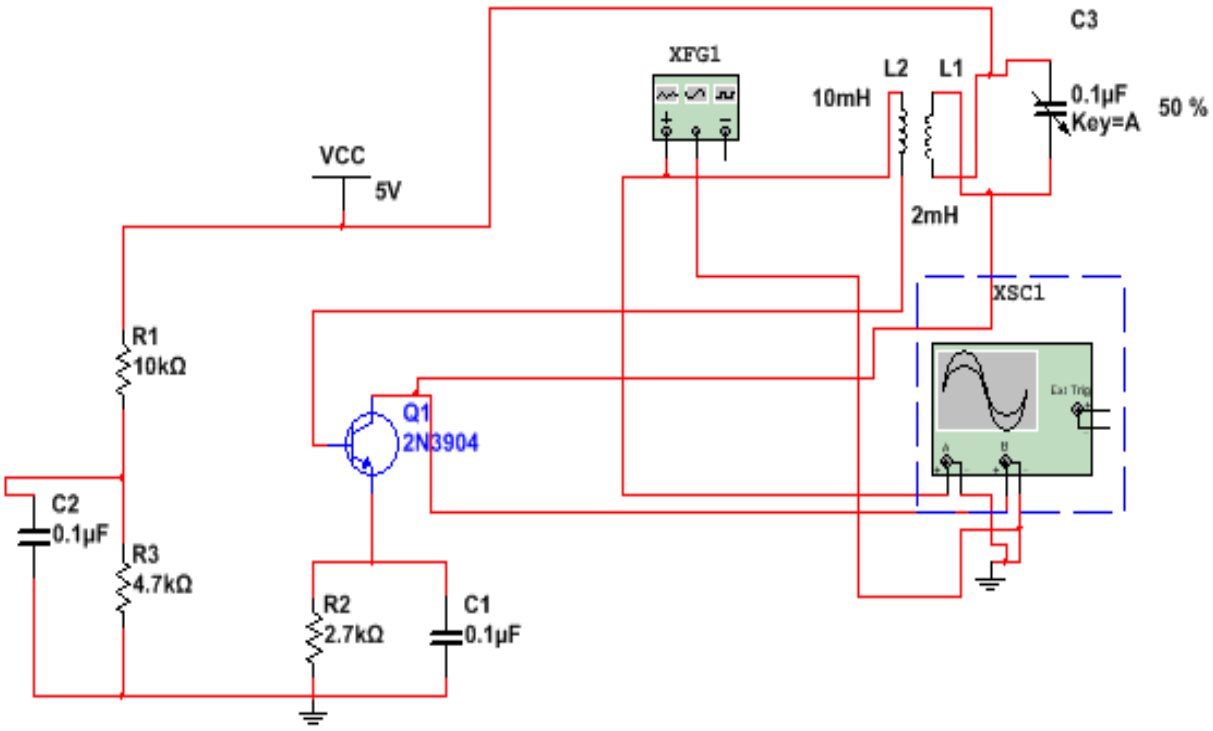
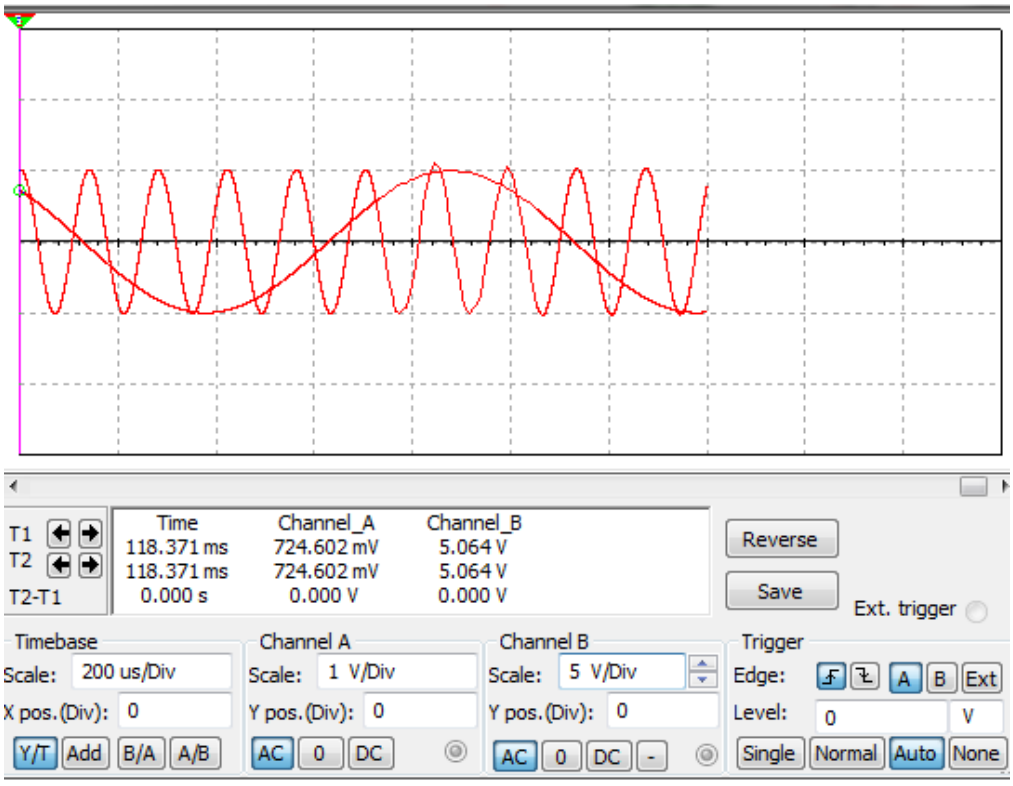
SIMULATION

EXPERIMENTS

SIMULATION DIAGRAM FOR TUNED COLLECTOR OSCILLATOR



SIMULATED OUTPUT WAVE FORM



Ex. No:
Date:

TUNED COLLECTOR OSCILLATOR

AIM

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

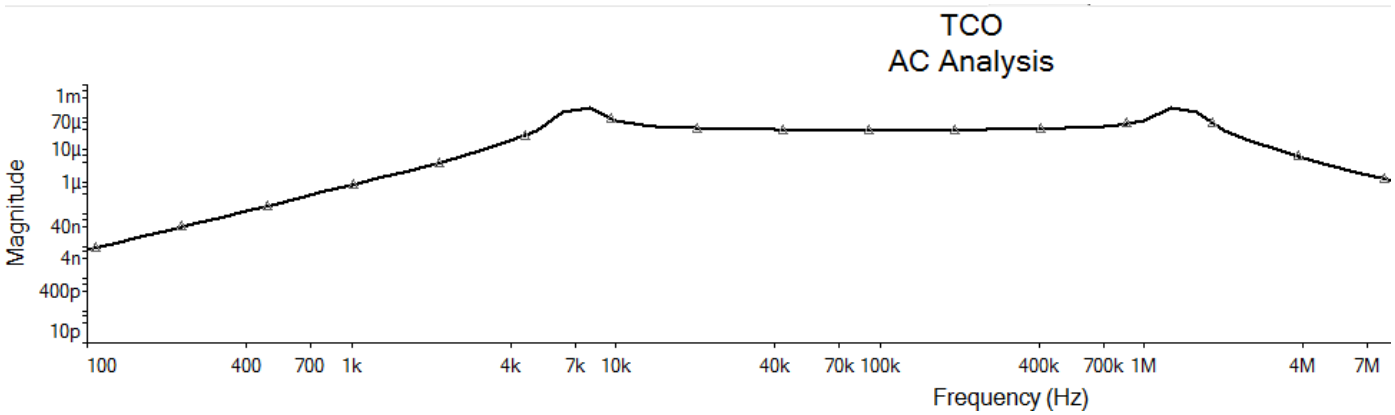
SOFTWARE TOOLS REQUIREMENT

1. PC with Multisim 12.0V

PROCEDURE:

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

FREQUENCY RESPONSE PLOT

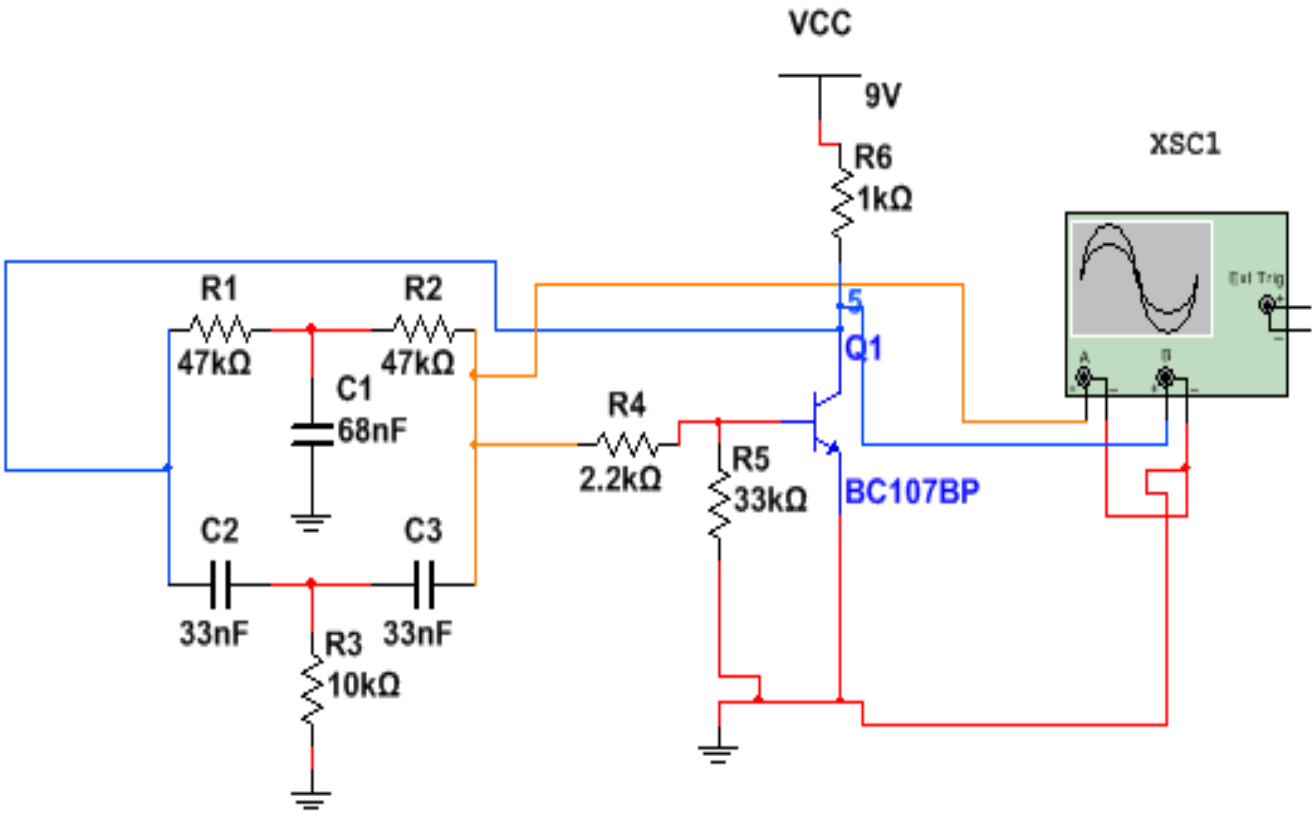


RESULT:

Thus the tuned collector oscillator is designed , simulated and wave forms are observed.

MARKS ALLOCATION		
Experimental Setup	**10**	
Execution	**10**	
Viva	**10**	
Total	**30**	

SIMULATION CIRCUIT FOR TWIN-T OSCILLATOR



Ex. No:
Date:

TWIN-T OSCILLATOR

AIM

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

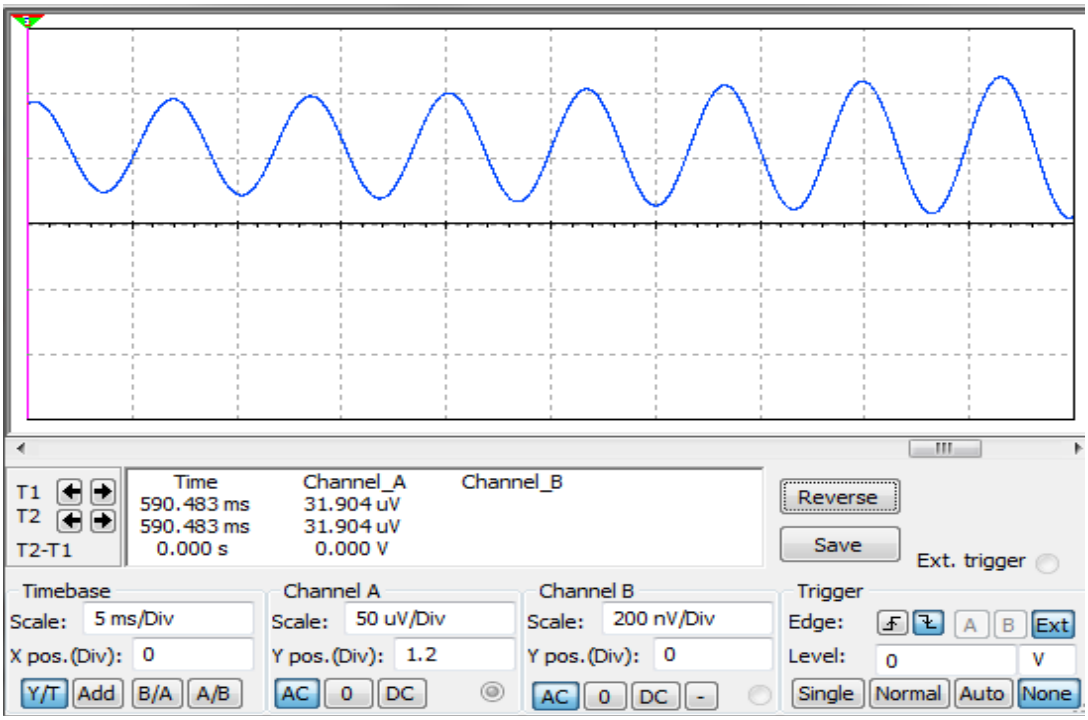
SOFTWARE TOOLS REQUIREMENT

PC with Multisim 12.0V

PROCEDURE:

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

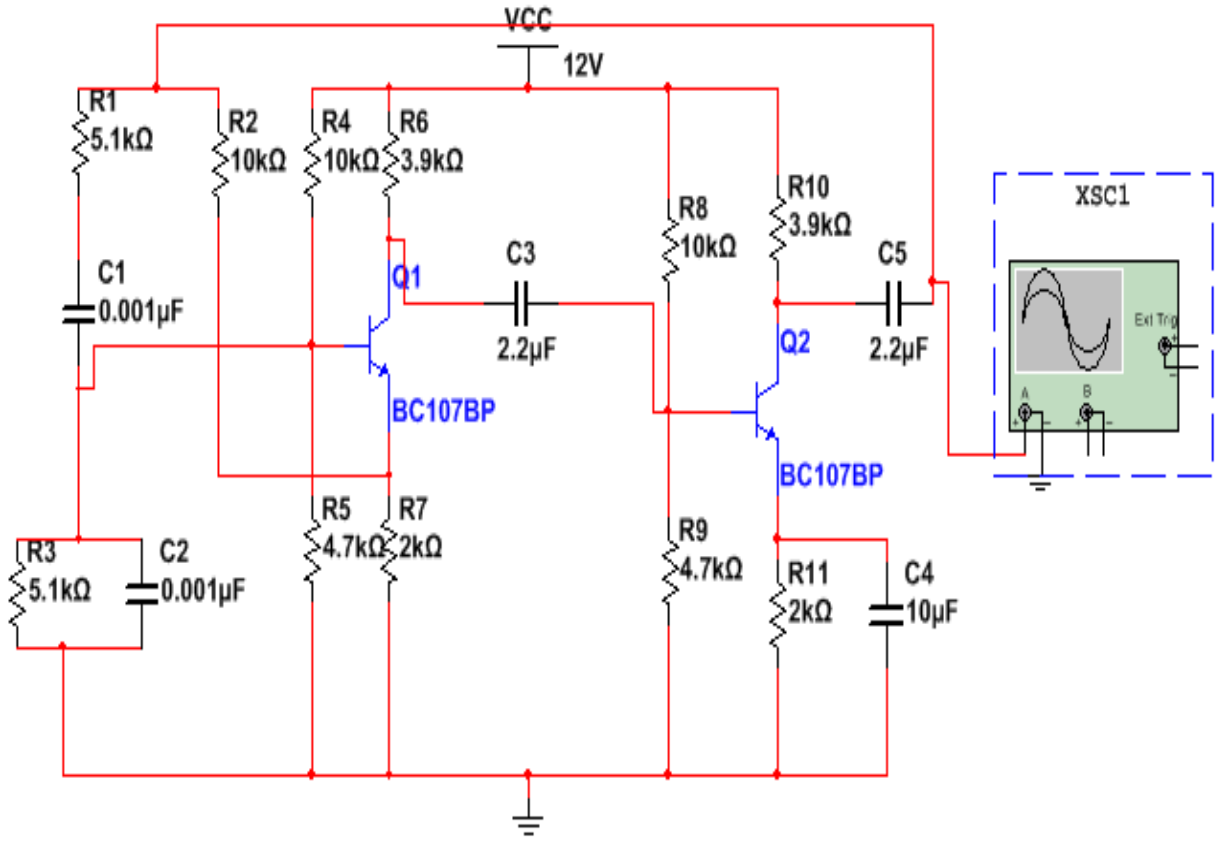


RESULT:

Thus the twin t oscillator is designed , simulated and wave forms are observed.

