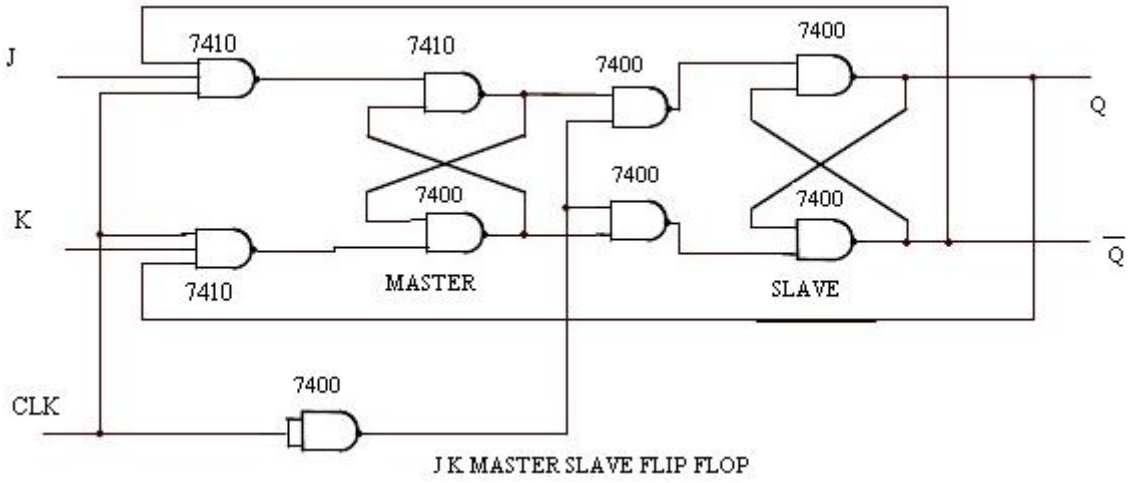
The single input version of the JK flip-flop is (toggle) flip-flop and it is obtained from a JK flip-flop if both inputs are tied together. The name T comes from the ability of the flip-flop to 'toggle' or change the state. Generally T flip-flop IC' are not available. It can be realized using JK, SR, or D flip-flop.

Fig 6.7. shows the logic diagram of a clocked T flip-flop; which has only one input referred to as T-input.

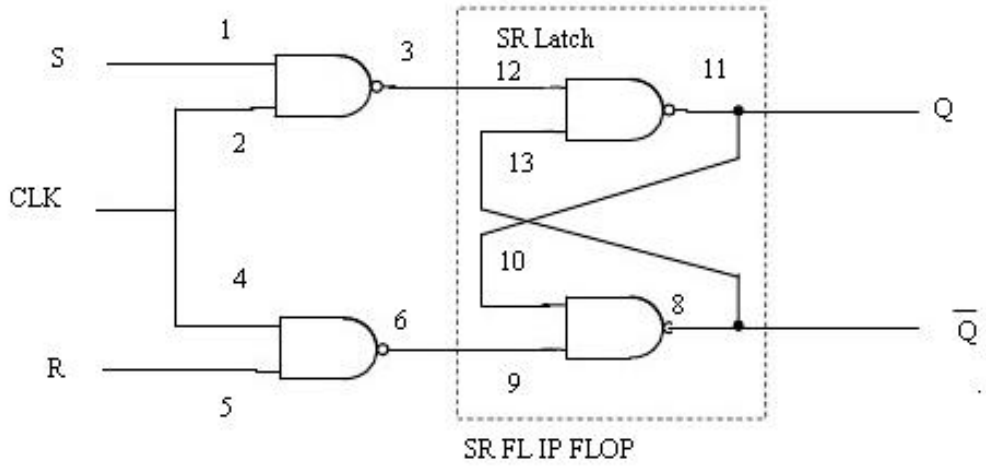


CLK	I/P (T)	O/P (Q)
0	0	Qn
1	0	No Change
0	1	No Change
1	1	Toggle

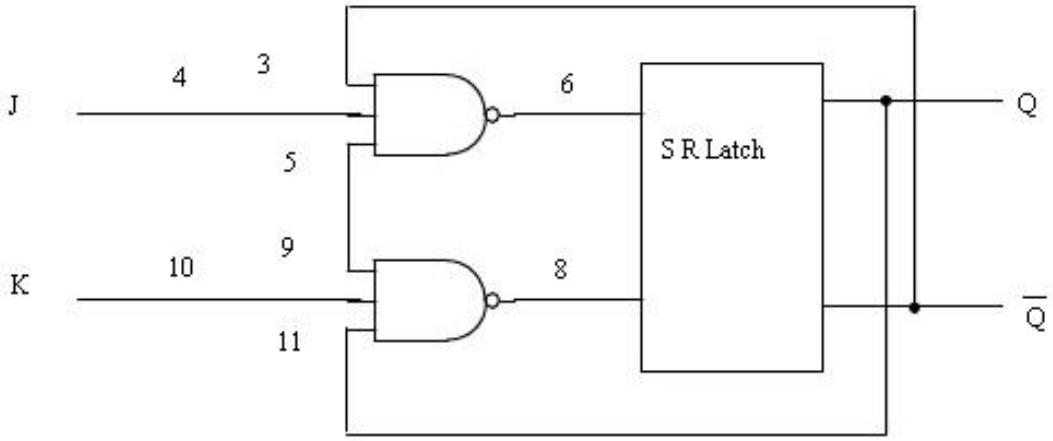
Figure 6.7: Logic symbol and Truth Table for T Flip-flop



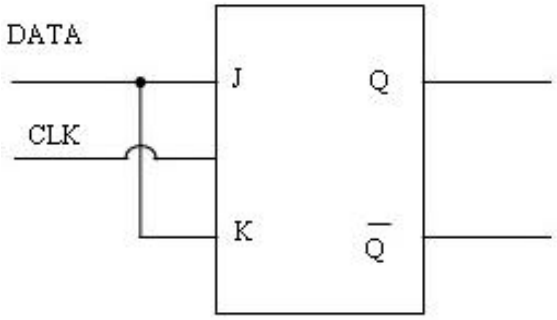
CIRCUIT DIAGRAMS:



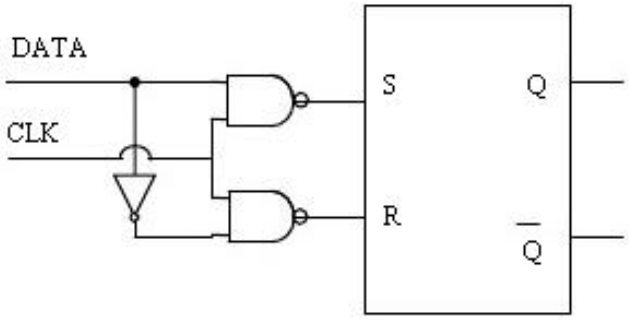
SR FL IP FLOP



J K Flip Flop



T Flip flop



D Flip flop

PROCEDURE:

1. Connect the circuit diagram as shown in the diagram.
2. Verify the truth table of the SR latch.
3. Connect the JK flip-flop circuit using 3-input NAND gates and verify the truth table.
4. Verify the truth table for T and D Flip-flops.

5. Verify the truth table for J-K MASTER-SLAVE FLIP-FLOP

QUESTIONS:

1. What is a sequential logic circuit? How is it different from a combinational system?
2. What is a binary? What is a flip-flop?
3. Explain the meaning of set and reset?
4. Explain the need of flip-flop?
5. What basic purpose does all flip-flops serve?
6. Is it possible to toggle a SR flip-flop? Explain?
7. Explain the meaning of race – around problem? How does it affect the behavior of a flip-flop?
8. Explain how the race – around problem is overcome in the JK Master Slave flip flop
9. Explain how a shift register can be constructed from SR flip-flop?

APPLICATIONS:

1. Data Storage
2. Data Transfer
3. Counters
4. Frequency division

GRAPH

EXPERIMENT NO:7 **Date:**

BISTABLE MULTIVIBRATOR

AIM: *To design a fixed bias Bistable Multivibrator and to measure the stable state voltages before and after triggering.*

COMPONENTS REQUIRED:

1. Resistors 2.2KΩ-2
   15 KΩ-2
   100 KΩ-2
2. Transistors 2N2369 – 2

APPARATUS:

1. Bread board
2. Power supply 0-30V
3. CRO
4. Connecting wires

THEORY:

A bistable multivibrator has two stable output states. It can remain indefinitely in any one of the two stable states, and it can be induced to make an abrupt transition to the other stable state by means of suitable external excitation. It would remain indefinitely in this stable state, until it is again induced to switch into the original stable state by external triggering.

Bistable multivibrators are also termed as 'Binary's or Flip-flops'**.** A binary is sometimes referred to as ***'Eccles-Jordan Circuit'***.

Principle of Operation of bistable multivibrator.

Consider the circuit as shown in the figure.7.1. The transistor Q_1 and Q_2 are n-p-n transistors. They are coupled to each other as shown in figure 7.1. It is evident that the output of each transistor is coupled to the input of the other transistor. Since the transistors are identical, there quiescent currents would be the same, unless the loop gain is greater than unity. When I_1 increases slightly, the voltage drop across the collector resistance R_{C1} increases. Since V_{CC} is fixed, the voltage of point C decreases. This has the effect of decreasing the base current of Q_2. This, in turn, decreases the collector current of Q_2 viz. I_2 decreases, the voltage drop I_2R_{C2} decreases. Hence the voltage of point *D* increases.

Due to increase of V_D, the base current of Q_1 increases. This increases the collector current of Q_1 viz I_1. Thus I_1 further increases. I_1R_{C1} drop further increases, V_C further decreases, the base current of Q_2 further decreases, with the result that I_2 further decreases. Thus it can easily seen that if the collector current I_1 increases even marginally, I_2 would go on progressively decreasing and as a result, I_1 would progressively increase. Eventually I_2 would become practically zero, cutting off the transistor Q_2, at the same time transistor Q_1 would conduct heavily with the result that it would be driven into saturation. Thus Q_2 becomes OFF and Q_1 becomes ON. It can similarly be shown that if I_2 increases even marginally similar sequence of operation would result and ultimately Q_2 would be ON and Q_1 OFF. Thus when Q_1 is ON, Q_2 is OFF and when Q_1 is OFF Q_2 is ON.

