# Varuvan Vadivelan Institute of Technology

Dharmapuri – 636 703

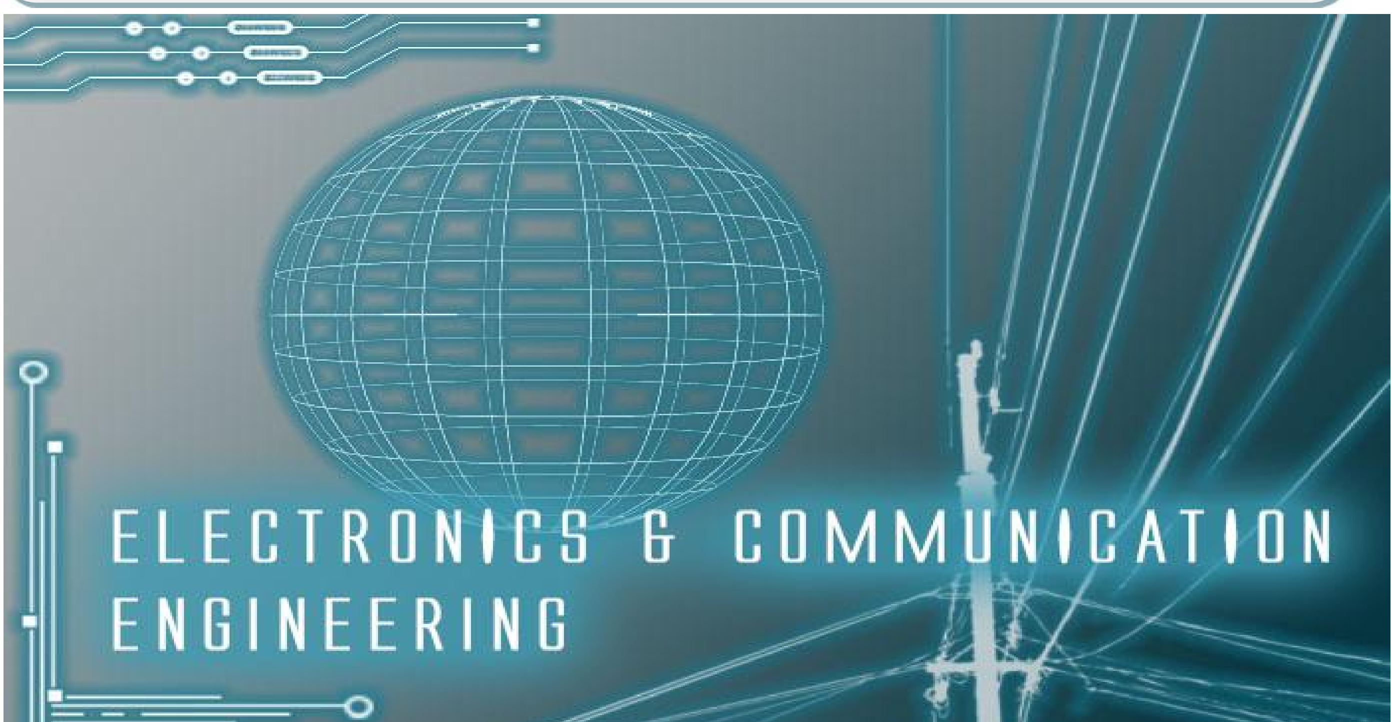
## LAB MANUAL

**Regulation** : 2013

**Branch** : *B.E.* – ECE

**Year & Semester** : II Year / IV Semester

# EC6411 - CIRCUITS AND SIMULATION INTEGRATED LABORATORY



**ANNA UNIVERSITY CHENNAI**
**Regulation 2013**

## EC6411 CIRCUITS AND SIMULATION INTEGRATED LABORATORY

**DESIGN AND ANALYSIS OF THE FOLLOWING CIRCUITS**

1. Series and Shunt feedback amplifiers-Frequency response, Input and output impedance calculation
2. RC Phase shift oscillator and Wien Bridge Oscillator
3. Hartley Oscillator and Colpits Oscillator
4. Single Tuned Amplifier
5. RC Integrator and Differentiator circuits
6. Astable and Monostable Multivibrators
7. Clippers and Clampers
8. Free running Blocking Oscillators

**SIMULATION USING SPICE (Using Transistor):**

9. Tuned Collector Oscillator
10. Twin -T Oscillator / Wein Bridge Oscillator
11. Double and Stager tuned Amplifiers
12. Bistable Multivibrator
13. Schmitt Trigger circuit with Predictable hysteresis
14. Monostable multivibrator with emitter timing and base timing
15. Voltage and Current Time base circuits

**TOTAL: 45 Periods**

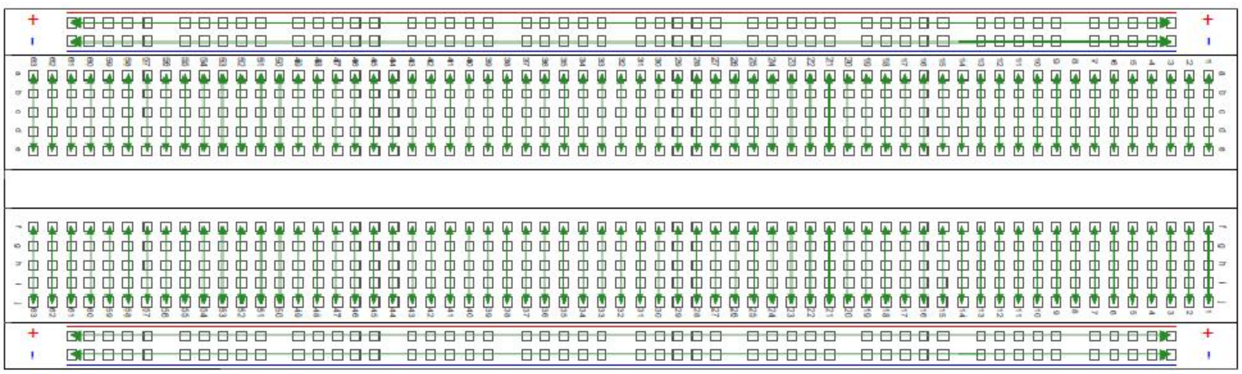
# INDEX

# INTRODUCTION ABOUT CIRCUITS AND SIMULATION LAB

## CIRCUITS LAB:

### Bread Board:

In order to build the circuit, a digital design kit that contains a power supply, switches for input, light emitting diodes (LEDs), and a breadboard will be used. Make sure to follow your instructor's safety instructions when assembling, debugging, and observing your circuit. You may also need other items for your lab such as: logic chips, wire, wire cutters, a transistor, etc.
A common breadboard, while Exhibit how each set of pins are tied together electronically. A fairly complex circuit built on a breadboard. For these labs, the highest voltage used in your designs will be five volts or +5V and the lowest will be 0V or ground.



Bread Board

The breadboard is typically a white piece of plastic with lots of tiny little holes in it. You stick wires and component leads into the holes to make circuits. Some of the holes are already electrically connected with each other. The holes are 0.1 inch apart, which is the standard spacing for leads on integrated circuit dual in-line packages. You will verify the breadboard internal connections in this lab.

### Key points to use the breadboard:

- Keep the power off when wiring the circuit.
- Make sure to keep things neat, as you can tell from Exhibit bread board, it is easy for designs to get complex and as a result become difficult to debug.
- Do not strip more insulation off of the wires used than is necessary. This can cause wires that are logically at different levels to accidentally touch each other. This creates a short circuit.

- Do not push the wires too far into each hole in the breadboard as this can cause two different problems.
- The wire can be pushed so far that only the insulation of the wire comes into contact with the breadboard, causing an open circuit.
- Too much wire is pushed into the hole; it curls under and ends up touching another component at a different logical level. This causes a short circuit.
- Use the longer outer rows for +5V on one side and ground on the other side.
- Wire power to the circuit first using a common color (say red) for +5V and another (black) for ground.

## CIRCUIT SIMULATION

A common tool (computer aided design or CAD / electronic design automation or EDA software) for the electronic circuit designer is circuit simulation software. Although most often called simply a simulator, it is a software application that typically may include many functions beyond electrical circuit simulation, including schematic capture, printed circuit board layout, and bill of materials generation.

Most circuit simulator software grew out of a public domain program called SPICE (Simulation Program with Integrated Circuit Emphasis) developed at UC Berkeley in the 1970s. The original SPICE program operated in a batch mode and was text based. That is, the user created a text file which described the circuit using special circuit netlist syntax. This file also included simulation directives which told the software what type of simulation is to be performed. The SPICE program read the input file, performed the appropriate analyses, and produced a text output file that contained the results.

Over time EDA companies began adding graphical “back-ends” that could produce better looking graphs and plots of the simulation results. A next obvious step was to add a graphical interface for building the circuit (GUI). This had the dual benefit of both describing the circuit for the simulation engine (generating the SPICE netlist) and allowing for the production of publication quality schematic diagrams. Some of the early popular graphical versions included PSpice and Electronics Workbench (EW being the precursor to Multisim).

More recent features include instrumentation simulation. That is, simulations of real world commercial measurement devices may be used as part of the circuit simulation. In this way, a sort of “virtual lab bench” may be created. With this feature, the circuit being designed will look very similar to the actual circuit sitting on your lab bench. That is, if a transistor is used in the simulation, it will look like a real transistor instead of the standard schematic symbol. While this may initially appear to be very useful, especially for beginners, in practical terms it sometimes slows down the design process by making the schematic less clear and more cluttered to the user.

**PSPICE SOFTWARE PROCEDURE.**

1. Run the CAPTURE (SPICE) program.
2. Select File/New/Project from the File menu.
3. On the New Project window select Analog or Mixed A/D, and give a name to your project then click OK.
4. The Create PSpice Project window will pop up, select Create a blank project, and then click OK.
5. Now you will be in the schematic environment where you are to build your circuit.
6. Select Place/Part from the Place menu.
7. Click ANALOG from the box called Libraries: then look for the part called R. You can do it either by scrolling down on the Part List: box or by typing R on the Part box. Then click OK.
8. Use the mouse to place the resistor where you want and then click to leave the resistor there. You can continue placing as many resistors as you need and once you have finished placing the resistors right-click your mouse and select end mode.

9. To rotate the components there are two options:
   - Rotate a component once it is placed: Select the component by clicking on it then Ctrl-R
   - Rotate the component before it is placed: Just Ctrl-R.

10. Select Place/Part from the Place menu.

11. Click SOURCE from the box called Libraries:, then look for the part called VDC. You can do it either by scrolling down on the Part List: box or by typing VDC on the Part box, and then click OK. Place the Source.

12. Repeat steps 10 - 12 to get and place a current source named $I_{DC}$.

13. Select Place/Wire and start wiring the circuit. To start a wire click on the component terminal where you want it to begin, and then click on the component terminal where you want it to finish. You can continue placing wires until all components are wired. Then right-click and select end wire.

14. Select Place/Ground from the Place menu, click on GND/CAPSYM. Now you will see the ground symbol. Type 0 on the Name: box and then click OK. Then place the ground. Wire it if necessary.

16. Now change the component values to the required ones. To do this you just need to double-click on the parameter you want to change. A window will pop up where you will be able to set a new value for that parameter.

17. Once you have finished building your circuit, you can move on to the next step – prepare it for simulation.

18. Select PSpice/New Simulation Profile and type a name, this can be the same name as your project, and click Create.

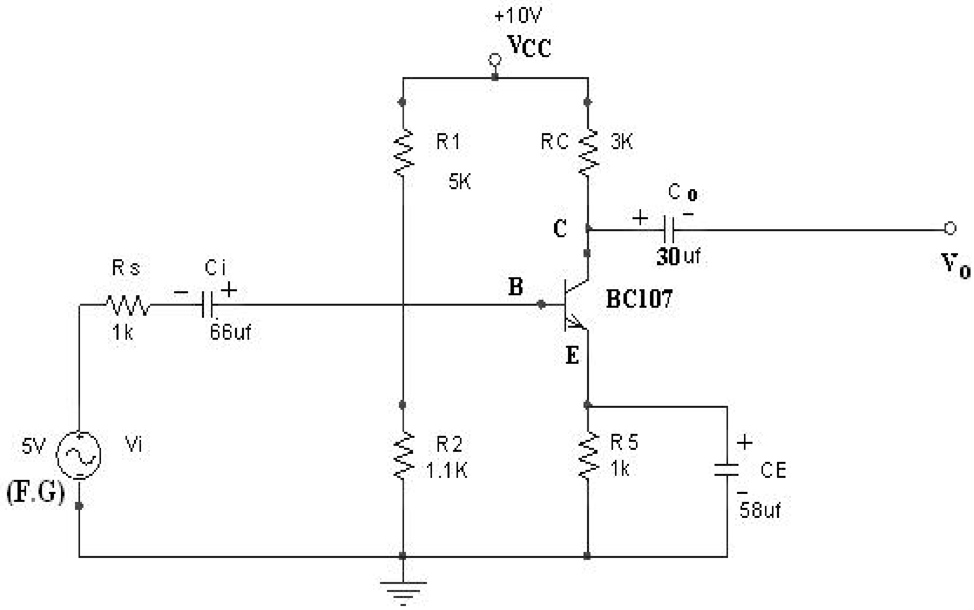
19. The Simulations Settings window will now appear. You can set up the type of analysis you want PSpice to perform. In this case it will be Bias Point. Click Apply then OK.

20. Now you are ready to simulate the circuit. Select PSpice/Run and wait until the PSpice finishes. Go back to Capture and see the voltages and currents on all the nodes.

21. If you are not seeing any readout of the voltages and currents then select PSpice/Bias Point/Enable Bias Voltage Display and PSpice/Bias Point/Enable Bias Current Display. Make sure that PSpice/ Bias Point/Enable is checked.

## CIRCUIT DIAGRAM: (VOLTAGE SHUNT FEEDBAK AMPLIFIER)

**(a) Without Feedback:**




## TABULATION:

**(Without feedback)**

| FREQUENCY | OUTPUT $V_O$(V) | $V_{in}$(V) | Gain= $20\log(V_o/V_{in})$ dB |
|---|---|---|---|
| | | | |

| Ex. No: 1 | VOLTAGE SHUNT FEEDBAK AMPLIFIER |
|---|---|
| Date: | |

**AIM:**

To design and study the frequency response of voltage shunt feedback amplifier for the given specifications $V_{CC}$ = 10V, $I_C$ =1.2mA, $A_V$= 30, $f_I$ = 1 kHz, S=2, $h_{FE}$= 150, β=0.4

**APPARATUS REQUIRED:**

| S. No | APPARATUS | RANGE | QUANTITY |
|---|---|---|---|
| 1 | FG | (0-3)MHz | 1 |
| 2 | CRO | (0-30)MHz | 1 |
| 3 | Resistors | 3kΩ, 1.1 kΩ,5kΩ,2.5 kΩ<br>1kΩ | Each 1<br>2 |
| 4 | Power supply | (0-30)V | 1 |
| 5 | Transistors | BC 107 | 1 |
| 6 | Capacitors | 66µF, 30µF,58 µf | 1 |

**Design example:**

Given specifications:

$V_{CC}$= 10V, $I_C$=1.2mA, $A_V$= 30, $f_I$ = 1 kHz, S=2, $h_{FE}$= 150, β=0.4

**(i) To calculate $R_C$:**

The voltage gain is given by,

$A_V$= -hfe ($R_C$|| $R_F$) / hie

hie = β re

Re = 26mV / $I_E$ = 26mV / 1.2mA = 21.6

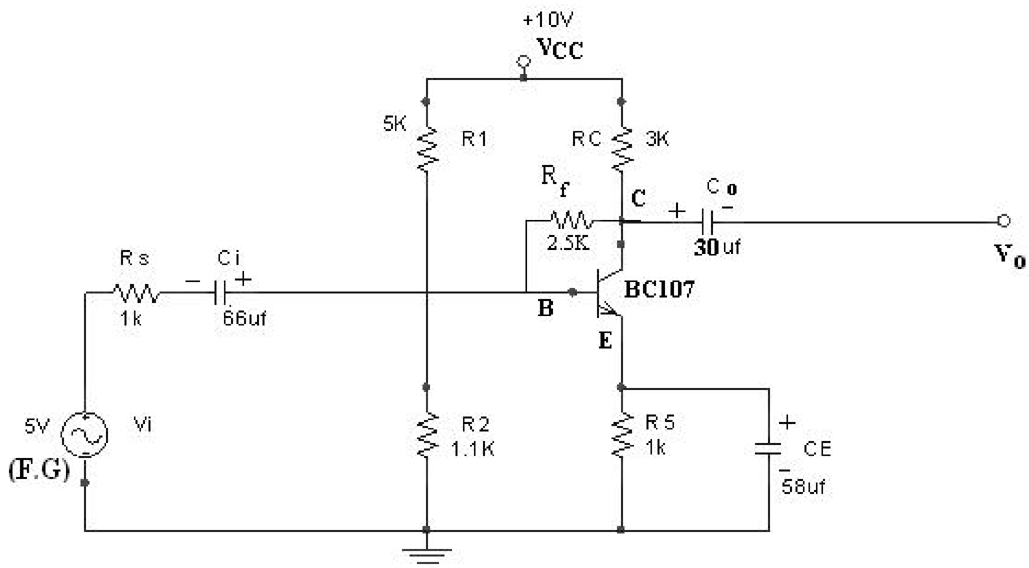
hie = 150 x 21.6 =3.2K

Apply KVL to output loop, $V_{CC}= I_C R_C + V_{CE}+ I_E R_E$ ----- (1)

Where $V_E = I_E R_E$ here ($I_C \sim I_E$)

**CIRCUIT DIAGRAM: (VOLTAGE SHUNT FEEDBAK AMPLIFIER)**

**(b)With Feedback:**




**TABULATION:**

**(With feedback)**

| FREQUENCY | OUTPUT $V_O$(V) | $V_{in}$(V) | Gain in dB= $20\log(V_o/V_{in})$ dB |
|---|---|---|---|
| | | | |

$V_E = V_{CC} / 10 = 1V$

Therefore $R_E = 1/1.2x10^{-3} = 0.8K = 1K\Omega$

$V_{CE} = V_{CC} / 2 = 5V$

From equation (1), $R_C = 3\ K\Omega$

**(ii) <u>To calculate $R_1$&$R_2$:</u>**

$S = 1 + (R_B/R_E)$

$R_B = (S-1)\ R_E = R_1 \parallel R_2 = 1K\Omega$

$R_B = R_1 R_2 / R_1 + R_2$------- (2)

$V_B = V_{BE} + V_E = 0.7 + 1 = 1.7V$

$V_B = V_{CC}\ R_2 / R_1 + R_2$ ------- (3)