MARKS ALLOCATION		
Experimental Setup	10	
Execution	10	
Viva	10	
Total	30	

SIMULATION CIRCUIT FOR WEIN BRIDGE OSCILLATOR:



Ex. No:
Date:

WEIN BRIDGE OSCILLATOR

AIM

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

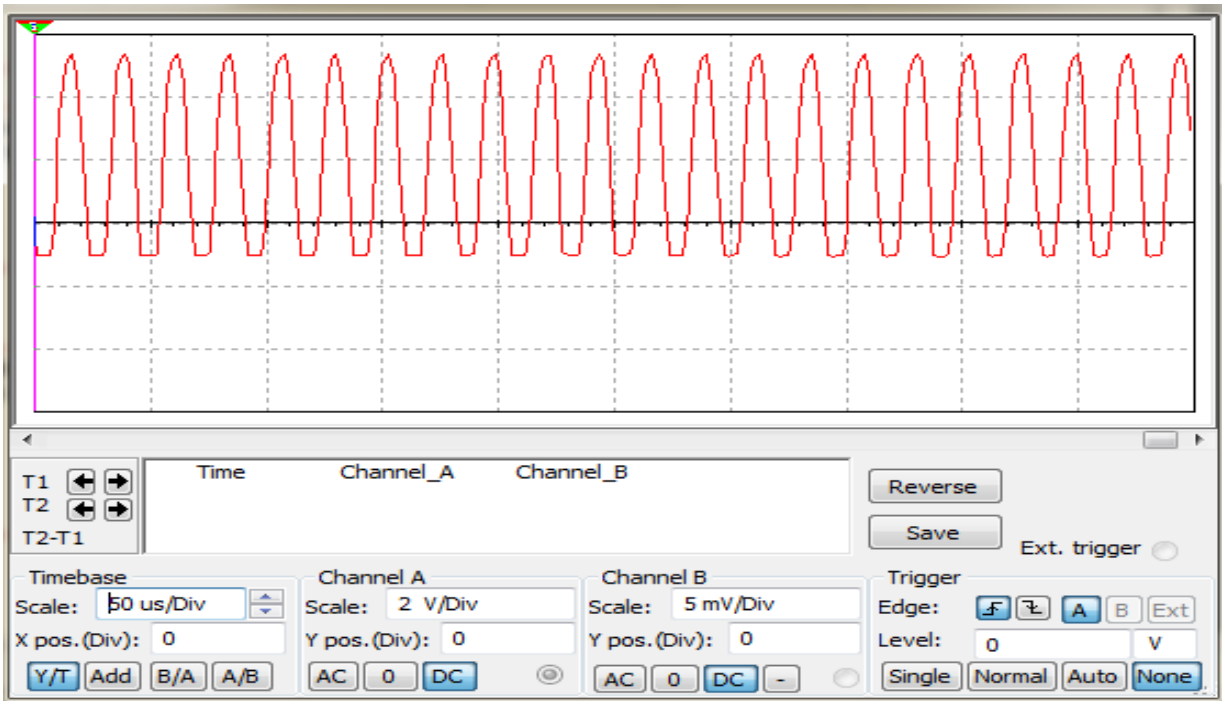
SOFTWARE TOOLS REQUIREMENT

PC with Multisim 12.0V

PROCEDURE:

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

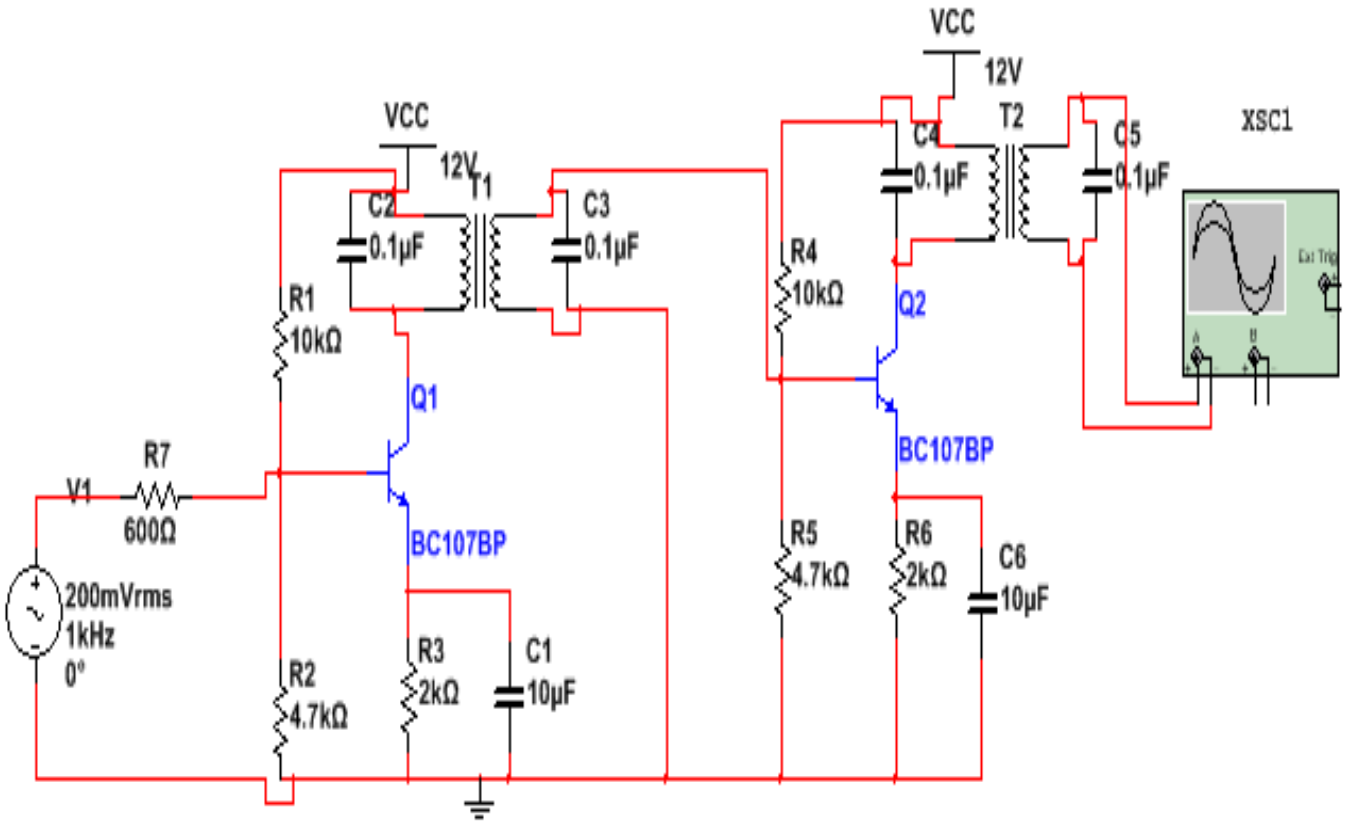


RESULT:

Thus the wein bridge oscillator is designed , simulated and wave forms are observed.

MARKS ALLOCATION		
Experimental Setup	**10**	
Execution	**10**	
Viva	**10**	
Total	**30**	

SIMULATION CIRCUIT FOR DOUBLE TUNED AMPLIFIER:



Ex. No:
Date:

DOUBLE TUNED AMPLIFIER

AIM

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

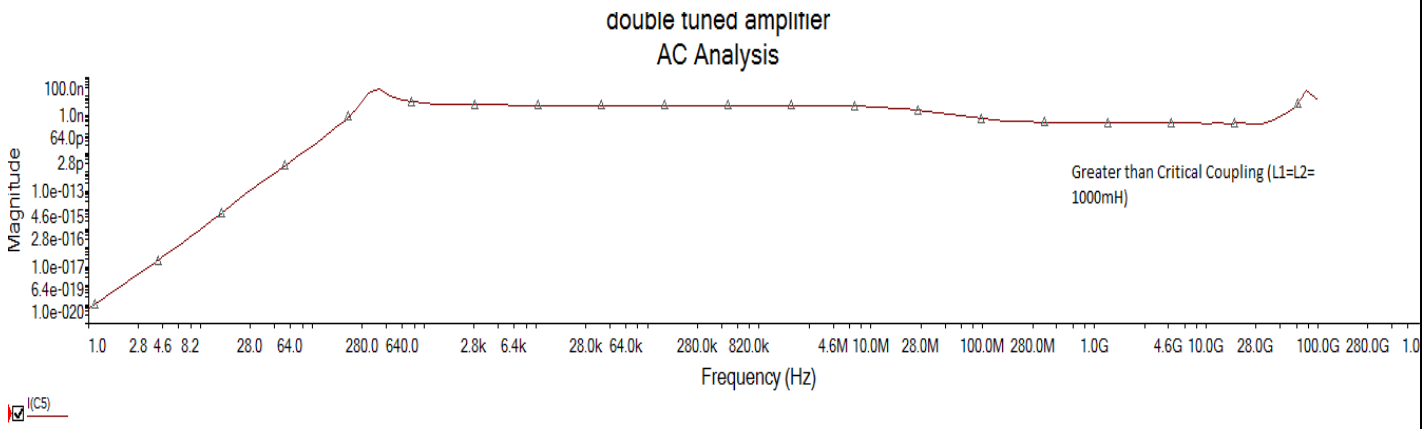
SOFTWARE TOOLS REQUIREMENT

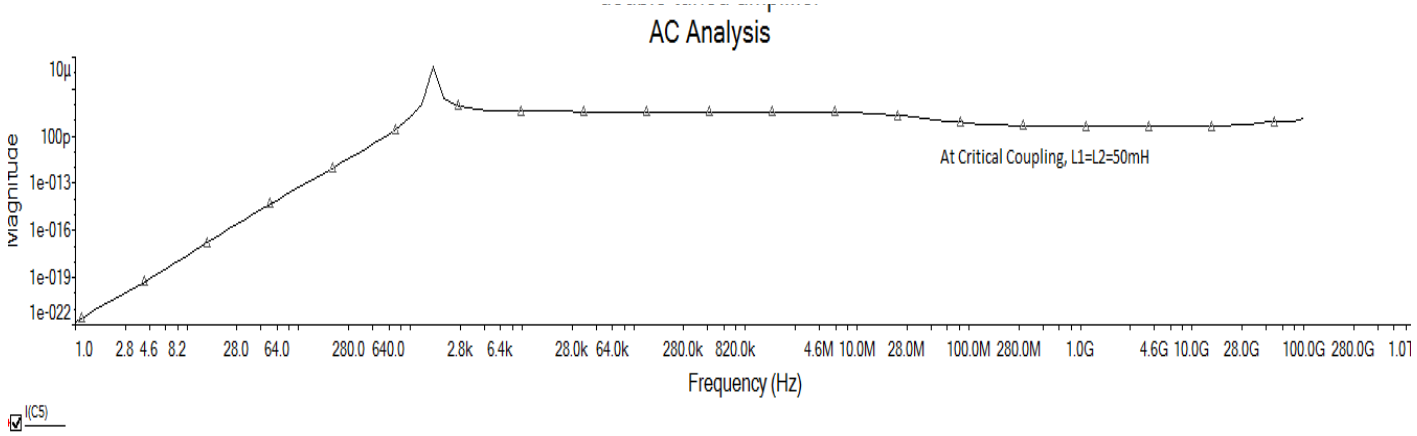
PC with Multisim 12.0V

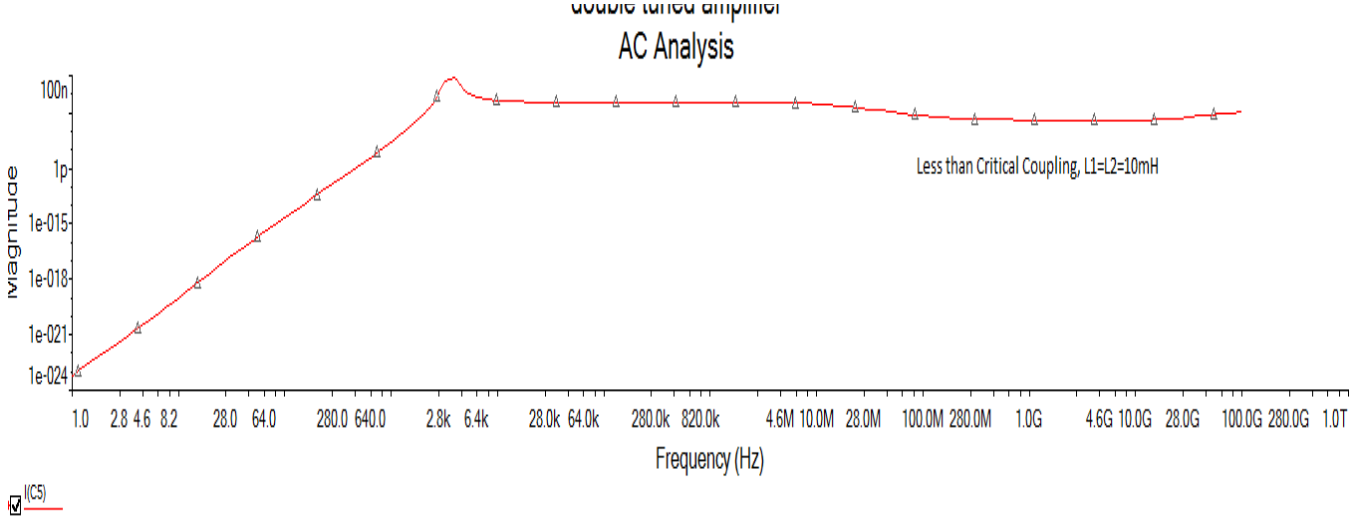
PROCEDURE:

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





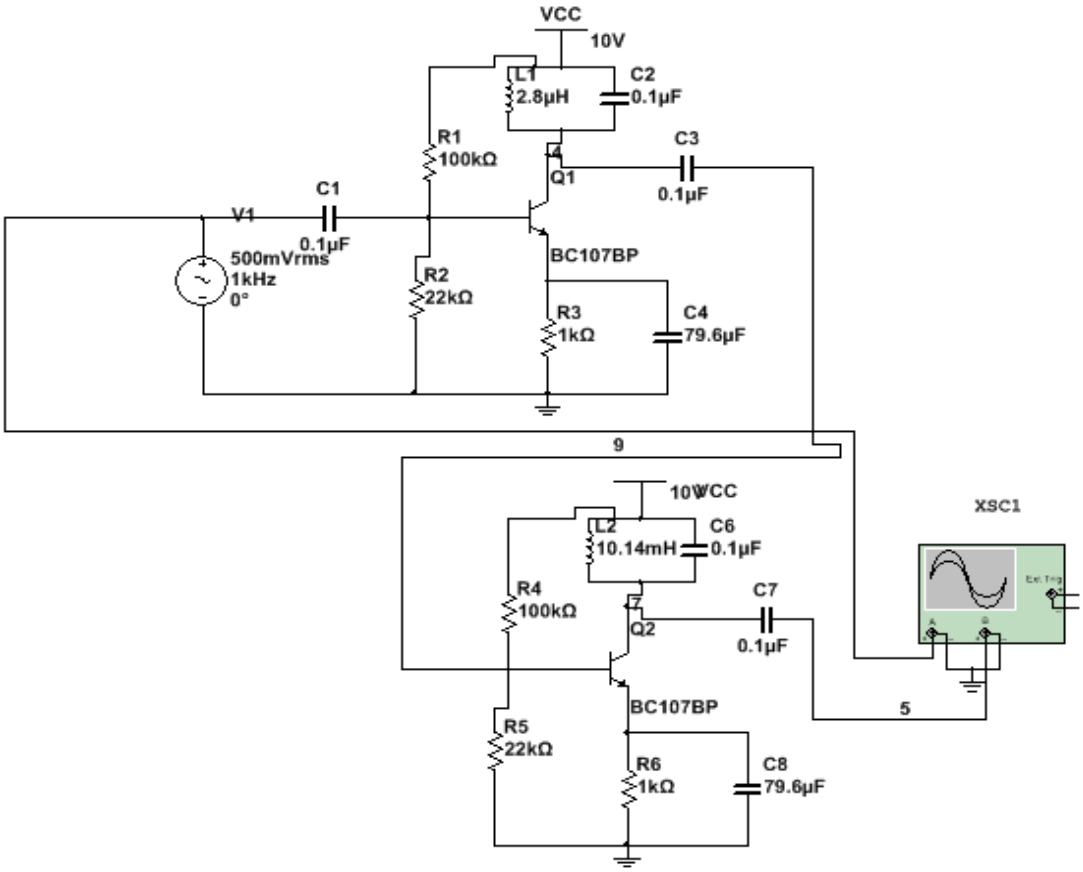


RESULT

Thus the double tuned amplifier is designed and the output waveform is simulated using multisim.

MARKS ALLOCATION		
Experimental Setup	**10**	
Execution	**10**	
Viva	**10**	
Total	**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.

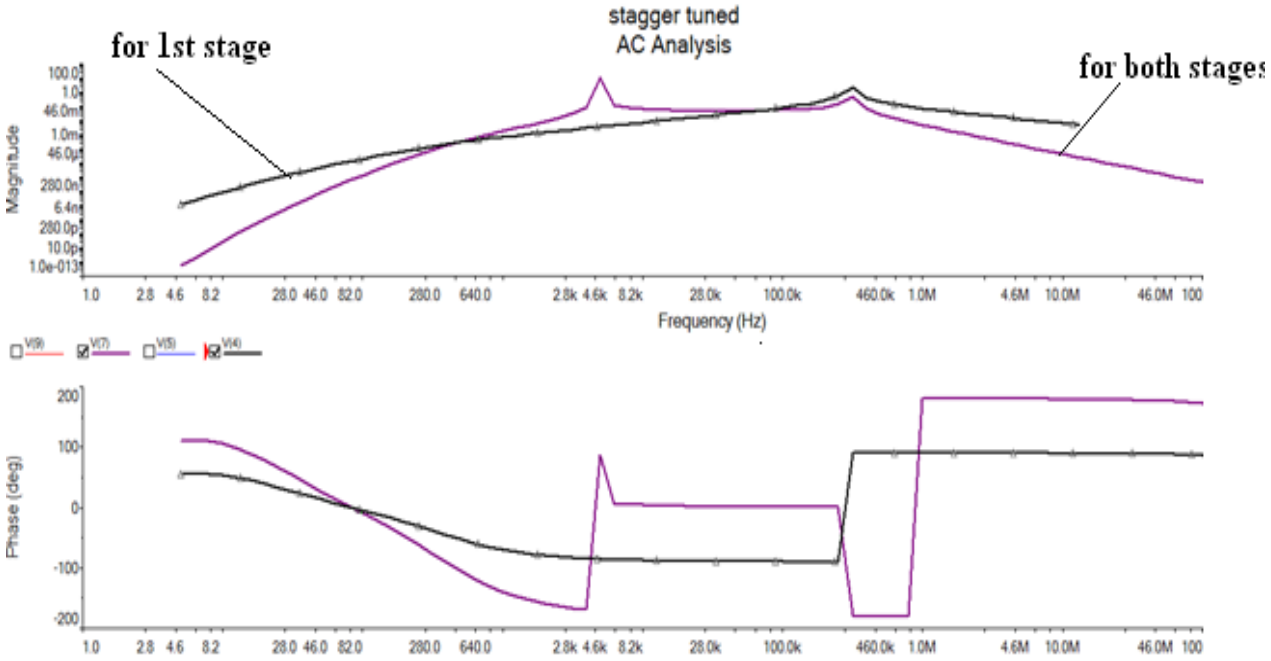
SOFTWARE TOOLS REQUIREMENT

PC with Multisim 12.0V

PROCEDURE:

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

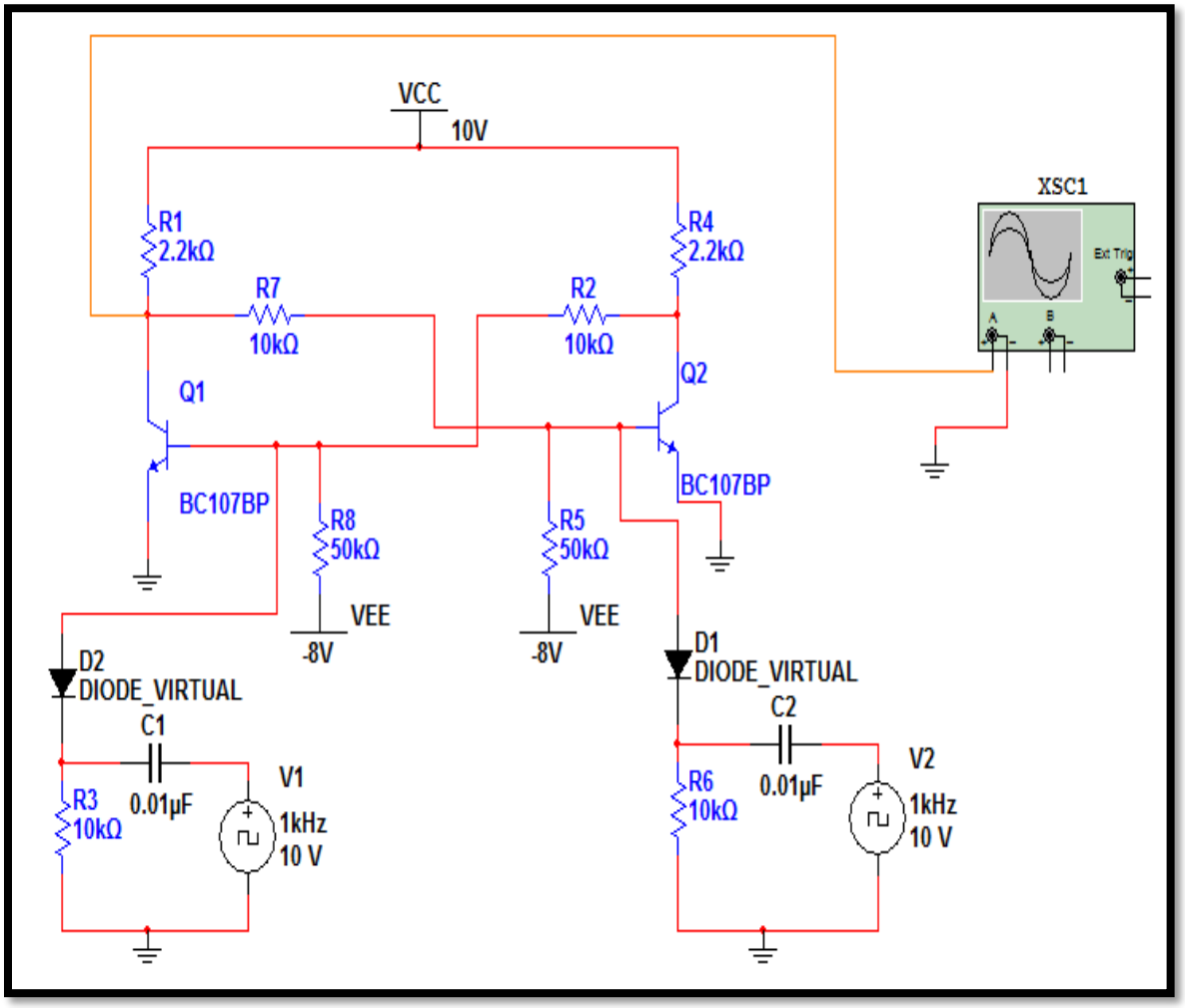


RESULT

Thus the stagger tuned amplifier is designed and the output waveform is simulated using multisim.

MARKS ALLOCATION		
Experimental Setup	**10**	
Execution	**10**	
Viva	**10**	
Total	**30**	

SIMULATION CIRCUIT FOR BISTABLE MULTIVIBRATOR



Ex. No:
Date:

BISTABLE MULTIVIBRATOR

AIM

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

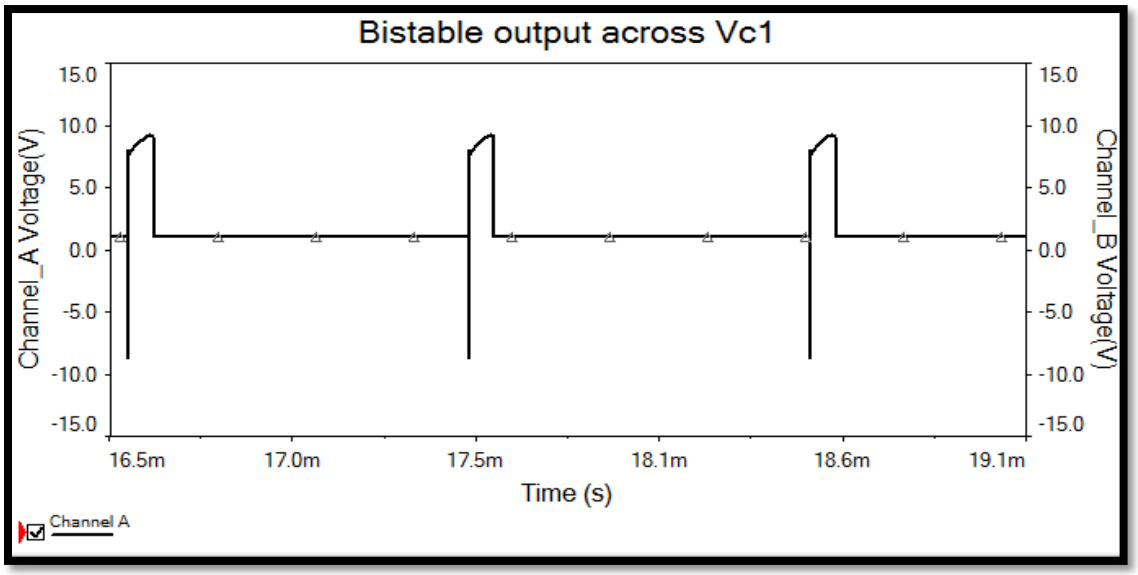
SOFTWARE TOOLS REQUIREMENT

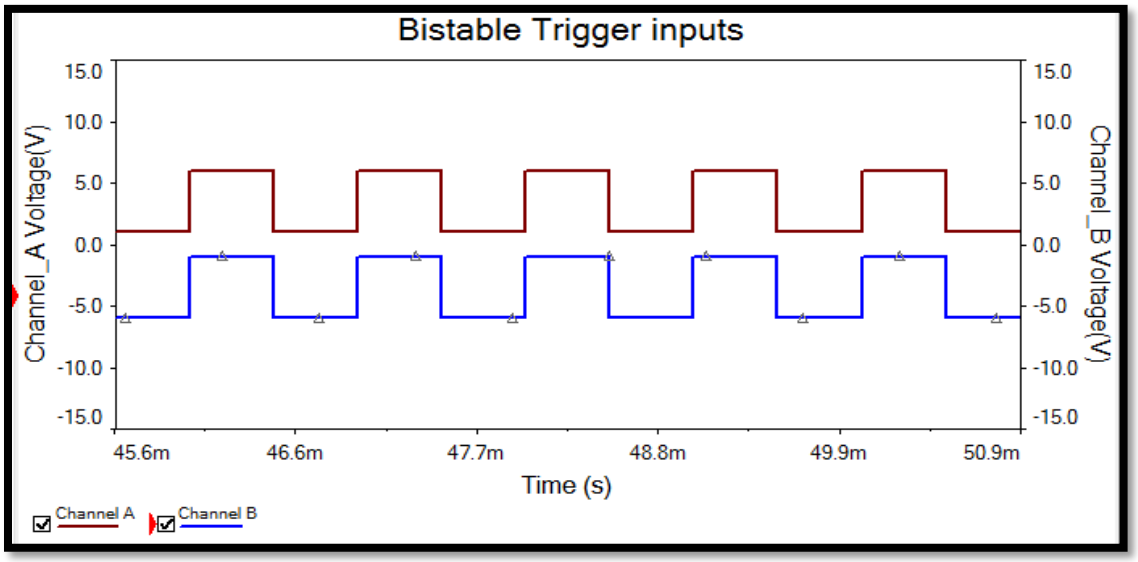
PC with Multisim 12.0V

PROCEDURE:

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



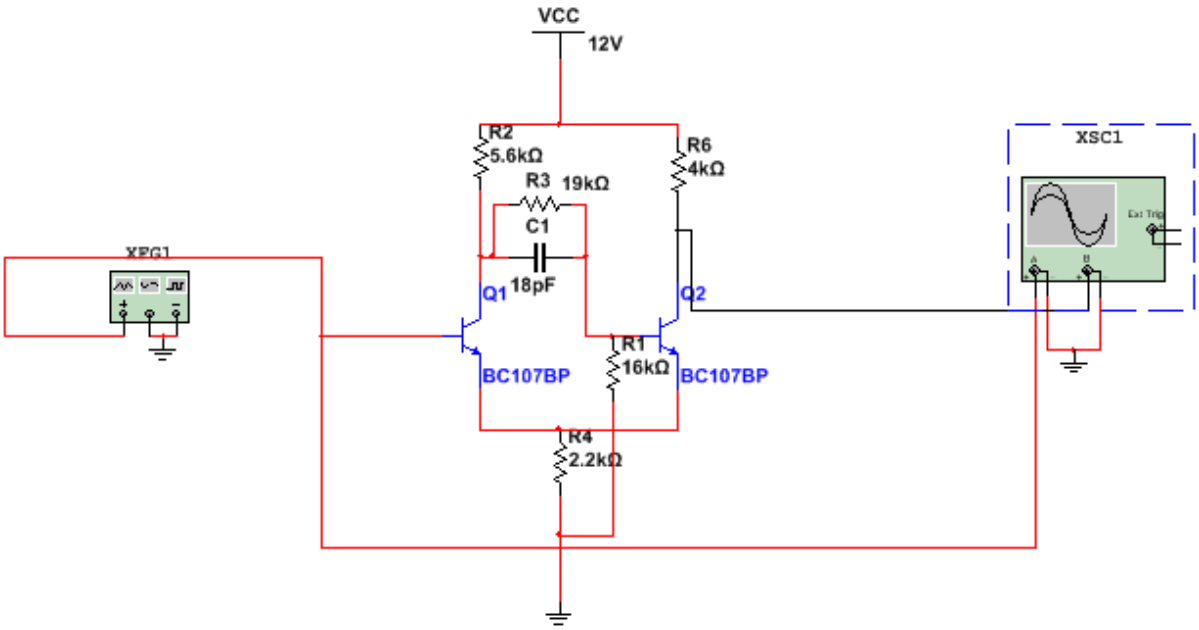


RESULT

Thus the bistable multivibrator is designed and the output waveform is simulated using multisim.

MARKS ALLOCATION		
Experimental Setup	**10**	
Execution	**10**	
Viva	**10**	
Total	**30**	

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:

$$V_{HT} = \frac{R_E}{R_E + R_{C2}} V_+$$

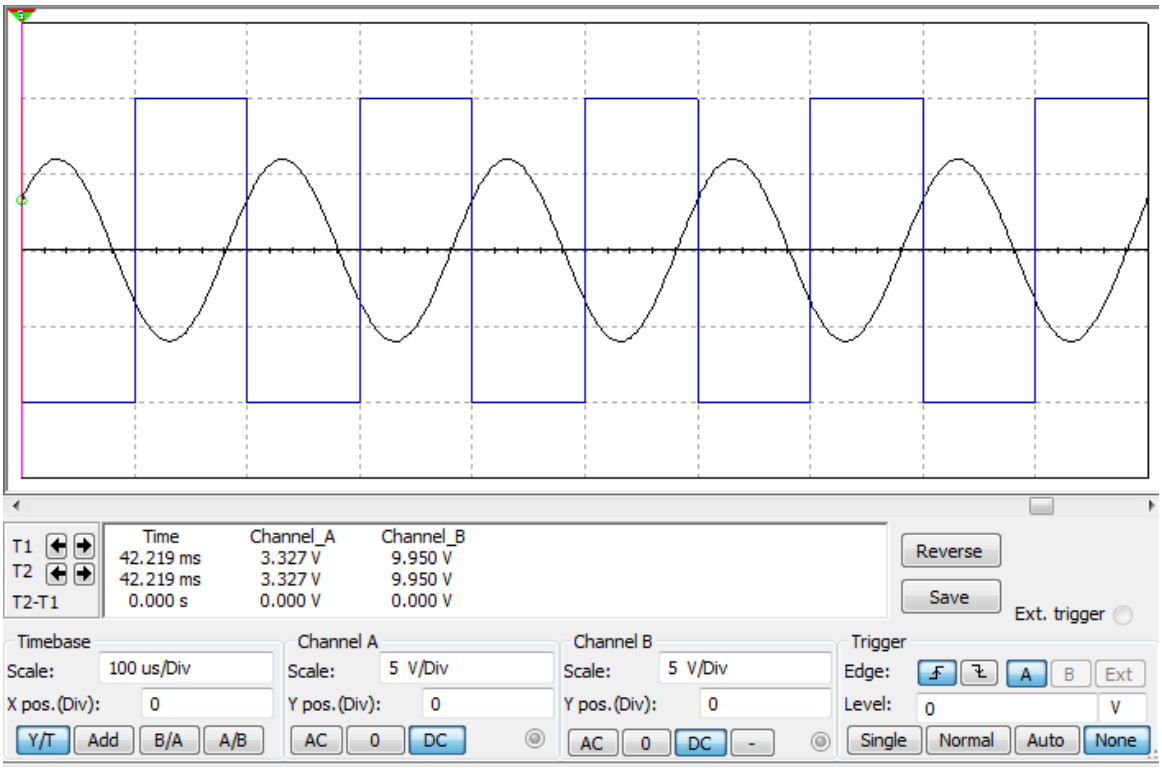
Low threshold:

$$V_{LT} = \frac{R_E}{R_E + R_{C1}} V_+$$

PROCEDURE

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

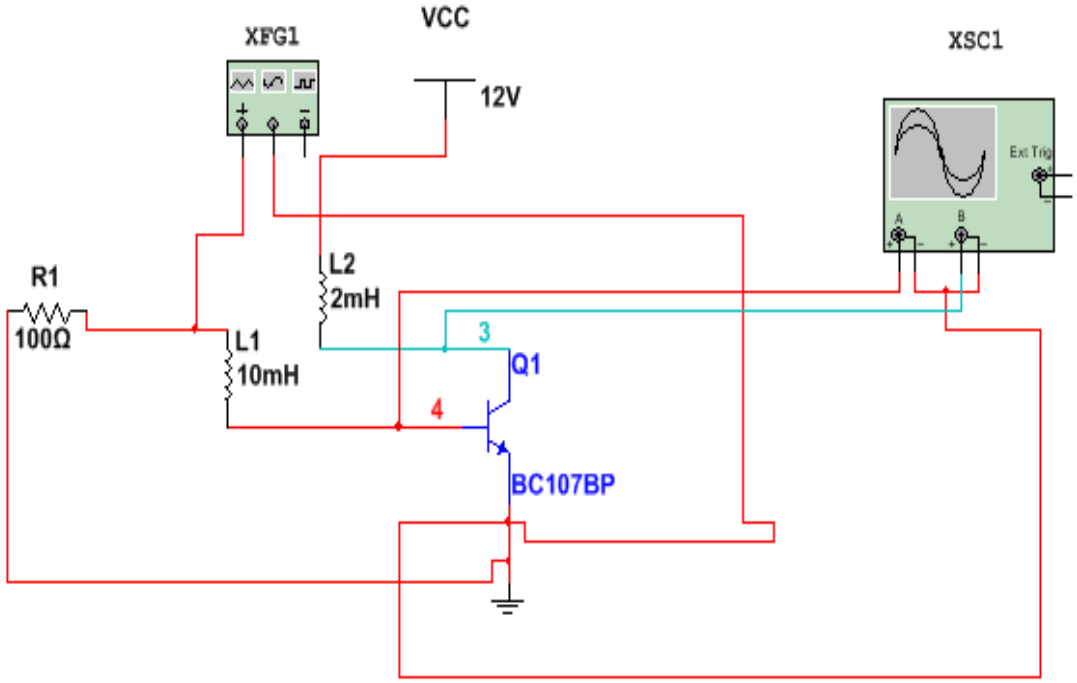


RESULT

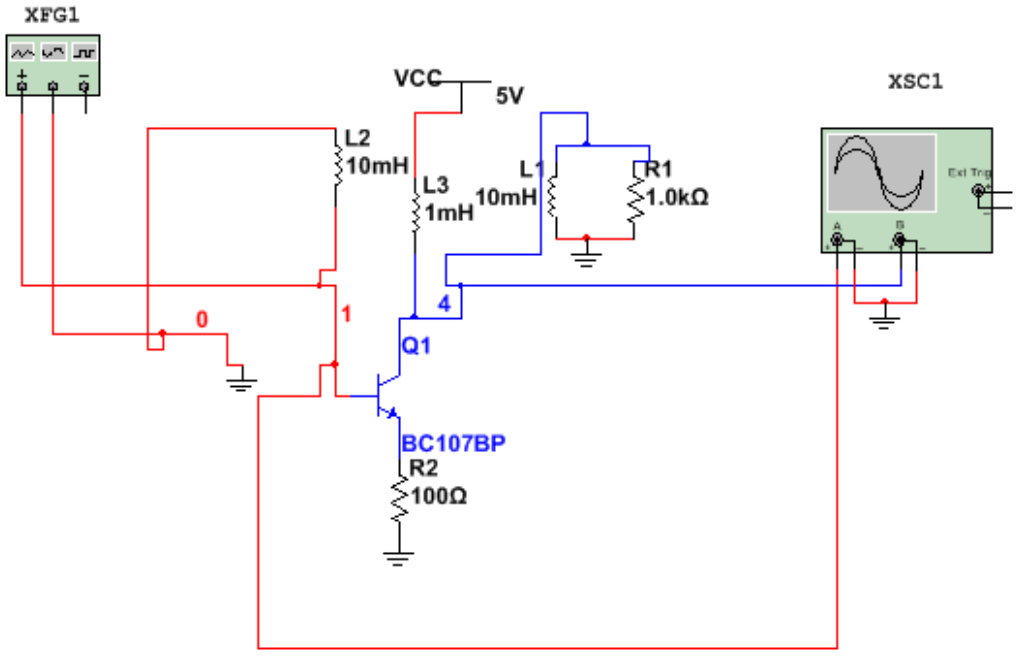
Thus the Schmitt Trigger is designed using transistor and the output waveform is simulated using multisim.

MARKS ALLOCATION		
Experimental Setup	**10**	
Execution	**10**	
Viva	**10**	
Total	**30**	

SIMULATION CIRCUIT FOR MONOSTABLE MULTIVIBRATOR WITH BASE TIMING



SIMULATION CIRCUIT FOR MONOSTABLE MULTIVIBRATOR WITH BASE 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

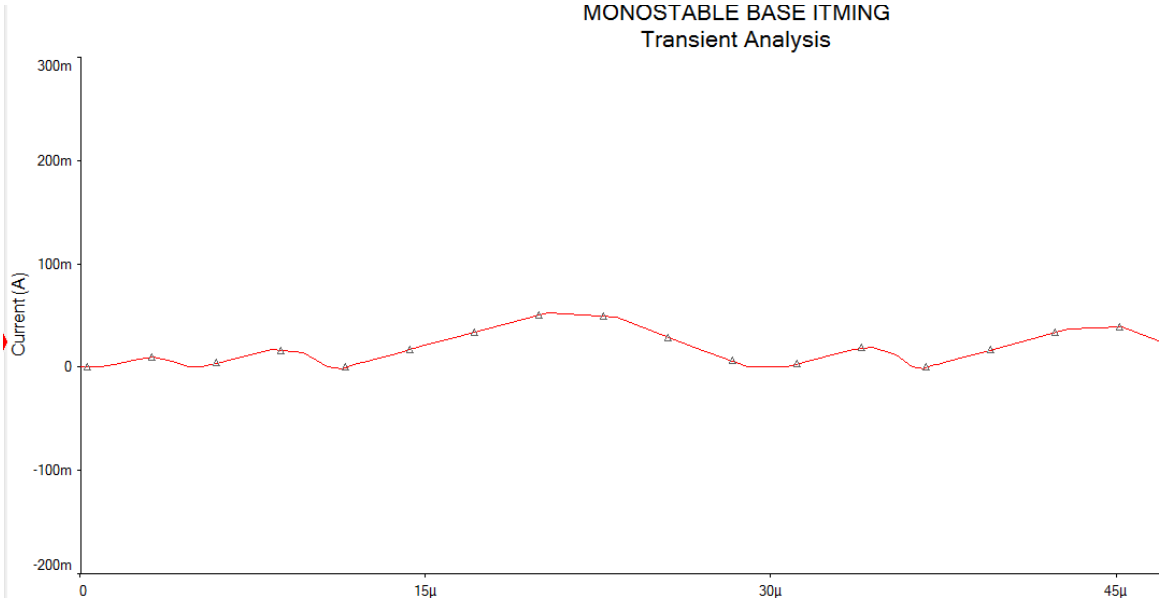
PROCEDURE

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

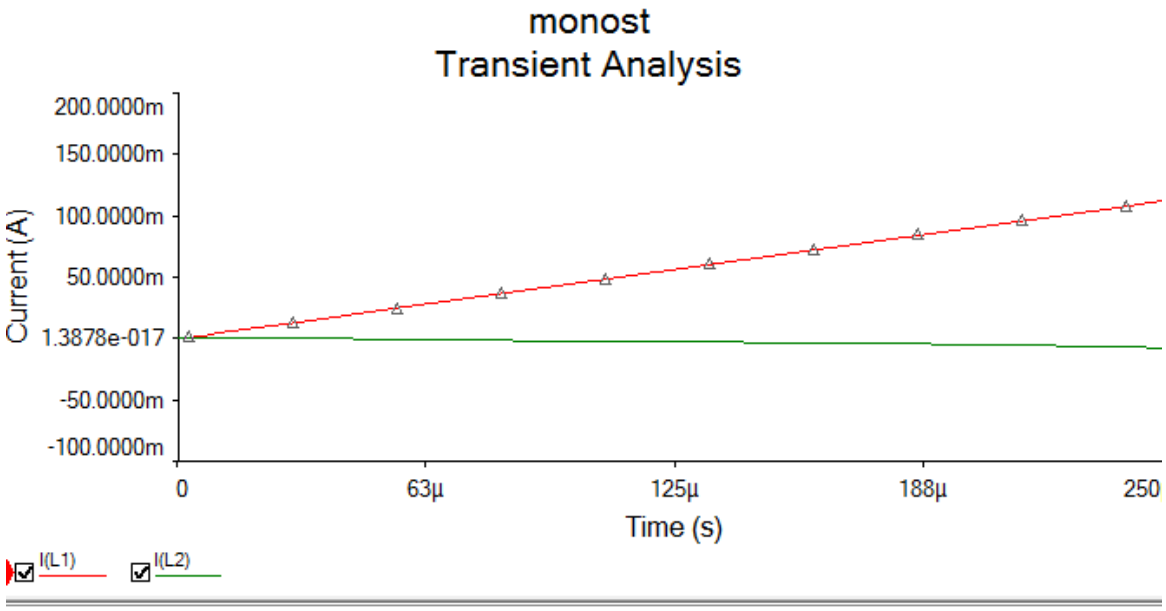
SIMULATED WAVEFORMS:

Transient analysis response(collector current vs time)

(i)MONOSTABLE MULTIVIBRATOR WITH BASE TIMING



(ii)MONOSTABLE MULTIVIBRATOR WITH EMITTER TIMING

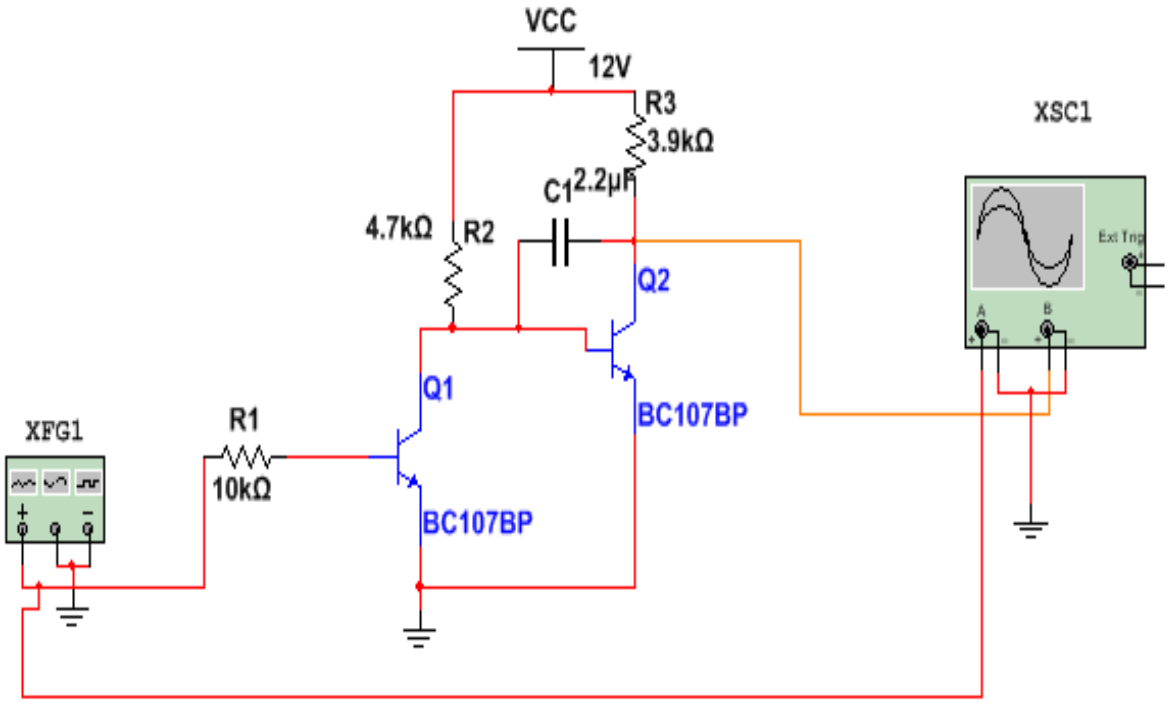


RESULT:

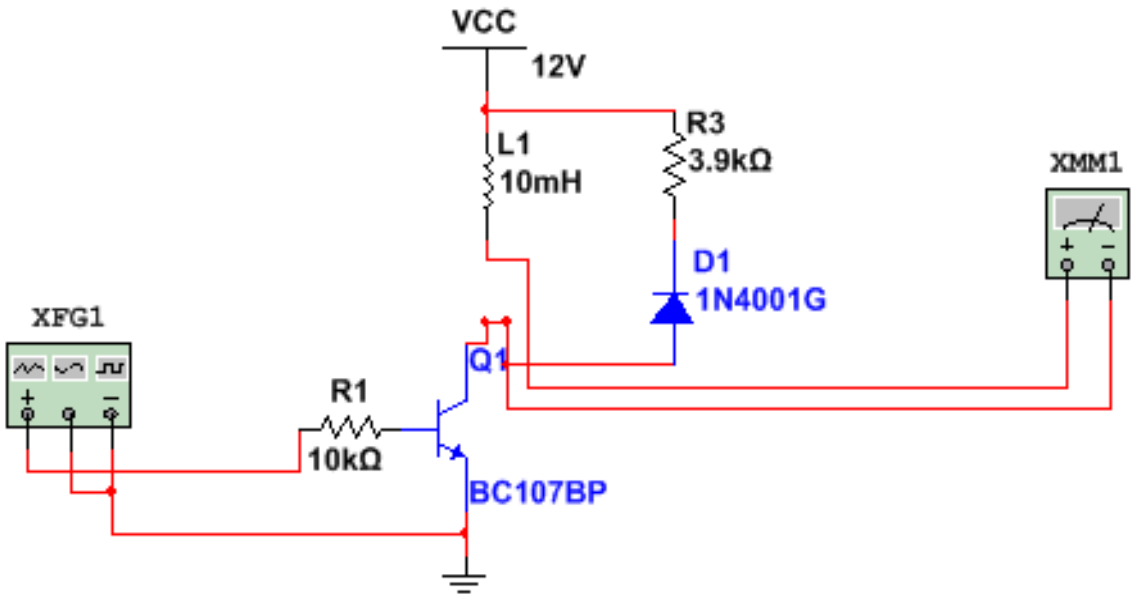
Thus the monostable multivibrator with emitter timing and base timing Circuits are designed using transistor and the output waveform is simulated using multisim.

MARKS ALLOCATION		
Experimental Setup	**10**	
Execution	**10**	
Viva	**10**	
Total	**30**	

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

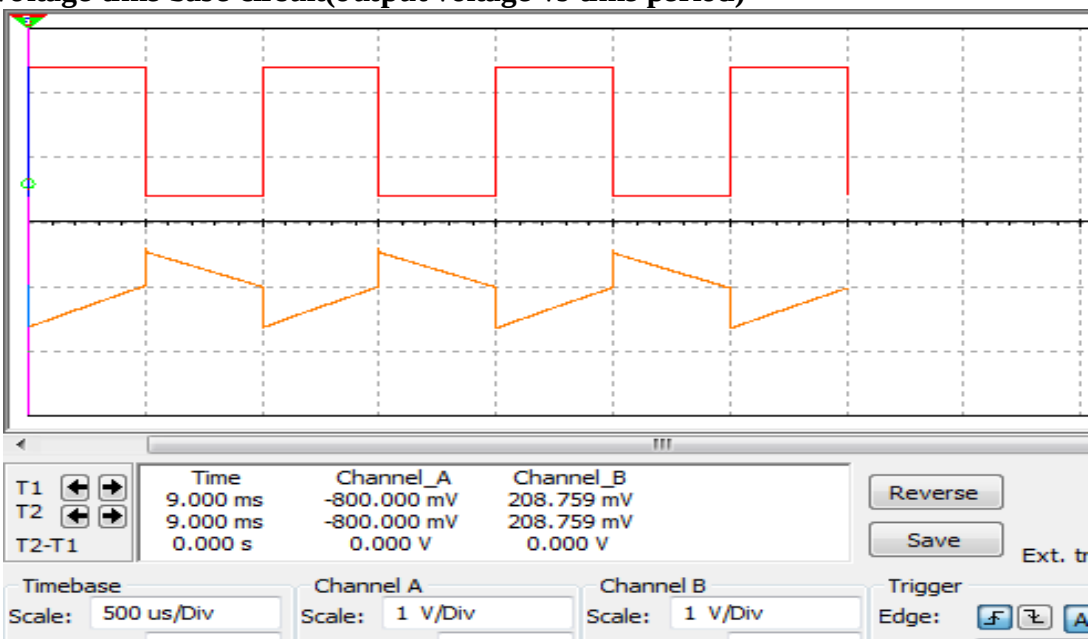
1. PC with Multisim 12.0

PROCEDURE

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

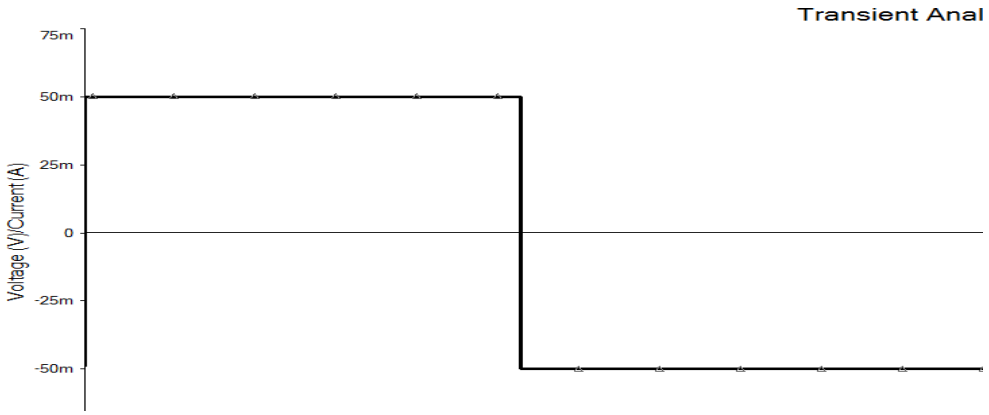
SIMULATED WAVEFORM

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

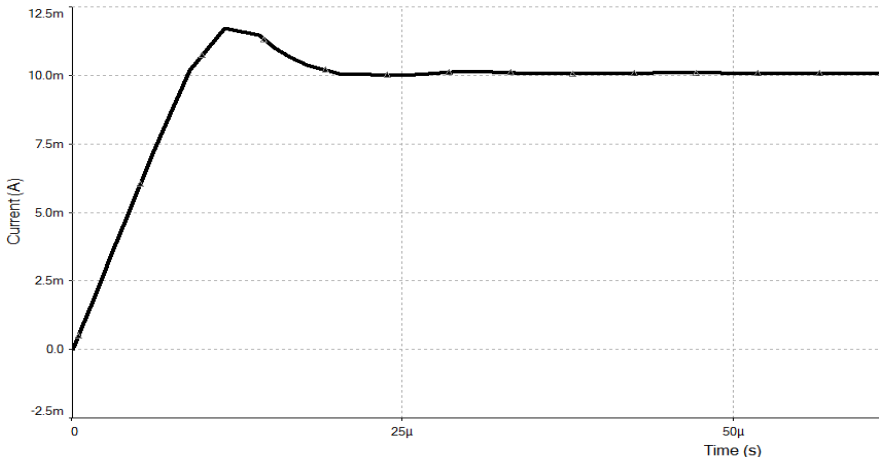


(ii)current time base circuit

Transient Analysis for Input:



Current Output Across Inductor Vs Time:



RESULT:

Thus the Current Time Base Circuit is designed using transistor and the output waveform is simulated using multisim.

MARKS ALLOCATION		
Experimental Setup	**10**	
Execution	**10**	
Viva	**10**	
Total	**30**	