CIRCUIT DIAGRAM:

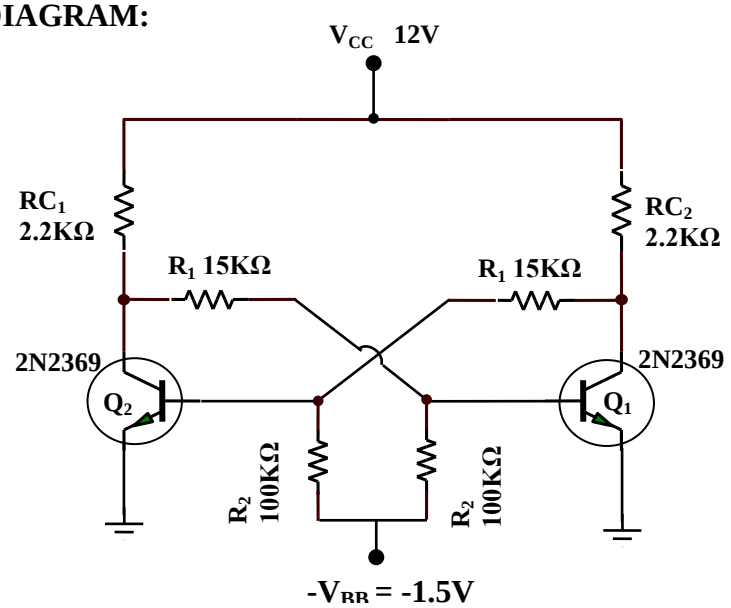
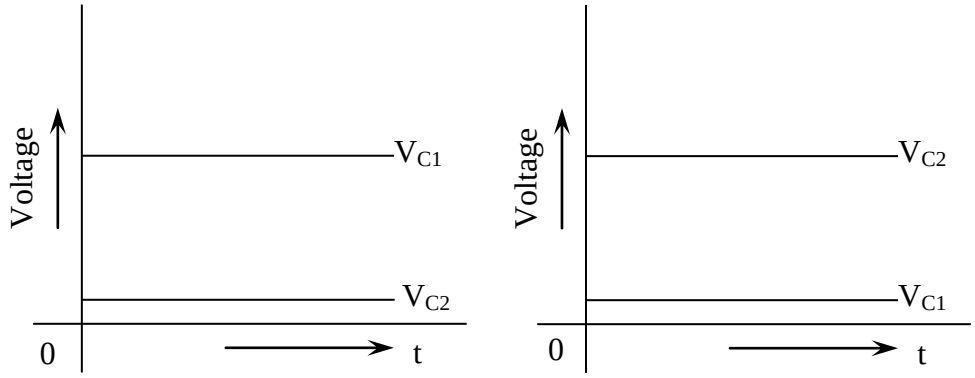


Figure 7.1.Bistable multivibrator

PROCEDURE:

1. Connect the circuit as shown in figure 2.
2. Observe the waveforms at V_{BE1}, V_{BE2}, V_{CE1}, V_{CE2}
3. Observe which transistor is in ON state and which transistor is in OFF state.
4. Apply –ve triggering at the base of the ON transistor and observe the voltages V_{C1}, V_{C2}, V_{B1}, and $V_{B2.}$
5. Apply + ve triggering at the base of the OFF transistor and observe the Voltages V_{C1}, V_{C2}, V_{B1}, $V_{B2.}$

EXPECTED WAVEFORMS:



Tabulations:

Stable State Voltages	VBE1	VBE2	VCE1	VCE2
Before Triggering				
After Triggering				

RESULT:

Hence we have designed a fixed bias Bistable Multivibrator and measured the Stable state voltages before and after triggering.

QUESTIONS:

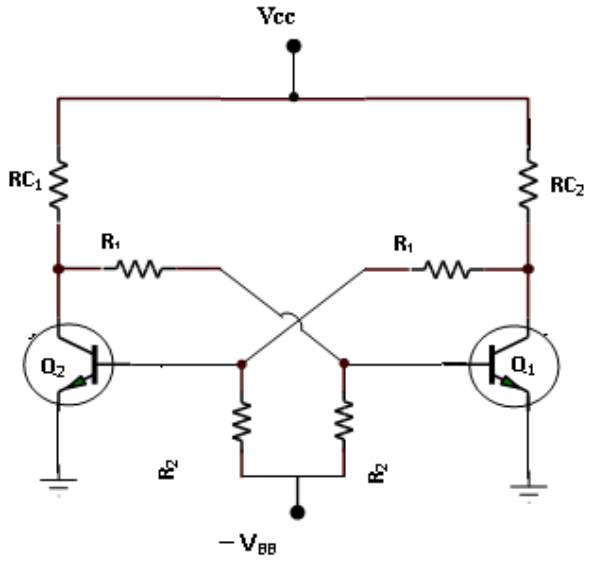
1. What is Multivibrator? Explain the principle on which it works? Why is it called a binary?
2. Explain the role of commutating capacitors in a Bistable Multivibrator?
3. Give the Application of a Binary.

APPLICATIONS:

1. used for the performance of many digital operations such as counting and storing binary information.
2. used in the generation and processing of pulse-type waveforms.

DESIGNING PROBLEM:

1. Silicon transistors with h_{fe}(min) equal to 20 are available. If $V_{cc}=V_{BB}$ = 10V, design the bistable multivibrator.



GRAPH

EXPERIMENT NO:8 **Date:**

ASTABLE MULTIVIBRATOR

AIM : *To design and test performance of an Astable Multivibrator to generate clockpulse for a given frequency.*

COMPONENTS REQUIRED:

1. Resistors 3.3KΩ-2
   10KΩ-2
2. Capacitors 0.1 μf - 2
3. Transistors 2N2369 – 2

APPARATUS :

1. CRO
2. Power supply 0-30V
3. Bread board
4. Connecting wires

THEORY:

An Astable multivibrator has two quasi-stable states, and it keeps on switching between these two states, by itself, No external triggering signal is needed. The astable multivibrator cannot remain indefinitely in any of these two states. The two amplifiers of an astable multivibrator are regeneratively cross-coupled by capacitor**.**

Principle:

A collector-coupled astable multivibrator using n-p-n transistor in figure 1. The working of an astable multivibrator can be studied with respect to the figure1.

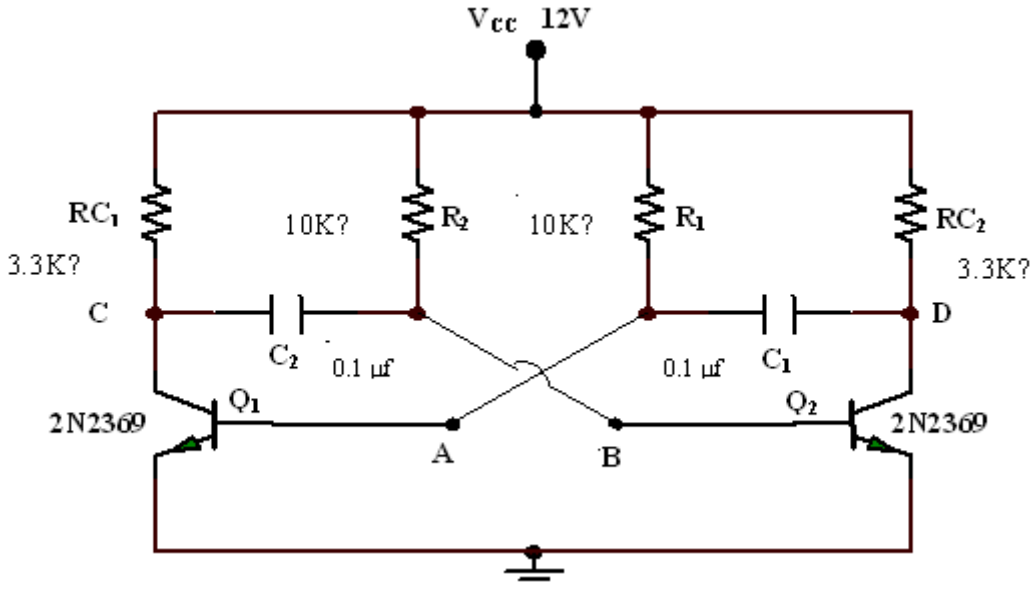


Figure 8.1: Astable Multivibrator

Let it be assumed that the multivibrator is already in action and is oscillating i.e., switching between the two states. Let it be further assumed that at the instant considered, Q_2 is ON and Q_1 is OFF.

i) Since Q_2 is ON, capacitor C_2 charges through resistor R_{C1}. The voltage across C_2 is V_{CC}.

ii) Capacitor C_1discharges through resistor R_1, the voltage across C_1 when it is about to start discharging is V_{CC}.(Capacitor C_1 gets charged to V_{CC} when Q_1 is ON).

As capacitor C_1 discharges more and more, the potential of point A becomes more and more positive (or less and less negative), and eventually V_A becomes equal to V_γ, the cut in voltage of Q_1. For $V_A > V_\gamma$, transistor Q_1 starts conducting. When Q_1 is ON Q_2 becomes OFF.

Similar operations repeat when Q_1 becomes ON and Q_2 becomes OFF.

Thus with Q_1 ON and Q_2 OFF**,** capacitor C_1 charges through resistor R_{C2} and capacitor C_2 discharges through resistor R_2. As capacitor C_2 discharges more and more , it is seen that the potential of point B becomes less and less negative (or more and more positive), and eventually V_B becomes equal to V_γ, the cut in voltage of Q_2. when $V_B > V_\gamma$, transistor Q_2 starts conducting. When Q_2 becomes On, Q_1 becomes OFF.

It is thus seen that the circuit keeps on switching continuously between the two quasi-stable states and once in operation, no external triggering is needed. Square wave voltage are generated at the collector terminals of Q_1 and Q_2 i.e., at points C and D.

DESIGN:

I_C max = 5 mA ; V_{CC} = 12 V; $V_{CE\,(SAT)}$ = 0.2V

$R_C = (V_{CC} - V_{CE(SAT)}) \ / \ I_{C\,MAX}$

Let C = 0.1 μf

and R= 10KΩ

$T = 0.69\,(R_1C_1+R_2C_2) = 0.69(2RC)$ $\quad \because \ (R_1=R_2 \ ; C_1=C_2)$

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

PROCEDURE:

1. Connect the circuit as shown in figure 1.
2. Observe the waveforms at V_{BE1}, V_{BE2}, V_{CE1}, V_{CE2} and find frequency.
3. Vary C from 0.01 to 0.001μF and measure the frequency at each step.
4. Keep the DC- AC control of the Oscilloscope in DC mode.

EXPECTED WAVEFORMS:

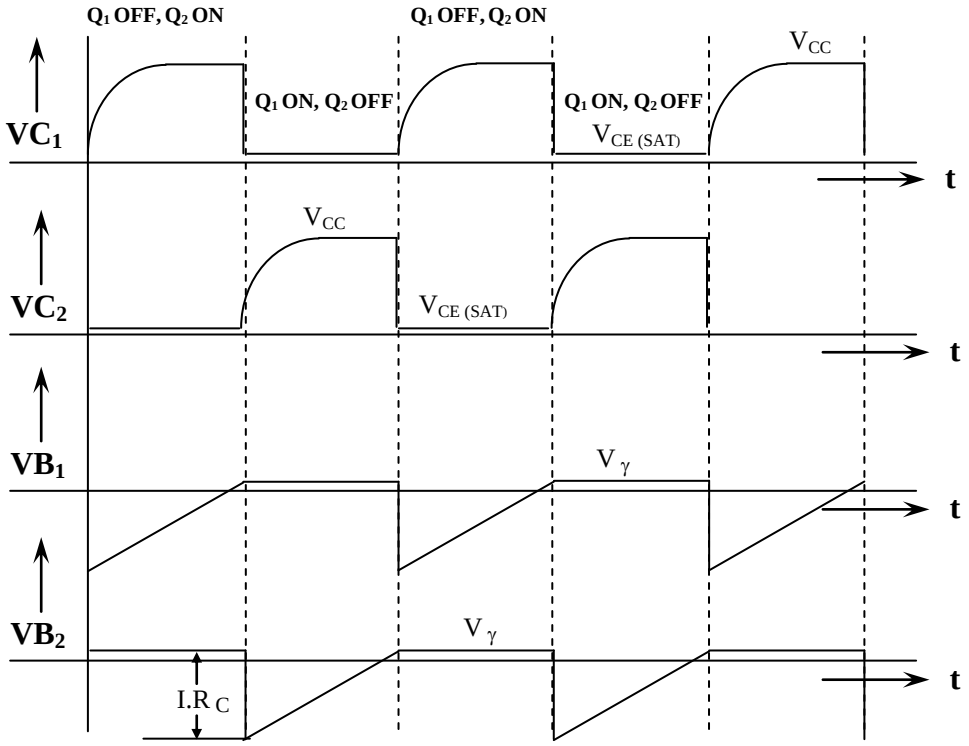


Figure 8.2. Expected Waveforms

RESULT:

T_{ON} = **T_{OFF} =** **$T(T_{ON} + T_{OFF})$ =**

Astable multivibrator is designed and its performance is tested.