Solving equation (2) & (3), $R_1 = 5\ K\Omega$ & $R_2 = 1.1K\Omega$

**(iii) <u>To calculate Resistance:</u>**

Output resistance is given by,

$R_O = R_C \parallel R_F$

$R_O = 1.3K\Omega$

Input impedance is given by,

$R_i = (R_B \parallel R_F) \parallel hie = 0.6K\Omega$

Trans-resistance is given by,

$R_m = -hfe\ (R_B \parallel R_F)\ (R_C \parallel R_F) / (R_B \parallel R_F) + hie$

$R_m = 0.06K\Omega$

**<u>AC parameter with feedback network:</u>**

**(i) <u>Input Impedance</u>**:

$R_{if} = R_i / D$ (where $D = 1 + \beta\ Rm$)

Therefore D = 25

$R_{if} = 24$

Input coupling capacitor is given by,

$Xci = R_{if} / 10 = 2.4$ (since $X_{Ci} << R_{if}$)

$Ci = 1/ 2\pi f X_{Ci} = 66\mu f$

**(ii) <u>Output impedance:</u>**

$R_{Of} = R_O / D = 52$

Output coupling capacitor:

$X_{CO} = Rof / 10 = 5.2$

$C_O = 1/ 2\pi f X_{CO} = 30\mu f$

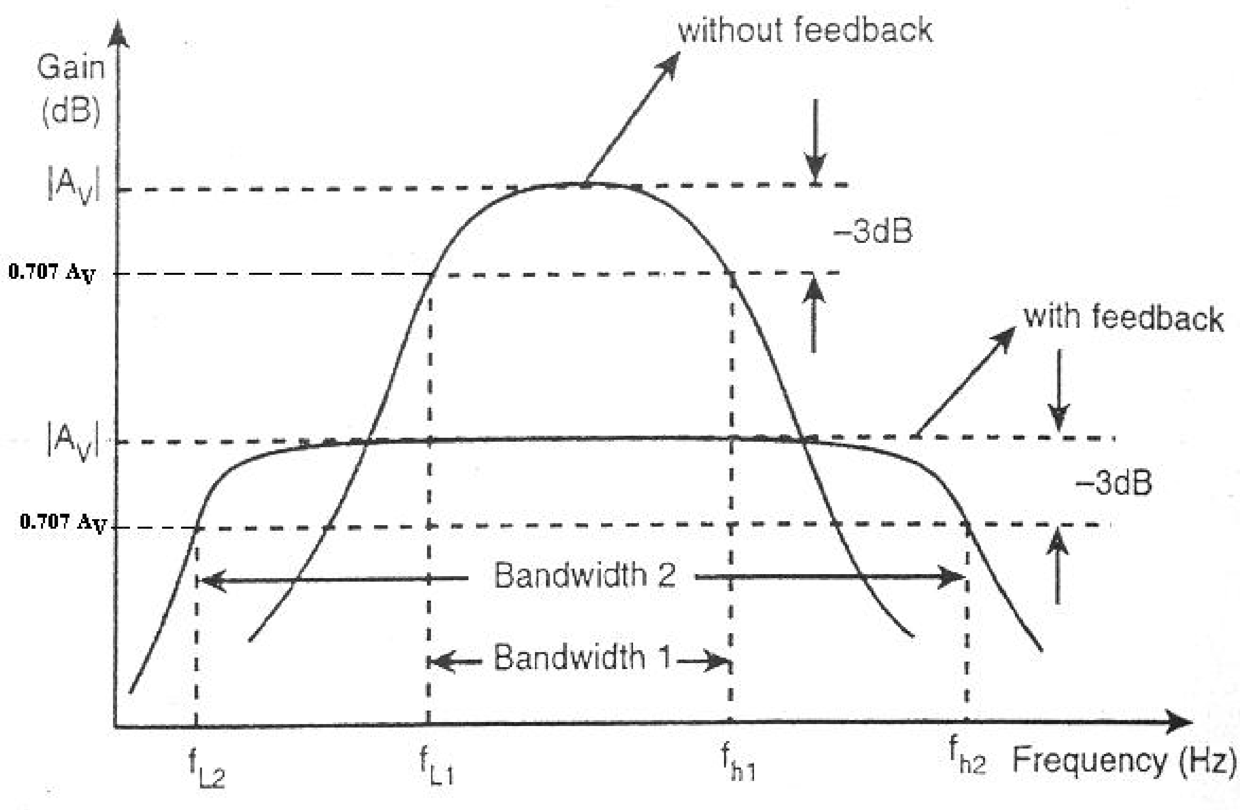
**(iii) <u>Emitter capacitance:</u>**

$X_{CE} << R'_E = R'/10$

$R'_E = R_E \parallel \{(hie + R_B) / (1+hfe)\}$

$X_{CE}$= 2.7 Therefore $C_E$= 58µf

**MODEL GRAPH: (VOLTAGE SHUNT FEEDBAK AMPLIFIER)**



**THEORY:**

Negative feedback in general increases the bandwidth of the transfer function stabilized by the specific type of feedback used in a circuit. In Voltage shunt feedback amplifier, consider a common emitter stage with a resistance R' connected from collector to base. This is a case of voltage shunt feedback and we expect the bandwidth of the Trans resistance to be improved due to the feedback through R'. The voltage source is represented by its Norton's equivalent current source $I_S$=Vs/Rs.

**PROCEDURE:**

1. Connect the amplifier without feedback circuit as per the circuit diagram.
2. Set $V_{CC}$ = 10V; set input voltage using audio frequency oscillator.
3. By varying audio frequency oscillator take down output frequency oscillator voltage for difference in frequency.
4. Calculate the gain in dB.
5. Plot gain Vs frequency curve in semi-log sheet.
6. Connect the amplifier with feedback circuit as per the circuit diagram.
7. Set $V_{CC}$ = 10V; set input voltage using audio frequency oscillator.
8. By varying audio frequency oscillator take down output frequency oscillator voltage for difference in frequency.
9. Calculate the gain in dB
10. Plot gain Vs frequency curve in semi-log sheet.
11. Compare response with respect to the amplifier with and without feedback.

**Abbreviations:**

FG – Function Generator.

CRO- Cathode Ray Oscilloscope.

$V_O$ – Output Voltage; Vi – Input Voltage.

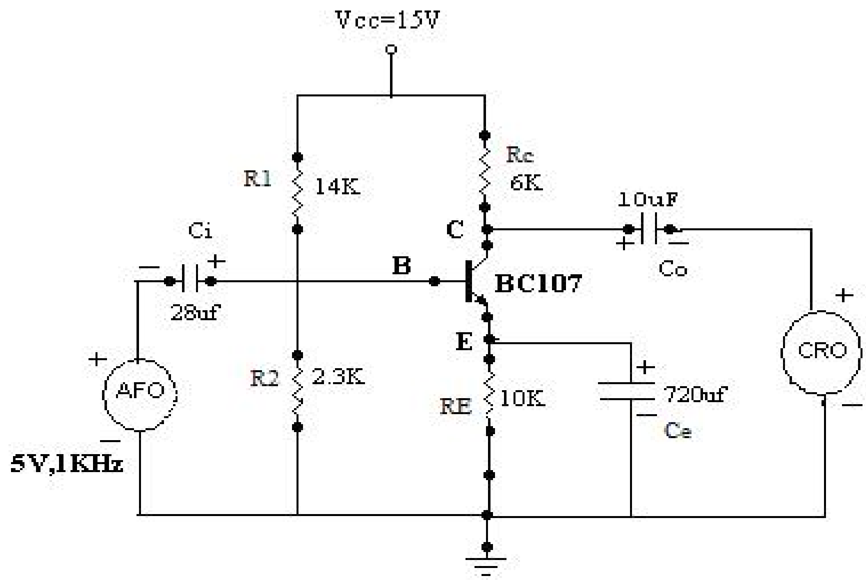
dB – Decibel unit

**RESULT:**

Thus voltage shunt feedback amplifier is designed and Bandwidth is calculated.

**CIRCUIT DIAGRAM: (CURRENT SERIES FEEDBACK AMPLIFER)**

(a) **Without Feedback:**




**TABULATION:**

**(Without feedback)**

**Vin = ----- (V)**

| FREQUENCY (in Hz) | OUTPUT $V_O$(V) | Gain in dB= $20\log(V_o/V_{in})$ dB |
|---|---|---|
| | | |

| **Ex. No: 2** | **CURRENT SERIES FEEDBACK AMPLIFER** |
|---|---|
| **Date:** | |

**AIM:**

To design a current series feedback amplifier for the given specifications $V_{CC}$= 15V, $I_C$=1mA, $A_V$= 30, $f_L$= 50Hz, S=3, $h_{FE}$= 100, hie= 1.1KΩ and draw its frequency response.

**APPARATUS REQUIRED:**

| S. No | APPARATUS | RANGE | QUANTITY |
|---|---|---|---|
| 1 | AFO | (0-3)MHz | 1 |
| 2 | CRO | (0-30)MHz | 1 |
| 3 | Resistors | 6KΩ,14kΩ,2.3KΩ,10KΩ | Each 1 |
| 4 | RPS | (0-30V) | 1 |
| 5 | Transistors | BC 107 | 1 |
| 6 | Capacitors | 28µF, 10µF,720µF | 1 |
| 7 | Connecting wires | - | few |

**Design example:**

$V_{CC}$= 15V, $I_C$=1mA, $A_V$= 30, $f_L$= 50Hz, S=3, $h_{FE}$= 100, hie= 1.1KΩ

Gain formula is,

$A_V = - h_{FE} R_{Leff} / hie$

Assume, $V_{CE} = V_{CC} / 2$ (transistor in active region)

$V_{CE}$ = 15 /2=7.5V

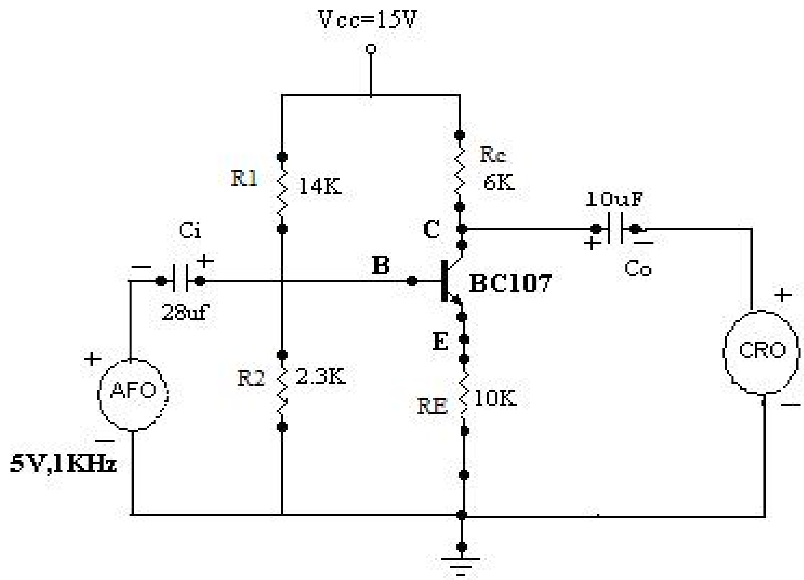
$V_E = V_{CC}$ / 10= 15/10=1.5V

Emitter resistance is given by, re =26mV/ $I_E$

Therefore $r_e$ =26 Ω

hie= hfe re hie =2.6KΩ

**WITH FEEDBACK: (CURRENT SERIES FEEDBACK AMPLIFER)**




**TABULATION:**

$V_{in}$ = ----- (V)

| FREQUENCY | OUTPUT $V_O$(V) | Gain $20\log(V_o/V_{in})$ dB |
|---|---|---|
| | | |

**(i) To calculate $R_C$:**

Applying KVL to output loop,

$V_{CC}= I_C R_C + V_{CE}+ I_E R_E$ ----- (1)

Where $R_E = V_E / I_E$ ($I_C= I_E$)

$R_E = 1.5 / 1x10-3= 1.5KΏ$

From equation (1),

$R_C= 6KΏ$

**(ii) To calculate $R_{B1}$&$R_{B2}$:**

Since $I_B$ is small when compared with $I_C$,

$I_C \sim I_E$

$V_B= V_{BE} + V_E= 0.7 + 1.5=2.2V$

$V_B= V_{CC} (R_{B2} / R_{B1}+ R_{B2})$ ----- (2)

$S=1+ (R_B / R_E)$

$R_B= 2KΏ$

We know that $R_B= R_{B1}\| R_{B2}$

$R_B= R_{B1}R_{B2}/ R_{B1}+R_{B2}$--------- (3)

Solving equation (2) & (3),

Therefore,

$R_{B1} = 14KΏ$

From equation (3), $R_{B2}= 2.3KΏ$

**(iii) To find input coupling capacitor ($C_i$):**

$X_{Ci} = (hie\| R_B) / 10$

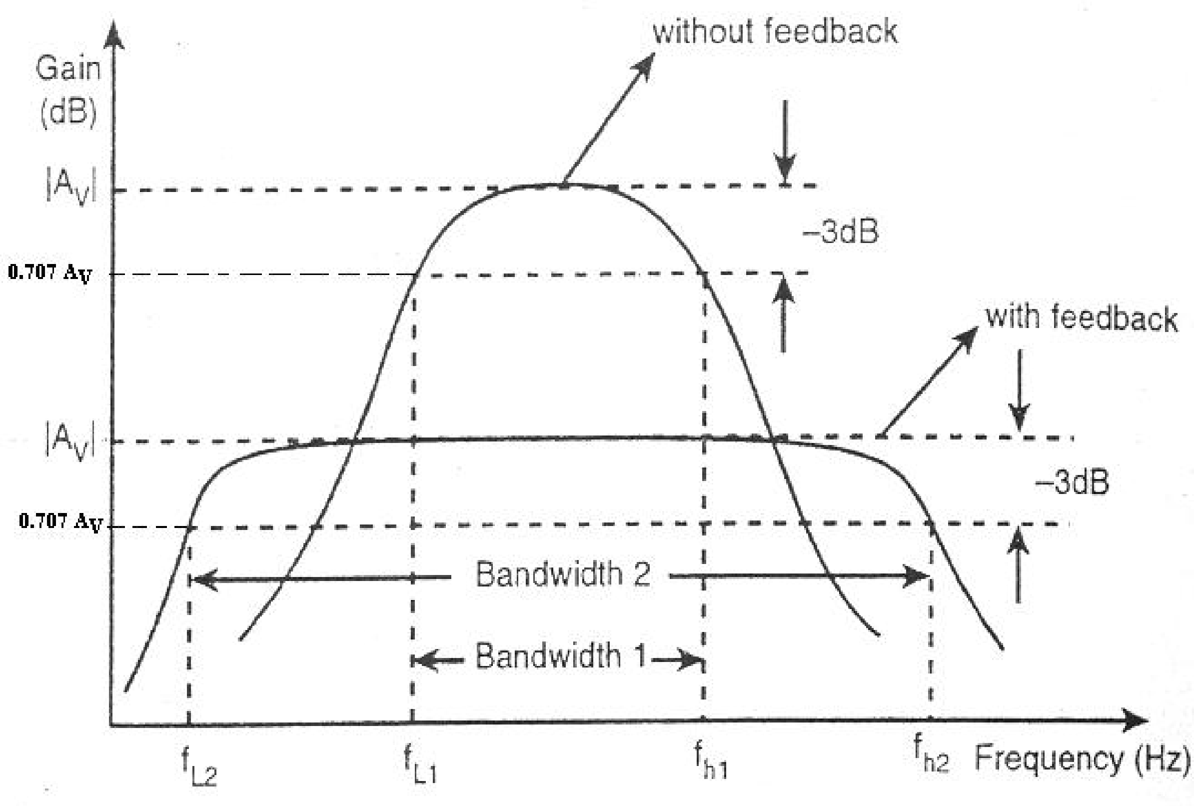
$X_{Ci} = 113$

$X_{Ci}= 1/ 2\pi f C_i$

$C_i = 1 / 2\pi f X_{Ci}$

$C_i = 1/ 2X3.14X 50 X 113=28\mu f$

**MODEL GRAPH( CURRENT SERIES FEEDBACK AMPLIFER)**



**(iv)<u>To find output coupling capacitor ($C_O$):</u>**

$X_{CO}$= ($R_C$ || $R_L$) / 10, (Assume $R_L$= 10KΩ)

$X_{CO}$= 375

$X_{CO}$= 1/ 2πf $C_O$

$C_O$ = 1/ 2x 3.14x 50 x 375=8µf =10 µf

**(v) <u>To find Bypass capacitor ($C_E$):</u>**

(Without feedback)

$X_{CE}$ = {($R_B$ + hie / 1+ hfe) || $R_E$}/ 10

$X_{CE}$ = 4.416

$C_E$= 1 / 2πf XCE

$C_E$ = 720 µf

**<u>Design with feedback:</u>**

To design with feedback remove the bypass capacitor ($C_E$).

Assume $R_E$ = 10K

**<u>THEORY:</u>**

Negative feedback in general increases the bandwidth of the transfer function stabilized by the specific type of feedback used in a circuit. In Voltage series feedback amplifier, consider a common emitter stage with a resistance R' connected from emitter to ground. This is a case of voltage series feedback and we expect the bandwidth of the Trans resistance to be improved due to the feedback through R' .The voltage source is represented by its Norton's equivalent current source Is=Vs/Rs.

**PROCEDURE:**

1. Connect the circuit as per the circuit diagram.
2. Set $V_{CC}$ = 10V; set input voltage using audio frequency oscillator.
3. By varying audio frequency oscillator take down output frequency oscillator voltage for difference in frequency.
4. Calculate the gain in dB
5. Plot gain Vs frequency curve in semi-log sheet.

**Abbreviations:**

AFO – Audio frequency oscillator.

CRO - Cathode Ray Oscilloscope.

$V_O$ – Output Voltage; Vi – Input Voltage.

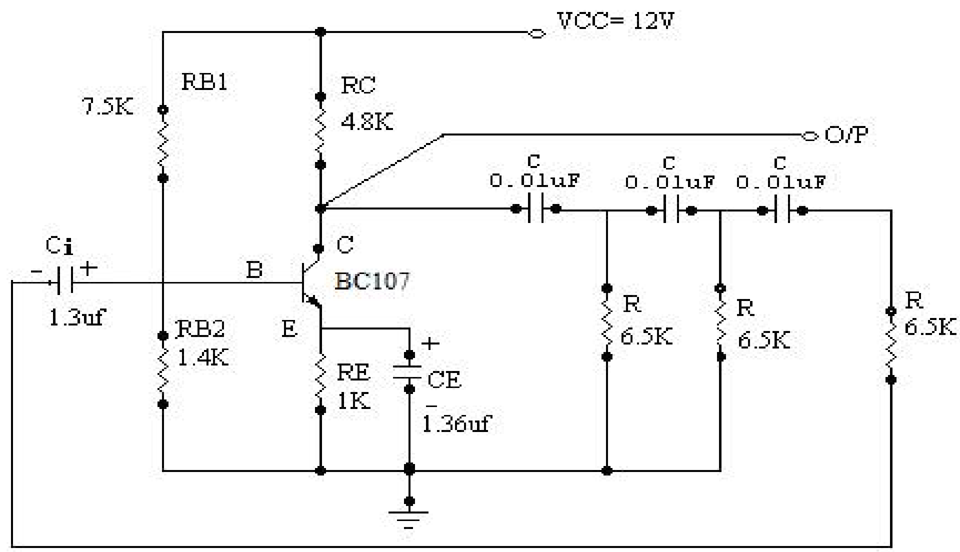
dB – Decibel unit

RPS – Regulated Power Supply.

