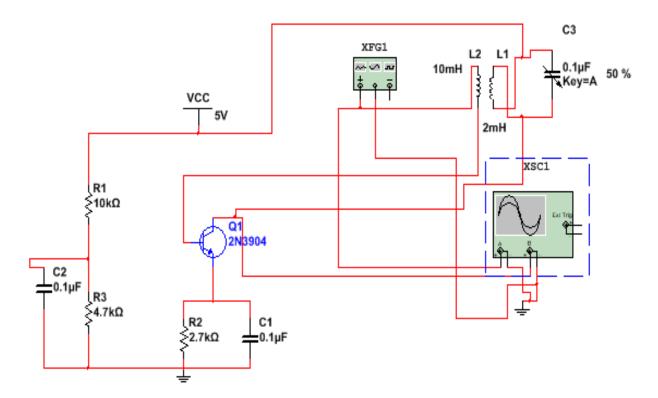
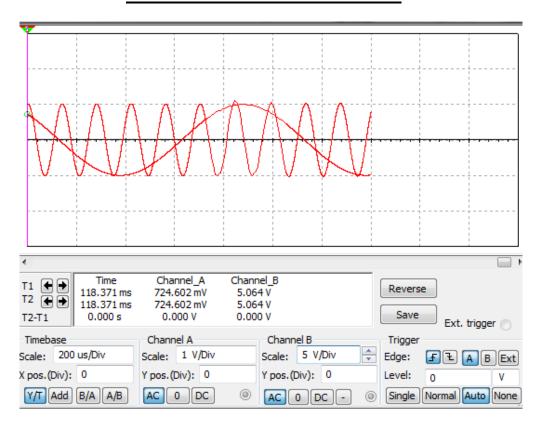


EC 6411 CIRCUITS AND SIMULATION INTEGRATED LABORATORY **SIMULATION EXPERIMENTS**

SIMULATION DIAGRAM FOR TUNED COLLECTOR OSCILLATOR



SIMULATED OUTPUT WAVE FORM



| Ex. No: | TUNED COLLECTOR OSCILLATOR |
|---------|----------------------------|
| Date: | TUNED COLLECTOR OSCILLATOR |

<u>AIM</u>

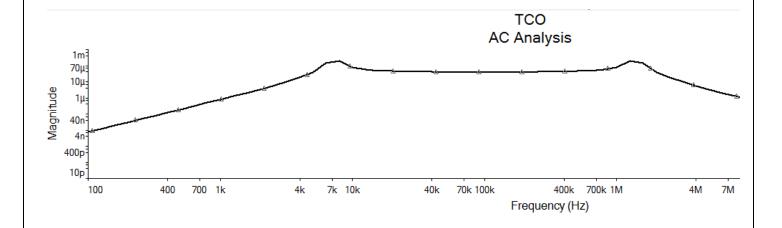
To design and simulate the tuned collector oscillator using transistor and obtain the output characteristics by using multisim.

SOFTWARE TOOLS REQUIREMENT

1. PC with Multisim 12.0V

- 1. Start the Multisim 12.0V
- 2. Open the new project
- 3. Click and drag the components required from master database directory
- 4. Connect the components as per circuit diagram
- 5. Save the project
- 6. Click the run symbol to simulate the circuit
- 7. Click the grapher to view the output and note down the parameter





| RESULT: | | |
|--|----------------------------|----------------|
| Thus the tuned collector oscillator is | designed, simulated and wa | ve forms are |
| observed. | | |
| | MARKS ALLO | |
| | Experimental Setup | 10 |
| | | 10 |
| | Execution | 10 |
| | | 10 10 30 |