QUESTIONS:

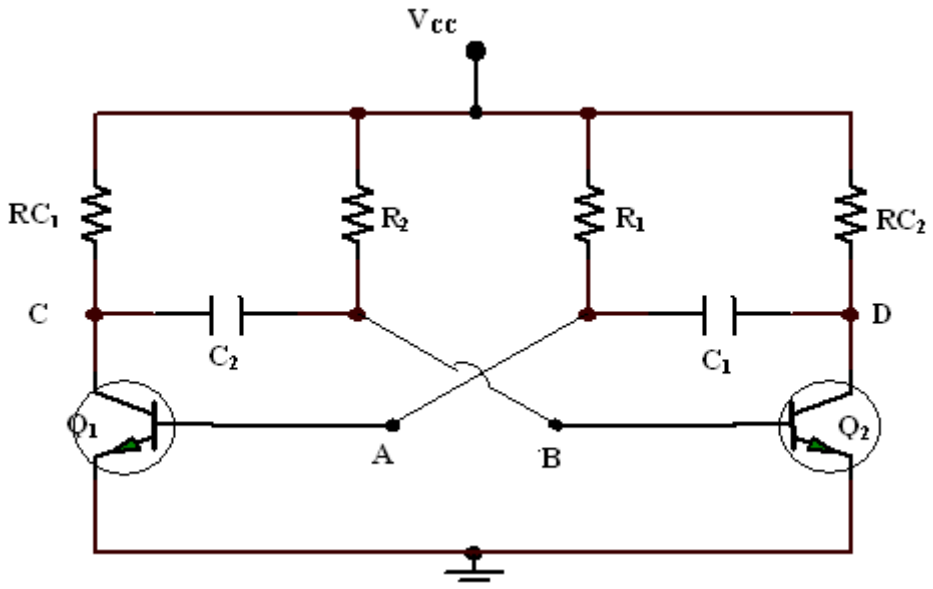
1. What is a switching circuit?
2. Justify that the Astable Multivibrator is a two stage RC coupled Amplifier using negative feedback. How does it generate square wave.
3. What is the difference between a switching transistor and an ordinary transistor?
4. What is the effect of slew rate on the working of an Op-amp Multivibrator?

APPLICATIONS:

1. Used anywhere, where we need the clock pulse train of low frequency, like a function generator.
2. Used to produce waveforms.
3. used in signal generation

DESIGNING PROBLEM:

1. Design an Astable multivibrator to generate a square wave of 1Khz.


Vcc
RC1
R2
R1
RC2
C
C2
C1
D
Q1
A
B
Q2

GRAPH

EXPERIMENT NO: 9 **Date:**

MONOSTABLE MULTIVIBRATOR

AIM : *To design and test performance of a monostable multivibrator to generate clock pulse for a given frequency. And obtain the waveforms.*

COMPONENTS REQUIRED:

1. Resistors 2.2KΩ-2
   10KΩ-1
   1KΩ-2
   1.5 KΩ-1

2. Capacitors- 0.1µF
   1 µF
3. Transistors 2N2369 – 2
4. Diode QA79-1

APPARATUS REQUIRED:

1. CRO
2. Power supply 0-30V
3. Bread board
4. Connecting wires

CIRCUIT DIAGRAM:

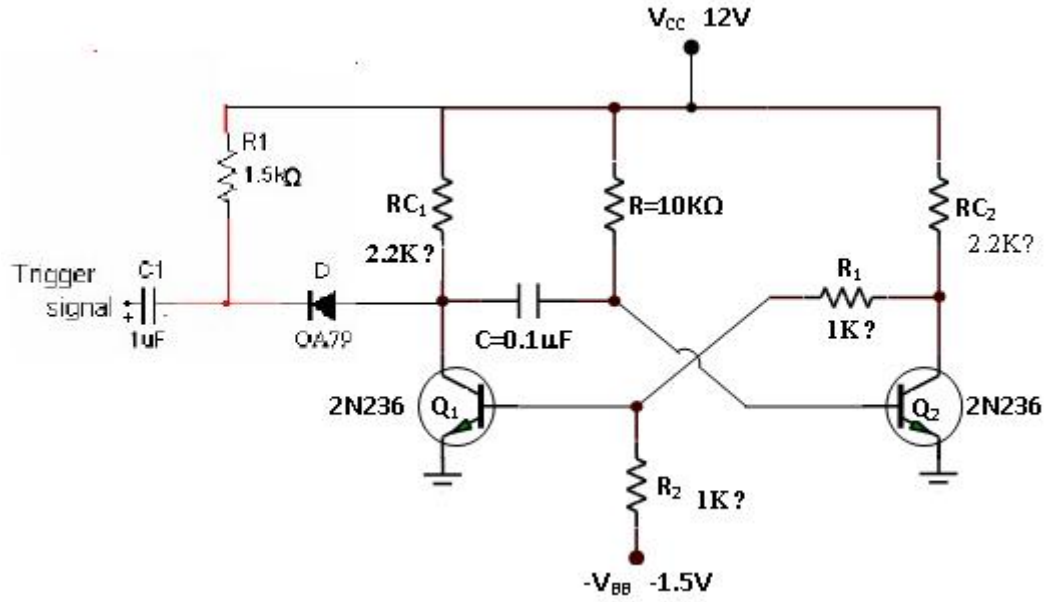


Figure 9.1. Monostable multivibrator

THEORY :

'A monostable multivibrator has only one stable state, the other state being quasi-stable. Normally the multivibrator is in the stable state, and when an external triggering pulse is applied, it switches from the stable to the quasi-stable state. It remains in the quasi-stable state fro a

short duration, but automatically reverts i.e. switches back to its original stable state, without any triggering pulse'.

Principle of operation

A collector-coupled Monostable multivibrator of the two transistors Q_1 and Q_2, Q_1 is normally OFF and Q_2 is Normally ON. Resistor R_1 and R_2 are connected to the normally OFF transistor, and the capacitor C is connected to the normally ON transistor.

It is seen from the circuit of the monostable multivibrator that, under normal conditions, the supply voltage V_{CC} provides enough base drive to the transistor Q_2 through resistor R, with the result that Q_2 goes into saturation. With Q_2 ON, Q_1 goes OFF, as already studied in the context of binary operation.

With Q_2 ON and Q_1 OFF, the capacitor finds a charging path. The voltage across the capacitor is V_{CC} with polarity. It is obvious that in the stable state of the multivibrator, Q_2 is ON and Q_1 is OFF.

If the negative triggering pulse is applied to the collector of Q_1, it is transmitted to the base of Q_2 through the capacitor, and hence makes the base of Q_2 negative. Immediately Q_2 goes OFF and Q_1 becomes ON. However, this is only a quasi-stable state as is obvious form the following observation.

With Q_1 ON and Q_2 OFF, the capacitor C finds a discharging path. As the capacitor discharges, it is seen that the potential at the base of the transistor Q_2 becomes less and less negative, and after a time, we have $V_B = V_\gamma$, the cut-in-voltage of Q_2.

As soon as V_B crosses the level of V_γ, Q_2 starts conducting and gets saturated. When Q_2 becomes ON, Q_1 becomes OFF. Thus the original stable state of the multivibrator is restored.

[In quasi-stable state: Q_1 is ON and Q_2 is OFF]

The interval during which the quasi-stable state of the multivibrator persists i.e., Q_2 remains OFF is dependent upon the rate at which the capacitor C discharges. This duration of the quasi-stable state is termed as delay time or pulse width or gate time. It is denoted as T. The wave forms of the voltage at base of the transistor Q_2 and C (Collector of Q_1)

DESIGN:

V_{CE} = 5.56v, V_{CC} = 6v, $V_{CE(sat)}$ = 0.3v, $V_{BE(sat)}$, = 0.7v, I_C = 6mA,V_F = -0.3v

Rc = $(V_{CC}–V_{CE(sat)})/I_C$.

$$V_F = \frac{-V_{BB}R_1}{R_1+R_2} + \frac{V_{CE(sat)}R_2}{R_1+R_2}$$

$$V_{CE} = \frac{V_{CC}R_1}{R_1+R_C} + \frac{V_{BE(sat)}R_C}{R_1+R_C}$$

Find the values of R_1 and R_2

PROCEDURE:

1. Connect the circuit as shown in figure.
2. With the help of a triggering circuit and using the condition T (trig) > T(Quasi) a pulse waveform is generated.
3. The output of the triggering circuit is connected to the base of the off transistor.
4. The Off transistor goes into ON state.
5. Observe the waveforms at V_{BE1}, V_{BE2}, V_{CE1}, V_{CE2}
6. Keep the DC- AC control of the Oscilloscope in DC mode.

EXPECTED WAVEFORMS:

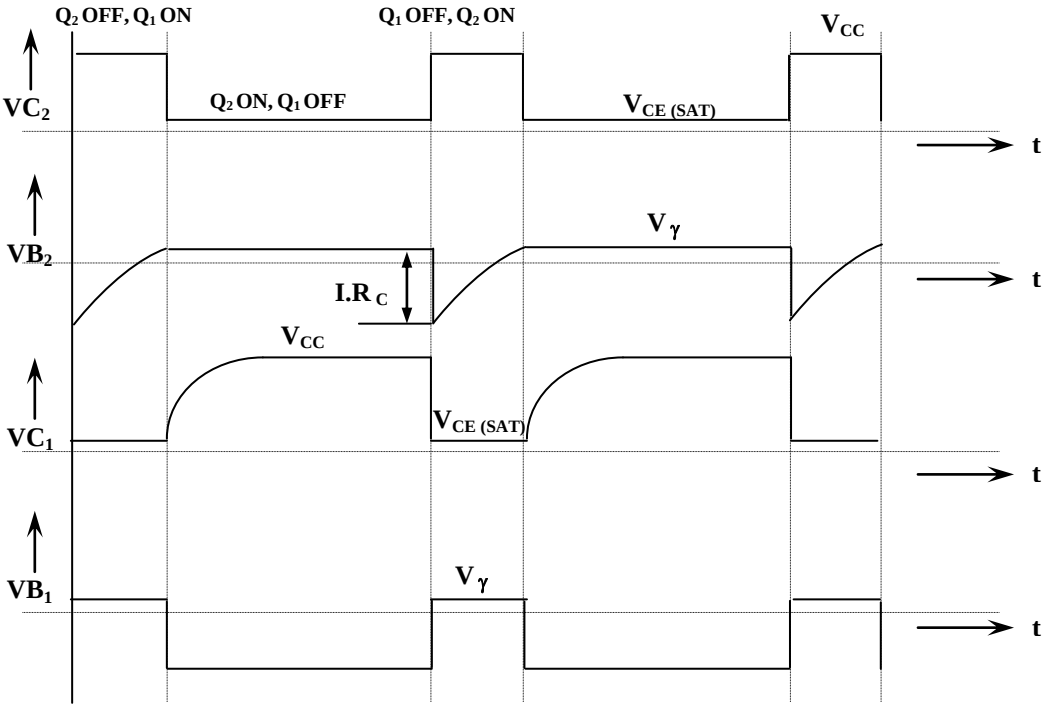


Figure 9.2. Expected Waveforms

RESULT:

T_{ON} =
T_{OFF} =
Total T (T_{ON} + T_{OFF}) =
Monostable multivibrator is designed and studied.