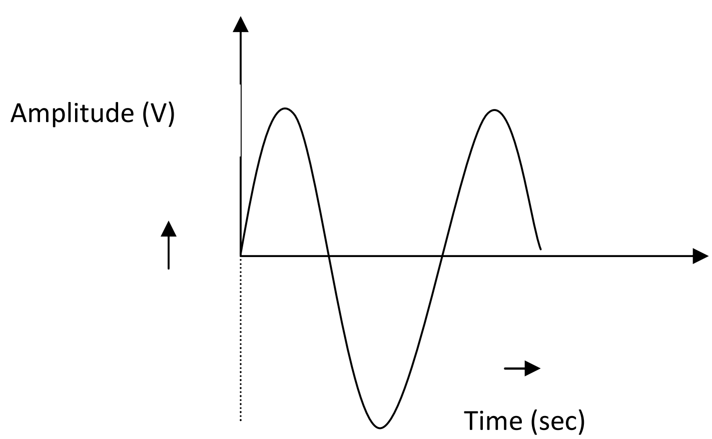
**RESULT:**

Thus current series feedback amplifier is designed and Bandwidth is calculated.

**CIRCUIT DIAGRAM:** **(RC PHASE SHIFT OSCILLATOR)**




**MODEL GRAPH:**



| Ex. No: 3<br>Date: | RC PHASE SHIFT OSCILLATOR |
|---|---|

**AIM:**

To design a RC phase shift oscillator for the given specifications: VCC = 12V, $I_Cq$ =1mA, β=100, Vce = 5V, f=1 KHz, S=10, C=0.01 µf, $h_{fe}$ = 330, $A_V$= 29 and to find the frequency of oscillation.

**APPARATUS REQUIRED:**

| S.No | APPARATUS REQUIRED | RANGE | QUANTITY |
|---|---|---|---|
| 1 | Transistor | BC107 | 1 |
| 2 | Resistors | 7.5kΩ,1.4 kΩ,4.8 kΩ ,1kΩ<br>6.5k | Each 1<br>6 |
| 3 | Power supply | (0-30)V | 1 |
| 4 | Capacitors | 1.3µf | 2 |
| 5 | CRO | (0-30)MHz | 1 |
| 6 | Bread board | - | 1 |
| 7 | Connecting wires | - | few |

**Design Example:**

**Specifications:**

$V_{CC}$ = 12V, $I_Cq$ =1mA, β=100, Vceq = 5V, f=1 KHz, S=10, C=0.01 µf, $h_{fe}$= 330, $A_V$= 29

**Design:**

**(i)To find R:**

Assume f=1 KHz, C=0.01µf

$f=1/2\Pi RC\sqrt{6}$

$R=1/2x3.14\ x1x10^{3}x0.01x10^{-6}\sqrt{6}$ =6.5KΩ

**TABULATION☹ RC PHASE SHIFT OSCILLATOR)**

| Amplitude (V) | Time period (m sec) | Frequency (Hz) |
|---|---|---|
| | | |

**(ii)To find $R_E$ & $R_C$:**

$V_{CE} = V_{CC} /2 = 6V$

$r_e = 26mV / I_E = 26\Omega$

$h_{ie} = h_{fe} r_e = 330 \times 26 = 8580\Omega$

On applying KVL to output loop,

$V_{CC} = I_C R_C + V_{CE} + I_E R_E$ ----- (1)

$V_E = I_E R_E$

$R_E = V_E / I_E = 1.2/ 10^{-3} = 1.2K\Omega$

From equation (1), $12 = 10^{-3}(R_C + 1200) + 6 = R_C = 4800\Omega = 4.8K\Omega$

**(iii)To calculate $R_1$ & $R_2$:**

$V_{BB} = V_{CC} R_2 / R_1 + R_2$ ------ (2)

$V_B = V_{BE} + V_E = 0.7+12 = 1.9V$

From equation (2), $1.9 = 12 R_2 / R_1 + R_2$

$R_2 / R_1 + R_2 = 0.158$ -------- (3)

$S = 1 + R_B / R_E = R_B = 1.2K\Omega$

$R_B = R_1 || R_2$

$0.15R_1 = 1.2 \times 10^{-3} = 7.5K\Omega$

$R_2 = 0.158 R_1 + 0.158 R_2, R_2 = 1.425K\Omega$

**(iv)<u>To calculate Coupling capacitors</u>:**

(i) $X_{Ci}= \{[h_{ie} + (1+h_{fe}) R_E] \parallel R_B \}/ 10 = 0.12K\Omega$

$X_{Ci}= 1 / 2\prod f\ C_i == 1.3\mu f$

(ii) $X_{CO}= R_{Leff} / 10$ $[\ A_V = -\ h_{fe}\ R_{Leff} / h_{ie}]$

$R_{Leff} = 0.74K\Omega,\ X_{CO}=0.075\ K\Omega$

$X_{CO}= 1 / 2\prod f\ C_O\ ,\ C_O = 2.1\mu f$

(iii) $X_{CE}= R_E / 10 = 1.326\ \mu f;\ X_{CE} = 1 / 2\prod f\ C_E=49.$

(iv) Feedback capacitor, $X_{CF} = R_f / 10\ ;C_f = 0.636\mu f = 0.01\mu f$

**<u>THEORY:</u>**

The low frequencies RC oscillators are more suitable. Tuned circuit is not an essential requirement for oscillation. The essential requirement is that there must be a $180^o$ phase shift around the feedback network and loop gain should be greater than unity. The $180^o$ phase shift in feedback signal can be achieved by suitable RC network.

**<u>PROCEDURE:</u>**

1. Connect the circuit as per the circuit diagram.
2. Set $V_{CC} = 15V$.
3. For the given supply the amplitude and time period is measured from CRO.
4. Frequency of oscillation is calculated by the formula f=1/T.
5. Amplitude Vs time graph is drawn.

**Abbreviations:**

Av – Voltage Gain.

B – Stability Factor

f - Frequency

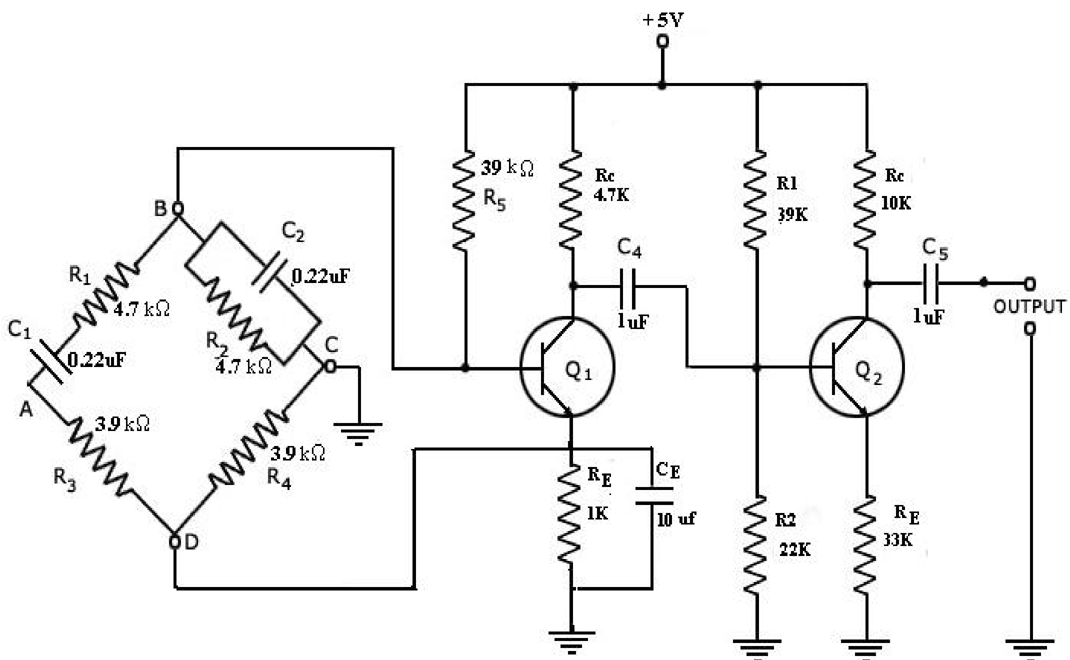
KVL – Kirchhoff Voltage Law

**<u>RESULT:</u>**

Thus the RC-phase shift oscillator is designed and constructed for the given frequency.

Frequency **:**

**CIRCUIT DIAGRAM ( WEIN- BRIDGE OSCILLATOR)**




**TABULATION:**

| Amplitude(V) | Time(msec) |
|---|---|
| | |

| Ex. No: 4 | WEIN- BRIDGE OSCILLATOR |
|---|---|
| Date: | |

**AIM:**

To design a Wein-bridge oscillator using transistors and to find the frequency of oscillation for the given operating frequency 1 KHz.

**APPARATUS REQUIRED:**

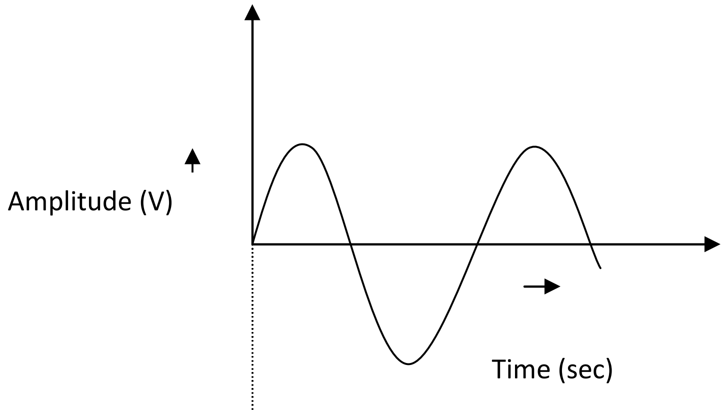
| S.No | APPARATUS REQUIRED | RANGE | QUANTITY |
|---|---|---|---|
| 1 | Resistors | 3.9 kΩ, 4.7 kΩ, 39kΩ, 1kΩ, 10kΩ,22KΩ,33kΩ | 2,3,2,1,1,1,1 |
| 2 | Power supply | (0-30)V | 1 |
| 3 | Transistor | BC107 | 2 |
| 4 | Capacitors | 0.22μF,1μF | Each 2 |
| 5 | CRO | (0-30)MHz | 1 |
| 6 | Bread board | - | 1 |
| 7 | Connecting wires | - | few |

**THEORY:**

Generally in an oscillator, amplifier stage introduces $180^o$ phase shift and feedback network introduces additional $180^o$ phase shift, to obtain a phase shift of $360^o$ around a loop. This is a condition for any oscillator. But Wein bridge oscillator ses a non-inverting amplifier and hence does not provide any phase shift during amplifier stage. As total phase shift requires is $0^o$ or $2n\pi$ radians, in Wein bridge type no phase shift is necessary through feedback. Thus the total phase shift around a loop is $0^o$.

The output of the amplifier is applied between the terminals 1 and 3, which are the input to the feedback network. While the amplifier input is supplied from the diagonal terminals 2 and 4, which is the output from the feedback network. Thus amplifier supplied its own output through the Wein Bridge as a feedback network.

**MODEL GRAPH:**



The two arms of the bridge, namely $R_1$, $C_1$ in series and $R_2$, $C_2$ in parallel are called frequency sensitive arms. This is because the components of these two arms decide the frequency of the oscillator.

**PROCEDURE:**

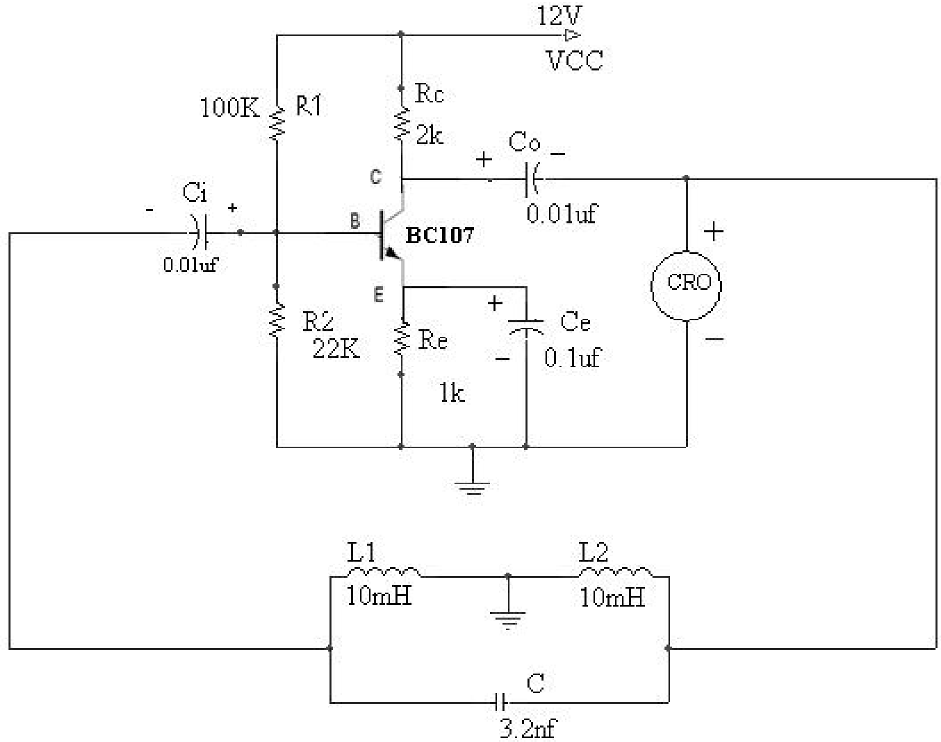
1. Connect the circuit as per the circuit diagram.
2. Set $V_{CC}$ = 5V.
3. For the given supply the amplitude and time period is measured from CRO.
4. Frequency of oscillation is calculated by the formula f=1/T
5. Amplitude Vs Time graph is drawn.

**RESULT:**

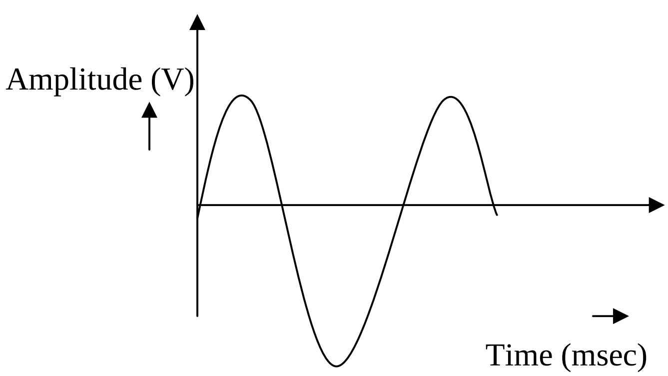
Thus the Wein – bridge oscillator is designed for the given frequency of oscillation.

Frequency **:**

**CIRCUIT DIAGRAM: (HARTLEY OSCILLATOR)**




**MODEL GRAPH**:



| Ex.No.5 | HARTLEY OSCILLATOR |
|---|---|
| Date: | |

**AIM:**

To design and construct a Hartley oscillator for the given specifications $L_1$= $L_2$=10mH, f=20 KHz, $V_{CC}$=12V, $I_C$=3mA, S=12

**APPARATUS REQUIRED:**

| S.No | APPARATUS REQUIRED | RANGE | QUANTITY |
|---|---|---|---|
| 1 | Resistors | 2kΩ,1KΩ,100 kΩ,22kΩ | Each one |
| 2 | RPS | (0-30)V | 1 |
| 3 | Transistor | BC107 | 1 |
| 4 | Capacitors | 3.2nf,0.1µF, 0.01µF | Each 1 |
| 5 | Inductor | 10mH | 2 |
| 6 | CRO | (0-30)MHz | 1 |
| 7 | Bread board | - | 1 |
| 8 | Connecting wires | - | few |

**Design Example:**

**Design of feedback Network:**

Given $L_1$= $L_2$=10mH, f=20 KHz, $V_{CC}$=12V, $I_C$=3mA, S=12

**Frequency Formula:**

$F=1/2\Pi\sqrt{LeqC}$ , Where, Leq = $L_1+L_2$

Therefore,

$F = 1/2\Pi\sqrt{(L1+L2)C}$

C= 3.2nf

## TABULATION( HARTLEY OSCILLATOR)

| Amplitude(V) | Time(msec) | Frequency(Hz) |
|---|---|---|
| | | |

**THEORY:**

Hartley oscillator is very popular and is commonly used as local oscillator in radio receivers. The collector voltage is applied to the collector through inductor L whose reactance is high compared with $X_2$ and may therefore be omitted from equivalent circuit, at zero frequency, however capacitor $C_b$ acts as an open circuit.

**PROCEDURE:**

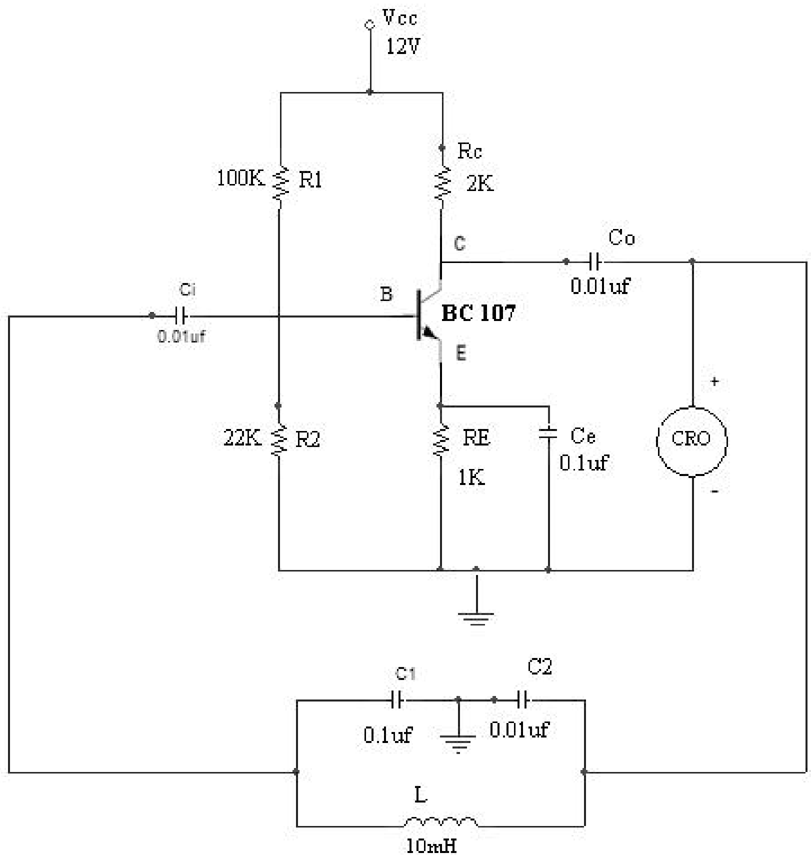
1. Connect the circuit as per the circuit diagram.
2. Set $V_{CC}$ = 12V.
3. For the given supply the amplitude and time period is measured from CRO.
4. Frequency of oscillation is calculated by the formula F=1/T
5. Verify it with theoretical frequency, F= 1/2Π ( $\sqrt{(L1+L2)C}$ )
6. Amplitude Vs time graph is drawn.

**RESULT:**

Thus the Hartley oscillator is designed and constructed for the given frequency.

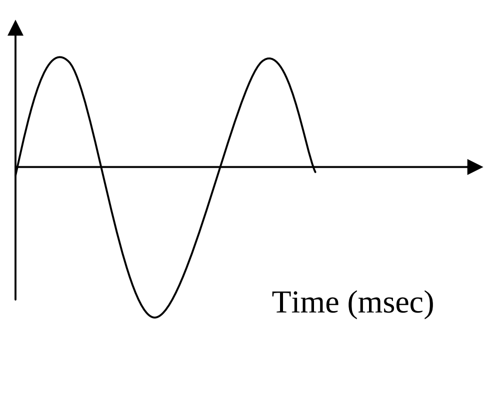
Frequency :

**CIRCUIT DIAGRAM: (COLPITT'S OSCILLTOR)**




**MODEL GRAPH:**

Amplitude (V)



Time (msec)

| Ex. No: 6 | COLPITT'S OSCILLTOR |
|---|---|
| Date: | |

**AIM:**

To design and construct a Colpitt's oscillator for the given specifications $C_1$= 0.1 μF, L=10mH, f=20 KHz, $V_{CC}$=12V,$I_C$=3mA, S=12.

**APPARATUS REQUIRED:**

| S. No | APPARATUS | RANGE | QUANTITY |
|---|---|---|---|
| 1 | Resistors | 2kΩ,1KΩ,100 kΩ, 22kΩ | Each One |
| 2 | RPS | (0-30)V | 1 |
| 3 | Transistor | BC107 | 1 |
| 4 | Capacitors | 0.1μF, 0.01μF | Each 2 |
| 5 | Inductor | 10mH | 1 |
| 6 | CRO | (0-30)MHz | 1 |
| 7 | Bread board | - | 1 |
| 8 | Connecting wires | - | few |

**Design of feedback Network:**

Given $C_1$= 0.1 μF, $C_2$= 0.01 μF, L=10mH, f=20 KHz, $V_{CC}$=12V.

**Frequency Formula:**

$$F = 1/\, 2\Pi \sqrt{LCeq} \quad \text{, Where } C_{eq} = \frac{C_1 C_2}{C_1 + C_2}$$

$$F = (1\,/\,2\Pi)\sqrt{\frac{C1 + C2}{LC1C2}} \quad , \quad C2 = 0.01\mu F$$

**TABULATION:**

| Amplitude(V) | Time( msec ) | Frequency(Hz) |
|---|---|---|
| | | |

## THEORY:

Colpitt's oscillator is very popular and is commonly used as local oscillator in radio receivers. The collector voltage is applied to the collector through inductor L whose reactance is high compared with $X_2$ and may therefore be omitted from equivalent circuit, at zero frequency. The circuit operates as Class C. the tuned circuit determines basically the frequency of oscillation.

## PROCEDURE:

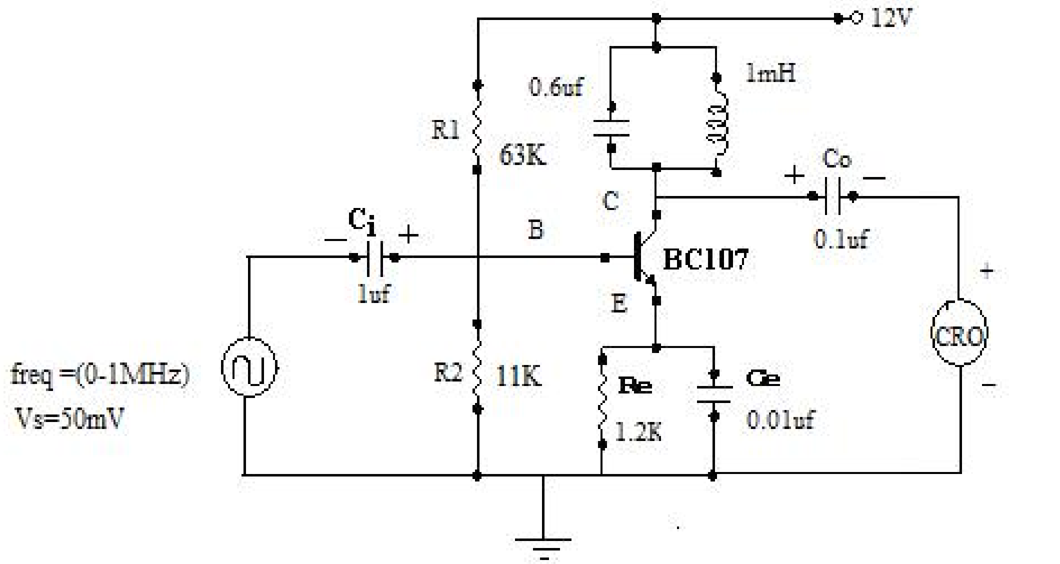
1. Connect the circuit as per the circuit diagram.
2. Set $V_{CC}$ = 12V.
3. For the given supply the amplitude and time period is measured from CRO.
4. Frequency of oscillation is calculated by the formula f=1/T
5. Amplitude Vs time graph is drawn.

## RESULT:

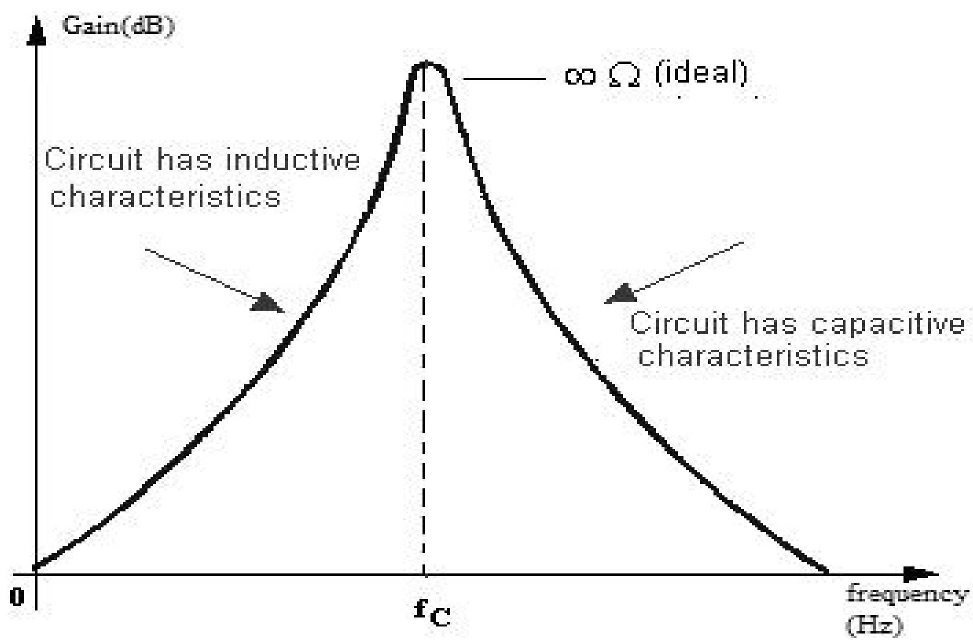
Thus the colpitt's oscillator is designed and constructed for the given frequency.

Frequency **:**

**CIRCUIT DIAGRAM:** (single tuned amplifier)




**MODEL GRAPH:**



| Ex.No:7 | SINGLE TUNED AMPLIFIER |
|---|---|
| Date: | |

**AIM:**

To design a single tuned amplifier for the given specifications Vcc = 12V, β = 100, $I_c$ = 1mA, L=1mH, f=2 KHz, S= [2-10] and to draw its frequency response.

**APPARATUS REQUIRED:**

| S. No | APPARATUS REQUIRED | RANGE | QUANTITY |
|---|---|---|---|
| 1 | Resistors | 11kΩ,63 kΩ,1.2kΩ | Each one |
| 2 | RPS | (0-30)V | 1 |
| 3 | Transistor | BC107 | 1 |
| 4 | Capacitors | 0.1μf ,0.6μf,1 μf , 0.01 μf | Each one |
| 5 | Inductance | 1mH | 1 |
| 6 | CRO | (0-30)MHz | 1 |
| 7 | Function generator | (0-3)MHz | 1 |
| 8 | Bread board | - | 1 |

**DESIGN EXAMPLE:**

**Given specifications:** Vcc = 12V, β = 100, Ic = 1mA, L=1mH, f=2 KHz, S= [2-10]

Assume, $V_{CE} = V_{CC} / 2 = 6V$

$V_E = V_{CC} / 10 = 1.2V$

**(i) To calculate C:**

$$F = 1/ 2\Pi \sqrt{LC}$$

Therefore C=0 .6μf

**TABULATION:**

| FREQUENCY (Hz) | OUTPUT $V_O$(V) | $A_V = V_o/V_{in}$ | Gain in dB= $20\log A_V$ (dB) |
|---|---|---|---|
| | | | |

**(ii) To calculate $R_E$:**

The voltage drop across emitter resistance is given as ,

$V_E = I_E R_E$ , Where $(I_C \sim I_E)$

$R_E = V_E / I_E = 1.2 / 1x10^{-3} = 1.2K\Omega$

Assume S= 10, S= 1+ $(R_B / R_E)$ & So, $R_B = 10.8\ K\Omega$

**(iii) To find $R_1$ & $R_2$:**

$R_B = R_1 \| R_2$

$R_B = R_1 R_2 / R1 + R_2$ ------------- (1)

By applying voltage divider rule,

$V_B = V_{CC}$ x $(R_2 / R_1 + R_2)$ ------ (2)

From equation 1 & 2

$R_B / R_1 = V_B / V_{CC}$ --------------- (3)

$V_B = V_{BE} + V_E$ ($V_{BE} = 0.7$ for silicon transistor)

$V_B = 0.7 + 1.2 = 1.9V$

Substitute values in equation 3, $R_1 = 63K$

From equation 2, $R_2 = 11K \approx 10K$

**(iv) To calculate coupling capacitors:**

**(i)Input capacitance ($C_i$):**

$X_{Ci} = \{[hie + (1+ h_{fe}) R_E] \| R_B\} /10$ **(OR)** $R_B /10$

$X_{Ci} = 10.8k/10 = 1.08k$

$X_{Ci} = 1/ 2\Pi f C_i$

Therefore $C_i = 0.7\ \mu f \approx 1\mu f$

**(ii) Output capacitance ($C_O$):**

$X_{CO} = R_C \parallel R_L / 10 \approx R_L / 10$

$X_{CO} = 1000$ [since $R_L = 10K$]

$X_{CO} = 1/ 2\Pi f C_O$

$C_O = 0.0796$ μf ≈ 0.1 μf

**(iii) Bypass capacitance ($C_E$):**

$X_{CE} = R_E / 10$

$X_{CE} = 120$

$X_{CE} = 1/ 2\Pi f C_E$

$C_E = 0.06$ μf≈ 0.01 μf

**THEORY:**

The single tuned amplifier selecting the range of frequency the resistance load replaced by the tank circuit. Tank circuit is nothing but inductors and capacitor in parallel with each other. The tuned amplifier gives the response only at particular frequency at which the output is almost zero. The resistor R1 and R2 provide potential diving biasing, Re and Ce provide the thermal stabilization. This it fixes up the operating point.

**PROCEDURE:**

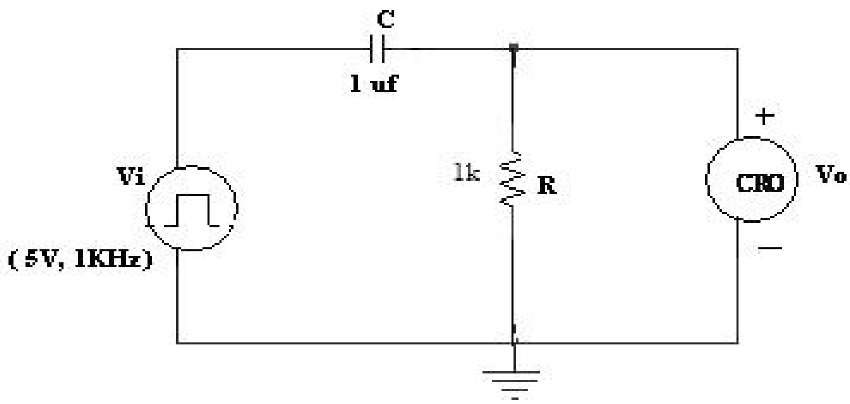
1. Connections are given as per the circuit diagram
2. By varying the frequency, amplitude is noted down
3. Gain is calculated in dB
4. Frequency response curve is drawn.

**RESULT:**

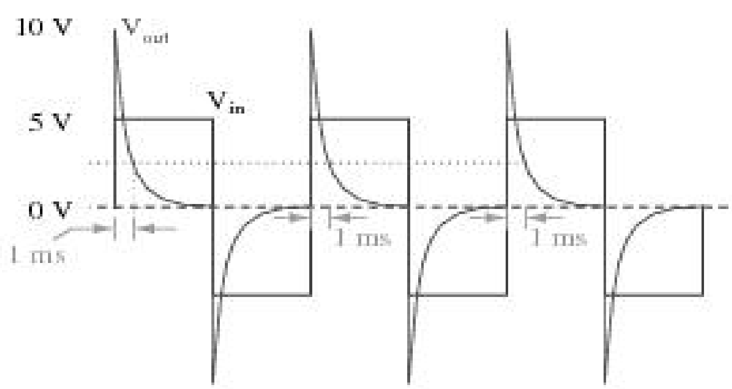
Thus the class – C single tuned amplifier is designed and frequency response is plotted.

## CIRCUIT DIAGRAM:

### (a) DIFFERENTIATOR:




## MODEL GRAPH:



| **Ex. No: 8** | **INTEGRATOR AND DIFFERENTIATOR** |
|---|---|
| **Date:** | |

**AIM:**

To design and construct, differentiator and integrator circuit.

**APPARATUS REQUIRED:**

| S. No. | APPARATUS REQUIRED | RANGE | QUANTITY |
|---|---|---|---|
| 1. | Function generator | (0-3)MHz | 1 |
| 2. | CRO | (0-30)MHz | 1 |
| 3. | Capacitor | 1µf | 1 |
| 4. | Resistor | 1KΩ | 1 |
| 5. | Bread board | - | 1 |

**THEORY:**

**Differentiator:**

Differentiator is a circuit which differentiates the input signal, it allows high order frequency and blocks low order frequency. If time constant is very low it acts as a differentiator. In this circuit input is continuous pulse with high and low value.

**Integrator:**

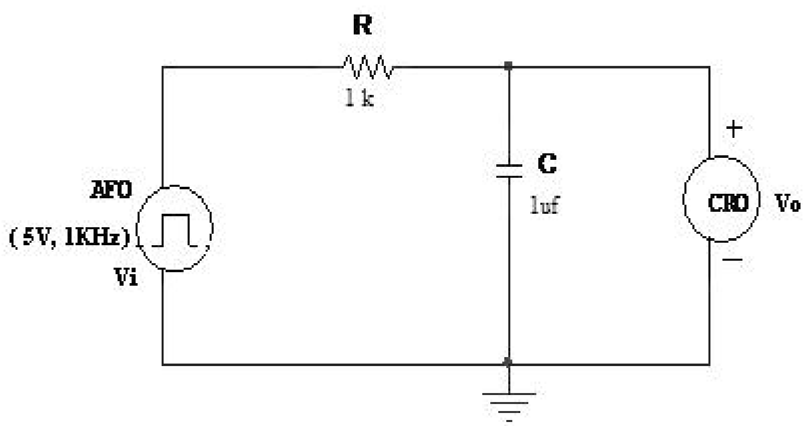
In a low pass filter when the time constant is very large it acts as an integrator. In this the voltage drop across C will be very small in comparison with the drop across resistor R. Therefore, the total input appears across the R.

**PROCEDURE:**

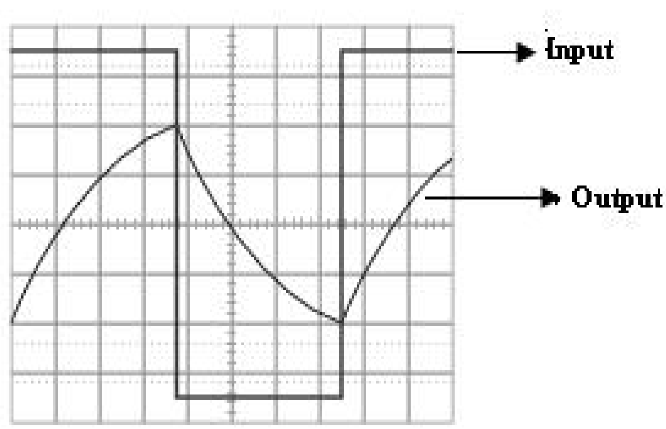
1. Connections are given as per the circuit diagram.
2. Set the signal voltage Vi (5V, 1KHz) in the function generator.
3. Observe the output waveform in the CRO.
4. Sketch the output waveform.

**CIRCUIT DIAGRAM:**

**(b) INTEGRATOR:**



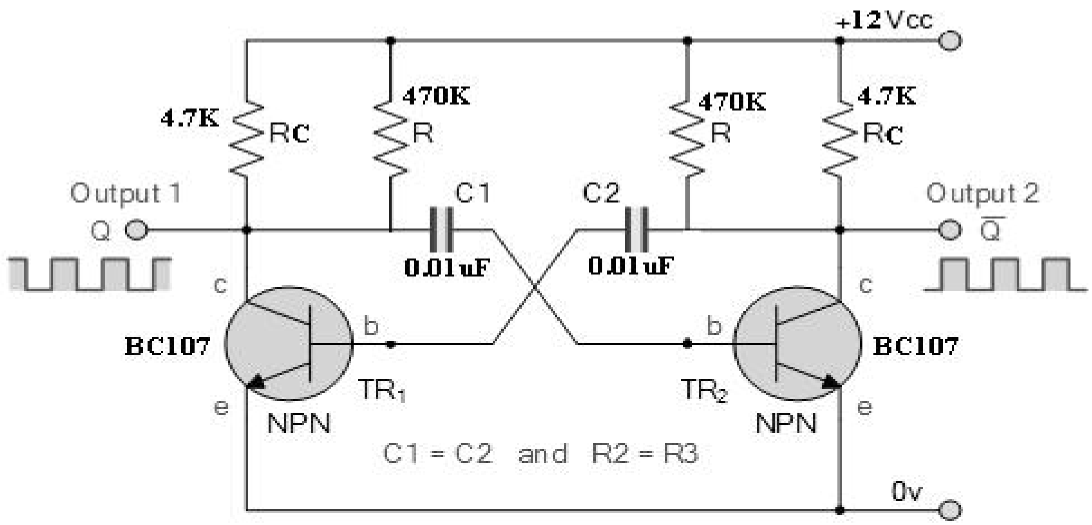

**MODEL GRAPH:**



**RESULT:**

Thus the integrator and differentiator circuit is constructed and output waveform is observed.

**CIRCUIT DIAGRAM( Astable Multivibrator)**




**TABULATION:**

| Amplitude(V) | Time period(msec) |
|---|---|
| | |

| **Ex. No: 9** | **ASTABLE MULTIVIBRATOR** |
|---|---|
| **Date:** | |

**AIM:**

To design an Emitter coupled Astable multivibrator, for the given specifications $V_{CC}$= 10V; hfe = 100; f=1 KHz; I *c* = 2mA; Vce (sat) = 0.2V; and to study the output waveform.

**APPARATUS REQUIRED:**

| S. No | APPARATUS | RANGE | QUANTITY |
|---|---|---|---|
| 1 | Resistors | 4.7kΩ,470kΩ | Each 2 |
| 2 | RPS | (0-30)V | 1 |
| 3 | Transistor | BC107 | 2 |
| 4 | Capacitors | 0.01μF | 2 |
| 5 | CRO | (0-30)MHz | 1 |
| 6 | Bread board | - | 1 |

**Design example:**

**Given specifications:**

$V_{CC}$= 10V; hfe = 100; f=1 KHz; I *c* = 2mA; Vce (sat) = 0.2v;

**To design $R_C$:**

$R \leq h_{FE} R_C$ $R_C = V_{CC} - V_{C2\,(Sat)} / I_C = 4.9\ k\Omega$

Since $R \leq h_{FE} R_C$

Therefore $R \leq 100 \times 4.7 \times 10^3 = 490\ k\Omega \approx 470\ k\Omega$

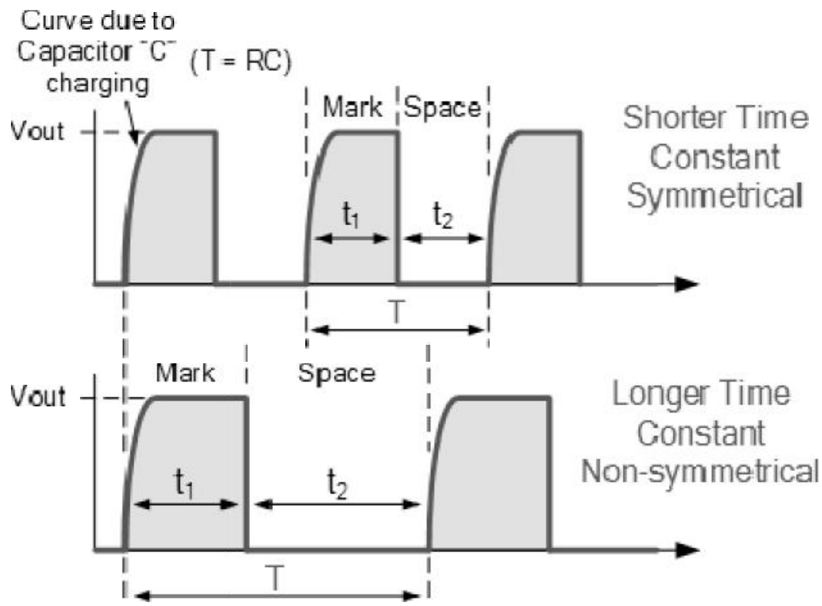
**To Design C:**

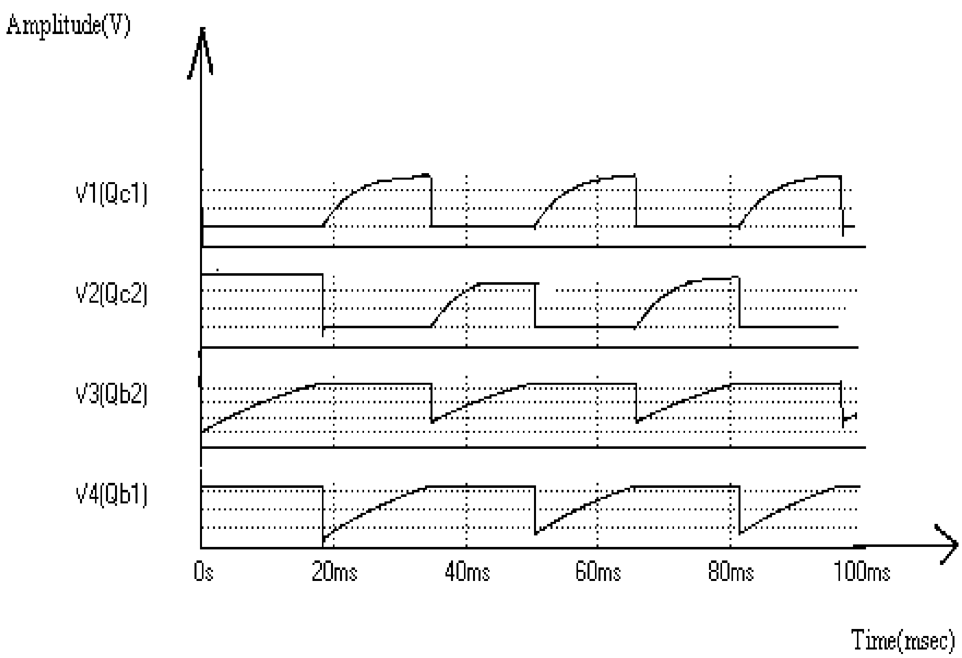
Since T= 1.38RC

$1 \times 10^{-3} = 1.38 \times 490 \times 10^3 \times C$

Therefore C=0.01μF

**MODEL GRAPH(Astable Multivibrator)**



**THEORY:**

The Astable multivibrator generates square wave without any external triggering pulse. It has no stable state, i.e., it has two quasi- stable states. It switches back and forth from one stable state to other, remaining in each state for a time depending upon the discharging of a capacitive circuit. When supply voltage +Vcc is applied, one transistor will conduct more than the other due to some circuit imbalance.

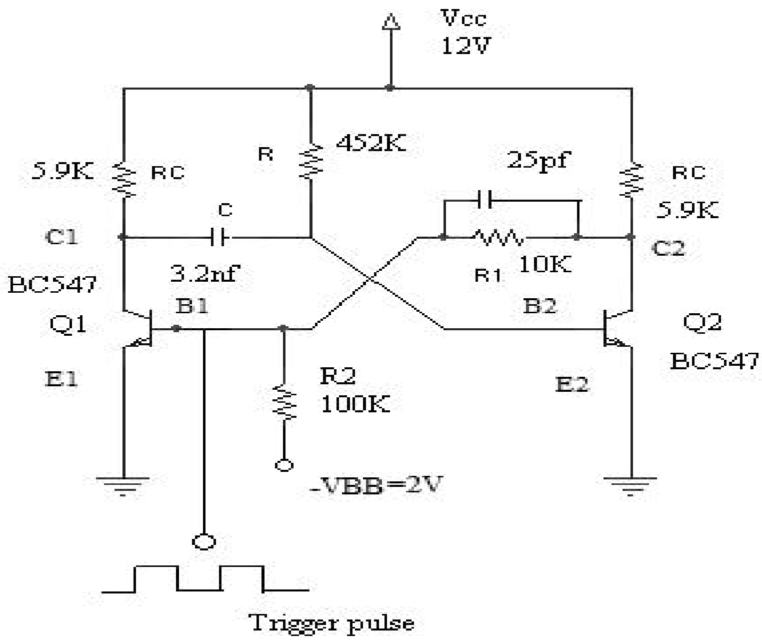
**PROCEDURE:**

1. Connect the circuit as per the circuit diagram.
2. Set $V_{CC}$ = 5V.
3. For the given supply the amplitude and time period is measured from CRO.
4. Frequency of oscillation is calculated by the formula f=1/T
5. Amplitude Vs time graph is drawn.

**RESULT:**

Thus the Astable Multivibrator is designed and output waveform is plotted.

**CIRCUIT DIAGRAM(Monostable Multivibrator)**




**TABULATION:**

| Amplitude(V) | Time period(msec) | |
|---|---|---|
| | $T_{ON}$ | $T_{OFF}$ |
| | | |

| Ex. No: 10 | MONOSTABLE MULTIVIBRATOR |
|---|---|
| Date: | |

## AIM:

To design and test the performance of Monostable Multivibrator for the given specifications $V_{CC}$= 12V; hfe = 200; f=1 KHz; Ic = 2mA; Vce (sat) = 0.2v; $V_{BB}$= - 2V and to obtain its output waveform.

## APPARATUS REQUIRED:

| S.No | APPARATUS | RANGE | QUANTITY |
|---|---|---|---|
| 1 | Resistors | 5.9 kΩ,452 kΩ,100 kΩ,10KΩ | 2,1,1,1 |
| 2 | RPS | (0-30)V | 1 |
| 3 | Transistor | BC547 | 2 |
| 4 | CRO | (0-30)MHz | 1 |
| 5 | Capacitor | 3.2nf,25pf | Each One |
| 6 | Bread board | - | 1 |

**To calculate R1 & R2:**

$V_{B1}$= {($V_{BB}$ R1/ R1 +R2) + ($V_{CE\ (sat)}$ R2 / R1+R2)}

Since Q1 is in off state, $V_{B1} \leq 0$

Then ($V_{BB}$ R1/ R1 +R2) = ($V_{CE\ (sat)}$ R2 / R1+R2)

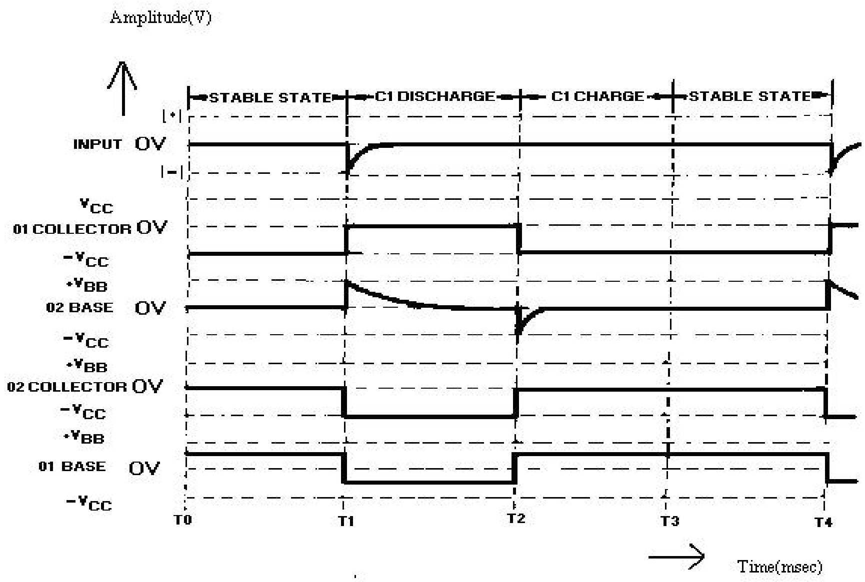
$V_{BB}$ R1 = $V_{CE\ (sat)}$ R2

2 R1 = 0.2 R2

Assume R1=10KΩ, then R2=100 KΩ

Consider, $C_1$= 25pf (commutative capacitor)

**MODEL GRAPH(Monostable Multivibrator)**




**Design Example:**

**Given specifications**: $V_{CC}$= 12V; hfe = 200; f=1 KHz; I $c$ = 2mA; Vce (sat) = 0.2v; $V_{BB}$= -2 V

**(i) To calculate $R_C$:**

$R_C = V_{CC}$ - Vce (sat) / $I_C$

$R_C = 12 – 0.2 / 2x10^{-3}$=5.9KΩ

**(ii)To calculate R:**

$I_{B2(min)}=I_{C2} / h_{fe}= 2x10^{-3} / 200 = 10\mu$ A

Select $I_{B2}>I_{B1(min)}$ (say 25µ A)

Then R=$V_{CC}$ – V $_{BE}$ (sat) / $I_{B2}$

Therefore R= $12-0.7/25x10^{-6}$=452KΩ

**(iii)To calculate C:**

T=0.69RC $1x10^{-3}= 0.69x452x10^3xC$ , Therefore C=3.2nf

**THEORY:**

The Monostable multivibrator has one stable state when an external trigger input is applied the circuit changes its state from stable to quasi -stable state. And then automatically after some time interval the circuit returns back to the original normal stable state. The time T is dependent on circuit components.

The capacitor $C_1$ is a speed-up capacitor coupled to base of Q2 through C. Thus DC coupling in Bistable multivibrator is replaced by a capacitor coupling. The resistor R, at input of Q2 is returned to $V_{CC}$.

The value of $R_2$, $V_{BB}$ are chosen such that transistor Q1 is off by reverse biasing it. Q2 is on. This is possible by forward biasing Q2 with the help of $V_{CC}$ and resistance R. Thus Q2-ON and Q1-OFF is normal stable state of circuit.

**PROCEDURE:**

1. Connect the circuit as per the circuit diagram.
2. Give a negative trigger input to the base of Q1.
3. Note the output of transistor Q2 and Q1.
4. Find the value of Ton and Toff.
5. Plot the response of the Monostable Multivibrator

**RESULT:**

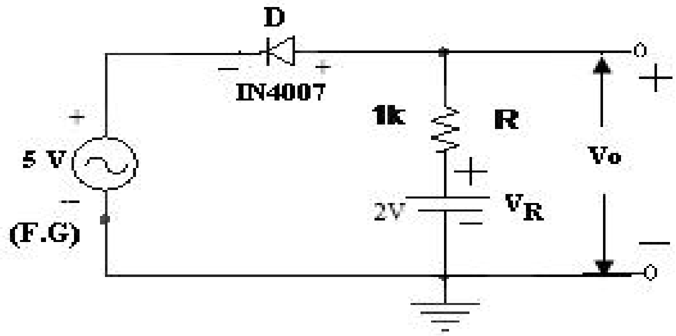
Thus the Monostable Multivibrator is designed and the performance is verified.

Theoretical period :

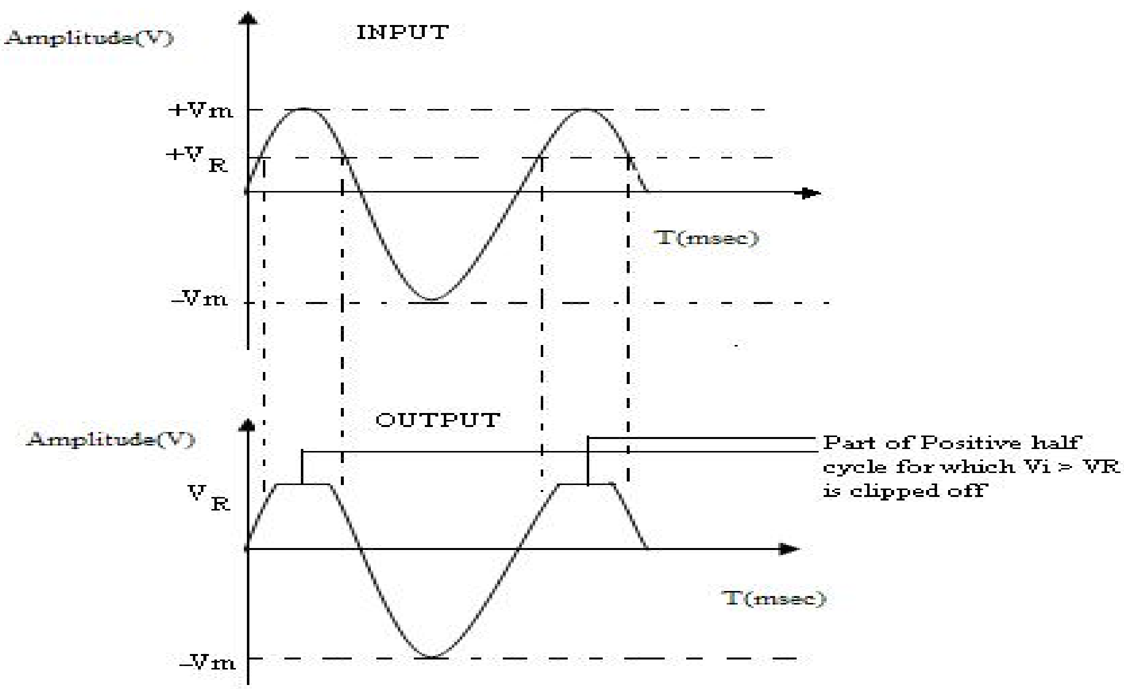
Practical period :

**CIRCUIT DIAGRAM:**

**(a) BIASED POSITIVE CLIPPER:**




**MODEL GRAPH:**



| **Ex. No: 11** | **CLIPPER AND CLAMPER CIRCUITS** |
|---|---|
| **Date:** | |

**AIM:**

To construct and design the clipper and clamper circuits using diodes at 1 KHz

**APPARATUS REQUIRED:**

| S.No | APPARATUS | RANGE | QUANTITY |
|---|---|---|---|
| 1 | Resistors | 1 kΩ | 1 |
| 2 | RPS | (0-30)V | 1 |
| 3 | Diode | IN4007 | 1 |
| 4 | CRO | (0-30)MHz | 1 |
| 6 | Function generator | (0-3)MHz | 1 |
| 7 | Capacitor | 1 μf | 2 |
| 8 | Bread board | - | 1 |

**DESIGN:**

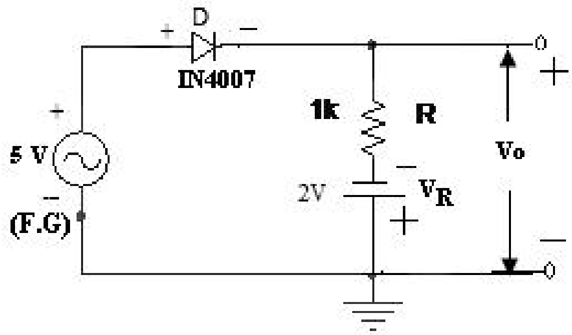
Given f=1 kHz,

$T=t=1/f=1x10^{-3}$ sec=RC

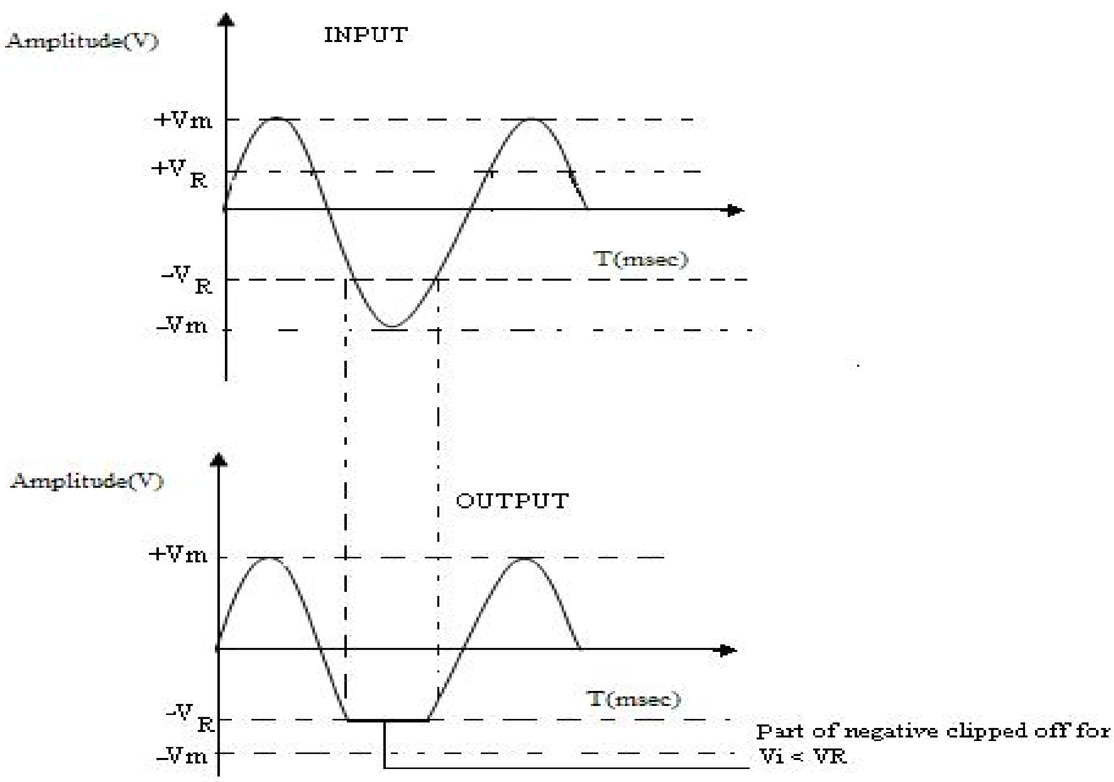
Assume, C=1uF

Then, R=1K

**(b) BIASED NEGATIVE CLIPPER:**




**MODEL GRAPH:**



## THEORY

**Clipper:**

A Clipper is a circuit that removes either the positive or negative part of a waveform. For a positive clipper only the negative half cycle will appear as output. Clipping circuits are also referred as voltage or current limiters, Amplitude selectors, or Slicers.

**Clamper:**

A Clamper circuit is a circuit that adds a dc voltage to the signal. A positive clamper shifts the ac reference level up to a dc level.

**Working:**

During the positive half cycle, the diode turns on and looks like a short circuit across the output terminals. Ideally, the output voltage is zero. But practically, the diode voltage is 0.7 V while conducting. On the negative half cycle, the diode is open and hence the negative half cycle appear across the output.

**Application:**

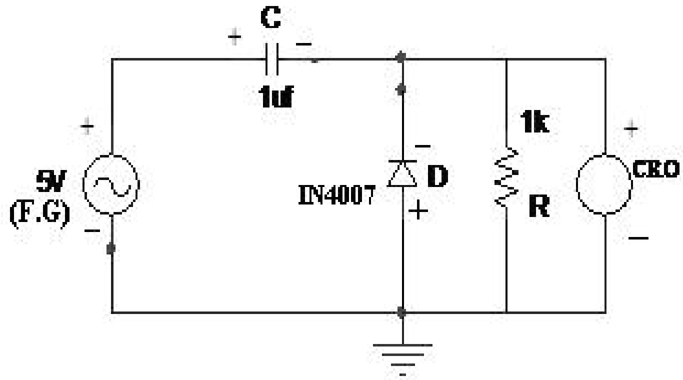
- Used for wave shaping
- To protect sensitive circuits

## PROCEDURE:

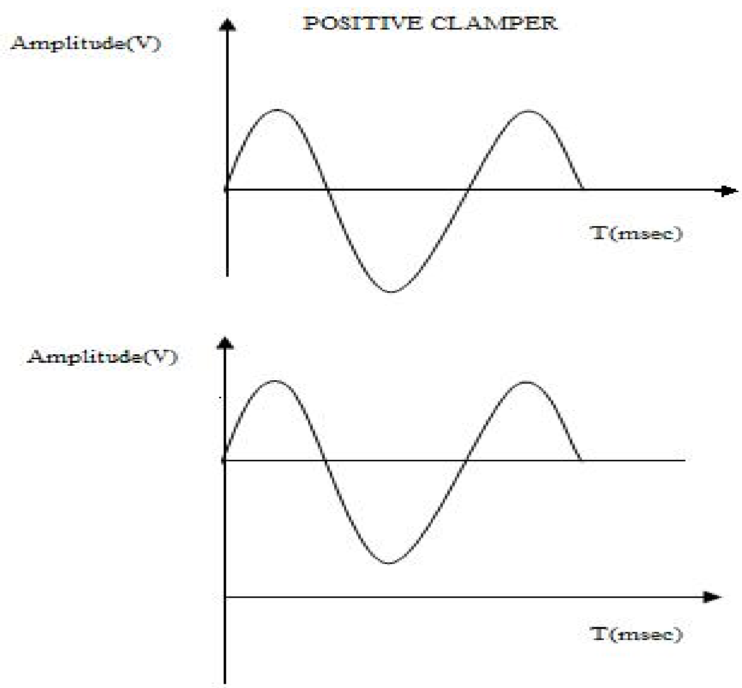
1. Connect as per the circuit diagram.
2. Set the signal voltage (say 5V, 1 KHz) using signal generator.
3. Observe the output waveform using CRO.
4. Sketch the output waveform.

## CIRCUIT DIAGRAM:

### (c) CLAMPER CIRCUIT (Positive Clamper)

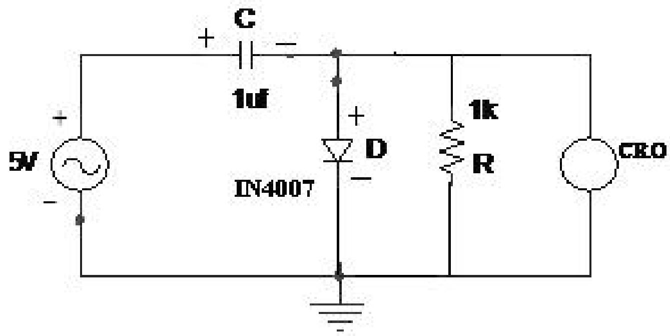



## MODEL GRAPH:

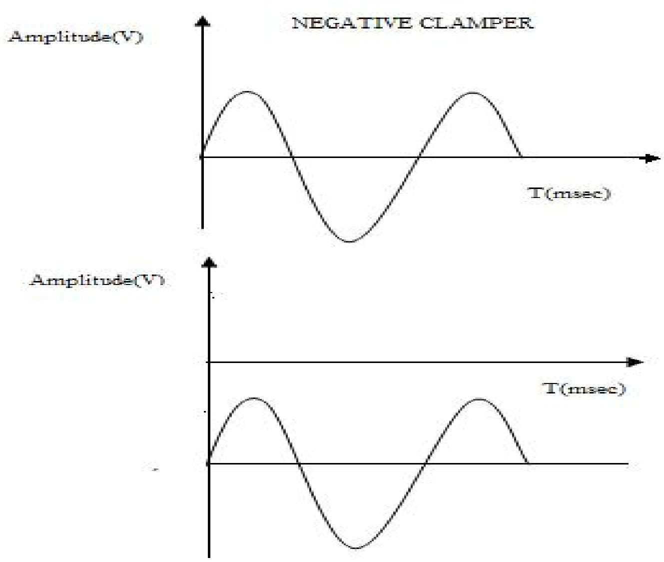


## CIRCUIT DIAGRAM:

(d) **Negative clamper:**



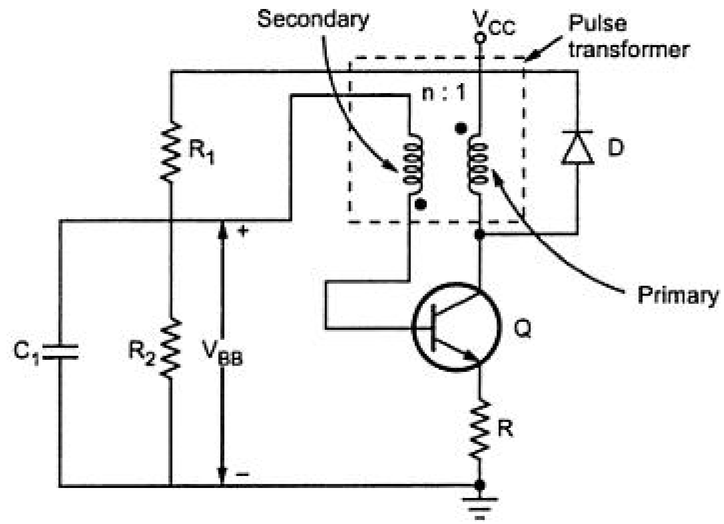

## MODEL GRAPH:



**RESULT:**

Thus, the output waveforms for Clipper and Clamper circuits were observed.

## CIRCUIT DIAGRAM(Free Running Blocking Oscillators)




## TABULATION:

| Amplitude(V) | Time period(msec) | |
|---|---|---|
| | $T_{ON}$ | $T_{OFF}$ |
| | | |

| Ex. No: 12 | FREE RUNNING BLOCKING OSCILLATORS |
|---|---|
| Date: | |

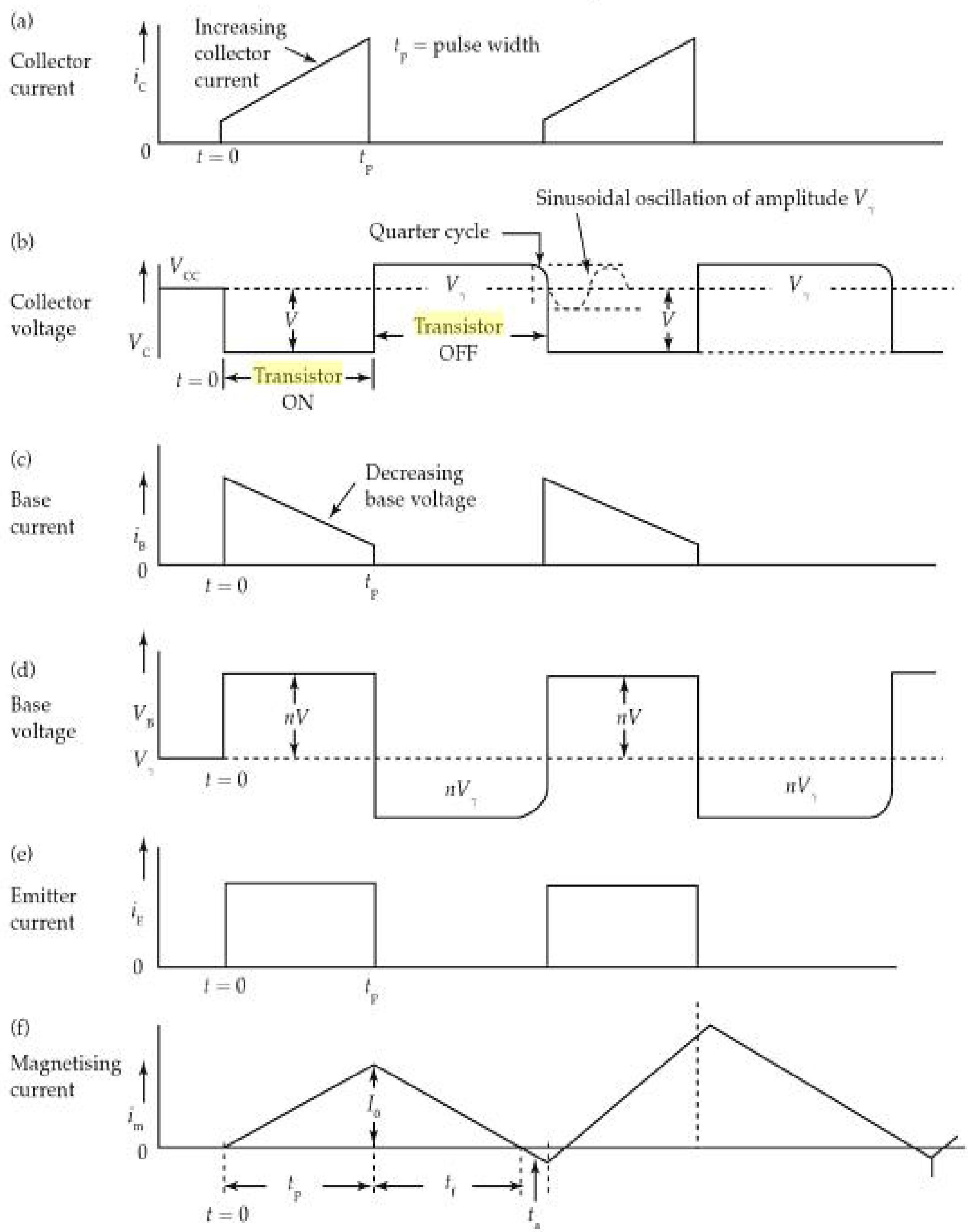
## AIM:

To design and test the performance of free running blocking oscillator and to obtain its output waveform.

## APPARATUS REQUIRED:

| S. No | APPARATUS REQUIRED | RANGE | QUANTITY |
|---|---|---|---|
| 1 | Resistors | 100 kΩ,10KΩ | 2,1 |
| 2 | RPS | (0-30)V | 1 |
| 3 | Transistor | BC547 | 2 |
| 4 | CRO | (0-30)MHz | 1 |
| 5 | Capacitor | 25pf | 1 |
| 6 | Transformer | 6V-0-6V | 1 |
| 7 | Diode | 1N4001 | 1 |
| 8 | Bread board | - | 1 |

**MODEL GRAPH(Free Running Blocking Oscillators)**



**THEORY:**

Astable blocking oscillator is also called free running blocking oscillator. It produces train of pulses, when triggered. The pulse width and the duty cycle of the blocking oscillator output can be controlled as per the requirement. There are two types of blockling oscillators available, which are,

1. Diode controlled Astable blocking oscillator.

2. RC controlled Astable blocking oscillator.

**Applications:**

1. Both Monostable and Astable blocking oscillators are used for generating pulses of large peak power.

2. It is used as a frequency divider or counter.

3. It is used to discharge a capacitor rapidly.

4. It may be used as a gating waveform with very small mark space ratio.

5. Both positive and negative pulses can be obtained from a blocking oscillator by using a tertiary winding.

**PROCEDURE:**

1. Connect the circuit as per the circuit diagram.
2. Note down the voltages and current at various base, emitter and collector.
3. Note the magnetizing current.
4. Draw the waveform for various values.

**RESULT:**

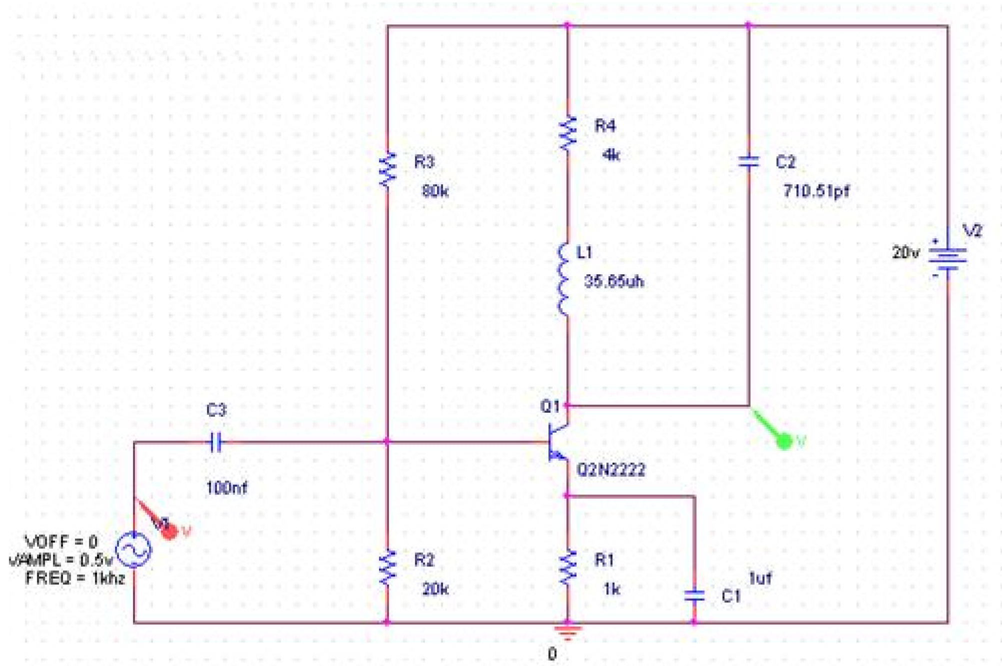
Thus the free running blocking oscillator is designed and the performance is verified.

# SIMULATION USING PSPICE

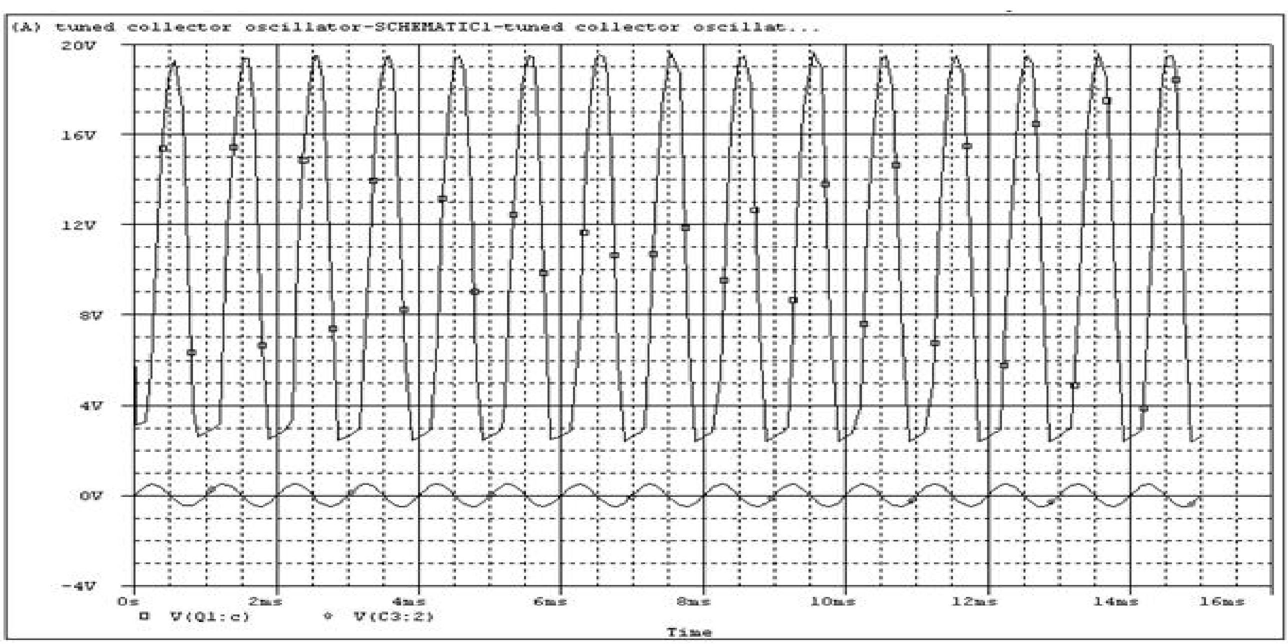
**(Using Transistor)**

**CIRCUIT DIAGRAM:**

## TUNED COLLECTOR OSCILLATORS:




**MODEL GRAPH:**



| Ex. No: 13 | TUNED COLLECTOR OSCILLATORS |
|---|---|
| Date: | |

**AIM:**

To simulate a tuned collector oscillation circuit using PSPICE

**APPARATUS REQUIRED:**

1. PC
2. PSPICE software

**THEORY:**

Tuned collector oscillator is a type of transistor LC oscillator where the tuned circuit (tank) consists of a transformer and a capacitor is connected in the collector circuit of the transistor. Tuned collector oscillator is of course the simplest and the basic type of LC oscillators. The tuned circuit connected at the collector circuit behaves like a purely resistive load at resonance and determines the oscillator frequency. The common applications of tuned collector oscillator are RF oscillator circuits, mixers, frequency demodulators, signal generators etc.

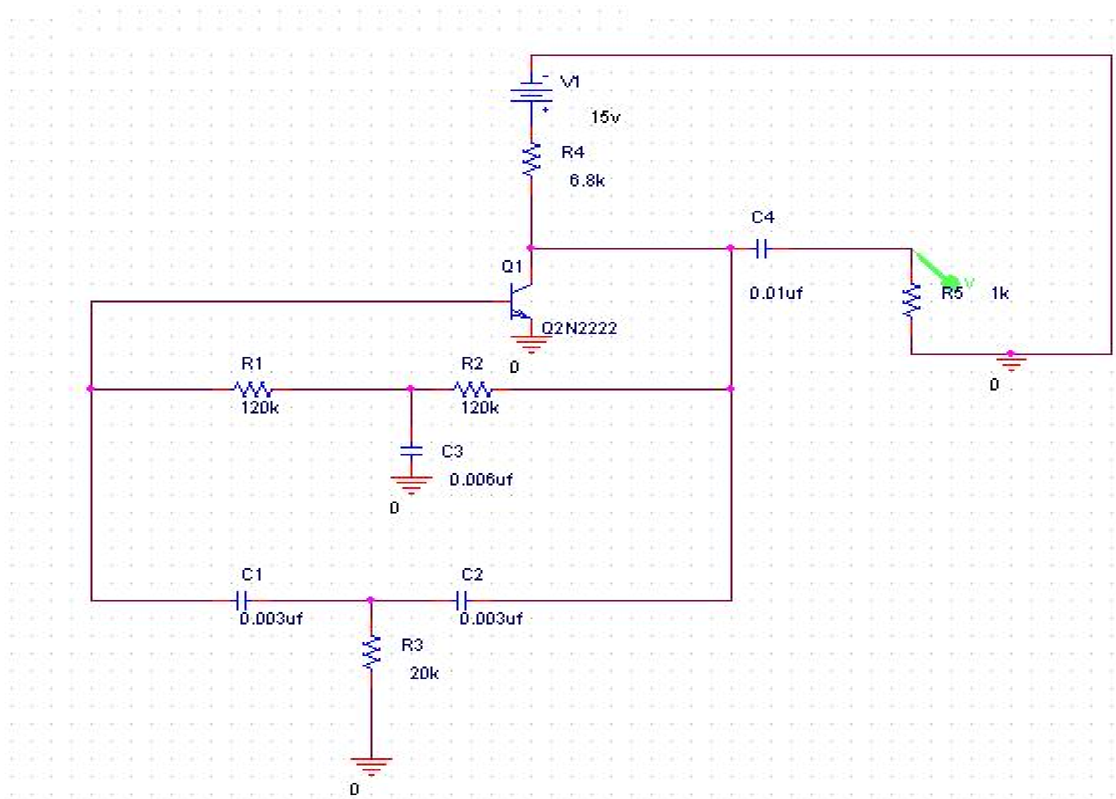
**PROCEDURE:**

1. Click on the start menu and select the Pspice simulation software
2. Select the parts required for the circuit from the parts menu and place them in the work space
3. Connect the parts using wires
4. Save the file and select the appropriate analysis
5. Simulate the circuit and observe the corresponding output waveforms

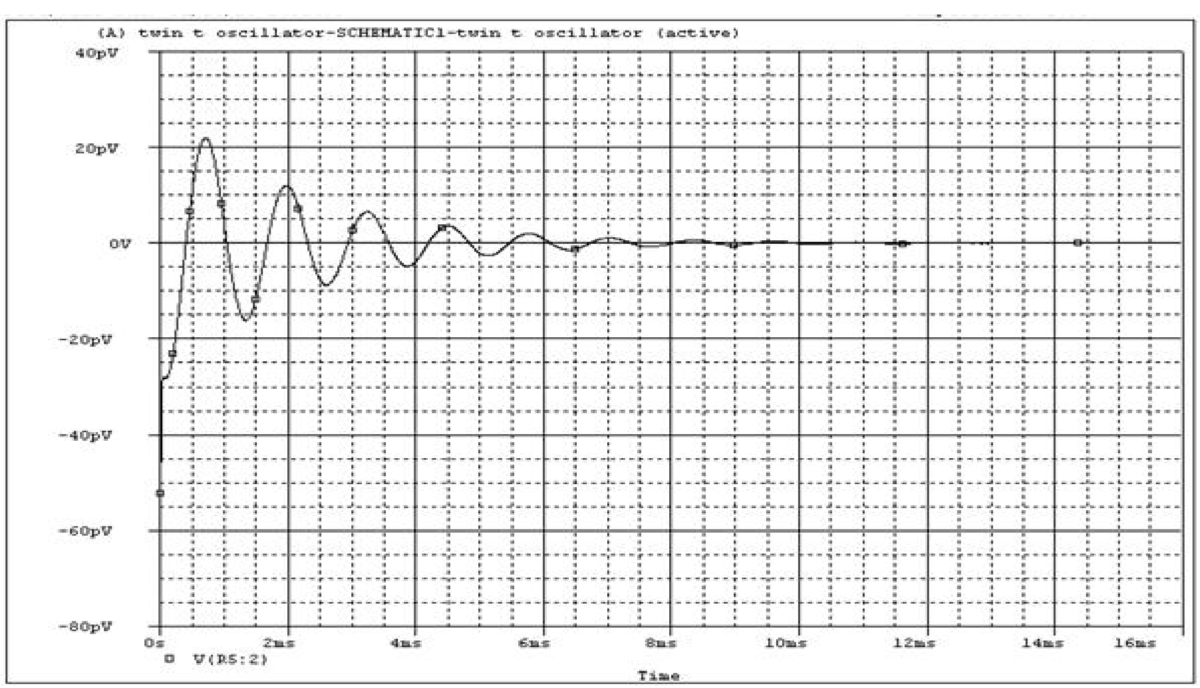
**RESULT:**

Thus, the tuned collector oscillator circuit is simulated using Pspice.

**CIRCUIT DIAGRAM(Twin-T Oscillator)**




**MODEL GRAPH:**



| Ex. No: 14 | TWIN-T OSCILLATOR |
|---|---|
| Date: | |

**AIM:**

To simulate a twin-T oscillation circuit using PSPICE

**APPARATUS REQUIRED:**

1. PC
2. PSPICE software

**THEORY:**

"Twin-T" oscillator uses two "T" RC circuits operated in parallel. One circuit is an R-C-R "T" which acts as a low-pass filter. The second circuit is a C-R-C "T" which operates as a high-pass filter. Together, these circuits form a bridge which is tuned at the desired frequency of oscillation. The signal in the C-R-C branch of the Twin-T filter is advanced, in the R-C-R - delayed, so they may cancel one another for frequency $f$=12$\pi RC$ if $x$=2; if it is connected as a negative feedback to an amplifier, and x>2, the amplifier becomes an oscillator.

**PROCEDURE:**

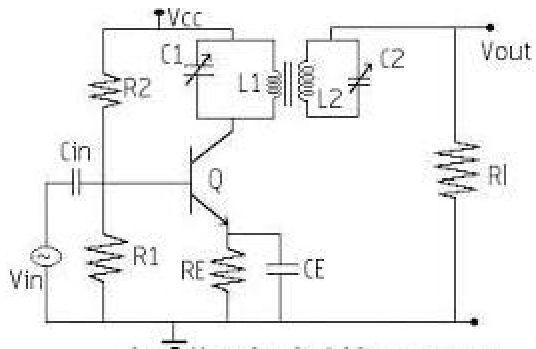
1. Click on the start menu and select the Pspice simulation software
2. Select the parts required for the circuit from the parts menu and place them in the work space
3. Connect the parts using wires
4. Save the file and select the appropriate analysis
5. Simulate the circuit and observe the corresponding output waveforms

**RESULT:**

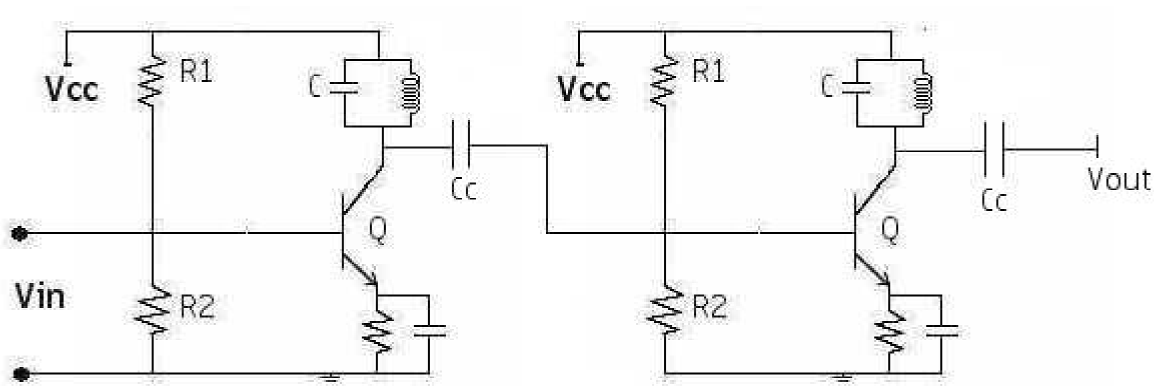
Thus, the twin-T oscillator oscillator circuit is simulated using Pspice.

**CIRCUIT DIAGRAM:**

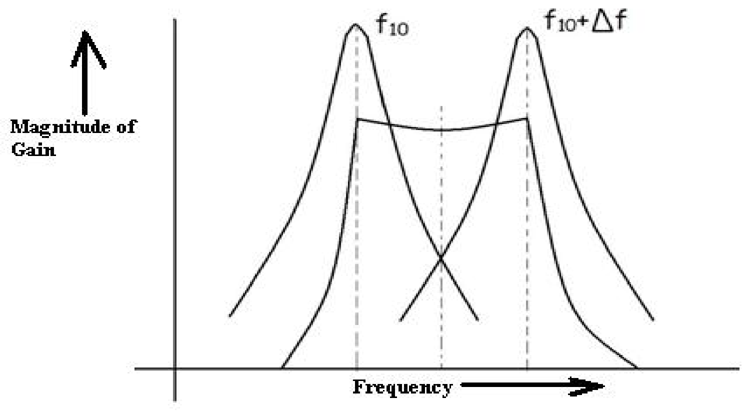
**(a) Double tuned Amplifiers**




**(b) Stager tuned Amplifiers**




**MODEL GRAPH:**



| Ex. No: 15<br>Date: | DOUBLE AND STAGER TUNED AMPLIFIERS |
|---|---|

**AIM:**

To simulate a double and stager tuned amplifiers circuit using PSPICE

**APPARATUS REQUIRED:**

1. PC
2. PSPICE software

**THEORY:**

Stagger Tuned Amplifiers are used to improve the overall frequency response of tuned Amplifiers. Stagger tuned Amplifiers are usually designed so that the overall response exhibits maximal flatness around the centre frequency. It needs a number of tuned circuits operating in union. The overall frequency response of a Stagger tuned amplifier is obtained by adding the individual response together. Since the resonant Frequencies of different tuned circuits are displaced or staggered, they are referred as stagger tuned amplifier.

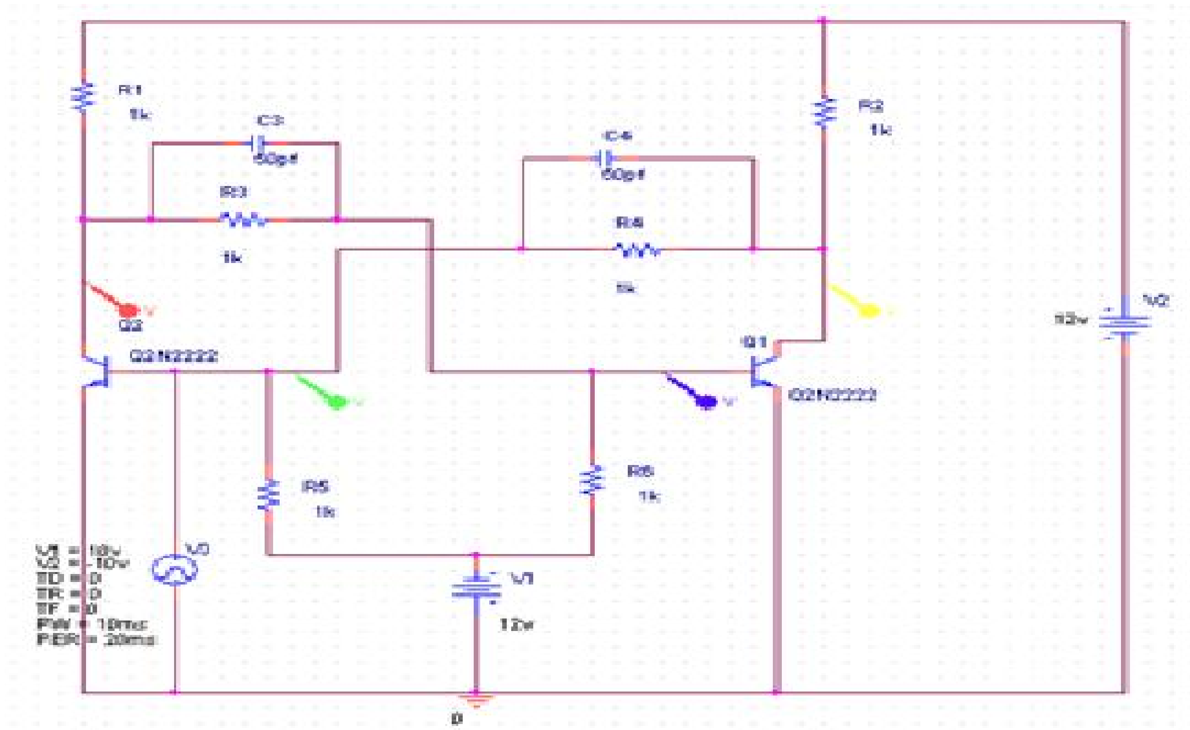
**PROCEDURE:**

1. Click on the start menu and select the Pspice simulation software
2. Select the parts required for the circuit from the parts menu and place them in the work space
3. Connect the parts using wires
4. Save the file and select the appropriate analysis
5. Simulate the circuit and observe the corresponding output waveforms

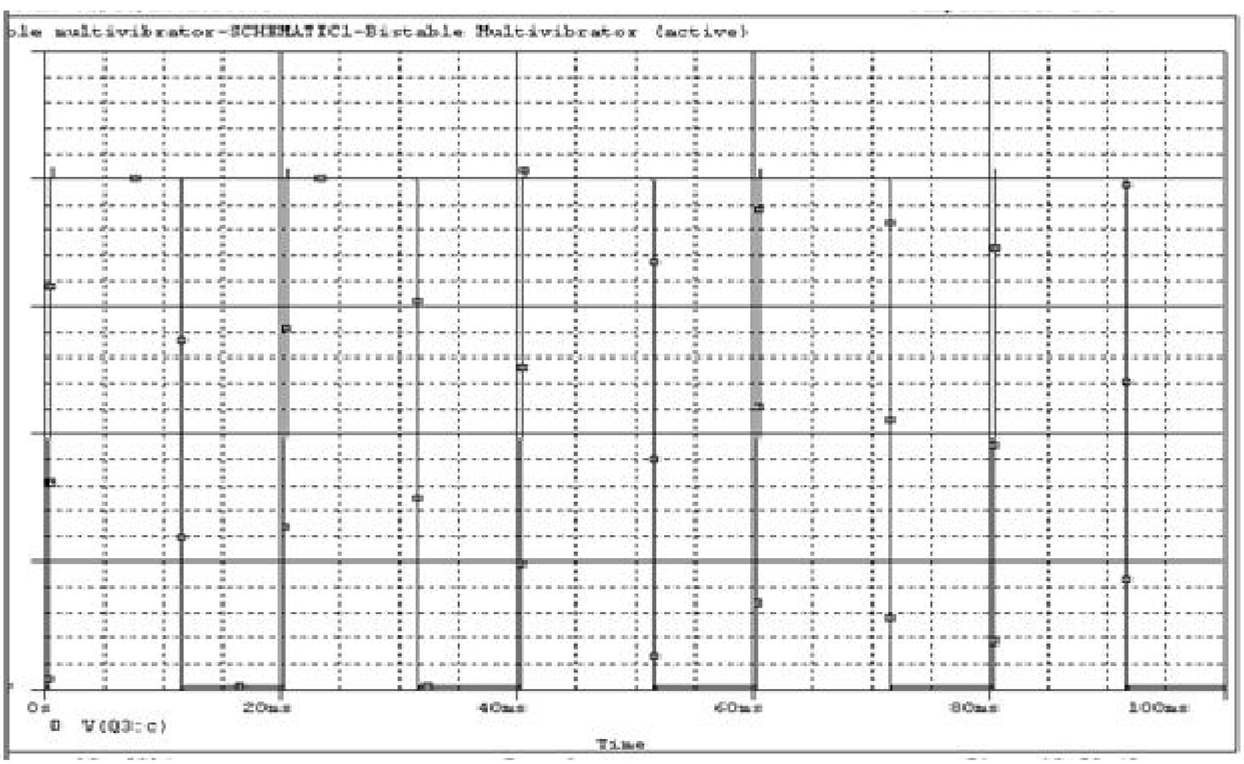
**RESULT:**

Thus, the double and stager tuned amplifier circuit is simulated using Pspice.

**CIRCUIT DIAGRAM(Bi-Stable Multivibrator)**




**MODEL GRAPH:**



| **Ex. No: 16** | **BI-STABLE MULTIVIBRATOR** |
|---|---|
| **Date:** | |

**AIM:**

To simulate an Bi-stable multivibrator using PSPICE

**APPARATUS REQUIRED:**

1. PC
2. PSPICE software

**THEORY:**

Bi- stable multivibrator contains two stable states and no quasi states. It requires two clock or trigger pulses to change the states. It is also called as flip flop, scale of two toggle circuit, trigger circuit.

**PROCEDURE:**

1. Click on the start menu and select the Pspice simulation software
2. Select the parts required for the circuit from the parts menu and place them in the work space
3. Connect the parts using wires
4. Save the file and select the appropriate analysis
5. Simulate the circuit and observe the corresponding output waveforms

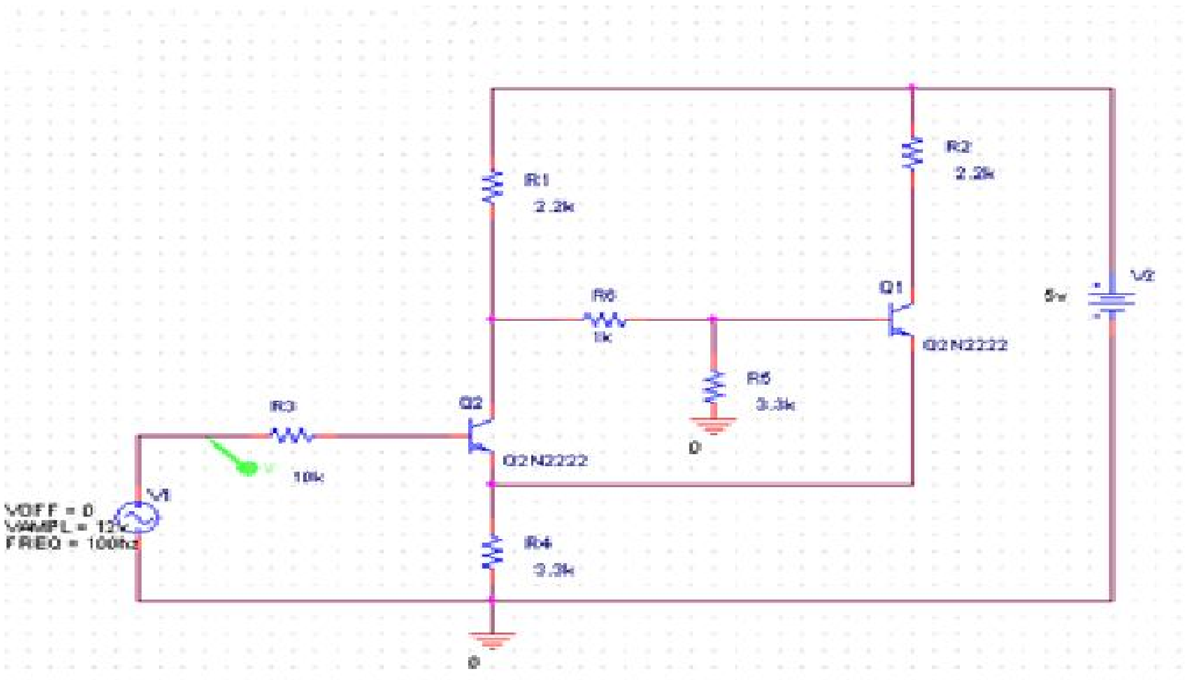
**APPLICATIONS:**

- It is used in digital operations like counting, storing data's in flip flops and production of square waveforms.

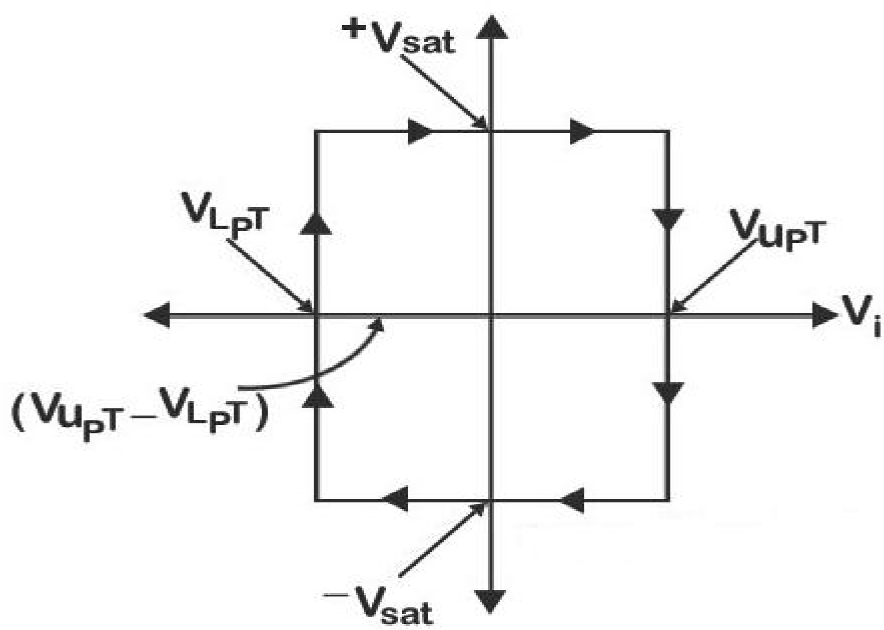
**RESULT:**

Thus, the Bi-stable multivibrator circuit is simulated using PSpice.

**<u>CIRCUIT DIAGRAM(</u>Schmitt Trigger Circuit)**




**<u>MODEL GRAPH:</u>**



| **Ex. No: 17** | **SCHMITT TRIGGER CIRCUIT WITH PREDICTABLE HYSTERESIS** |
|---|---|
| **Date:** | |

**AIM:**

To simulate a schmitt trigger circuit with predictable hysteresis using PSPICE

**APPARATUS REQUIRED:**

1. PC
2. PSPICE software

**THEORY:**

A Schmitt trigger is a comparator circuit with hysteresis, implemented by applying positive feedback to the non-inverting input of a comparator or differential amplifier. It is an active circuit which converts an analog input signal to a digital output signal. The circuit is named a "trigger" because the output retains its value until the input changes sufficiently to trigger a change. In the non-inverting configuration, when the input is higher than a certain chosen threshold, the output is high. When the input is below a different (lower) chosen threshold, the output is low, and when the input is between the two levels, the output retains its value. This dual threshold action is called *hysteresis* and implies that the Schmitt trigger possesses memory and can act as a bistable circuit (latch or flip-flop). There is a close relation between the two kinds of circuits: a Schmitt trigger can be converted into a latch and a latch can be converted into a Schmitt trigger.

Schmitt trigger devices are typically used in signal conditioning applications to remove noise from signals used in digital circuits, particularly mechanical switch bounce. They are also used in closed loop negative feedback configurations to implement relaxation oscillators, used in function generators and switching power supplies.

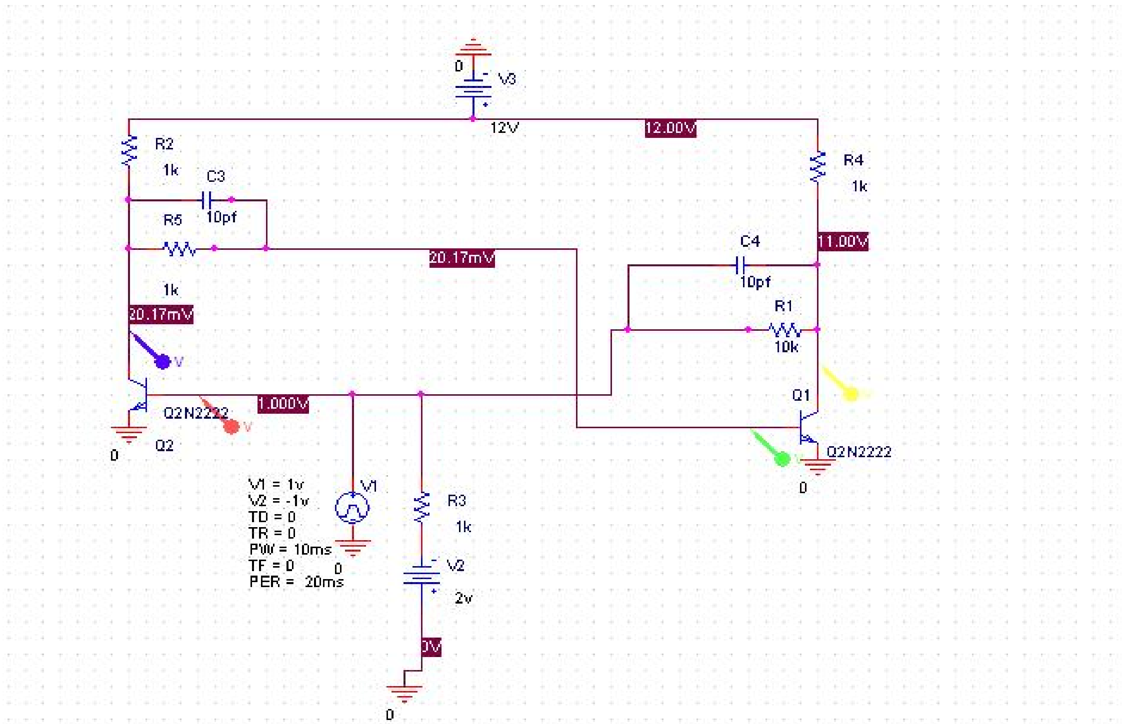
**PROCEDURE:**

1. Click on the start menu and select the Pspice simulation software
2. Select the parts required for the circuit from the parts menu and place them in the work space
3. Connect the parts using wires
4. Save the file and select the appropriate analysis
5. Simulate the circuit and observe the corresponding output waveforms

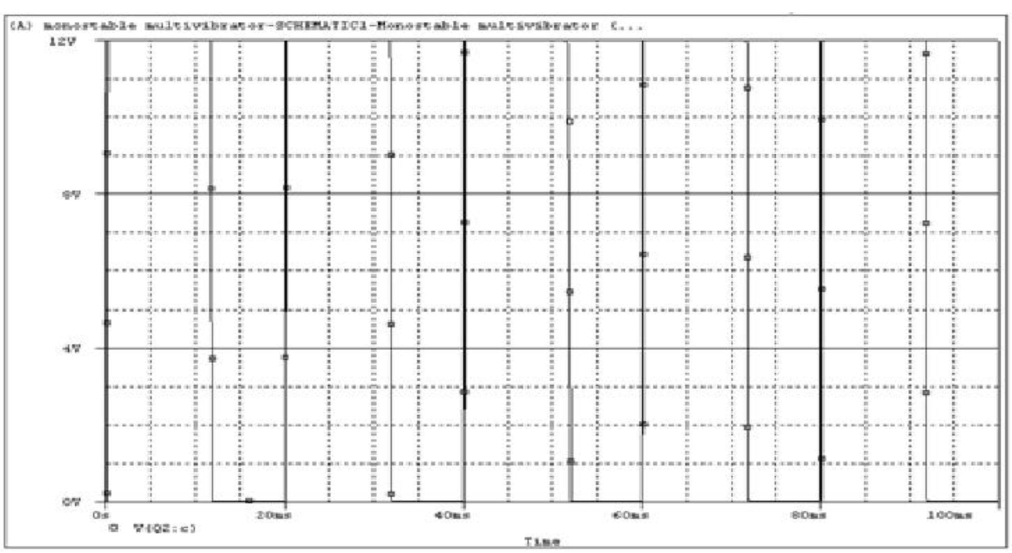
**RESULT:**

Thus, the Bi-stable multivibrator circuit is simulated using PSpice.

**CIRCUIT DIAGRAM(Mono Stable Multivibrator)**




**MODEL GRAPH:**



| **Ex. No: 18** | **MONO STABLE MULTIVIBRATOR** |
|---|---|
| **Date:** | |

**AIM:**

To simulate an Monostable multivibrator circuit using PSPICE

**APPARATUS REQUIRED:**

1. PC
2. PSPICE software

**THEORY:**

Monostable multivibrator is an electronic circuit which has one stable state and one quasi stable state. It needs external pulse to change their stable state to quasi state and return back to its stable state after completing the time constant RC. Thus the RC time constant determines the duration of quasi state. Also called as one-shot, single shot and one swing multivibrator.

**PROCEDURE:**

1. Click on the start menu and select the Pspice simulation software
2. Select the parts required for the circuit from the parts menu and place them in the work space
3. Connect the parts using wires
4. Save the file and select the appropriate analysis
5. Simulate the circuit and observe the corresponding output waveforms

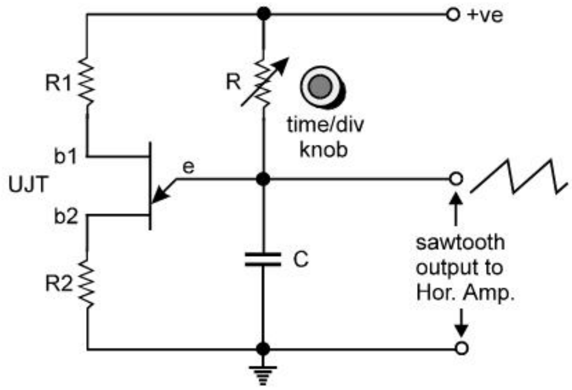
**Applications:**

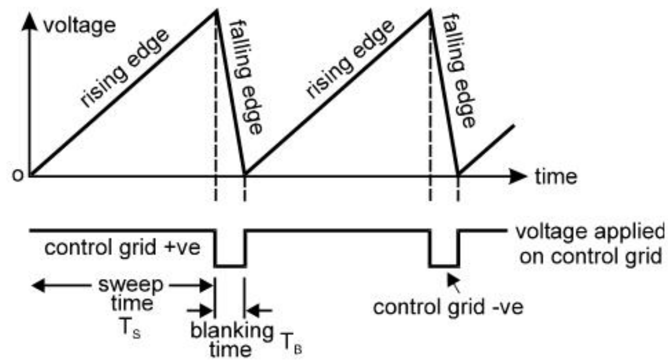
- Used as triggering circuit for some circuits like timer circuit, delay circuits etc.

**RESULT:**

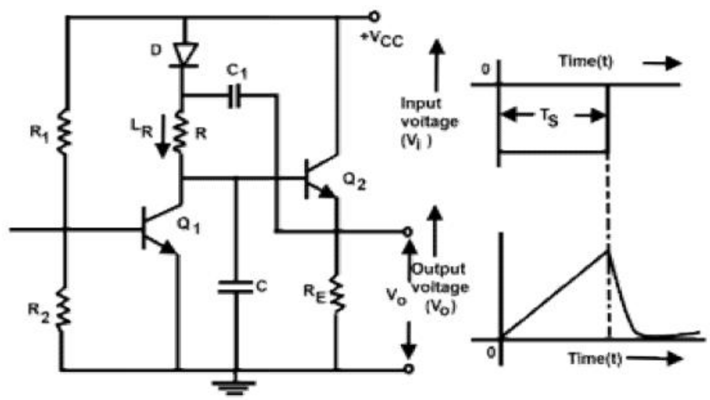
Thus, the Monostable Multivibrator circuit is simulated using Pspice.

**CIRCUIT DIAGRAM(Voltage Time Base Circuits)**

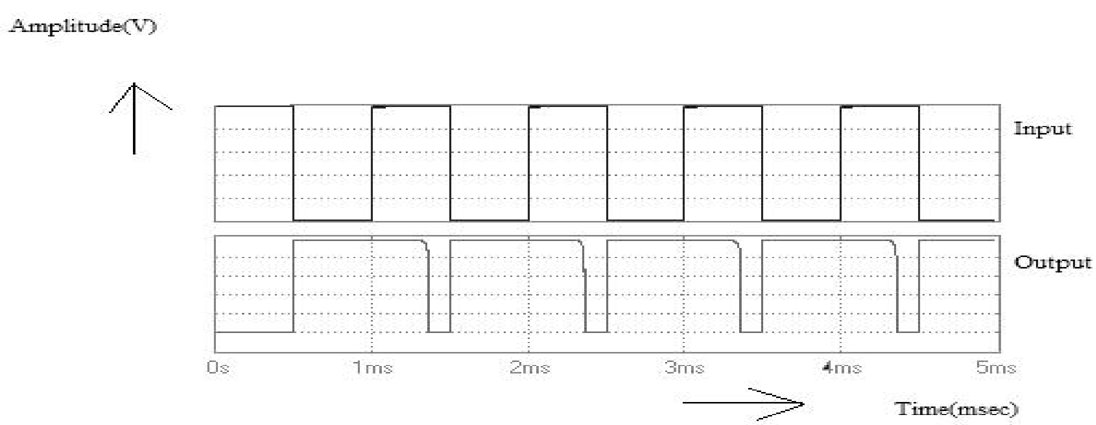



**CIRCUIT DIAGRAM (Current Time Base Circuits)**




**MODEL GRAPH: (Current Time Base Circuits)**



| Ex. No: 19 | VOLTAGE AND CURRENT TIME BASE CIRCUITS |
|---|---|
| Date: | |

**AIM:**

To simulate a voltage and current time base circuits using PSPICE

**APPARATUS REQUIRED:**

1. PC
2. PSPICE software

**THEORY:**

A voltage and current time base circuit is an electronic circuit which has one stable state and one quasi stable state. It needs external pulse to change their stable state to quasi state and return back to its stable state after completing the time constant RC. Thus the RC time constant determines the duration of quasi state.

**PROCEDURE:**

1. Click on the start menu and select the Pspice simulation software
2. Select the parts required for the circuit from the parts menu and place them in the work space
3. Connect the parts using wires
4. Save the file and select the appropriate analysis
5. Simulate the circuit and observe the corresponding output waveforms

**Applications:**

- Used as triggering circuit for some circuits like timer circuit, delay circuits etc.

**RESULT:**

Thus, the voltage and current time base circuit is simulated using Pspice.