SIMULATION CIRCUIT FOR TWIN-T OSCILLATOR VCC 9٧ XSC1 R6 -1kΩ R1 R2 C1 ^{47kΩ} 47kΩ R4 68nF $2.2k\Omega$ ≲33kΩ BC107BP C2 C3 R3 33nF 33nF **≶10kΩ**

| Ex. No: | TWIN-T OSCILLATOR |
|---------|--------------------|
| Date: | I WIN-I OSCILLATOR |

<u>AIM</u>

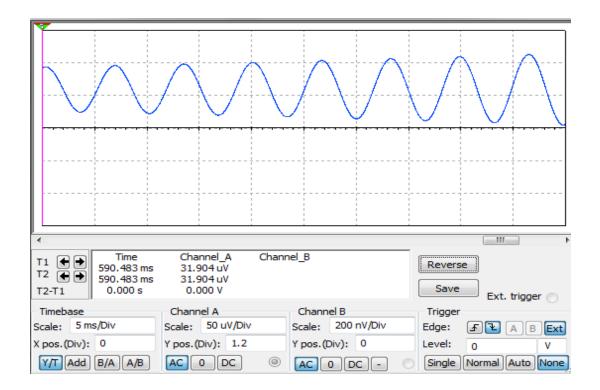
To design and simulate the Twin T oscillator using transistor and obtain the output characteristics by using multisim.

SOFTWARE TOOLS REQUIREMENT

PC with Multisim 12.0V

- 1) Start the Multisim 12.0V
- 2) Open the new project
- 3) Click and drag the components required from master database directory
- 4) Connect the components as per circuit diagram
- 5) Save the project
- 6) Click the run symbol to simulate the circuit
- 7) Click the grapher to view the output and note down the parameter

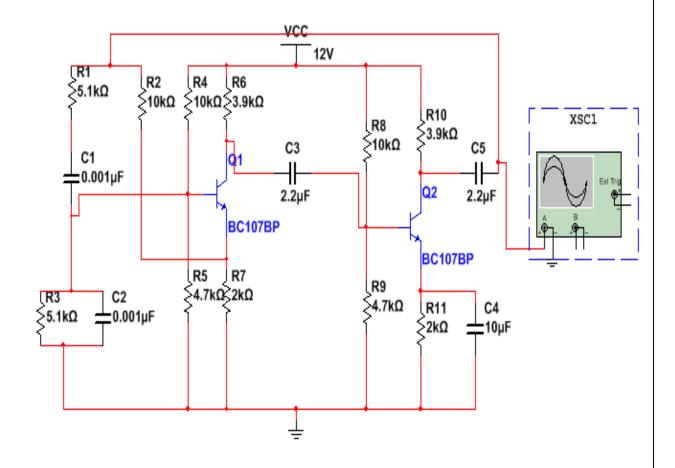
SIMULATED OUTPUT



EC 6411 CIRCUITS AND SIMULATION INTEGRATED LABORATORY

| RESULT: | | |
|---|--------------------------|--------------|
| Thus the twin t oscillator is designed, | cimulated and wave forms | are observed |
| Thus the twin to semator is designed, | MARKS ALLO | |
| | Experimental Setup | 10 |
| | Execution | 10 |
| | Execution | |
| | Viva | 10 |

SIMULATION CIRCUIT FOR WEIN BRIDGE OSCILLATOR:



| Ex. No: | WEIN BRIDGE OSCILLATOR |
|---------|-------------------------|
| Date: | WEIN BRIDGE OF CIEENTON |

<u>AIM</u>

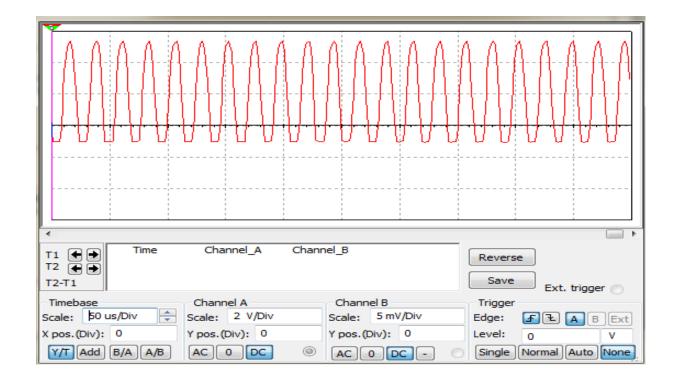
Design the bistable multivibrator using transistor. Obtain the output characteristics by using multisim

SOFTWARE TOOLS REQUIREMENT

PC with Multisim 12.0V

- 1. Start the Multisim 12.0V
- 2. Open the new project
- 3. Click and drag the components required from master database directory
- 4. Connect the components as per circuit diagram
- 5. Save the project
- 6. Click the run symbol to simulate the circuit
- 7. Click the grapher to view the output and note down the parameter

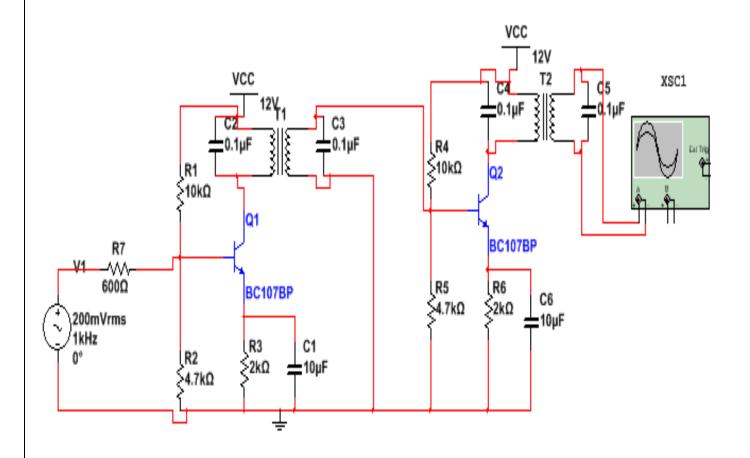
SIMULATED OUTPUT



EC 6411 CIRCUITS AND SIMULATION INTEGRATED LABORATORY

| RESULT: | | |
|---|--------------------------|-----------|
| | ad simulated and wave t | forma ara |
| Thus the wein bridge oscillator is designed observed. | ed, simulated and wave i | orns are |
| | MARKS ALLO | CATION |
| | Experimental Setup | 10 |
| | Execution | 10 |
| | Vi va | 10 |
| | Total | 30 |

SIMULATION CIRCUIT FOR DOUBLE TUNED AMPLIFIER:



| Ex. No: | DOUBLE TUNED AMPLIFIER |
|---------|-------------------------|
| Date: | DOODLE TONED ANTIERTIER |

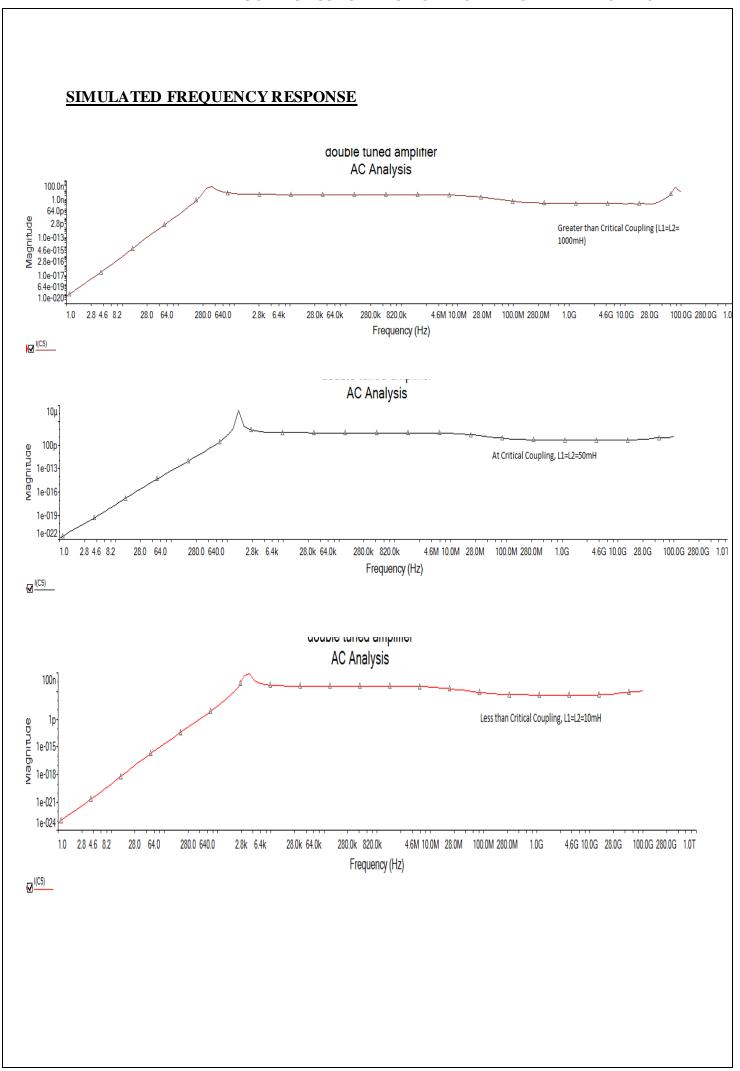
<u>AIM</u>

To design the double tuned amplifier and to obtain its frequency response at different coupling levels by using multisim.

SOFTWARE TOOLS REQUIREMENT

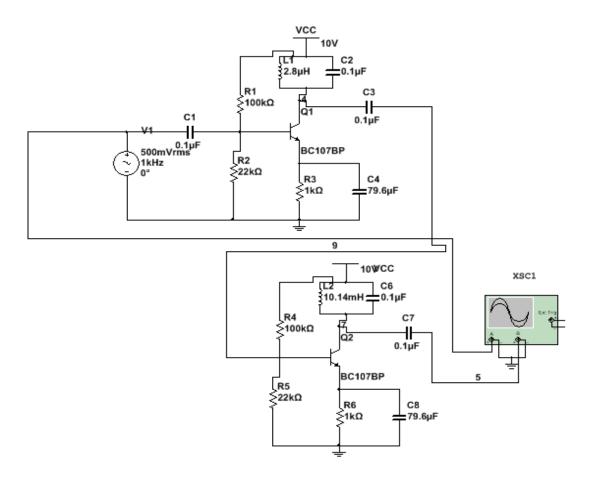
PC with Multisim 12.0V

- 1. Start the Multisim 12.0V
- 2. Open the new project
- 3. Click and drag the components required from master database directory
- 4. Connect the components as per circuit diagram
- 5. Save the project
- 6. Click the run symbol to simulate the circuit
- 7. Click the grapher to view the output and note down the parameter



| RESULT | | |
|--|---------------------|-------------|
| Thus the double tuned amplifier is designing simulated using multisim. | gned and the output | waveform is |
| | | |
| | MARKS ALLOCA | TION |
| | perimental Setup | 10 |
| | ecution | 10 |
| | va | 10 30 |
| To | otal | 1-30 |

SIMULATION CIRCUIT FOR STAGGER TUNED AMPLIFIER



| Ex. No: | |
|---------|-------------------------|
| Date: | STAGGER TUNED AMPLIFIER |

AIM

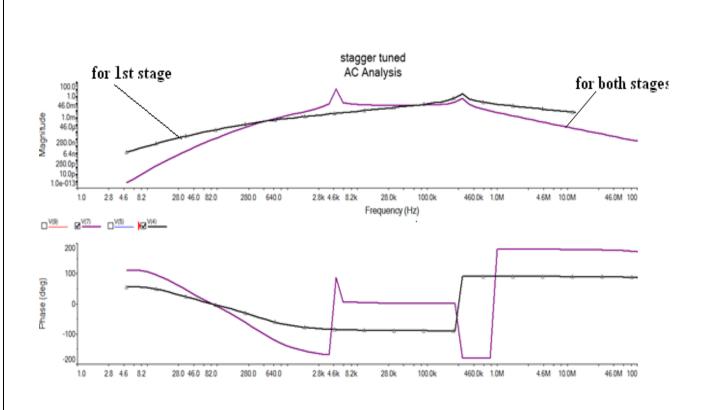
To design a stagger tuned amplifier and to obtain its frequency response for each stage by using multisim.

SOFTWARE TOOLS REQUIREMENT

PC with Multisim 12.0V

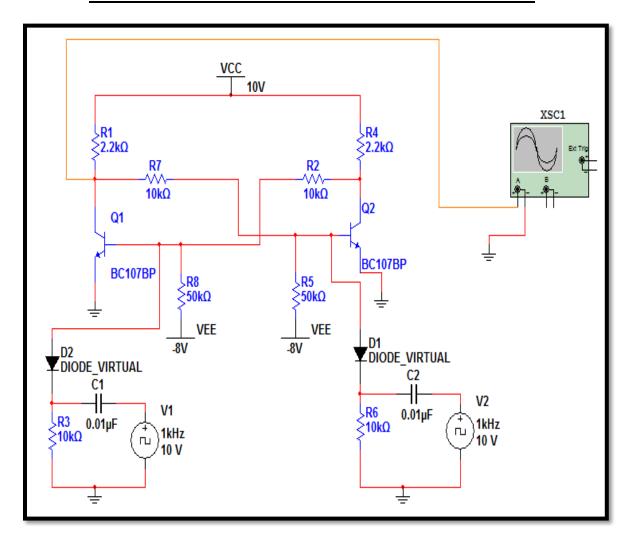
- 1. Start the Multisim 12.0V
- 2. Open the new project
- 3. Click and drag the components required from master database directory
- 4. Connect the components as per circuit diagram
- 5. Save the project
- 6. Click the run symbol to simulate the circuit
- 7. Click the grapher to view the output and note down the parameter

SIMULATED FREQUENCY RESPONSE



| DEGLE | Tr. | | | | |
|---------|-----------------------------------|-------------------|--------------------------|----------------|--------|
| RESUL | | | | | |
| using m | Thus the stagger tun nultisim. | ed amplifier is d | esigned and the output w | aveform is sim | ulated |
| | | | MA DYO | HOCATION | |
| | | | Experimental Setup | ALLOCATION 10 | |
| | | | Execution | 10 | |
| | | | Viva | 10 | |
| | | | Total | 30 | |

SIMULATION CIRCUIT FOR BISTABLE MULTIVIBRATOR



| Ex. No: | |
|---------|--|
| Date: | |

BISTABLE MULTIVIBRATOR

<u>AIM</u>

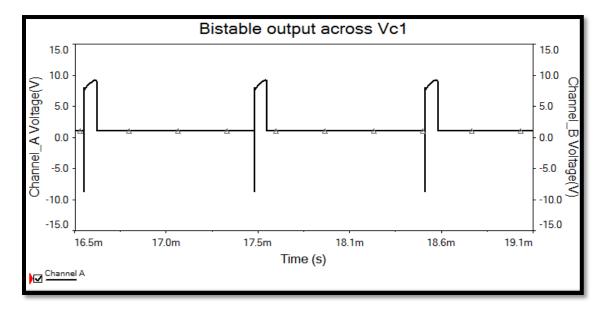
To design a bistable multivibrator using transistorand to Obtain its output characteristics by using multisim

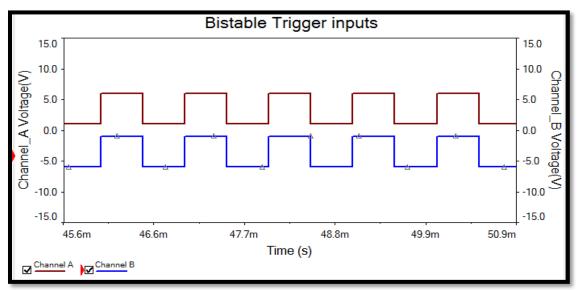
SOFTWARE TOOLS REQUIREMENT

PC with Multisim 12.0V

- 1. Start the Multisim 12.0V
- 2. Open the new project
- 3. Click and drag the components required from master database directory
- 4. Connect the components as per circuit diagram
- 5. Save the project
- 6. Click the run symbol to simulate the circuit
- 7. Click the grapher to view the output and note down the parameter

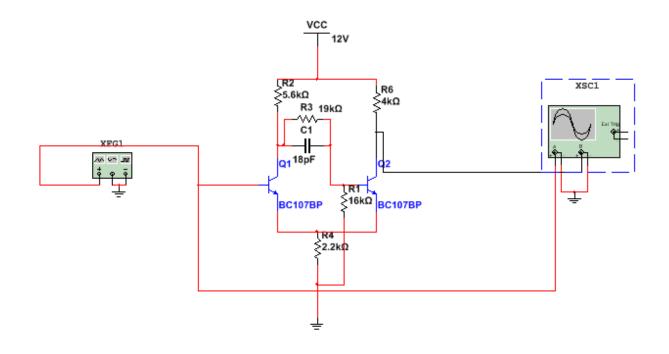
SIMULATED OUTPUT





| DECLIE TO | | |
|--|---|------------------|
| RESULT | | |
| RESULT Thus the bistable multivibrator is deusing multisim. | esigned and the output wavefo | orm is simulated |
| Thus the bistable multivibrator is do | esigned and the output wavefo | orm is simulated |
| Thus the bistable multivibrator is do | esigned and the output wavefo | |
| Thus the bistable multivibrator is do | | |
| Thus the bistable multivibrator is de | MARKS ALLO | CATION |
| Thus the bistable multivibrator is do | MARKS ALLO Experimental Setup | CATION 10 |
| Thus the bistable multivibrator is do | MARKS ALLO Experimental Setup Execution | CATION 10 |

SIMULATION CIRCUIT FOR SCHMITT TRIGGER



Ex. No:
Date:

SCHMITT TRIGGER WITH PREDICTABLE HYSRERISIS

AIM

To design the schmitt trigger using transistor and to obtain the output characteristics by using multisim

SOFTWARE TOOLS REQUIREMENT

1. PC with Multisim 12.0V

Hysteresis condition:

High threshold:

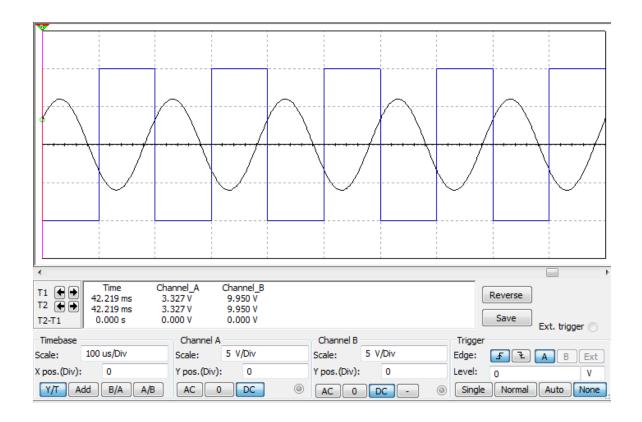
$$V_{\mathrm{HT}} = \frac{R_{\mathrm{E}}}{R_{\mathrm{E}} + R_{\mathrm{C2}}} V_{+}$$

Low threshold:

$$V_{\mathrm{LT}} = \frac{R_{\mathrm{E}}}{R_{\mathrm{E}} + R_{\mathrm{C1}}} V_{+}$$

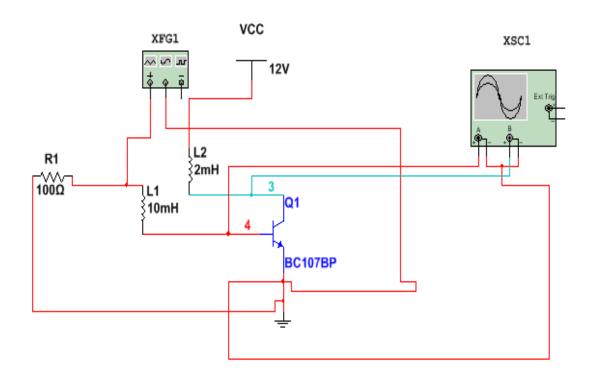
- 1. Start the Multisim 12.0V
- 2. Open the new project
- 3. Click and drag the components required from master database directory
- 4. Connect the components as per circuit diagram
- 5. Save the project
- 6. Click the run symbol to simulate the circuit
- 7. Click the grapher to view the output and note down the parameter

SIMULATED WAVEFORM

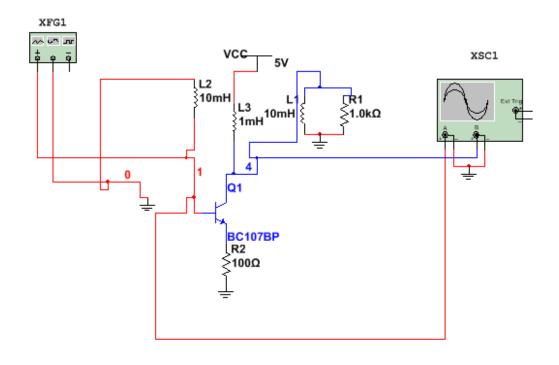


| RESULT | | |
|--|-------------------------------------|-----------------|
| Thus the Schmitt Trigger is designed simulated using multisim. | l using transistor and the out | put waveform is |
| | MARKS ALLO | CATION |
| | | 10 |
| | Experimental Setup | |
| | Experimental Setup Execution Viva | 10 |

SIMULATION CIRCUIT FOR MONOSTABLE MULTIVIBRATOR WITH BASE TIMING



$\frac{\textbf{SIMULATION CIRCUIT FOR MONOSTABLE MULTIVIBRATOR WITH BASE}}{\textbf{TIMING}}$



| Ex. No: | |
|---------|--|
| Date: | |

MONOSTABLE MULTIVIBRATOR WITH BASE TIMING AND EMITTER TIMING

AIM

To design the monostable multivibrator with base timing and emitter timing using transistor and to obtain the output characteristics by using multisim

SOFTWARE TOOLS REQUIREMENT

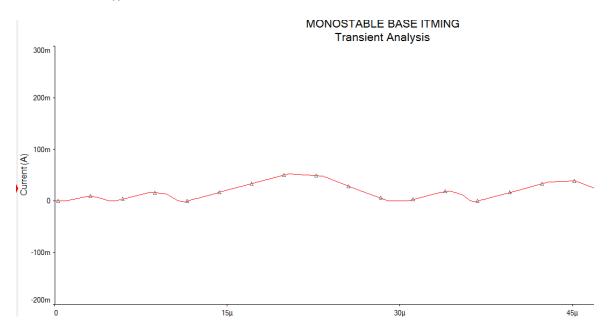
PC with Multisim 12.0V

- 1. Start the Multisim 12.0V
- 2. Open the new project
- 3. Click and drag the components required from master database directory
- 4. Connect the components as per circuit diagram
- 5. Save the project
- 6. Click the run symbol to simulate the circuit
- 7. Click the grapher to view the output and note down the parameter

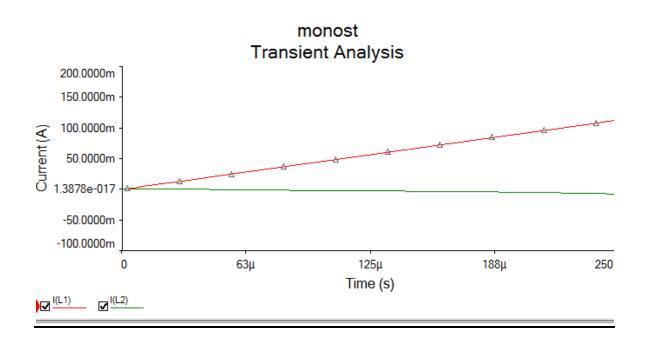
SIMULATED WAVEFORMS:

Transient analysis response(collector current vs time)

(i)MONOSTABLE MULTIVIBRATOR WITH BASE TIMING



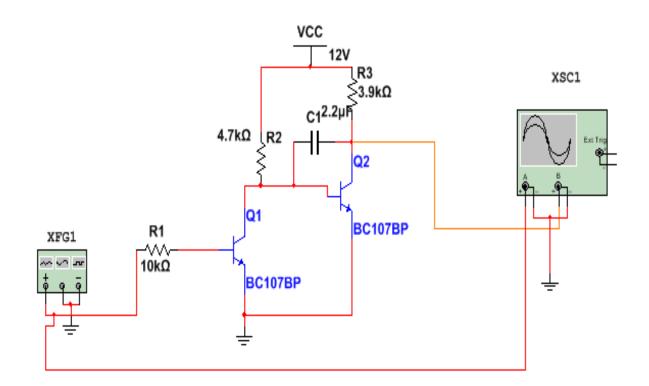
(ii)MONOSTABLE MULTIVIBRATOR WITH EMITTER TIMING



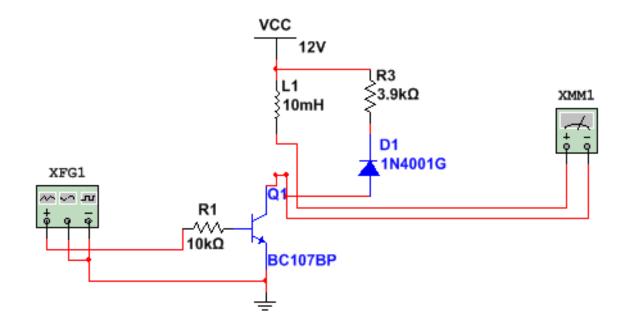
EC 6411 CIRCUITS AND SIMULATION INTEGRATED LABORATORY

| RESULT: | | | |
|--|------------------------------------|----------------|----|
| Thus the monostable multivibr | rator with emitter timing and base | timing Circuit | ts |
| 1 | atput waveform is simulated using | multisim. | |
| are designed using transistor and the ou | | | |
| are designed using transistor and the ou | | | |
| are designed using transistor and the ou | MARKS ALLO | CATION | |
| are designed using transistor and the ou | Experimental Setup | 10 | |
| are designed using transistor and the ou | Experimental Setup Execution | 10 | |
| are designed using transistor and the ou | Experimental Setup | 10 | |

SIMULATION CIRCUIT FOR VOLTAGE TIME BASE CIRCUIT:



IMULATION CIRCUIT FOR CURRENT TIME BASE CIRCUIT:



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

AIM

To design a voltage and current time base circuits using transistor and to obtain its output characteristics by using multisim.

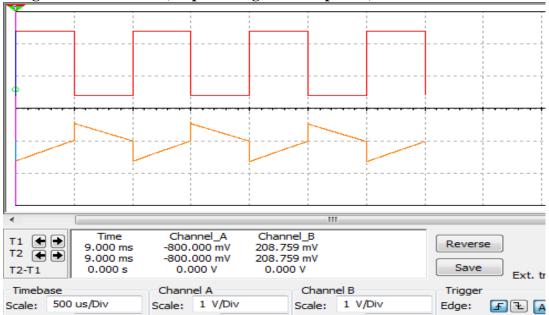
SOFTWARE TOOL REQUIREMENT

1. PC with Multisim 12.0

- 1. Start the Multisim 12.0V
- 2. Open the new project
- 3. Click and drag the components required from master database directory
- 4. Connect the components as per circuit diagram
- 5. Save the project
- 6. Click the run symbol to simulate the circuit
- 7. Click the grapher to view the output and note down the parameter

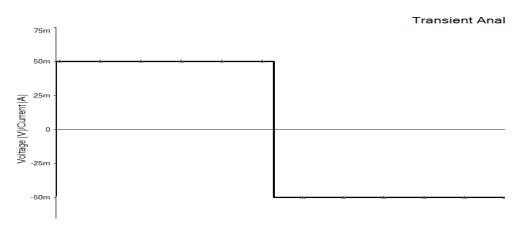
SIMULATED WAVEFORM

(i)voltage time base circuit(output voltage vs time period)

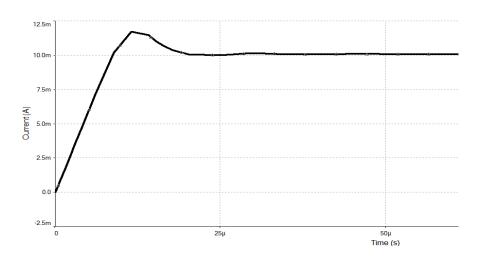


(ii)current time base circuit

Transient Analysis for Input:



Current Output Across Inductor Vs Time:



| RESULT: | |
|---|------|
| RESULT: Thus the Current Time Base Circuit is designed using transistor and the out waveform is simulated using multisim. | tput |
| Thus the Current Time Base Circuit is designed using transistor and the out waveform is simulated using multisim. | tput |
| Thus the Current Time Base Circuit is designed using transistor and the out waveform is simulated using multisim. MARKS ALLOCATION | tput |
| Thus the Current Time Base Circuit is designed using transistor and the out waveform is simulated using multisim. | tput |
| Thus the Current Time Base Circuit is designed using transistor and the out waveform is simulated using multisim. MARKS ALLOCATION Experimental Setup 10 | tput |