QUESTIONS:

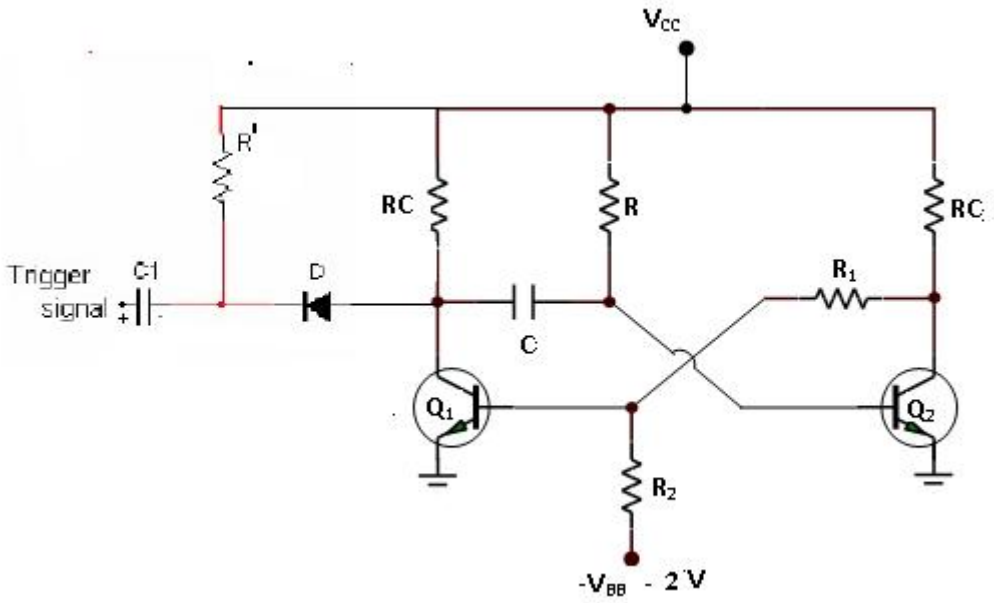
1. Explain the operation of collector coupled Monostable Multivibrator?
2. Derive the expression for the gate width of a transistor Monostable Multivibrator?
3. Give the application of a Monostable Multivibrator.

APPLICATIONS:

1. Frequency divider
2. Pulse width modulation
3. Linear ramp generator
4. Missing pulse detector

DESIGNING PROBLEM:

1. Design a collector coupled one shot with a gate width of 3ms, using n-p-n transistors.


Vcc
R'
RC
R
RC
Trigger signal
C1
D
C
R1
Q1
Q2
R2
-VBB - 2 V

GRAPH

EXPERIMENT NO: 10 **Date:**

SCHMITT TRIGGER

AIM: *To design and analyze Schmitt trigger and to observe the waveforms.*

COMPONENTS REQUIRED:

1. Resistors 100 KΩ-1
   2.2 KΩ-2
   3.3 KΩ-1
   3.9 KΩ-1
   6.8 K Ω-1
2. Capacitors 0.01μF-1
2. Transistors BC 107 – 2

APPARATUS:

1. Bread board
2. Power supply 0-30V
3. Signal generator
4. CRO
5. Connecting Wires.

CIRCUIT DIAGRAM:

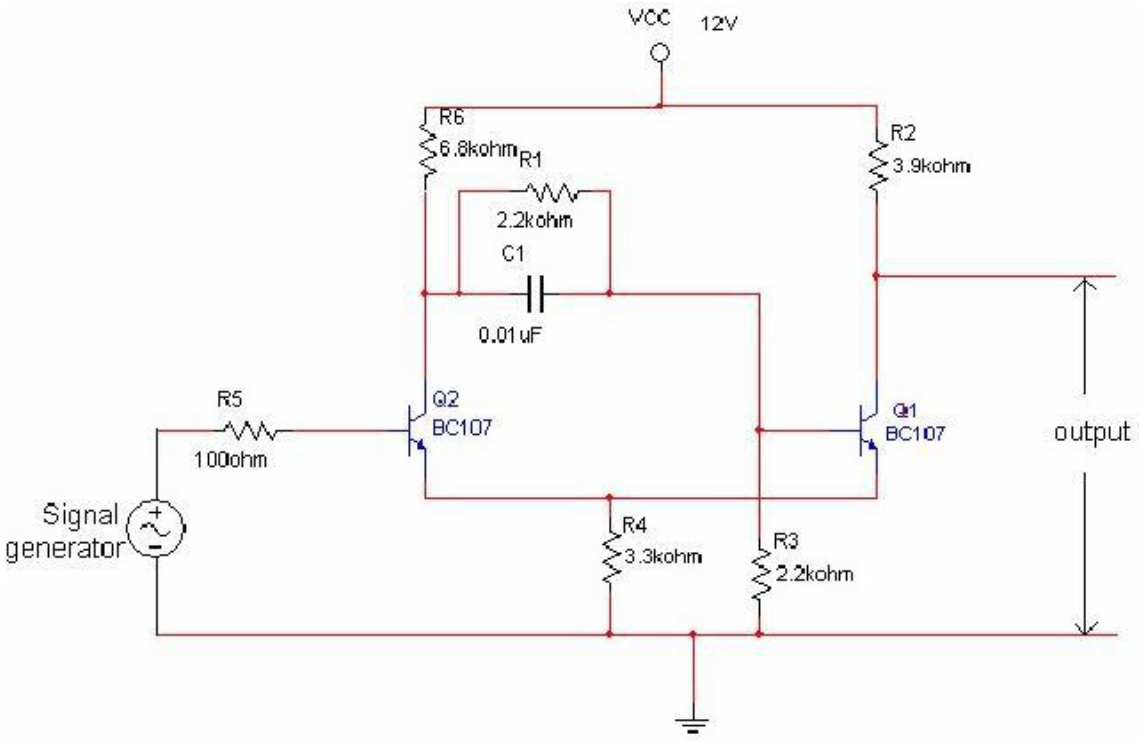


Figure: 10.1. Schmitt trigger

THEORY:

The most important application of Schmitt Trigger circuit are amplitude comparator and squaring circuit are amplitude comparator and squaring circuit. The circuit is used to obtain a square waveform from any arbitrary input waveform. The loop gain is to be less than unity.

If Q_2 is conducting there will be voltage drop across R_Z which will elevate the emitter of Q_1. Consequently if V is small enough in voltage, Q_1 will be cut-off with Q_1 conducting, the circuit amplifies and since the gain is positive, the output to rise, V_2 continues to fall and Z_2 continues to rise. Therefore a value of V will be reached where Q_2 is turned OFF. At the point the output no longer responds to the input.

Here the input signal is arbitrary except that it has large enough excursion to carry input beyond the limits of hysteresis range, $V_H = (V_1 - V_2)$.

The output is a square wave whose amplitude is independent of the amplitude of the input waveform.

DESIGN:

$I_{C2} = 5mA$
$(Rc_2 + R_E) = V_{CC} / I_{C2}$
U.T.P = $V_{E2} = 5V$
$V_{E2} = (R_E \times V_{CC}) / (Rc_2 + R_E)$
$I_2 = 0.1 \times I_{C2}$
L.T.P = $V_{E1} = 3V$
$R_2 = E_{R2i} / I_2 = V_{E1} / I_2 = L.T.P / I_2$
$Rc_1 = \{(R_E \times V_{CC}) / V_{E1}\} - R_E$
$I_{B2} = I_{C2} / hfe(min)$
$(V_{CC} - V_{E2}) / (R_1 + R_{L1}))) = (V_{E2}/R_2) + I_{B2}$
$R_B = (h_{fe} \times R_E) / 10$
Find R_1, R_2, R_E, $Rc_1$and Rc_2 from the above equations

PROCEDURE:

1. Connect the circuit as shown in figure 1 with designed values.
2. Apply V_{CC} of 12V and an input frequency of 1KHz with an amplitude more than the designed UTP.
3. Now note down the output wave forms
4. Observe that the output comes to ON state when input exceeds UTP and it comes to OFF state when input comes below LTP
5. Observe the waveforms at V_{C1}, V_{C2}, V_{B2} and V_E and plot graphs.
6. Keep the DC- AC control of the Oscilloscope in DC mode.

MODEL GRAPHS:

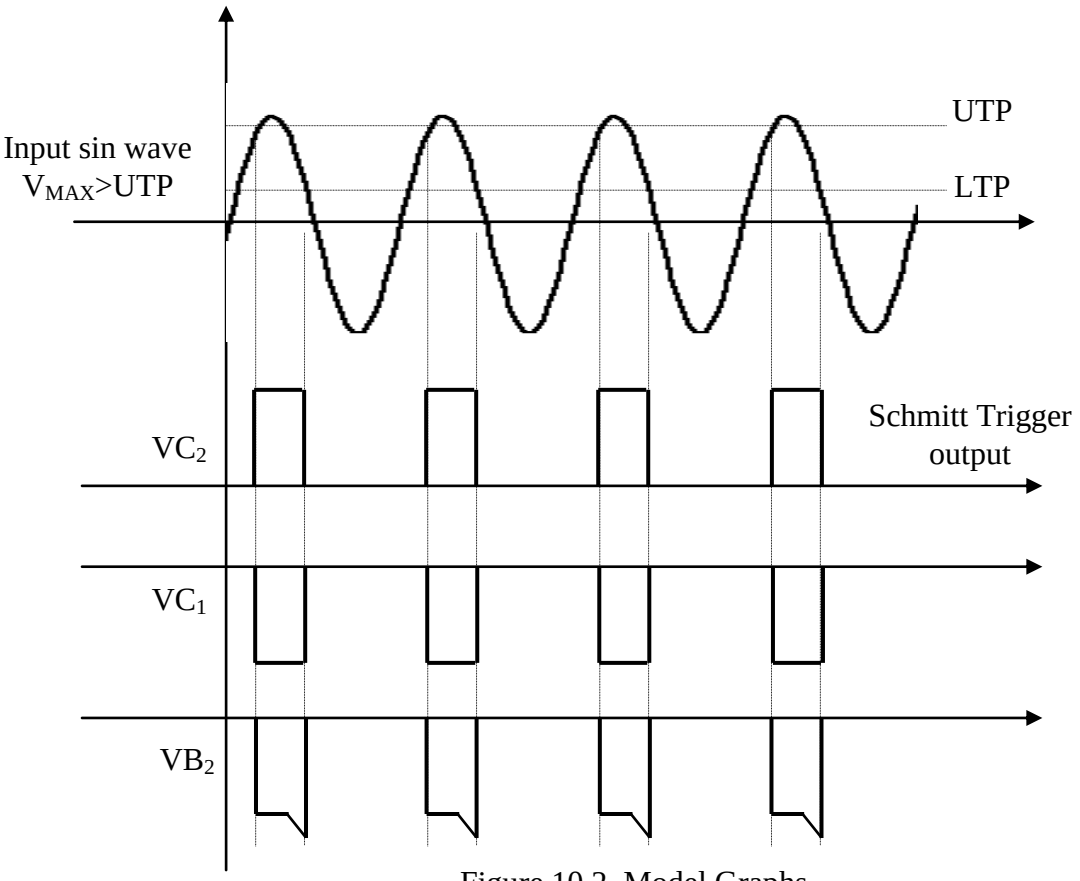


Figure 10.2. Model Graphs

RESULT: Schmitt Trigger circuit is designed and studied.

QUESTIONS:

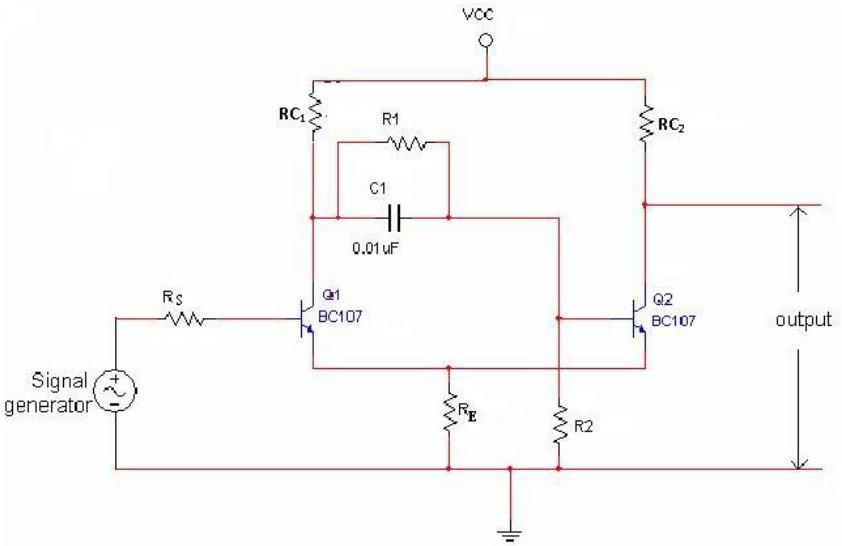
1. Explain how a Schmitt trigger acts as a comparator?
2. Derive its expressions for UTP & LTP.

APPLICATIONS:

1. squarewave generator
2. on/off controllers
3. Used as a comparator

DESIGNING PROBLEM:

1. Design a Schmitt trigger circuit for the specifications: UTP = 8v, LTP = 5V, I_C(sat) = 2mA, h_{FE}(min) =25.


VCC
RC1
R1
RC2
C1
0.01uF
RS
Q1
BC107
Q2
BC107
output
Signal
generator
RE
R2

GRAPH

EXPERIMENT NO: 11 **Date:**

UJT RELAXATION OSCILLATOR

AIM : *To obtain a saw tooth waveform using UJT and test its performance as an oscillator and also obtain the error.*

COMPONENTS:

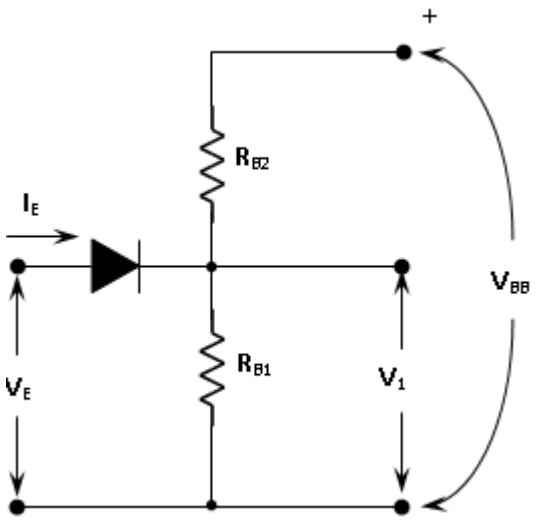
1. Resistors 47kΩ-1
   100Ω-1
2. Capacitor 0.1μF-1
3. UJT 2n2646-1

APPARATUS

1. Bread board
2. Power supply 0-30V
3. CRO

THOERY :

A Unijunction transistor (UJT), as the very implies, has only one p-n junction, unlike a BJT which has two p-n junctions'.



11.1. Equivalent Circuit of UJT Relaxation Oscillator

The equivalent circuit of the UJT is as shown in figure 1.

R_{B1} is the resistance between base B_1 and the emitter, and it is basically a variable resistance, its value being dependent upon the emitter current I_E.

R_{B2} is the resistance between base B_2 and the emitter, and the value is fixed.
Consider the circuit as shown in figure 1.
Let $I_E = 0$. Due to the applied voltage V_{BB} a current I results as shown.

We have $V_1 = iR_{B1}$.

But $i = \dfrac{V_{BB}}{R_{B1} + R_{B2}}$

$\therefore\ V_1 = \left(\dfrac{V_{BB}}{R_{B1} + R_{B2}}\right) R_{B1}$

$= \left(\dfrac{R_{B1}}{R_{B1} + R_{B2}}\right) V_{BB}$

The ratio $\left(\dfrac{R_{B1}}{R_{B1} + R_{B2}}\right)$ is termed as the ***intrinsic stand off ratio*** and it is denoted as η.

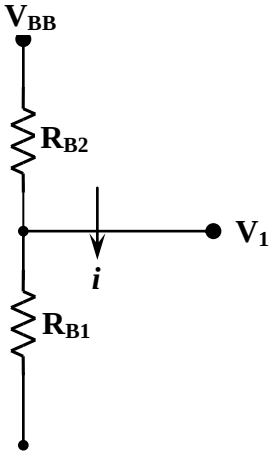


Figure 11.2

$\therefore$

$$\left[\eta = \frac{R_{B1}}{R_{B1} + R_{B2}} \text{ when }, I_E = 0\right] V_1 = \eta V_{BB}.$$

Form the equivalent circuit, it is evident that the diode cannot conduct unless the emitter voltage $V_E = V_\gamma + V_1$, where V_γ is the cutin voltage of the diode.

This value of the emitter voltage which makes the diode conduct is termed as ***peak voltage***, and it is denoted as V_P.

We have $V_E = V_\gamma + V_1$,

or since $\boxed{\boldsymbol{V_P = V_\gamma + \eta V_{BB}}}$ $V_1 = \eta V_{BB}$.

It is obvious that if $V_E < V_P$, the UJT is OFF, and

if $V_E < V_P$, the UJT is ON.

Figure 11.3. Shows the emitter characteristics of a UJT (plot of V_E vs I_E)

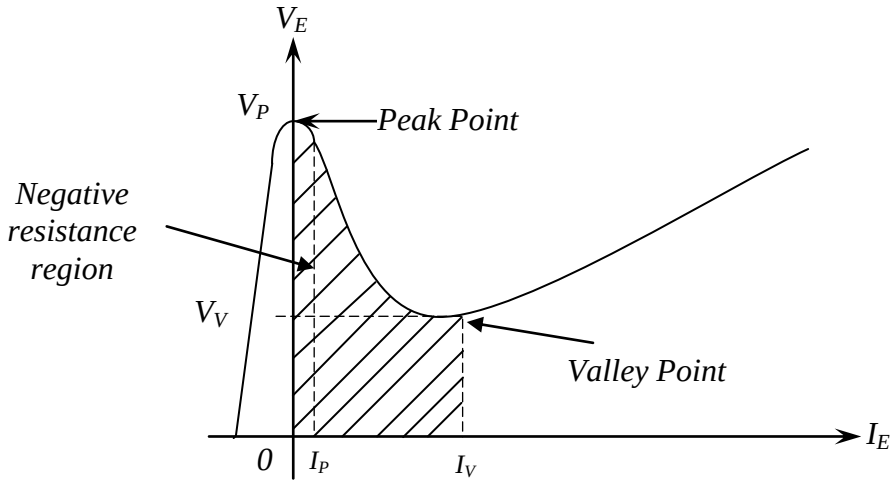


Figure 11.3. Emitter characteristics of a UJT

The main application of UJT is in switching circuits wherein rapid discharging of capacitor is very essential.

Having understood the basic of UJT, we shall next study the working of UJT relaxation oscillator.

Working of UJT relaxation oscillator (OR UJT sweep circuit)

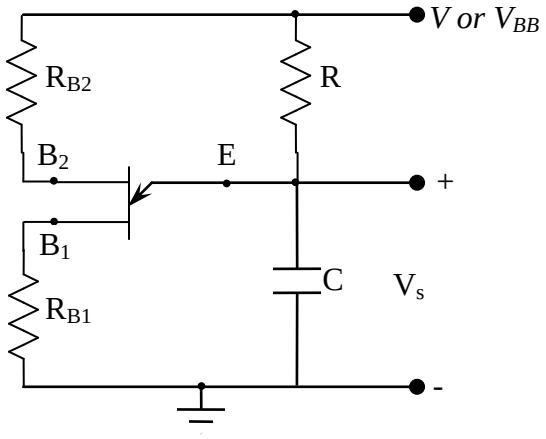


Figure:11.4. UJT relaxation oscillator

The UJT sweep circuit shown in the figure 4 consists of a UJT, a capacitor and a resistor arranged as shown.

We studied that a UJT is OFF as long as $V_E < V_P$, the peak voltage. Hence initially when the UJT is OFF, the capacitor *C* charges through the resistance *R* from the supply voltage *V*.
Let V_S = capacitor voltage.

It is seen that when the capacitor voltage V_S rises to the value V_P the UJT readily conducts. When the UJT becomes ON, the capacitor discharges and its voltage falls. When the voltage falls to the valley point V_V, the UJT becomes OFF and the capacitor charges again to V_P.

This cycle of charging and discharging of the capacitor *C* repeats, and as a result, a saw tooth wave form of voltage across *C* is generated.

CIRCUIT DIAGRAM :

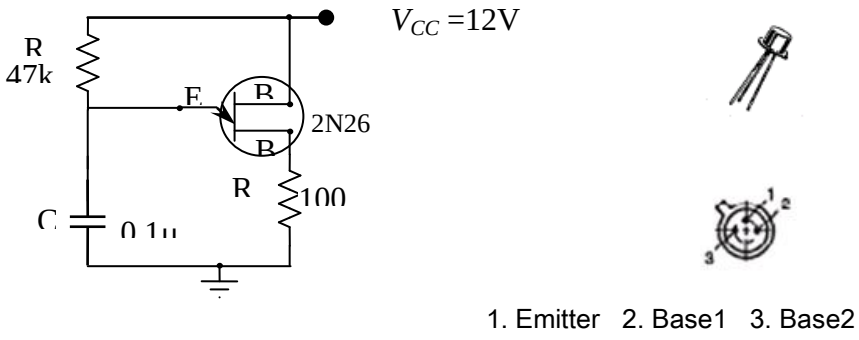


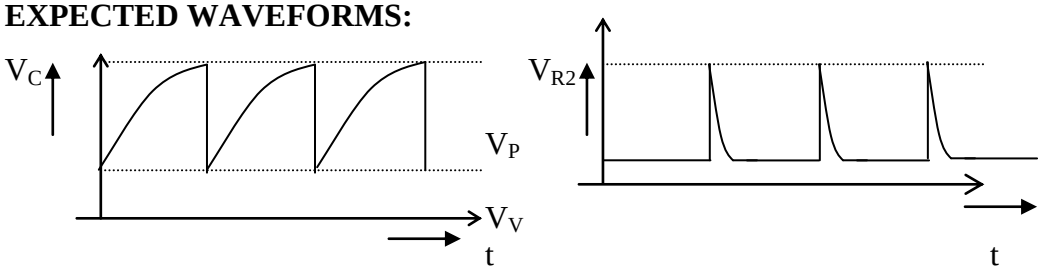
Figure:11.5. Circuit Diagram of UJT relaxation oscillator

PROCEDURE:

1. Connect the circuit as shown in figure with designed values.
2. Note down the voltages and frequencies across C& R_2 .
3. The time period of the output wave form is noted and is compared with theoretical value $T = R_1 \times C[\ \ln \{(V_{BB} - V_V) / (V_{BB} - V_P)\}]$

4. Plot the graphs of Vc and $V_{R1.}$

EXPECTED WAVEFORMS:



RESULT:

Theoretical T = Practical T =

Sawtooth waveform is obtained using UJT and its performance as an oscillator is studied.

QUESTIONS:

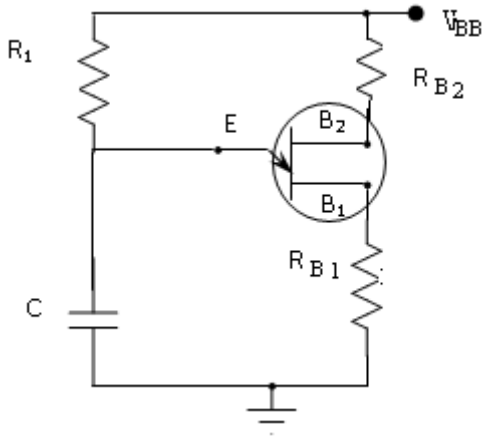
1. Describe some important applications of a UJT?
2. Is the name UJT appropriate?
3. Write short notes on UJT as a relaxation oscillator?
4. Discuss the concept of –ve resistance?
5. Define the intrinsic stand off ratio and explain its importance?

APPLICATIONS:

1. Sawtooth wave generator
2. Used in Phase Shifters.

DESIGNING ROBLEM:

1. The specifications of UJT are given as $\eta = 0.6$, $V_v = 2V$, $R_{BB} = 5K\Omega$, $I_v = 1.5mA$, $I_p = 8\mu A$ and $V_{BB} = 18V$. Calculate the component values of the UJT sweep circuit to generate an output sweep frequency of 10KHZ with swee amplitude of 12V.



GRAPH

EXPERIMENT NO: 12 **Date:**

BOOT STRAP SWEEP CIRCUIT

AIM: *To design and test the performance of bootstrap sweep circuit.*

COMPONENTS REQUIRED:

1. Resistors	100KΩ-1
	5.6KΩ-1
	10KΩ-1
2.Capacitors	0.1μF-1,
	10μF-1
	100μF-1
3. Diode	IN4007-1
4. Transistors	2N2369– 2

APPARATUS REQUIRED:

1. Bread board
2. Power supply
3. CRO
4. Signal generator
5. Connecting Wires.

CIRCUIT DIAGRAM:

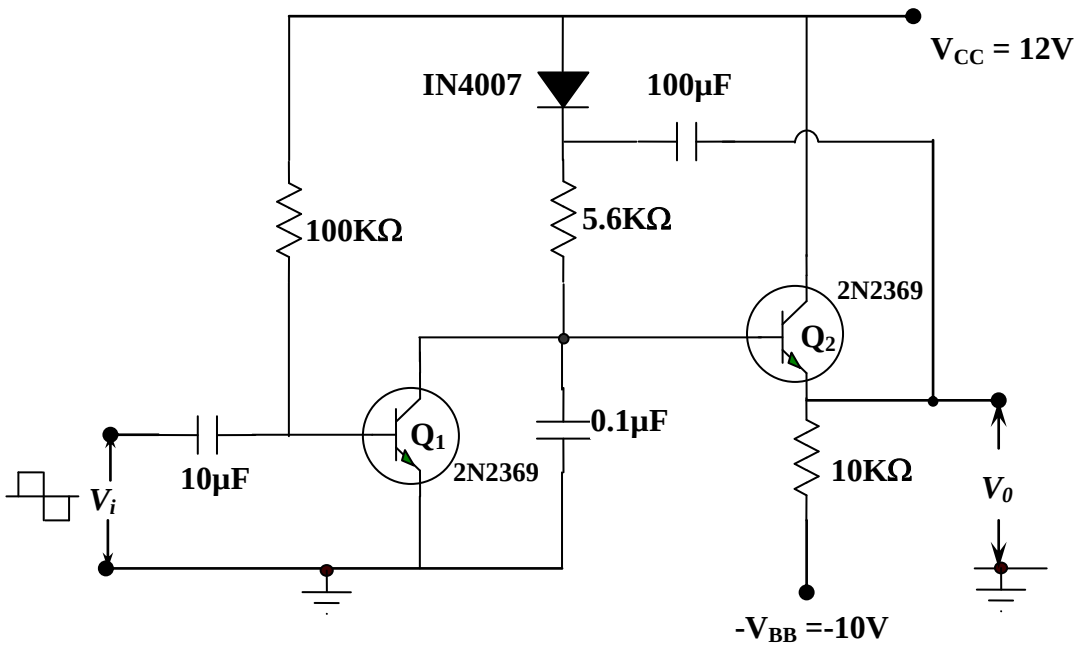


Figure 12.1. Boot Strap Sweep Circuit

DESIGN:

$$V_S = \frac{V_{CC} \times T_g}{RC}$$

$$T_R = \frac{\left[(C \times V_S)/V_{CC}\right]}{\left[(h_{fe}/R_B) - (1/R)\right]}$$

Sweep time = $T_S = RC$

THEORY:

The input to Q_1 is the gating waveform. Before the application of the gating waveform, at t = 0, transistor Q_1 is in saturation. The voltage across the capacitor C and at the base of Q_2 is $V_{CE(sat)}$. To ensure Q_1 to be in saturation for t = 0, it is necessary that its current be at least equal to i_{CE} / h_{FE} so that $R_b < h_{fe}R$.

With the application of the gating waveform at t = 0, Q_1 is driven OFF. The current i_{C1} now flow into C and assuming units gain in the emitter follower $V_0 = \frac{V_{CC}t}{RC}$. When the sweep starts, the diode is reverse biased, as already explained above, the current through R is supplied by C_1. The current V_{CC} / R through C and R now flows from base to emitter of Q_2.if the output V_0 reaches the voltage V_{CC} in a time T_S / T_g, then from above we have $T_S = RC$.

If the sweep amplitude is less than V_{CC}, then the maximum ramp voltage is given by

$$V_S = \frac{V_{CC}T_g}{RC}$$

PROCEDURE:

1. Connect the circuit as shown in figure.
2. Apply the square wave or rectangular wave form at the input terminals.
3. Connect the CRO at output terminals now plug the power card into line switch on and observe the power indication.
4. As mentioned in circuit practical calculation. Observe and record the output waveforms from CRO and compare with theoretical values.

EXPECTED WAVEFORMS:

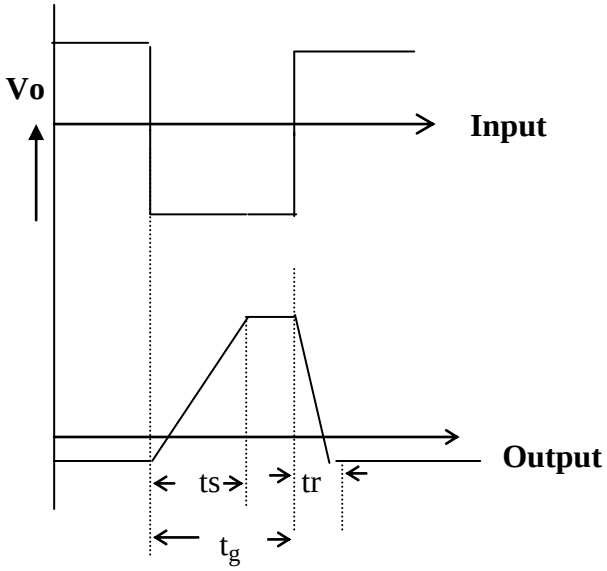


Figure 12.2. Expected Waveforms

RESULT:

T_S =

T_G=

T_R =

Bootstrap sweep circuit is designed and studied

QUESTIONS:

1 .What are the other methods of sweep generator?
2. Compare bootstrap and miller sweep generator?
3. Describe the operation a single transistor Boot strap time base voltage waveform generator making use of its related circuit diagrams?
4. Explain the principle of working of Boot strap circuit?

APPLCATIONS:

1.Used In analog circuit designs to alter the input impedance of a circuit.
2.Used in dc-dc converters to serve as top-side bias voltage supplies.

DESIGNING PROBLEM:

1. In the transistor bootstrap circuit of figure above , V_{CC} = 25V, V_{EE} = -15V,R = 10KΩ, R_E = 15KΩ, C = 0.05μF and C_1 = 100μF. The gating waveforms has a duration, Tg = 300μs. The transistor parameters are hie = 1.1KΩ, hre = $2.5x10^{-4}$ kΩ, hfe = 50, hoe = 1/40KΩ.
   a) Draw the waveforms of ic1 and Vo, labeling all current and voltage levels.
   b) What is the slope error of the sweep.
   c) What is the sweep speed and the minimum value of the sweep voltage
   d) What is the retrace time T, for C to discharge completely.
   e) Calculate the recovery time T1 for C1 to recharge completely.

GRAPH

EXPERIMENT NO: 13 **Date:**

SAMPLING GATES

AIM: *To construct and verify the response of sampling gate by using diode.*

COMPONENTS:

1. Resistors (1 kΩ, 10 kΩ)
2. Capacitor (0.047µf)
3. Diode 1N4007

APPARATUS:

1. Function generator
2. CRO
3. Connecting wires

CIRCUIT DIAGRAM:

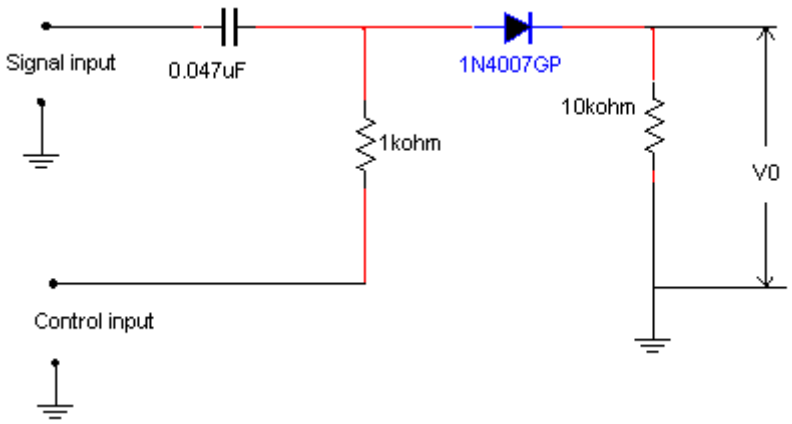


Figure 13.1. Sampling Gate

MODEL WAVEFORMS:

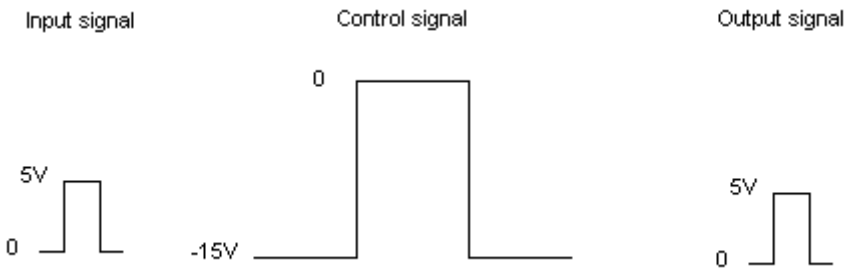


Figure 13.2. Circuit Diagram Of Sampling Gate and its model waveforms

THEORY:

An ideal sampling gate is a transmission circuit that produces an output signal identical to the input signal during a selected time interval.the output of the sampling gate is zero outside this selected time interval.the sampling gate is open during the sampling interval and it is closed at all other times.the time interval for transmission is monitored by a control input signal,which is usually rectangular in shape.in practice the idealized transmission gate is not realized.as long as the output is produced at the correct time the performance of the practical sampling gates available is treated to be quite satisfactory.

PROCEDURE:

1. Connect the circuit as per circuit diagram
2. Apply both inputs (signal input and control input) simultaneously to the circuit.
3. Repeat the second step by varying input signal and putting control signal fixed.
4. Note down the output waveforms for various range of input signals.

PRECAUTIONS:

1. Connections should be tight.
2. Take care when applying the control signal.

RESULT: Hence verified the response of sampling gate by using diode.

QUESTIONS:

1. What is sampling gate?
2. Define control signal?
3. What is the other name for the control signal?
4. What is the difference between logic gates and sampling gates?
5. What is the necessity of the sampling gate?

APPLICATIONS:

Used in

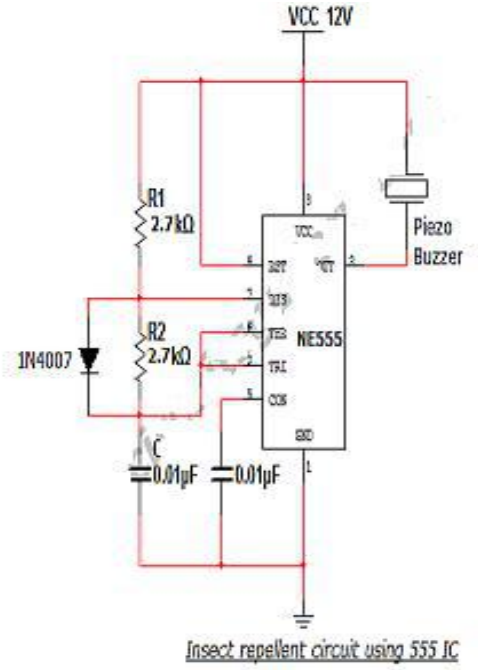
1. Multiplexers
2. D/A converters
3. Chopper stabilized amplifiers
4. Sampling scopes.
5. Sampling and hold circuits.

GRAPH

PROJECTS

MOSQUITO CONTROL CIRCUIT:

circuit daigram



Components required

1. Power supply (12V)
2. Resistors (2.7kΩ x 2)
3. Capacitors (0.01μF x 2)
4. NE 555 timer
5. IC Diode (1N4007)
6. Piezo Buzzer

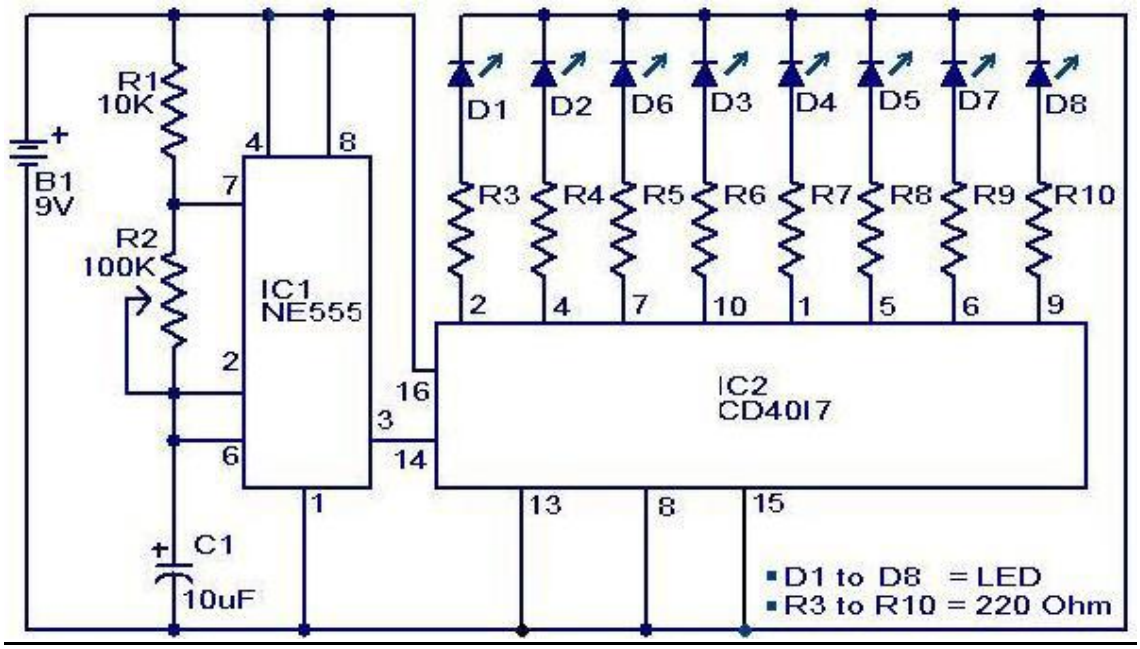
Working of bug repellent circuit

Here the 555 timer is configured in astable multivibrator mode, specifically devised for generating sound at the ultra frequency levels.The frequency of this circuit is set to 25 kHz, this frequency falls in the ultrasound region and it is not audible to human beings. The 1N4007 diode is used to get 50% duty cycle for the frequency generator.The piezo buzzer converts the output of 555 astable to

ultrasonic sound that can be heard by the insects.This ultrasound frequency cause some irritations and uneasiness to the insects such as mosquitoes. However it will not disturb you if you have installed this circuit of insect repellent for your home.

DANCING LIGHT

circuit daigram



Cponents required

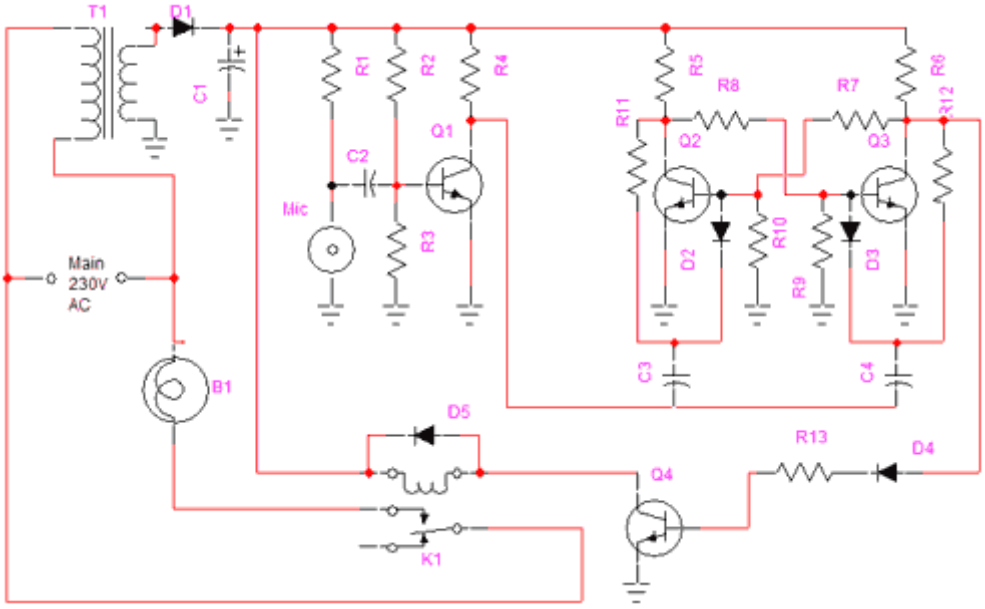
As indicated in the circuit diagram

Working principal

Here is a simple dancing light circuit Based on NE555(IC1) & CD4017(IC2). The IC1 is wored as an astable multivibrator to provide the clock pulses for the CD4017.for each clock pulse receiving at the clock input (pin14) of IC CD4017,the outputs Q0 to Q9 becomes high one by one alternatively.the leds connected to these pins glow in the same fashion to give a Dancing effect.the speed of the dancing LEDs depend on the frequency of the clock pulses generated by the IC1.

CLAP SWITCH

Circuit diagram



Components required

Resistors

R1=15KΩ, R5,R6=1.5KΩ

R2,R11,R12=2.2MΩ, R13=2.2KΩ

R3=270KΩ, R4=3.3KΩ

R7,R8=10KΩ, R9,R10=27KΩ

Capacitors

C1=1000µf/16v

C2=.01µf,C3,C4=.047µf

Semi Conductors

Q1,Q2,Q3= BC548

D2,D3,D4= IN 4148

D1,D5=IN 4007, Q4=BC368

Misc

T1=12v/500mA Transformer

Mic= Condenser Microphone

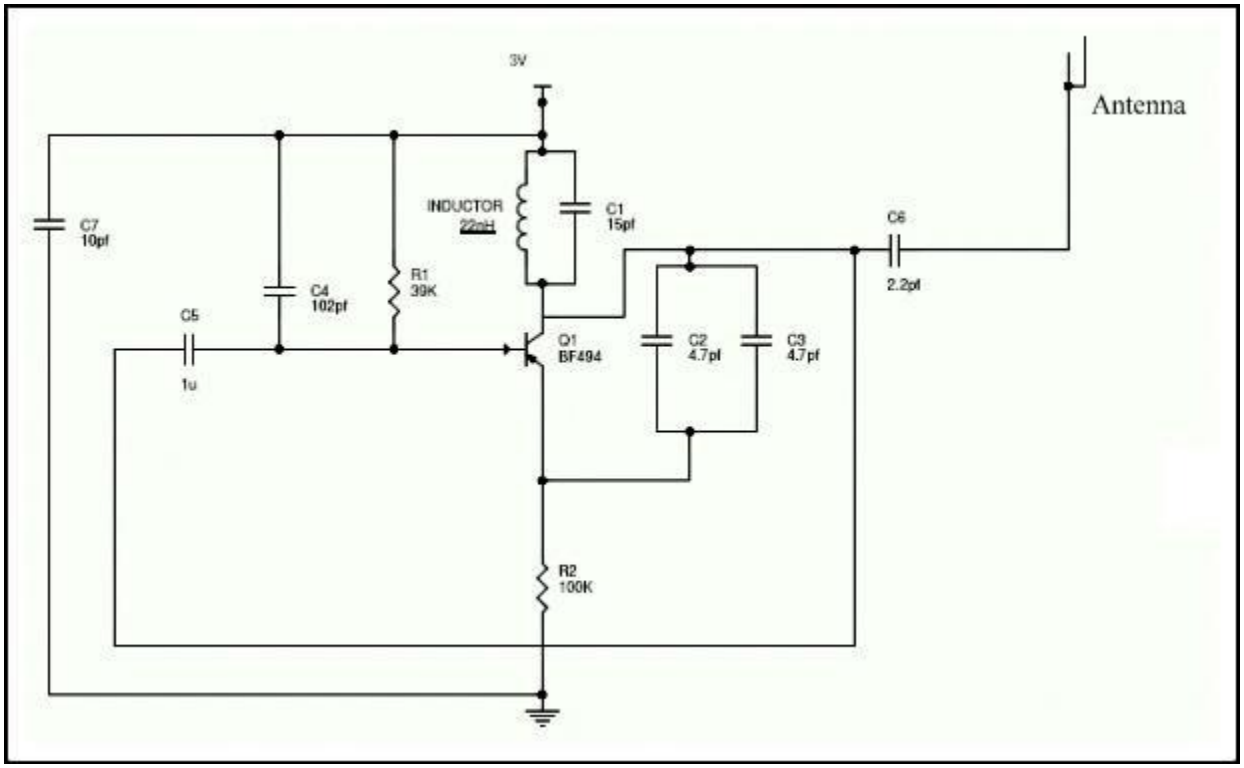
K1= 12V Relay, B1= Bulb or Load

Working principle

Here is a Hobby Circuit for electronics hobbyists that can switch on & off a light, Fan, Radio etc., by the sound of clap. The sound of clap is received by a small microphone that is shown biased by resistor R1 in the circuit. The microphone changes sound wave in to electrical wave, which is further amplified by Q1. Transistor Q1 is used as common emitter circuit to amplify weak signals received by the microphone. Amplified output from the collector of transistor Q1 is feed to the Bistable Multivibrator circuit also known as flip-flop circuit.

Mobile Jammer Circuit

Simple Mobile Jammer Circuit Diagram:



Simple Mobile Jammer Circuit Diagram

Cell Phone Jammer Circuit Explanation:

- If you understand the above circuit, this circuit analysis is simple and easy. For any jammer circuit, remember that there are three main important circuits. When they are combined together, the output of that circuit will work as a jammer. The three circuits are
  - RF amplifier.
  - Voltage controlled oscillator.
  - Tuning circuit.

So the transistor Q1, capacitors C4 & C5 and resistor R1 constitute the RF amplifier circuit. This will amplify the signal generated by the tuned circuit. The amplification signal is given to

the antenna through C6 capacitor. Capacitor C6 will remove the DC and allow only the AC signal which is transmitted in the air.

When the transistor Q1 is turned ON, the tuned circuit at the collector will get turned ON. The tuned circuit consists of capacitor C1 and inductor L1. This tuned circuit will act as an oscillator with zero resistance.

This oscillator or tuned circuit will produce the very high frequency with minimum damping. The both inductor and capacitor of tuned circuit will oscillate at its resonating frequency.

The tuned circuit operation is very simple and easy to understand. When the circuit gets ON, the voltage is stored by the capacitor according to its capacity. The main function of capacitor is to store electric energy. Once the capacitor is completely charged, it will allow the charge to flow through inductor. We know that inductor is used to store magnetic energy. When the current is flowing across the inductor, it will store the magnetic energy by this voltage across the capacitor and will get decreased, at some point complete magnetic energy is stored by inductor and the charge or voltage across the capacitor will be zero. The magnetic charge through the inductor will decreased and the current will charge the capacitor in opposite or reverse polarity manner. Again after some period of time, capacitor will get completely charged and magnetic energy across the inductor will be completely zero. Again the capacitor will give charge to the inductor and becomes zero. After some time, inductor will give charge to capacitor and become zero and they will oscillate and generate the frequency.

This circle run upto the internal resistance is generated and oscillations will get stop. RF amplifier feed is given through the capacitor C5 to the collector terminal before C6 for gain or like a boost signal to the tuned circuit signal. The capacitors C2 and C3 are used for generating the noise for the frequency generated by the tuned circuit. Capacitors C2 and C3 will generate the electronic pulses in some random fashion (technically called noise).

The feedback back or boost given by the RF amplifier, frequency generated by the tuned circuit, the noise signal generated by the capacitors C2 and C3 will be combined, amplified and transmitted to the air.

Cell phone works at the frequency of 450 MHz frequency. To block this 450MHz frequency, we also need to generate 450Mhz frequency with some noise which will act as simple blocking signal, because cell phone receiver will not be able to understand to which signal it has been received. By this, we can able to block the cell phone signal from reaching the cell phones.

So here in the above circuit, we generated the 450 MHz frequency to block the actual cell phone signal. That’s what the above circuit will act as a jammer for blocking the actual signal.

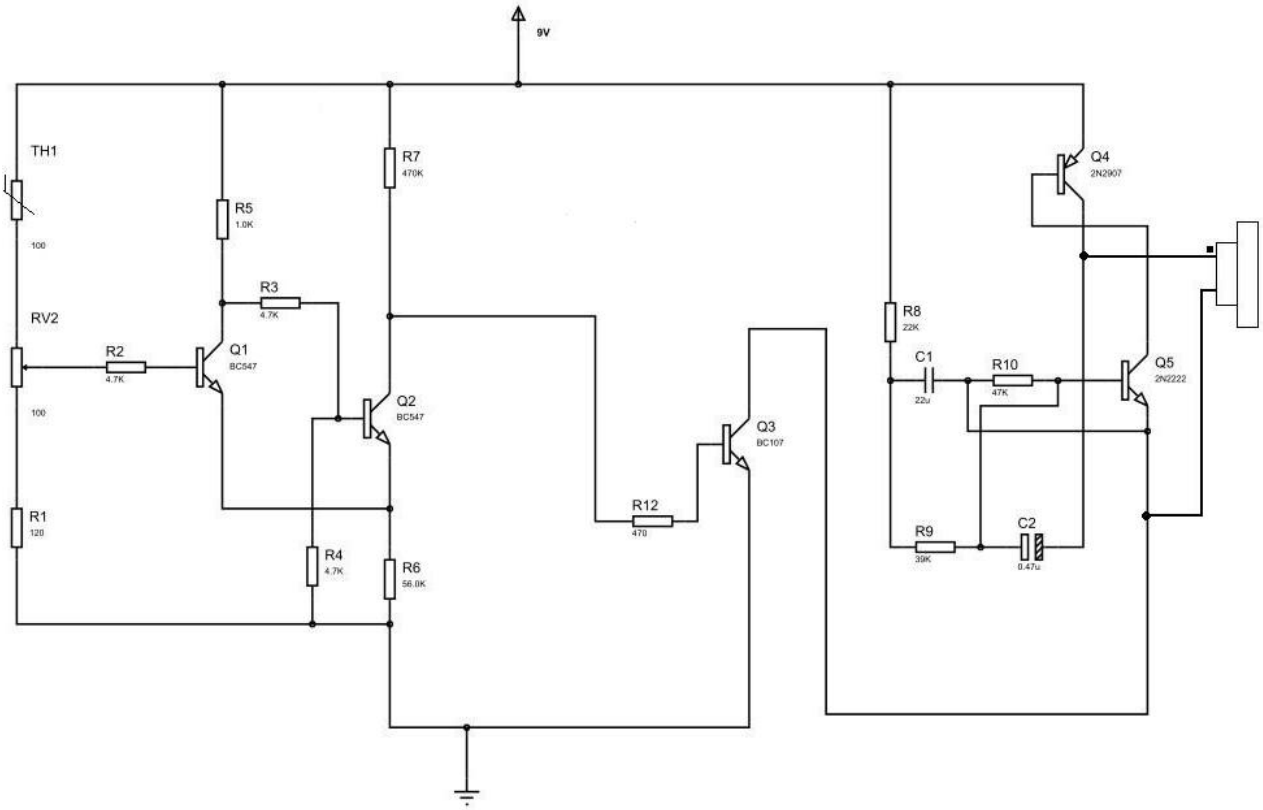
Fire Alarm with Siren Sound

Fire Alarm Circuit:

This circuit alerts us when there is a fire accident at home by ringing a siren sound. You might have seen fire alarms earlier but this is quite different as it generates a siren sound instead of a buzzer and also it uses basic components to generate that siren sound.

We are aware that there are many integrated circuits which can be used to generate the siren effect but we preferred to use basic electronics components like resistors, capacitors and transistors to generate it so that you will clearly understand the internal working of it and it will be much useful for you as you will gain more knowledge by analyzing it instead of simply going or pre designed integrated circuits.

Fire Alarm Circuit Diagram:



Description:

This circuit uses a thermistor to sense the temperature. When it senses that the temperature of the environment is increasing above a given threshold, then it gives a signal. The temperature at which the circuit detects fire can be adjusted by using the potentiometer arrangement at RV2.

When the temperature increases above the set value, the potentiometer arrangement produces a high voltage. This voltage is then given to BC547 transistor in common emitter mode. It is an NPN general purpose transistor. When the base is given a high input, it gets turned on. When the transistor is turned on, its collector voltage is reduced to low as the collector to emitter voltage decreases. The collector output voltage of the first transistor is given to the base as an input to the second BC 547 NPN transistor. This transistor too is in common emitter mode and as the input is low when the temperature threshold is reached, the output at the collector will rise high. In this state, it will turn on the next transistor, i.e BC107. This transistor will now act as a switch for the siren circuit. This transistor can bear power quite larger than the BC547 and it is also equipped with a heat sink for that purpose.

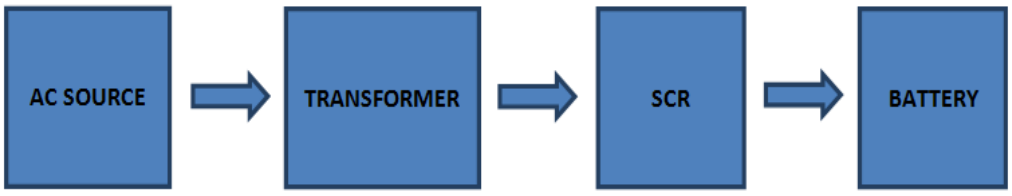
When the BC107 transistor turns on, it allows current to pass from power supply to ground through collector thereby acting as an electronically controlled switch. When the current is passing, the siren circuit which is assembled as the load to the circuit is turned ON. Then you can hear the siren sound through the buzzer. The capacitors used in the circuit are the main components in producing the siren effect. The principle involved in generating the siren effect is to make an oscillator with an envelope which periodically increases and decreases so as to generate that effect.

Battery Charger Circuit Using SCR

Introduction to Battery Charger Using SCR:

The battery is charged with small amount of AC voltage or DC voltage. So if you want to charge your battery with AC source then should follow these steps, we need first limit the large AC voltage, need to filter the AC voltage to remove the noise, regulate and get the constant voltage and then give the resulting voltage to the battery for charging. Once charging is completed the circuit should automatically turned off.

Block Diagram of Battery Charger Using SCR:



Block Diagram of Battery Charger Using SCR

The AC source is given to the step down transformer which converts the large AC source into limited AC source, filter the AC voltage and remove the noise and then give that voltage to the SCR where it will rectify the AC and give the resulting voltage to the battery for charging.

Circuit Diagram Explanation:

- The AC main voltage is given to the step down transformer the voltage should be down to 20V approx. the step down voltage is given to the SCR for rectification and SCR rectifies AC main voltage. This rectified voltage is used to charge battery.
- When the battery connecter to the charging circuit, the battery will not be dead completely and it will get discharged this will give the forward bias voltage to the transistor through the diode D2 and resistor R7 which will get turned on. When the transistor is turned on the SCR will get off.
- When the battery voltage is dropped the forward bias will be decreased and transistor gets turned off. When the transistor is turned off automatically the diode D1 and resistor R3 will get the current to the gate of the SCR, this will triggers the SCR and gets conduct. SCR will rectifies the AC input voltage and give to the battery through Resistor R6.
- This will charge the battery when the voltage drop in the battery decreases the forward bias current also gets increased to the transistor when the battery is completely charged the Transistor Q1 will be again turned on and turned off the SCR.
-

Circuit Diagram of Battery Charger Using SCR:

Circuit diagram of the Battery Charger Circuit using SCR can be seen below: