MARRI LAXMAN REDDY
INSTITUTE OF TECHNOLOGY AND MANAGEMENT
(AN AUTONOMOUS INSTITUTION)
(Approved by AICTE, New Delhi & Affiliated to JNTUH, Hyderabad)
Accredited by NBA and NAAC with 'A' Grade & Recognized Under Section2(f) & 12(B)of the UGC act,1956

LABORATORY MANUAL
ELECTRONIC CIRCUITS ANALYSIS LAB

II B. TECH -I SEMESTER (ECE)
MLRS-R25 REGULATION

AY-2025-2026

Prepared by
Lab In-charge:

Ms. R. Anuja
Mr. G.Siva Sankar Varma

DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

CERTIFICATE

This is to certify that this manual is a bonafide record of practical work in the ***Electronic Circuits Laboratory*** in **Second Semester of II-Year B. Tech (ECE) Programme** during the academic year **2025-2026.** This book is prepared by **Ms. R. Anuja, (Assistant Professor) and Mr. G. Siva Sankar Varma (Assistant Professor)** Department of Electronics and Communication Engineering.

Lab I/C **Head of the Department**

PREFACE

This laboratory lays the foundation for the Electronics and Communication Engineering students during second year of their course.

Circuits and components of EC Lab can be divided into two batches: Hardware & Software. In Hardware lab, the students design, construct and test the working of analog circuits such as transistor amplifiers, and oscillators and also pulse circuits like multivibrators and Time Base generators. In Software, the students will execute verify the outputs of few experiments using Multisim software. After performing all the experiments included in this Laboratory, it is hoped the student receives good training to handle any electronic equipment available in electronics field.

Basic electronic equipment used in electronics laboratory

- Cathode Ray Oscilloscope to view and measure AC waveforms.
- Function Generator used to design, develop and to trouble shoot electronic equipments
- Regulated power supply to supply DC voltage to electronic circuits.
- Breadboard to mount the components temporarily for experimental work.

By,
R. ANUJA,
G. SIVA SANKAR VARMA

ACKNOWLEDGEMENT

It was really a good experience, working with ***Electronic Circuits*** lab. First we would like to thank **Dr. Srinivas Nallagonda, Professor, HOD** of Department of Electronics and Communication Engineering, Marri Laxman Reddy Institute of technology & Management for his concern and giving the technical support in preparing the document.

We are deeply indebted and gratefully acknowledge the constant support and valuable patronage of **Dr. Ravi Prasad, Dean Academics,** Marri Laxman Reddy Institute of technology & Management for giving us this wonderful opportunity for preparing the with ***Electronic Circuits*** laboratory manual.

We express our hearty thanks to **Dr. R. Murali Prasad, Principal,** Marri Laxman Reddy Institute of Technology and Management for timely corrections and scholarly guidance.

We wish to express deepest gratitude and thanks to **Dr. P. Sridhar, Director,** Marri Laxman Reddy Institute of Technology and Management, for his constant support and encouragement in providing all the facilities in the college.

At last, but not the least I would like to thanks the entire ECE Department faculty those who had inspired and helped us to achieve our goal.

By,

R. ANUJA
G. SIVA SANKAR VARMA

GENERAL INSTRUCTIONS

1. Students are instructed to come to ***Electronic Circuits*** laboratory on time. Late comers are not entertained in the lab.
2. Students should be punctual to the lab. If not, the conducted experiments will not be repeated.
3. Students are expected to come prepared at home with the experiments which are going to be performed.
4. Students are instructed to display their identity cards before entering into the lab.
5. Students are instructed not to bring mobile phones to the lab.
6. Any damage/loss of equipment's like transformers, transistors, CRO's etc., during the lab session, it is student's responsibility and penalty or fine will be collected from the student.
7. Students should update the records and lab observation books session wise. Before leaving the lab the student should get his lab observation book signed by the faculty.
8. Students should submit the lab records by the next lab to the concerned faculty members in the staffroom for their correction and return.
9. Students should not move around the lab during the lab session.
10. If any emergency arises, the student should take the permission from faculty member concerned in written format.
11. The faculty members may suspend any student from the lab session on disciplinary grounds.
12. Never copy the output from other students. Write down your own outputs.

INSTITUTION VISION AND MISSION

VISION

To establish as an ideal academic institution in the service of the nation, the world and the humanity by graduating talented engineers to be ethically strong, globally competent by conducting high quality research, developing breakthrough technologies, and disseminating and preserving technical knowledge.

MISSION

To foster a transformative learning environment that empowers students to excel in engineering, innovation, and leadership.

To produce skilled, ethical, and socially responsible engineers who contribute to sustainable technological advancements and address global challenges.

To shape future leaders through cutting-edge research, industry collaboration, and community engagement.

DEPARTMENT VISION, MISSION, PROGRAMME EDUCATIONAL OBJECTIVES AND SPECIFIC OUTCOMES

Vision and Mission

Our Vision

To provide quality technical education in Electronics and Communication Engineering through research, innovation, striving for global recognition in specified domain, leadership, and sustainable societal solutions.

Our Mission

To develop a strong centre of excellence for education and research with excellent infrastructure and well qualified faculties to instill in them a passion for knowledge.

To achieve the Mission the department will

1. Establish a unique learning environment to enable the students to face the challenges of the Electronics and Communication Engineering field.
2. Promote the establishment of centre of excellence in niche technology areas to nurture the spirit of innovation and creativity among faculty and students.
3. Provide ethical and value based education by promoting activities addressing the societal needs.
4. Enable students to develop skills to solve complex technological problems of current times and also provide a framework for promoting collaborative and multidisciplinary activities.

PEO's & PO's

PROGRAMME EDUCATIONAL OBJECTIVES

PEO 1: Have successful careers in Industry.

PEO 2: Show excellence in higher studies/ Research.

PEO 3: Show good competency towards Entrepreneurship

PROGRAM OUTCOMES

a.An ability to apply knowledge of Science, Mathematics, Engineering & Computing fundamentals for the solutions of Complex Engineering problems.

b. An ability to identify, formulates, research literature and analyze complex engineering problems using first principles of mathematics and engineering sciences.

c.An ability to design solutions to complex process or program to meet desired needs.

d.Ability to use research-based knowledge and research methods including design of experiments to provide valid conclusions.

e.An ability to use appropriate techniques, skills and tools necessary for computing practice.

f.Ability to apply reasoning informed by the contextual knowledge to assess social issues, consequences & responsibilities relevant to the professional engineering practice.

g.Ability to understand the impact of engineering solutions in a global, economic, environmental, and societal context with sustainability.

h.An understanding of professional, ethical, Social issues and responsibilities.

i.An ability to function as an individual, and as a member or leader in diverse teams and in multidisciplinary settings.

j.An ability to communicate effectively on complex engineering activities within the engineering community.

k.Ability to demonstrate and understanding of the engineering and management principles as a member and leader in a team.

l.Ability to engage in independent and lifelong learning in the context of technological change.

PROGRAM SPECIFIC OUTCOMES

PSO1 Analyze and design analog & digital circuits or systems for a given specification and function.

PSO2 Implement functional blocks of hardware-software co-designs for signal processing and communication applications.

PSO3 Gain the hands-on competency skills in Computing Tools for electronics and communication systems for the entry level position to meet the requirements of the Employer.

L	T	P	C
0	0	2	1

2440473: ELECTRONIC CIRCUITS ANALYSIS LABORATORY

Course Objectives:
The students will try to learn

- Single stage and multi stage amplifiers
- The feedback amplifiers and oscillators through simulation.
- Frequency response of Power Amplifiers
- Implementation of circuits for linear and nonlinear wave shaping
- The characteristics of different multivibrators

Course Outcomes:
At the end of the laboratory work, students will be able to

- Compare the frequency of oscillations – Hartley oscillator and Colpitts oscillator RC phase shift oscillator and Wein Bridge Oscillator
- Analyze the bandwidth of power amplifiers
- Determine all multivibrator circuits
- Design Schmitt trigger
- Demonstrate about the output waveforms of Miller Sweep Circuit and Bootstrap Time Base generator.

LIST OF EXPERIMENTS:
Experiments marked with * has to be designed, simulated and verify in hardware laboratory.

1. Two Stage RC Coupled Amplifier(*).
2. Cascade amplifier Circuit/ Darlington Pair Circuit (*).
3. Current Shunt Feedback amplifier Circuit(*).
4. Voltage Series Feedback amplifier Circuit (*).
5. RC Phase shift Oscillator Circuit (*).
6. Hartley and Colpitt's Oscillators Circuit(*).
7. Class A power amplifier(Transformer less) (*).
8. Class B Complementary symmetry amplifier (*).
9. Single Tuned Amplifier Circuit(*).
10. Monostable Multivibrator(*).
11. Bistable Multivibrator(*).

12. Astable Multivibrator(*).
13. Schmitt Trigger using transistor(*).
14. Verify the output characteristics of Miller Sweep Circuit.
15. Verify the output characteristics of Bootstrap Time Base Generator.

NOTE: Minimum of 12 experiments to be conducted

COURSE STRUCTURE, OBJECTIVES & OUTCOMES

COURSE STRUCTURE

Level	Credits	Periods/week	Prerequisites
UG	1	3	Entire subject of Electronic Devices and Circuits.

Evaluation Scheme:

MID (Internal Lab) Semester Test	30 marks
Day to day evaluation	10 marks
End Semester Lab external Examination	60 marks

The end semester examination shall be conducted with an external examiner and internal examiner.

The external examiner shall be appointed by the principal / Chief Controller of examinations.

DEPARTMENT OF ELECTRONICS AND COMMUNICATION

ELECTRONIC CIRCUITS ANALYSIS LABORATORYLABORATORYLABORATORY

Virtual lab details

Name of the Virtual Lab: NA
Virtual Lab Host Institute: NA
URL/Link to Lab: NA
Academic Year: NA
Semester: NA

MARRI LAXMAN REDDY
INSTITUTE OF TECHNOLOGY AND MANAGEMENT
(AN AUTONOMOUS INSTITUTION)
(Approved by AICTE, New Delhi & Affiliated to JNTUH, Hyderabad)
Accredited by NBA and NAAC with 'A' Grade & Recognized Under Section2(f) & 12(B)of the UGC act,1956

DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING
ELECTRONIC CIRCUITS LAB

LAB PLANNER

S.No	Experiment	CO	Virtual Lab Availabilty	Date planned	Date conducted
1	Two Stage RC Coupled Amplifier	CO1	NA		
2	Cascade Amplifier circuit / Darlington Pair circuit	CO1	NA		
3	Current Shunt Feedback Amplifier	CO2	NA		
4	Voltage Series Feedback Amplifier	CO2	NA		
5	RC Phase Shift Oscillator using Transistors	CO2	NA		
6	Hartley and Colpitts's Oscillator circuit	CO3	NA		
7	LAB INTERNAL-1				
8	Class A Power Amplifier (Transformer less)	CO3	NA		
9	Class B Complementary Symmetry Amplifier	CO3	NA		
10	Single Tuned Amplifier circuit	CO4	NA		
11	Monostable Multivibrator	CO4	NA		
12	Bistable Multivibrator	CO5	NA		
13	Astable Multivibrator	CO5	NA		
14	LAB INTERNAL-1				

DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING
ELECTRONIC CIRCUITS LAB

LAB PLANNER

Date planed																																
Date conducted																																
Roll Number	Exp No	CO	VL*	Exp No	CO	VL*	Exp No	CO	VL*	Exp No	CO	VL*	Exp No	CO	VL*		Exp No	CO	VL*	Exp No	CO	VL*	Exp No	CO	VL*	Exp No	CO	VL*	Exp No	CO	VL*	
247Y1A0401	1	1	Y	2	1	N	3	2	Y	4	2	N	5	3	Y		6	4	Y	7	4	Y	8	4	Y	9	5	N	10	5	Y	
247Y1A0402	2	1	N	3	2	Y	4	2	N	5	3	Y	6	4	Y	M	7	4	Y	8	4	Y	9	5	N	10	5	Y	1	1	Y	**M**
247Y1A0403	3	2	Y	4	2	N	5	3	Y	6	4	Y	7	4	Y	I	8	4	Y	9	5	N	10	5	Y	1	1	Y	2	1	N	**I**
247Y1A0404	4	2	N	5	3	Y	6	4	Y	7	4	Y	8	4	Y	D	9	5	N	10	5	Y	1	1	Y	2	1	N	3	2	Y	D
247Y1A0405	5	3	Y	6	4	Y	7	4	Y	8	4	Y	9	5	N	-	10	5	Y	1	1	Y	2	1	N	3	2	Y	4	2	N	**-**
247Y1A0406	6	4	Y	7	4	Y	8	4	Y	9	5	N	10	5	Y	I	1	1	Y	2	1	N	3	2	Y	4	2	N	5	3	Y	**II**

Note:VL*-Virtual Lab Availabilty

MARRI LAXMAN REDDY
INSTITUTE OF TECHNOLOGY AND MANAGEMENT
(AN AUTONOMOUS INSTITUTION)
(Approved by AICTE, New Delhi & Affiliated to JNTUH, Hyderabad)
Accredited by NBA and NAAC with 'A' Grade & Recognized Under Section2(f) & 12(B)of the UGC act,1956

DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

ELECTRONIC CIRCUITS LAB

RUBRICS USED TO ASSESS LEARNINGS IN LABORATORIES

1. RUBRICS FOR DAY TO DAY EVALUATION

Parameter	Max Marks	Level-1 (Very Poor)	Level-2 (Poor)	Level-3 (Average)	Level-4 (Good)	Level-5 (Excellent)
Observation Book	05	No observations or irrelevant data. (0–1)	Incomplete or incorrect data. (2)	Basic values with some errors. (3)	Mostly correct with good format. (4)	Fully correct, clear, and well-formatted. (5)
Record Writing	05	Not submitted. (0-1)	Submitted but mostly incomplete. (2)	Submitted with some missing/wrong parts. (3)	Submitted with minor issues. (4)	Fully complete, correct algorithm & flowchart. (5)
Result	05	No result or major errors. (0–1)	Result partially obtained. (2)	Acceptable result with limited error. (3)	Near-correct result and reasonable error. (4)	Accurate result. (5)
Viva-Voce	05	Did not answer any questions. (1)	Answered very few questions. (2)	Answered some questions with help. (3)	Answered most questions correctly. (4)	Answered all questions accurately. (5)

MARRI LAXMAN REDDY
INSTITUTE OF TECHNOLOGY AND MANAGEMENT
(AN AUTONOMOUS INSTITUTION)
(Approved by AICTE, New Delhi & Affiliated to JNTUH, Hyderabad)
Accredited by NBA and NAAC with 'A' Grade & Recognized Under Section2(f) & 12(B)of the UGC act,1956

DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

ELECTRONIC CIRCUITS LAB

2.RUBRICS FOR INTERNAL EVALUATION

Criterion	Max Marks	Level-1 *(Very Poor)*	Level-2 *(Poor)*	Level-3 *(Average)*	Level-4 *(Good)*	Level-5 *(Excellent)*
Design/Tool/Apparatus Selection	2 Marks	Incorrect tool/design and no reasoning. **(0)**	Tool/design selection attempted with unclear logic. **(0.5)**	Satisfactory selection with partial justification. **(1)**	Correct selection and proper analysis with few errors. **(1.5)**	Smart selection with accurate, relevant analysis. **(2)**
Execution (Code/Debug/Run) /Analysis/Method Used	4 Marks	Did not attempt or completely failed to execute. **(0)**	Attempted but unable to proceed or with major errors. **(1)**	Partial execution with some logic/syntax errors. **(2)**	Mostly correct execution with minimal help. **(3)**	Fully correct and independently executed program. **(4)**
Results& Documentation	2 Marks	Incomplete or poorly presented. **(0)**	Basic structure but lacks clarity or formatting. **(0.5)**	Complete but generic or with formatting issues. **(1)**	Well-structured and mostly clear. **(1.5)**	Well-organized, professional, and engaging documentation. **(2)**
Viva-Voce (Understanding of Concepts)	2 Marks	No understanding; could not answer questions. **(0)**	Answered a few with difficulty. **(0.5)**	Answered half the questions with basic clarity. **(1)**	Good understanding with confident answers. **(1.5)**	Answered all questions with clarity and depth. **(2)**

MARRI LAXMAN REDDY
INSTITUTE OF TECHNOLOGY AND MANAGEMENT
(AN AUTONOMOUS INSTITUTION)
(Approved by AICTE, New Delhi & Affiliated to JNTUH, Hyderabad)
Accredited by NBA and NAAC with 'A' Grade & Recognized Under Section2(f) & 12(B)of the UGC act,1956

DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

ELECTRONIC CIRCUITS LAB

3. RUBRICS FOR SEMESTER END EXAMINATIONS

Criterion	Max Marks	Level-1 (Very Poor *(0–2 marks)*	Level-2 (Poor) *(3–4 marks)*	Level-3 (Average) *(5–6 marks)*	Level-4 (Good) *(7–9 marks)*	Level-5 (Excellent) *(10–12 marks)*
Preparedness for the Experiment	12 marks	No clarity on objective or procedure. Unable to explain basics.	Limited idea of the objective/procedure. Needed prompting.	Has basic understanding; minor gaps in concept or preparation.	Well-prepared, with clear understanding of steps and background.	Fully prepared with strong conceptual clarity and confident explanation.
Performance in the Laboratory	12 marks	Unable to perform experiment. Relied entirely on examiner's help.	Performed with multiple errors and constant support.	Performed with some errors; required occasional help.	Performed mostly independently with minimal support.	Performed independently, efficiently, and with precision.
Calculations & Graphs	12 marks	No or incorrect calculations. Graphs missing or irrelevant.	Multiple calculation errors. Graphs/plots inaccurate or poorly labeled.	Calculations partially correct. Graphs present but with some flaws.	Correct calculations and graphs with minor errors.	Accurate calculations and well-labeled graphs with proper interpretation.
Results & Error Analysis	12 marks	No result or invalid result. No error analysis attempted.	Incorrect result with vague or no error discussion.	Acceptable result. Error analysis attempted but limited.	Correct result with sound error discussion.	Accurate result with detailed and relevant error analysis.
Viva-Voce (Subject Knowledge)	12 marks	Unable to answer any questions. No conceptual understanding.	Answered few questions with poor logic.	Answered half of the questions with average understanding.	Answered most questions with clarity and confidence.	Answered all questions with depth, clarity, and reasoning.

INDEX

EXP NO: 1

TWO STAGE RC COUPLED AMPLIFIER (Software)

PRELAB:

1. Study the purpose of using multistage amplifiers.
2. Learn the different types of coupling methods.
3. Study the effect of cascading on Bandwidth.
4. Identify all the formulae you will need in this Lab.
5. Study the procedure of using Multisim tool (Schematic & Circuit File).

OBJECTIVE:

1. To simulate the Two Stage RC Coupled Amplifier in Multisim and study the transient and frequency response.
2. To determine the phase relationship between the input and output voltages by performing the transient analysis.
3. To determine the maximum gain, 3dB gain, lower and upper cutoff frequencies and bandwidth of Two Stage RC Coupled Amplifier by performing the AC analysis.
4. To determine the effect of cascading on gain and bandwidth.

SOFTWARE TOOL: Multisim 13.0

APPARATUS:

1.Regulated power supply	-	1 No.
2. Function generator	-	1 No.
3. CRO	-	1 No.
4. Transistor (BC 107 or 2N2222)	-	2 No.
5. Resistors (5KΩ,47 KΩ,2 KΩ, ,1 KΩ)	-	2 No. each
6. Resistor (10 KΩ)	-	4 Nos.
7. Capacitors (10 μF, 1 μF)	-	2,3No. each
8. Bread Board	-	1 No.
9. Connecting wires		

CIRCUIT DIAGRAM:

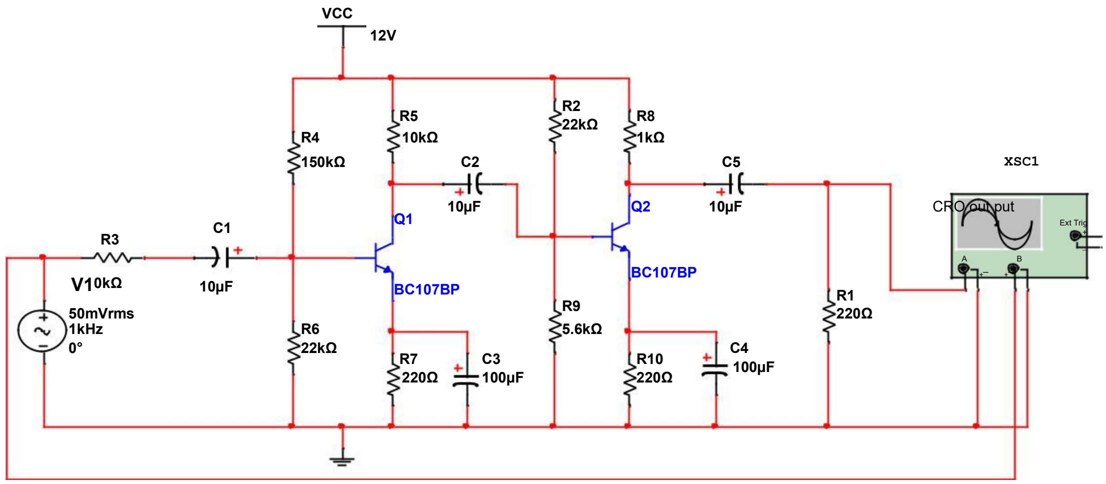


Fig: 1.a Two Stage RC Coupled Amplifier Circuit Diagram

THEORY:

An amplifier is the basic building block of most electronic systems. Just as one brick does not make a house, a single-stage amplifier is not sufficient to build a practical electronic system. The gain of the single stage is not sufficient for practical applications. The voltage level of a signal can be raised to the desired level if we use more than one stage. When a number of amplifier stages are used in succession (one after the other) it is called a multistage amplifier or a cascade amplifier. Much higher gains can be obtained from the multi-stage amplifiers. In a multi-stage amplifier, the output of one stage makes the input of the next stage. We must use a suitable coupling network between two stages so that a minimum loss of voltage occurs when the signal passes through this network to the next stage. Also, the dc voltage at the output of one stage should not be permitted to go to the input of the next. If it does, the biasing conditions of the next stage are disturbed.

Figure shows how to couple two stages of amplifiers using RC coupling scheme. This is the most widely used method. In this scheme, the signal developed across the collector resistor R_C (R2)of the first stage is coupled to the base of the second stage through the capacitor C_C.(C2) The coupling capacitor blocks the dc voltage of the first stage from reaching the base of the second stage. In this way, the dc biasing of the next stage is not interfered with. For this reason, the capacitor C_C (C2)is also called a blocking capacitor.

As the number of stages increases, the gain increases and the bandwidth decreases.

RC coupling scheme finds applications in almost all audio small-signal amplifiers used in record players, tape recorders, public-address systems, radio receivers, television receivers, etc.

PROCEDURE:

1. Open Multisim Software to design Two stage RC coupled amplifier circuit
2. Select on New editor window and place the required component CS amplifier on the circuit window.
3. Make the connections using wire and check the connections and oscillator.
4. Go for simulation and using Run Key observe the output waveforms on CRO
5. Indicate the node names and go for AC Analysis with the output node
6. Observe the Transient response and Ac Analysis for the first stage and second stage separately and draw the magnitude response curve
7. Calculate the bandwidth of the amplifier

OBSERVATIONS/GRAPHS:

TRANSIENT RESPONSE:

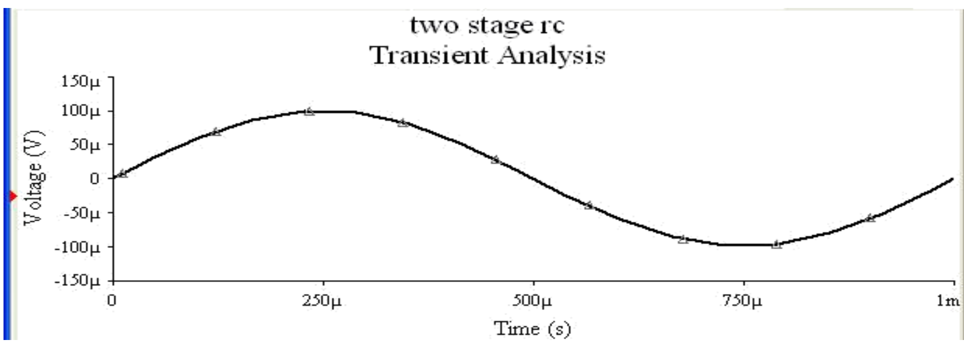


Fig 1.b: Transient response of First stage in Two stage RC coupled amplifier

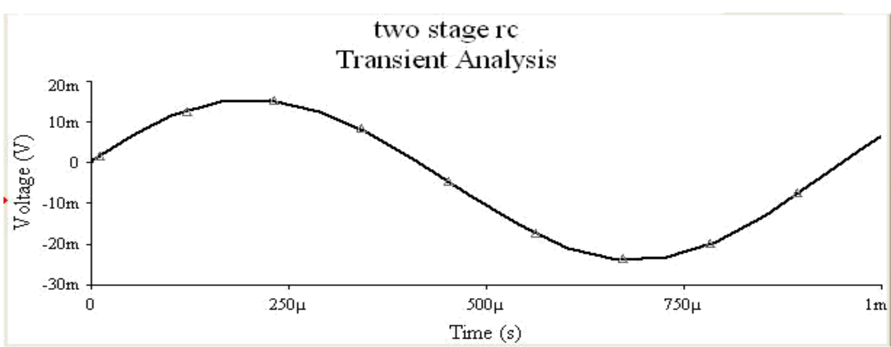


Fig 1.c: Transient response of second stage in Two stage RC coupled amplifier

FREQUENCY RESPONSE:

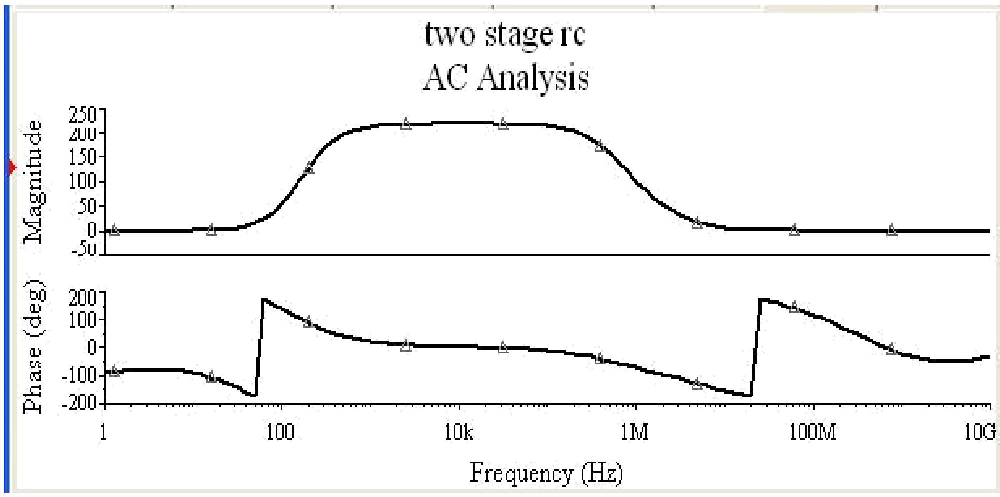


Fig 1.d: Frequency response of Two stage RC coupled amplifier

RESULTS :

1. From the transient analysis, it is observed that,____________________________
2. From the frequency response curve the following results are calculated.
3. From the AC response, it is observed that, ______________________________

S. No.	**Parameter**	**Value**
1	Max. Gain in dB	
2	3dB Gain	
3	Lower Cutoff Frequency	
4	Upper Cutoff Frequency	
5	Bandwidth of I stage	
6	Bandwidth of 2 stage	

EXPT NO :1

TWO STAGE RC COUPLED AMPLIFIER (Hardware)

AIM: -

1. Plot the frequency response of a Two Stage Amplifier.
2. Calculate gain.
3. Calculate bandwidth.

COMPONENTS & EQUIPMENTS REQUIRED: -

S.No	Device	Range/Rating	QTY
1.	(a) DC supply voltage	12V	1
	(b) Transistor	BC107	1
	(c) Capacitors	10 F	2
		100 F	1
	(d) Resistors	100 ,470	1
		4.7K ,8.2k	1
2.	Signal generator	0.1Hz-1MHz	1
3.	CRO	0Hz-20MHz	1
4.	Connecting wires	5A	4

CIRCUIT DIAGRAM:

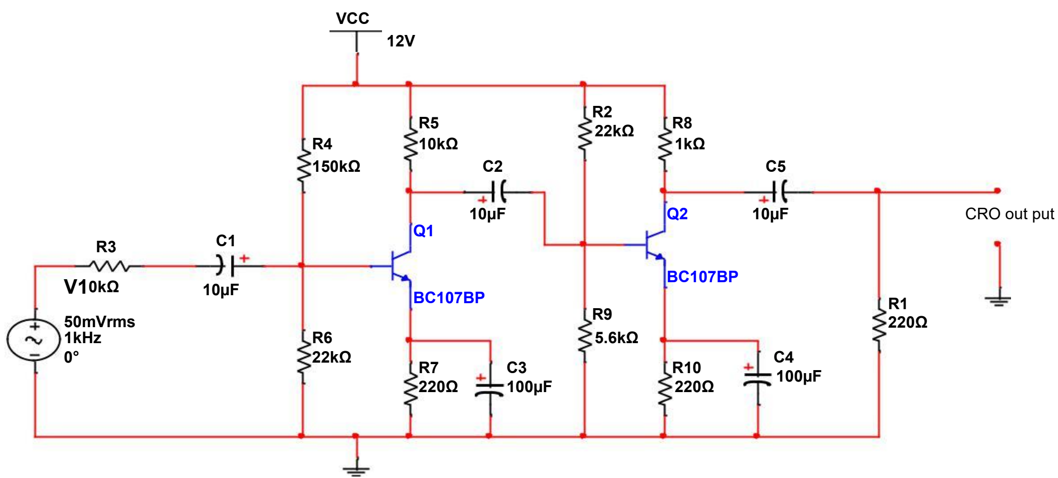


Fig: 1.e: circuit diagram of Two Stages RC coupled amplifier

PROCEDURE :

1. Connect the circuit diagram as shown in figure.
2. Adjust input signal amplitude in the function generator and observe an amplified voltage at the output without distortion.
3. By keeping input signal voltage, say at 50mV, vary the input signal frequency from 0 to 1MHz in steps as shown in tabular column and note the corresponding output voltages.

PRECAUTIONS:

Avoid loose connections and give proper input Voltage

TABULAR COLUMN:

Input = 50mV

Frequency(in Hz)	Output Voltage (V_o)		Gain $A_v=V_o/V_i$		Gain(in dB) $=20\log_{10}(V_o/V_i)$	
	With feedback	Without feedback	With feedback	Without feedback	With feedback	Without feedback
20						
40						
80						
100						
1K						
10k						
50k,100K						
1M						

RESULT: -

1. Frequency response of Two stage RC coupled amplifier graph is plotted.
2. Gain = _______dB (maximum).
3. Bandwidth= $f_H - f_L$ = _________Hz. At stage I
4. Bandwidth= $f_H - f_L$ = _________Hz. At stage 2

EXPECTED GRAPH:

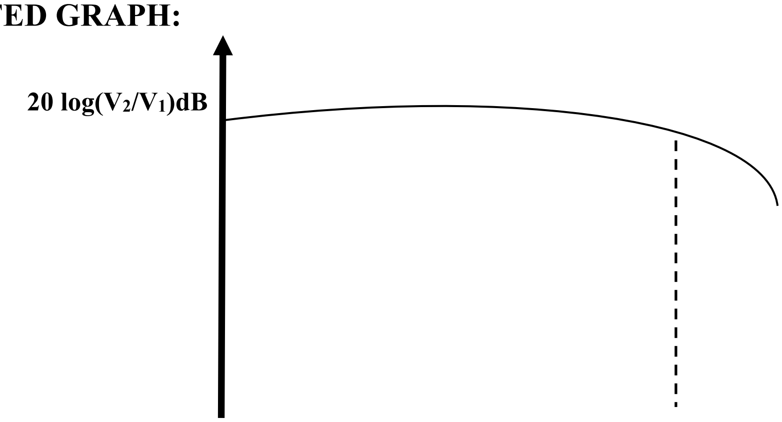


Fig: 1.f: Expected waveform of Two Stages RC coupled amplifier

VIVA QUESTIONS:

1. Why do you need more than one stage of amplifiers in practical circuits?
2. What is the effect of cascading on gain and bandwidth?
3. What happens to the 3dB frequencies if the number of stages of amplifiers increases?
4. Why we use a logarithmic scale to denote voltage or power gains, instead of using the simpler linear scale?
5. What is loading effect in multistage amplifiers?
6. What is the necessity of cascading?
7. What is 3dB bandwidth?
8. Why RC coupling is preferred in audio range?
9. Which type of coupling is preferred and why?
10. Explain various types of Capacitors?
11. What is loading effect?
12. Why it is known as RC coupling?
13. What is the purpose of emitter bypass capacitor?
14. Which type of biasing is used in RC coupled amplifier?
15. What is difference between Amplifier and Attenuator?
16. Which Amplifier will amplify voltage and current?
17. What are the advantages over single stage amplifier?
18. What are the classifications of multistage amplifiers?
19. What are the different BJT multistage amplifier configurations?
20. Define cut off frequency?
21. What Is the pin configuration on bread board used in the lab?
22. What is two stages RC coupled amplifier?
23. Which type of coupling is preferred and why?
24. Explain various types of Capacitors?
25. What is two stages RC coupled amplifier?
26. Plot the frequency response of BC 107 amplifier with C_1 = 5 µF.
27. Plot the frequency response of BC 547 amplifier with C_2 = 5 µF with i/p.
28. Plot the frequency response of BC 2N2222 amplifier with R_1 = 4.1 K.
29. Plot the frequency response of BC 107 amplifier with R_2 = 9.4 K i/p.
30. Plot frequency response of BC 107 amplifier R_1 = 4.1 K, R_2 = 9.4 K with i/p.

EXERCISE PROBLEMS:

1. Plot the frequency and amplitude response of BC 107 amplifier with C_1 = 5 μF.
2. Plot the frequency response of amplifier with C_2 = 5 μF, i/p.
3. Plot the frequency response of BC 107 amplifier with R_1 = 4.1 K.
4. Plot the frequency response of BC107 amplifier with R_2 = 9.4 K i/p.
5. Plot frequency response of BC107 R1 = 4.1 K, R_2 = 9.4 K with i/p.
6. Plot the frequency and amplitude response of BC107 amplifier with C_1 = 10 μF.
7. Plot the frequency response of BC 107amplifier with C_2 = 2 μF with i/p.
8. Plot the frequency response of BC 107 amplifier with R_1 = 2.1 K.
9. Plot the frequency response of BC 10 7amplifier with R_2 = 5.4 K i/p.
10. Plot frequency response of BC 107amplifier R_1 = 2.1 K, R_2 = 2.4 K with i/p.
11. Plot the frequency and amplitude response of BC 107amplifier with C_1 = 2 μF.
12. Plot the frequency response of amplifier with C_2 = 2 μF, i/p.
13. Plot the frequency response of BC 107 amplifier with R_1 = 2.1 K.
14. Plot the frequency response of BC 107 amplifier with R_2 = 2.4 K i/p.
15. Plot frequency response of R1 = 2.1 K, R_2 = 9.4 K with i/p

APPLICATIONS:

1. They are widely used as voltage amplifiers (ie. In the initial stages of public address systems) because of their excellent audio-fidelity over a wide range of frequency. However because of poor impedance matching this type of coupling transistor circuits is rarely employed in final stages impedance matching voltage amplifier in initial stage of public addressing system.

2. To increase the power gain, high input impedance, low output impedance, and increase the weaken signal.

3. To increase the power gain, high input impedance, low output impedance, and increase the weaken signal.

4. In a two stage RC coupled amplifier, the two transistors are identical and a common power supply is used. The input is provided to the first stage of the amplifier where it is amplified and this output is used as input for the second stage.

5. This is amplified once again by the other transistor in the second stage and the final output is obtained.

EXP NO 2: CASCADE AMPLFIER CIRCUIT /DARLINGTON PAIR CIRCUIT

EXPT NO: 2.A

CASCADE AMPLIFIER (Software)

AIM: To study the frequency response of amplifier, calculate voltage gain and bandwidth from the response.

APPARATUS:

1.Regulated power supply	-	1 No.
2.Function generator	-	1 No.
3.CRO	-	1 No.
4.Transistor (BC 547BG or 2N2222)	-	2 No.
5.Resistors (22KΩ,1.2KΩ , 4.7KΩ,300Ω)	-	2 No. each
6.Capacitors (0.1 µF, 4.7 µF)	-	2,1No. each
7.Bread Board	-	1 No.
8.Connecting wires		

SOFTWARE USED: Multisim V13.0

CIRCUIT DIAGRAM:

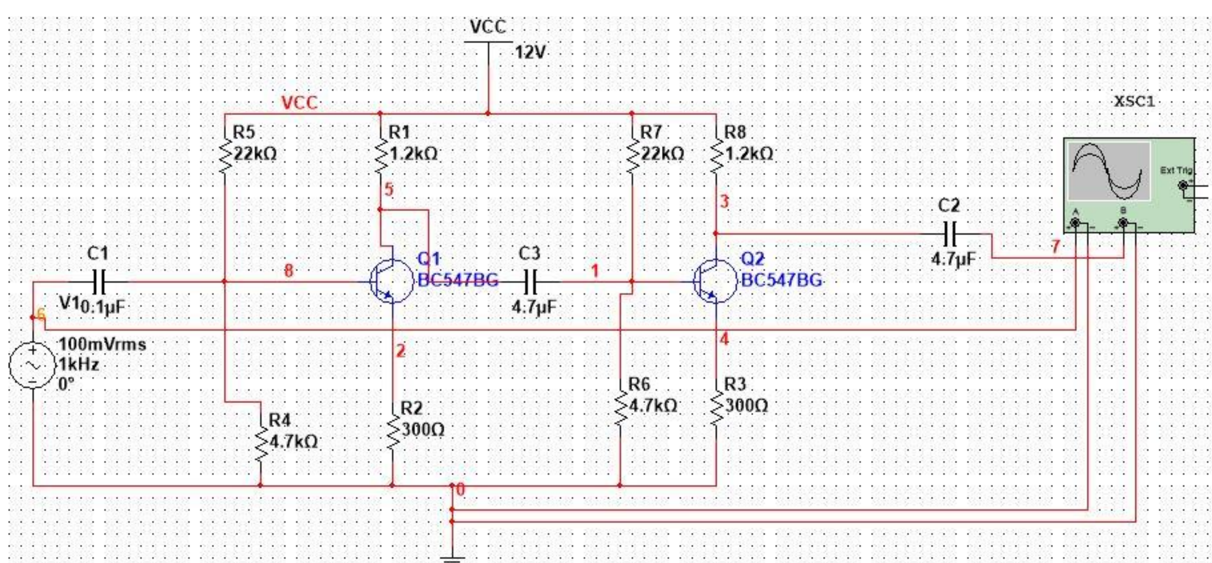


Fig: 2A.a Cascade Amplifier circuit diagram

THEORY:

A cascade amplifier is a multi-stage amplifier configuration where the output of one amplifier stage is connected to the input of the next stage. This arrangement is used to achieve a higher overall gain than what a single stage could provide. Each stage typically consists of a common-emitter or common-source configuration in the case of BJTs or MOSFETs, respectively. The main advantage of cascading amplifier stages is the multiplication of individual stage gains, allowing for significant amplification of weak signals. Additionally, cascade amplifiers improve input and output impedance matching, which enhances signal transfer efficiency between stages. However, careful design is required to minimize issues like phase shift, distortion, and bandwidth limitations that can arise due to the interaction between stage

PROCEDURE:

1. Switch ON the computer and open the multisim software.
2. Check whether the icons of the instruments are activated and enable.
3. Now connect the circuit using the designed values of each and every component.
4. Connect the function generator with sine wave of 50mVp-p as input at the input of terminals of the circuit.
5. Connect the Cathode Ray Oscilloscope (CRO) to the out put terminals of the circuit.
6. Go to simulation button click it for simulation process.
7. From the CRO note the following values
    1. Input voltage V_i =
    2. Output voltage V_0 =
    3. Voltage gain $A_V = V_0/V_i$ =
    4. Phase shift θ =
8. To study the frequency response click the AC analysis, so that a screen displays the following options
    1. Start frequency
9. Assign the proper values for start frequency, stop frequency and vertical scale according to the circuit requirements and observe the frequency response.
10. From the frequency response calculate the
    maximum gain A_{Vmax} =

    lower cutoff frequency (f1) at A_{Vmax}-3dB (decibel scale) value at $A_{Vmax}/\sqrt{2}$ (linear scale) =

Higher cutoff frequency (f2) at A_{Vmax}-3dB (decibel scale) valu
at $A_{Vmax}/\sqrt{2}$ (linear scale) =

OBSERVATIONS:

From CRO:

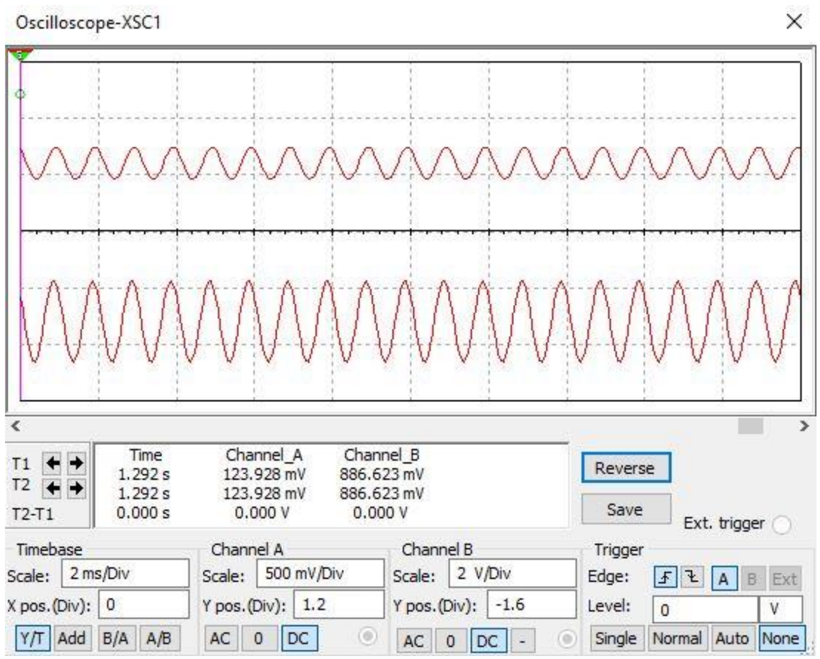


Fig: 2A.b Output waveform of cascade amplifier

1. Input voltage V_i =
2. Output voltage V_0 =
3. Voltage gain $A_V = V_0/V_i$ =
4. Phase shift θ =

From Frequency response:

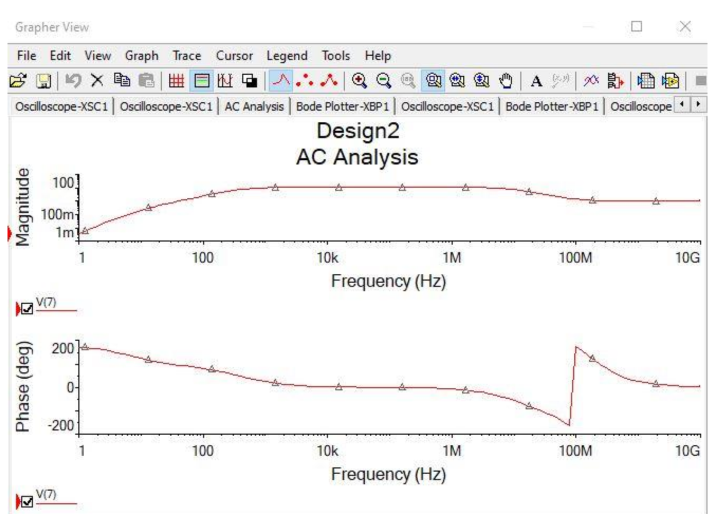


Fig: 2A.c Frequency response of cascade amplifier

1. Maximum gain A_{Vmax} =
2. Lower cutoff frequency(f1) at A_{Vmax}-3dB (decibel scale) value

   at $A_{Vmax}/\sqrt{2}$ (linear scale) =

3. Higher cutoff frequency(f2) at A_{Vmax}-3dB (decibel scale) value

   at $A_{Vmax}/\sqrt{2}$ (linear scale) =

CALCULATIONS:

Band width (BW) = $f_2 - f_1$

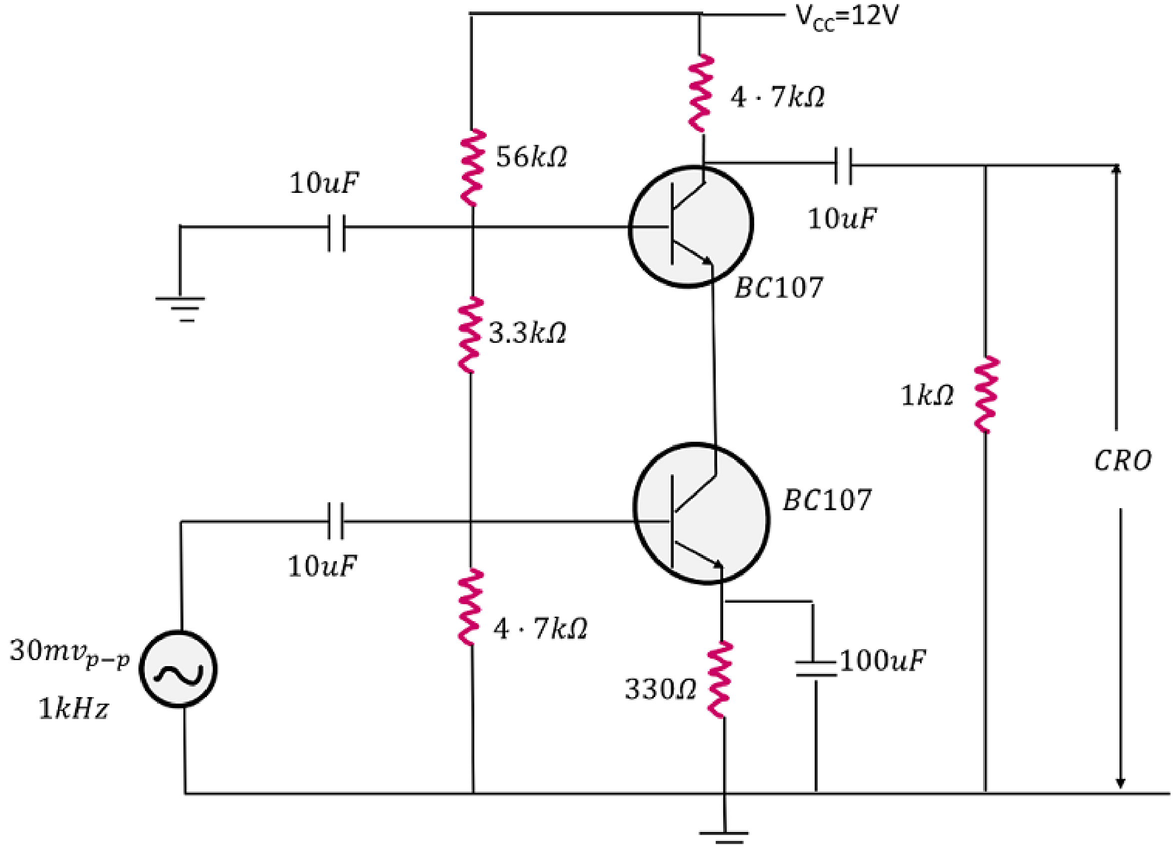
RESULT: Cascade amplifier is design with the given specifications and from observed frequency response, gain and band width are calculated.

APPLICATION:

1. It is used in RF tunner.
2. Used in Amplitude Modulation.

EXP NO: 2.A

CASCADE AMPLIFIER (HARDWARE)



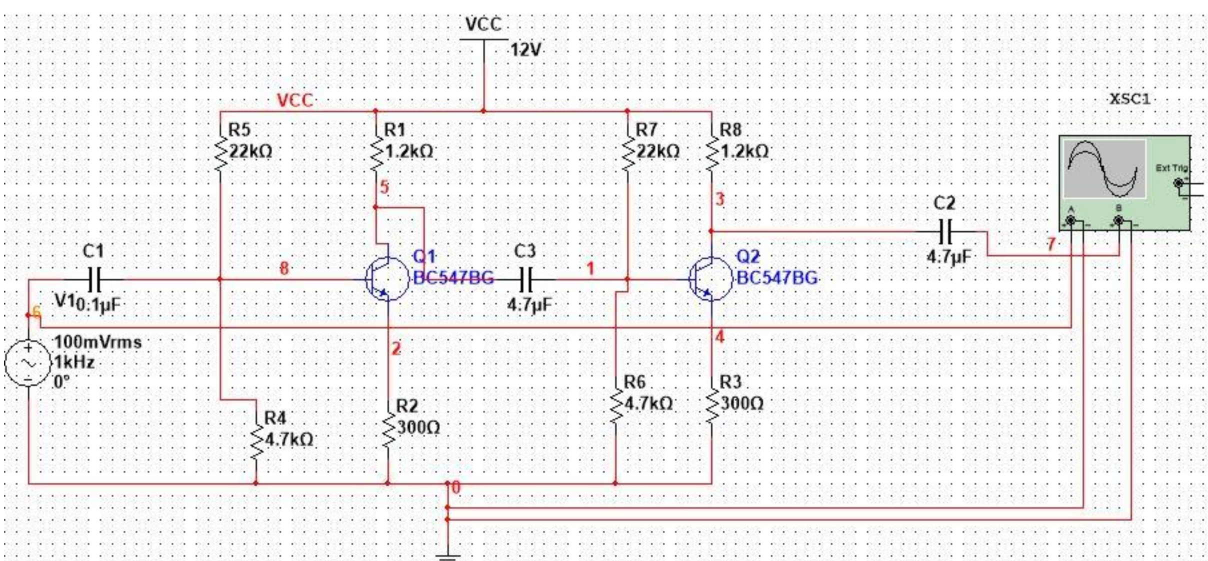


Fig: 2A.d Cascade Amplifier circuit diagram

PROCEDURE: -

1. Connect the circuit diagram as shown in figure.
2. Adjust input signal amplitude in the function generator and observe an amplified voltage at the output without distortion.
3. By keeping input signal voltage, say at 50mV, vary the input signal frequency from 0 to 1MHz in steps as shown in tabular column and note the corresponding output voltages.

PRECAUTIONS:

Avoid loose connections and give proper input Voltage

TABULAR COLUMN :

Input V_i =50mV

Frequency(Hz)	Output voltage V_o	Gain $=V_o/V_i$	Gain in dB=$20\log(V_0/V_i)$
50Hz			
100Hz			
1KHz			
10KHz			
50KHz			
100KHz			
500KHz			
1MHz			

EXPECTED GRAPH:

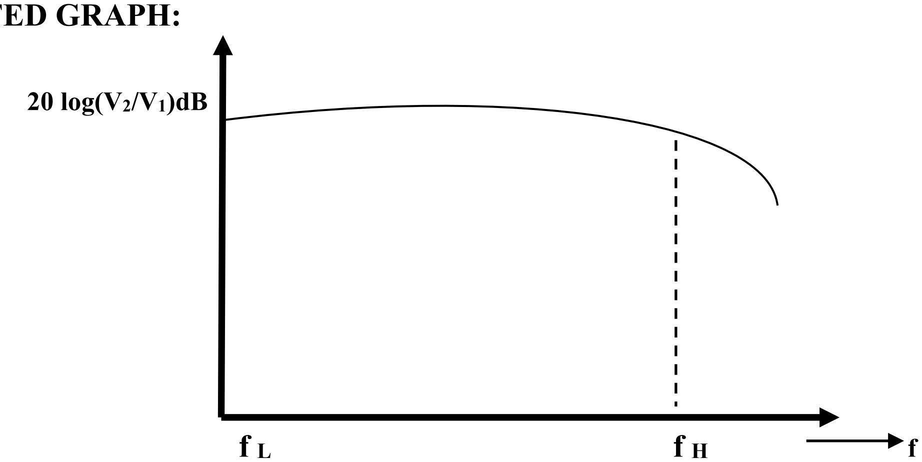


Fig: 2.e: Expected waveform of Cascade Amplifier

RESULT:

1. Frequency response of Cascade amplifier graph is plotted.
2. Gain = _______dB (maximum).
3. Bandwidth= fH--fL = ______

EXP NO:2B

DARLINGTON PAIR AMPLIFIER (SOTWARE)

.

AIM: To study the frequency response of amplifier, calculate voltage gain and bandwidth from the response.

APPARATUS:

1.Regulated power supply	-	1 No.
2.Function generator	-	1 No.
3.CRO	-	2 No.
4.Transistor (BC 107 or 2N2222)	-	2 No.
5.Resistors (57.3KΩ,3.15KΩ , 300Ω,1KΩ)	-	1 No. each
6.Resistor (4.7KΩ)	-	2 Nos.
7.Capacitors (10 µF, 100 µF)	-	3,1No. each
8.Bread Board	-	1 No.
9.Connecting wires		

SOFTWARE USED: Multisim V13.0

CIRCUIT DIAGRAM:

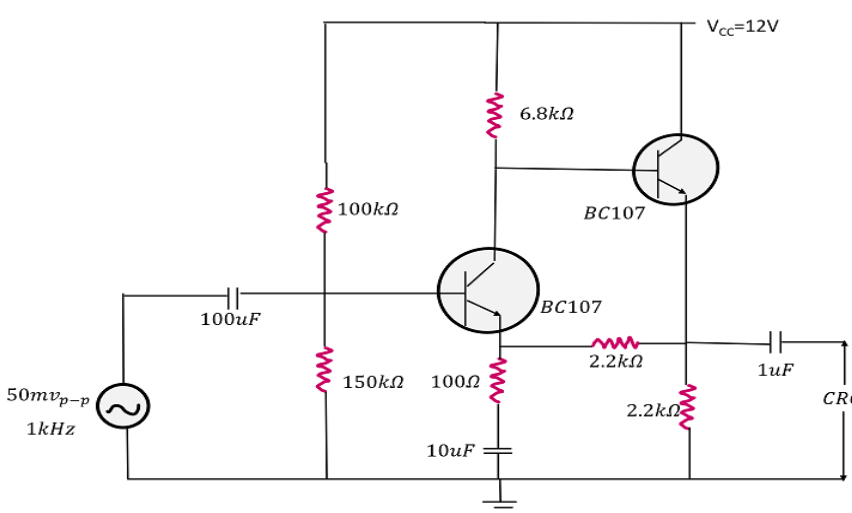


Fig: 2B.a circuit diagram of Darlington pair amplifier

THEORY:

The Darlington pair is a configuration of two bipolar junction transistors (BJTs) connected in such a way that the current amplified by the first transistor is further amplified by the second transistor. In this setup, the emitters of both transistors are connected together, with the collector of the first transistor connected to the base of the second. The result is a composite transistor with a much higher current gain (β) than a single transistor, often in the range of several thousand. This makes the Darlington pair highly effective in applications requiring large current amplification, such as in power amplifiers and switching circuits. However, this configuration also has a higher base-emitter voltage drop (approximately 1.2V instead of the typical 0.6V for a single BJT) and slower response times, which can limit its use in high-speed applications. Additionally, the increased gain can make the circuit more susceptible to noise and instability if not properly managed.

PROCEDURE:

1. Switch ON the computer and open the multisim software.
2. Check whether the icons of the instruments are activated and enable.
3. Now connect the circuit using the designed values of each and every component.
4. Connect the function generator with sine wave of 50mVp-p as input at the input of terminals of the circuit.
5. Connect the Cathode Ray Oscilloscope (CRO) to the out put terminals of the circuit.
6. Go to simulation button click it for simulation process.
7. From the CRO note the following values
   a. Input voltage V_i =
   b. Output voltage V_0 =
   c. Voltage gain $A_V = V_0/V_i$ =
   d. Phase shift θ =
8. To study the frequency response click the AC analysis, so that a screen displays the following options
   a. Start frequency
   b. Stop frequency
   c. Vertical scale
9. Assign the proper values for start frequency, stop frequency and vertical scale according to the circuit requirements and observe the frequency response.

10. From the frequency response calculate the
    maximum gain A_{Vmax} =
    a. lower cutoff frequency (f1) at A_{Vmax}-3dB (decibel scale) value
        a. at $A_{Vmax}/\sqrt{2}$ (linear scale) =
    b. Higher cutoff frequency (f2) at A_{Vmax}-3dB (decibel scale) value
        a. at $A_{Vmax}/\sqrt{2}$ (linear scale) =

OBSERVATIONS:

From CRO :

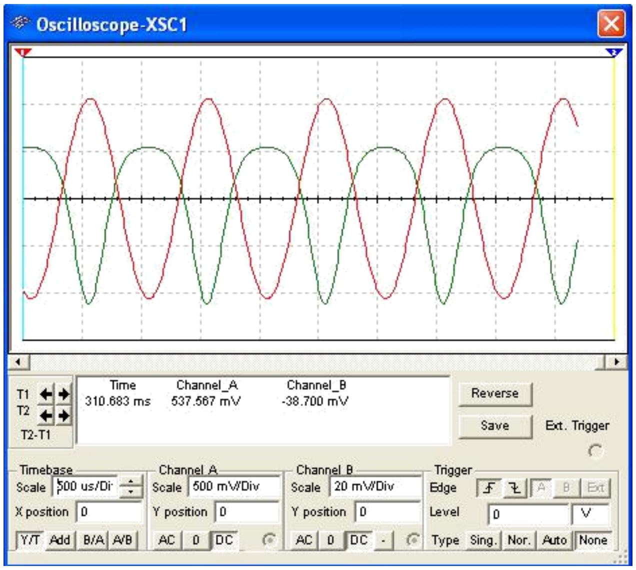


Fig 2B.b output waveform of Darlington Pair circuit

1. Input voltage V_i =
2. Output voltage V_0 =
3. Voltage gain $A_V = V_0/V_i$ =
4. Phase shift θ =

From Frequency response:

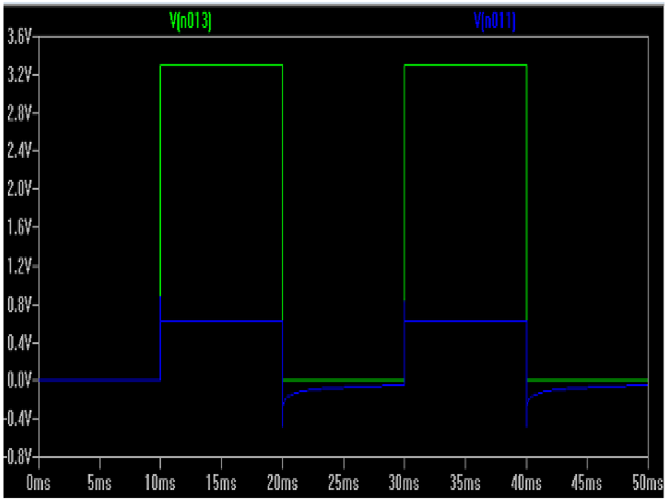


Fig 2B.c Frequency response of Darlington Pair circuit

1. Maximum gain A_{Vmax} =
2. Lower cutoff frequency(f1) at A_{Vmax}-3dB (decibel scale) value

   at $A_{Vmax}/\sqrt{2}$ (linear scale) =

3. Higher cutoff frequency(f2) at A_{Vmax}-3dB (decibel scale) value

   at $A_{Vmax}/\sqrt{2}$ (linear scale) =

CALCULATIONS:

Band width (BW) = f2 - f1

= Hz

RESULT: Darlington pair amplifier is design with the given specifications and from observed frequency response, gain and band width are calculated.

EXP NO:2B

DARLINGTON PAIR CIRCUIT (HARDWARE)

AIM: To study the frequency response of amplifier, calculate voltage gain and bandwidth from the response.

COMPONENTS & EQUIPMENTS REQUIRED: -

S.No	Device	Range/Rating	Qty
1.	(a) DC supply voltage (b) BJT (c) Capacitors (d) Resistors	12V BC107 BP 100 F,10 F 5.6K ,10k ,22K ,,1k 220 ,	1 2 2,1 Each 1NO
2.	Signal generator	0.1Hz-1MHz	1
3.	CRO	0Hz-20MHz	1
4.	Connecting wires	5A	4

CIRCUIT DIAGRAM:

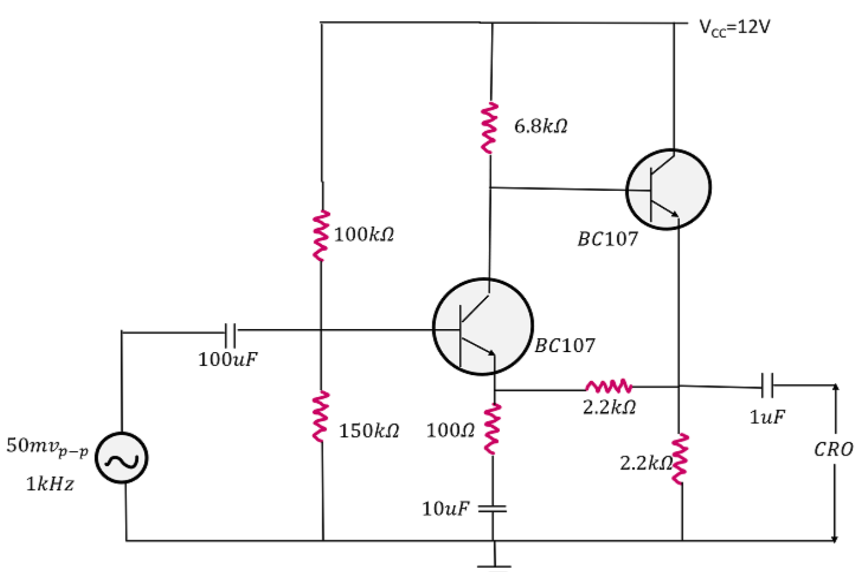


Fig: 2B.a circuit diagram of Darlington pair amplifier

PROCEDURE:

1. Connect the circuit as per the circuit diagram
2. Now connect the function generator to the input terminals of the amplifier circuit and keep the input voltage constant i.e., 30 mV, 1KHz.
3. Connect the CRO to the output of the amplifier
4. By varying the input frequency from 50 Hz to 100 MHz in steps take the output voltages from CRO.
5. Then calculate Voltage gain $A_V = \frac{V_O}{V_i}$

   In dB magnitude $A_V = 20\log\frac{V_O}{V_i}$

6. Then plot frequency Vs gain in dB on semilog sheet.
7. From semilog sheet find bandwidth (f_H - f_L)

   Band width = $f_H - f_L$

Expected Graph:

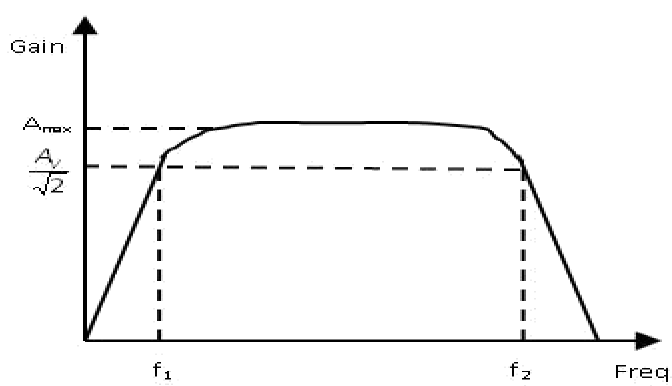


Fig: 2B.b: Expected waveform of Darlington pair amplifier

OBSERVATION TABLE:

Input Voltage V_{in} = 50 mV(constant)

Freq (Hz)	**Output voltage V_0 (V)**	**Gain $A_V = V_0/V_i$**	**Gain in dB = 20log(A_V)**
50Hz			
100Hz			
500Hz			
1kHz			
10kHz			
50kHz			
100kHz			
500kHz			
1MHz			

APPLICATIONS

1. Power Regulators.
2. Audio Amplifier o/p stages.
3. Controlling of motors.
4. Display drivers.
5. Controlling of Solenoid.
6. Light and touch sensors.

RESULT: -

1. Frequency response of Darlington pair amplifier Graph is plotted.
2. Gain = _______dB (maximum).
3. Bandwidth= $f_H - f_L$ = _________Hz

VIVA QUESTIONS:

1. Why the cascade amplifier provides a phase reversal?
2. In the dc equivalent circuit of an amplifier, how are capacitors treated?
3. What is the effect of bypass capacitor on frequency response?
4. Define lower and upper cutoff frequencies for an amplifier.
5. State the reason for fall in gain at low and high frequencies.
6. What is meant by unity gain frequency?
7. Define Bel and Decibel.
8. What do we represent gain in decibels?
9. Why do you plot the frequency response curve on a semi-log paper?
10. Explain the function of emitter bypass capacitor C_E?
11. What is the equation for voltage gain?
12. What is cut off frequency? What is lower 3dB and upper 3dB cut off frequency?
13. What are the applications of Case code amplifier?
14. What is active region?
15. What is Bandwidth of an amplifier?
16. What is the importance of gain bandwidth product?
17. Draw h parameter equivalent circuit of Case cade amplifier.
18. What is the importance of coupling capacitors in Case cade amplifier?
19. What is the importance of emitter by pass capacitor?
20. What type of feedback is used in case cade amplifier?
21. What are the various types of biasing a Transistor?
22. What is Q point of operation of the transistor? What is the region of operation of the transistor when it is working as an amplifier?
23. Why frequency response of the amplifier is drawn on semi-log scale graph?
24. If Q point is not properly selected, then what will be the effect on the output waveform?
25. What are the typical values of the input impedance and output impendence of case code amplifier?
26. What is meant by unity gain frequency?
27. In the dc equivalent circuit of an amplifier, how are capacitors treated?
28. What is the effect of bypass capacitor on frequency response?
29. Define lower and upper cutoff frequencies for an amplifier.
30. State the reason for fall in gain at low and high frequencies.
31. What is meant by unity gain frequency?
32. What is active region?
33. What is Bandwidth of an amplifier?
34. What is the importance of gain bandwidth product?
35. Draw h parameter equivalent circuit of Case cade amplifier.
36. What is the importance of coupling capacitors in Case cade amplifier?
37. What is the importance of emitter by pass capacitor?
38. What type of feedback is used in case cade amplifier?
39. Why the case code amplifier provides a phase reversal?
40. In the dc equivalent circuit of an amplifier, how are capacitors treated?

EXERCISE PROBLEMS:

1. Plot the frequency and amplitude response of BC 107 amplifier with C_1 = 5 μF.
2. Plot the frequency response of BC 107amplifier with C_2 = 5 μF with triangular i/p.
3. Plot the amplitude response of BC 107 amplifier with R_1 = 4.1 K.
4. Plot the amplitude response of BC 107 amplifier with R_2 = 9.4 K triangular i/p.
5. Plot frequency response of BC 107 amplifier R_1 = 4.1 K, R_2 = 9.4 K with square i/p.
6. Plot the frequency and amplitude response of BC 107 amplifier with C_1 = 5 μF.
7. Plot frequency response of BC107 R1 = 4.1 K, R_2 = 9.4 K with square i/p.
8. Plot the frequency and amplitude response of BC107 amplifier with C_1 = 10 μF.
9. Plot the frequency response of BC 107amplifier with C_2 = 2 μF with i/p.
10. Plot the amplitude response of BC 107 amplifier with R_1 = 2.1 K.
11. Plot the amplitude response of BC 10 7amplifier with R_2 = 5.4 K i/p.
12. Plot frequency response of BC 107amplifier R_1 = 2.1 K, R_2 = 2.4 K with i/p.
13. Plot the frequency and amplitude response of BC 107amplifier with C_1 = 2 μF.
14. Plot the frequency response of amplifier with C_2 = 2 μF, i/p.
15. Plot the amplitude response of BC 107 amplifier with R_1 = 2.1 K.
16. Plot the amplitude response of BC 107 amplifier with R_2 = 2.4 K i/p.
17. Plot frequency response of R1 = 2.1 K, R_2 = 9.4 K with i/p.
18. Plot the frequency and amplitude response of BC 107 amplifier with C_1 = 5 μF.
19. Plot the frequency response of BC 107 amplifier with C_2 = 5 μF with i/p.
20. Plot the amplitude response of BC 107 amplifier with R_1 = 4.1 K.

EXP NO: 3

CURRENT SHUNT FEED BACK AMPLIFIER (SOFTWARE)

AIM: -

1. Study the concept of feedback in amplifiers.
2. Study the characteristics of current shunt feedback amplifier.
3. Identify all the formulae will need in this experiment.

OBJECTIVE:

1. To simulate the Current Shunt Feedback Amplifier in Multisim and study the transient and frequency response.
2. To determine the maximum gain, 3dB gain, lower and upper cutoff frequencies and bandwidth of Current Shunt Feedback Amplifier by performing the AC analysis.
3. To determine the effect of feedback on gain and bandwidth and compare with Multisim results.

REQUIREMENTS:

1. Transistor – 2n2222(2)
2. Resistors – as per circuit diagram
3. Capacitors – as per circuit diagram
4. RPS – 0-30V.
5. CRO.
6. Breadboard.
7. Connecting wires and Probes.

CIRCUIT DIAGRAM:

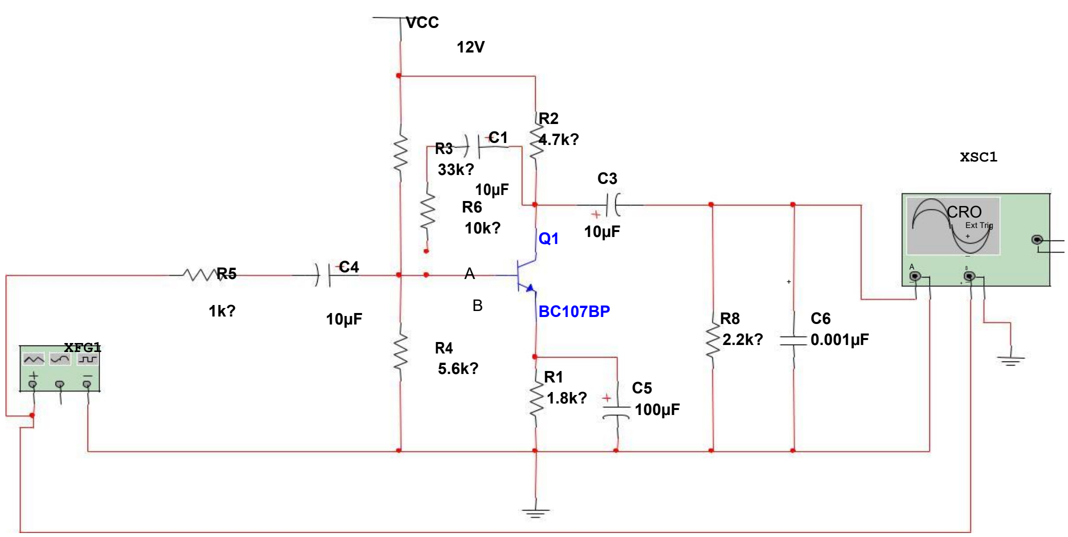


Fig: 3.a Current shunt feedback Amplifier circuit diagram

THEORY:

Feedback plays a very important role in electronic circuits and the basic parameters, such as input impedance, output impedance, current and voltage gain and bandwidth, may be altered considerably by the use of feedback for a given amplifier.

A portion of the output signal is taken from the output of the amplifier and is combined with the normal input signal and thereby the feedback is accomplished.

There are two types of feedback. They are i) Positive feedback and ii) Negative feedback. Negative feedback helps to increase the bandwidth, decrease gain, distortion, and noise, modify input and output resistances as desired.

A current shunt feedback amplifier circuit is illustrated in the figure. It is called a series-derived, shunt-fed feedback. The shunt connection at the input reduces the input resistance and the series connection at the output increases the output resistance. This is a true current amplifier.

PROCEDURE:

1. Open Multisim Software to design Common Emitter amplifier circuit
2. Select on New editor window and place the required component on the circuit window.
3. Make the connections using wire and set oscillator (FG) frequency & amplitude.
4. Check the connections and the specification of components value properly.
5. Go for simulation using Run Key observe the output waveforms on CRO
6. Indicate the node names and go for AC Analysis with the output node
7. Observe the Ac Analysis and draw the magnitude response curve
8. Calculate the bandwidth of the amplifier

Observations/Graphs:

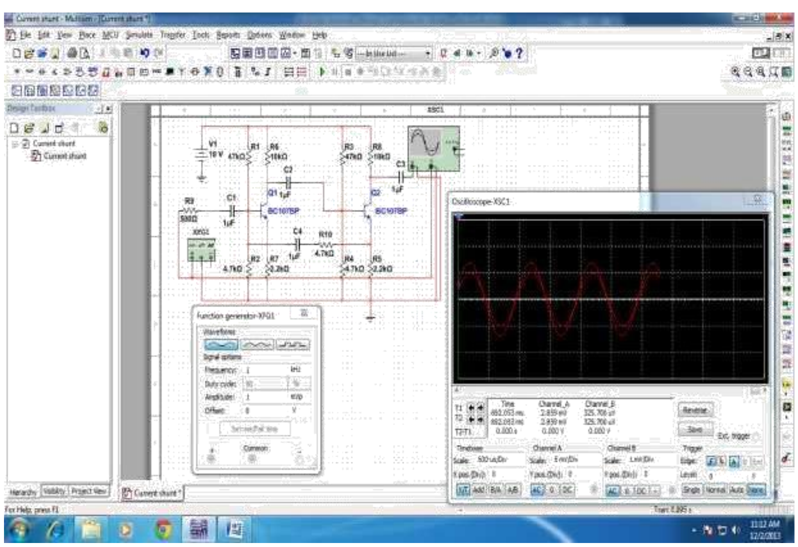


Fig: 3.b Output waveform Current shunt feedback Amplifier

i) Transient Response:

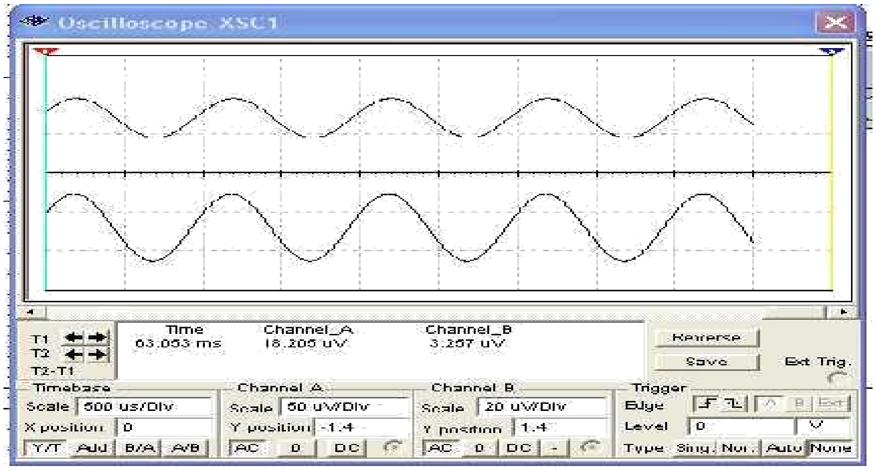


Fig: 3.b Transient response of Current shunt feedback Amplifier

ii) Frequency Response:

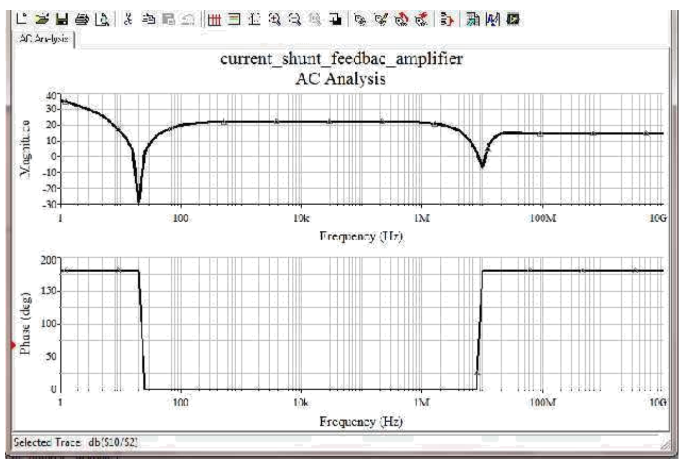


Fig: 3.b Frequency response of Current shunt feedback Amplifier

RESULTS:

1. From the frequency response curve the following results are calculated:

S. No.	Parameter	Value
1	Max. Gain in dB	
2	3dB Gain	

2. From the AC response, it is observed that, ______________________________.

EXP NO:3

CURRENT SHUNT FEED BACK AMPLIFIER (HARDWARE)

AIM:

1. Plot the frequency response of a Current Shunt Feedback Amplifier.
2. Calculate gain.
3. Calculate bandwidth.

COMPONENTS & EQUIPMENTS REQUIRED:

S.No	Device	Range/Rating	QTY
1.	(a) DC supply voltage	12V	1
	(b) Transistor	BC107	1
	(c) Capacitors	10µF	2
		100 µF	1
	(d) Resistors	3.3KΩ ,5.6KΩ 4.7KΩ	1
		,1.8KΩ,10KΩ	1
2.	Signal generator	0.1Hz-1MHz	1
3.	CRO	0Hz-20MHz	1
4.	Connecting wires	5A	4

CIRCUIT DIAGRAM:

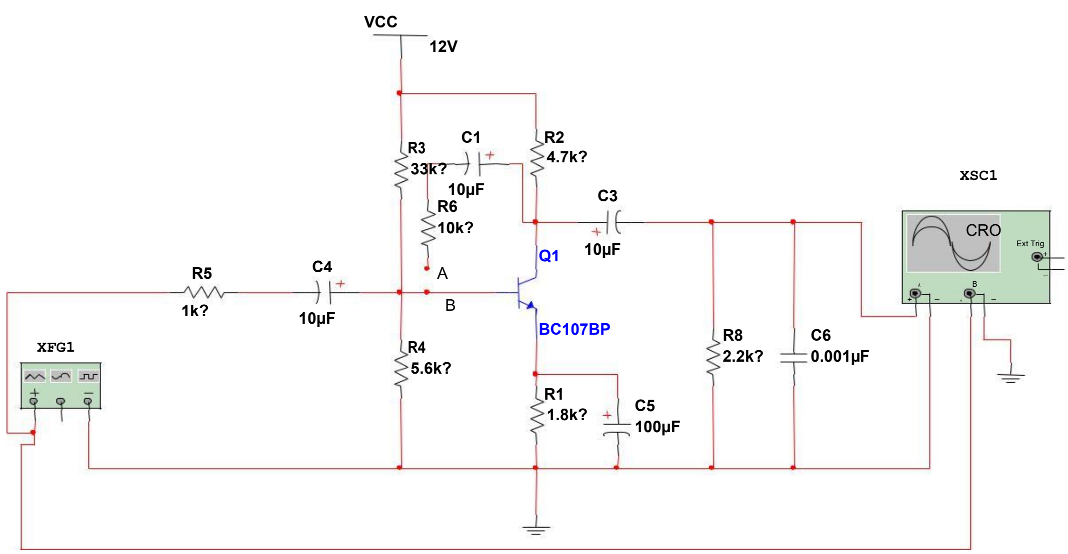


Fig: 3.a Current shunt feedback Amplifier circuit diagram

PROCEDURE :

1. Connect the circuit diagram as shown in figure.
2. Adjust input signal amplitude in the function generator and observe an amplified voltage at the output without distortion.
3. By keeping input signal voltage, say at 50mV, vary the input signal frequency from 0 to 1MHz in steps as shown in tabular column and note the corresponding output voltages.

PRECAUTIONS:

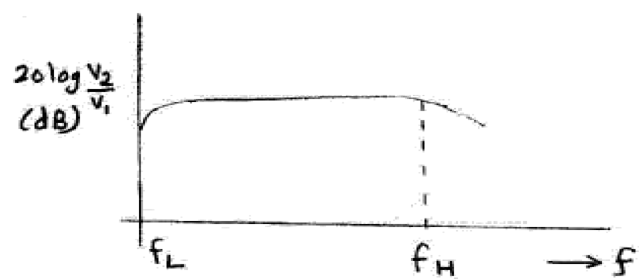
Avoid loose connections and give proper input Voltage

TABULAR COLUMN :

Input V_i =50mV

Frequency(Hz)	Output voltage V_o	Gain $=V_o/V_i$	Gain in dB=$20\log(V_0/V_i)$
50Hz			
100Hz			
1KHz			
10KHz			
50KHz			
100KHz			
500KHz			
1MHz			

EXPECTED GRAPH:



RESULT:

1. Frequency response of Current Shunt feedback amplifier is plotted.
2. Gain = _______dB (maximum).
3. Bandwidth= $f_H - f_L$ = _________Hz

. . . .

VIVA QUESTIONS :

1. State the merits and demerits of negative feedback in amplifiers.
2. If the bypass capacitor C_E in an RC coupled amplifier becomes accidentally open circuited, what happens to the gain of the amplifier? Explain.
3. When will a negative feedback amplifier circuit be unstable?
4. What is the parameter which does not change with feedback?
5. What type of feedback has been used in an emitter follower circuit?
6. Define Current shunt feedback amplifier?
7. Draw the current shunt feedback amplifier?
8. When will a negative feedback amplifier circuit be unstable?
9. What is the parameter which does not change with feedback?
10. What type of feedback has been used in an emitter follower circuit?
11. Transistor when it is working as an amplifier?
12. Why frequency response of the amplifier is drawn on semi-log scale graph?
13. If Q point is not properly selected, then what will be the effect on the output waveform?
14. What is active region?
15. Why is common base configuration used as current buffer even though it has properties of current amplifier?
16. What does the current shunt feedback amplifier amplify? And how?
17. If the bypass capacitor C_E in an RC coupled amplifier becomes accidentally open circuited, what happens to the gain of the amplifier? Explain.
18. When will a negative feedback amplifier circuit be unstable?
19. What is the parameter which does not change with feedback?
20. Transistor when it is working as an amplifier?
21. Why frequency response of the amplifier is drawn on semi-log scale graph?
22. If Q point is not properly selected, then what will be the effect on the output waveform?
23. What is active region?
24. What is Bandwidth of an amplifier?
25. Why is common base configuration used as current buffer even though it has properties of current amplifier?
26. Plot the frequency and amplitude response of BC 107amplifier with C_1 = 2 μF.
27. Plot the frequency response of amplifier with C_2 = 2 μF, i/p.
28. Plot the frequency response of BC 107 amplifier with R_1 = 2.1 K.
29. Plot the frequency response of BC 107 amplifier with R_2 = 2.4 K i/p.
30. Plot frequency response of R1 = 2.1 K, R_2 = 9.4 K with i/p.

APPLICATIONS:

1. Voltage series feedback (Af= Vo/Vs) –Voltage amplifier
2. Voltage shunt feedback (Af= Vo/Is) –Trans-resistance amplifier
3. Current series feedback (Af= Io/Vs) -Trans-conductance amplifier
4. Current shunt feedback (Af= Io/Is) -Current amplifier

EXERCISE PROBLEMS:

1. Plot the frequency response of BC 107 amplifier with C_1 = 5 μF.
2. Plot the frequency response of BC 547 amplifier with C_2 = 5 μF with i/p.
3. Plot the frequency response of BC 2N2222 amplifier with R_1 = 4.1 K.
4. Plot the frequency response of BC 107 amplifier with R_2 = 9.4 K i/p.
5. Plot frequency response of BC 107 amplifier R_1 = 4.1 K, R_2 = 9.4 K with i/p.
6. Plot the frequency and amplitude response of BC 107 amplifier with C_1 = 5 μF.
7. Plot the frequency response of amplifier with C_2 = 5 μF, i/p.
8. Plot the frequency response of BC 107 amplifier with R_1 = 4.1 K.
9. Plot the frequency response of BC107 amplifier with R_2 = 9.4 K i/p.
10. Plot frequency response of BC107 R1 = 4.1 K, R_2 = 9.4 K with i/p.
11. Plot the frequency and amplitude response of BC107 amplifier with C_1 = 10 μF.
12. Plot the frequency response of BC 107amplifier with C_2 = 2 μF with i/p.
13. Plot the frequency response of BC 107 amplifier with R_1 = 2.1 K.
14. Plot the frequency response of BC 10 7amplifier with R_2 = 5.4 K i/p.
15. Plot frequency response of BC 107amplifier R_1 = 2.1 K, R_2 = 2.4 K with i/p.

EXP NO: 4

VOLTAGE SERIES FEED BACK AMPLIFIER (SOFTWARE)

AIM: To study the frequency response of amplifier, calculate voltage gain and bandwidth from the response.

OBJECTIVE:

1. To simulate the Voltage series feedback Amplifier in Multisim and study the transient and frequency response.
2. To determine the phase relationship between the input and output voltages by performing the transient analysis.
3. To determine the maximum gain, 3dB gain, lower and upper cutoff frequencies and bandwidth of Voltage series feedback Amplifier by performing the AC analysis.
4. To determine the effect of cascading on gain and bandwidth.

SOFTWARE USED:

Multisim V 13.0

CIRCUIT DIAGRAM:

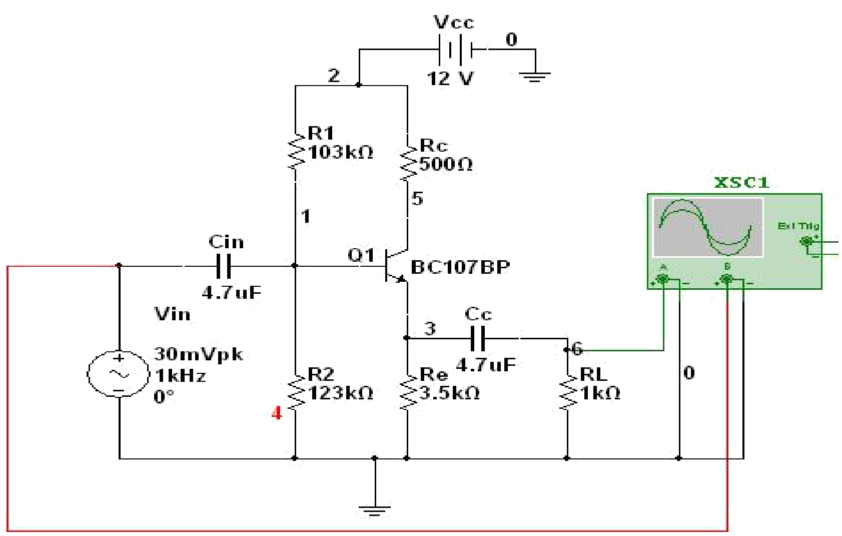


Fig:4.a Voltage series feedback Amplifier circuit diagram

THEORY:

A voltage series feedback amplifier is a type of feedback amplifier where a portion of the output voltage is fed back to the input in series with the signal source. In this configuration, the feedback is applied in such a way that it stabilizes the gain, reduces distortion, and improves the bandwidth of the amplifier. The feedback network typically consists of resistors or a combination of resistors and capacitors, and it is designed to ensure that the feedback voltage is proportional to the output voltage. This negative feedback lowers the overall gain of the amplifier, but it enhances linearity and reduces the effects of variations in transistor parameters, temperature changes, and power supply fluctuations. Additionally, voltage series feedback improves the input impedance of the amplifier, making it more suitable for interfacing with high-impedance sources, while slightly lowering the output impedance, which aids in better signal transfer to the load.

PROCEDURE:

1. Switch ON the computer and open the multisim software.
2. Check whether the icons of the instruments are activated and enable.
3. Now connect the circuit using the designed values of each and every component.
4. Connect the function generator with sine wave of 50 mV p-p as input at the input of terminals of the circuit.
5. Connect the Cathode Ray Oscilloscope (CRO) to the out put terminals of the circuit.
6. Go to simulation button click it for simulation process.
7. From the CRO note the following values
    1. Input voltage V_i =
    2. Output voltage V_0 =
    3. Voltage gain $A_V = V_0/V_i$ =
    4. Phase shift θ =
8. To study the frequency response click the AC analysis, so that a screen displays the following options
    1. Start frequency
    2. Stop frequency
    3. Vertical scale
9. Assign the proper values for start frequency, stop frequency and vertical scale according to the circuit requirements and observe the frequency response.
10. From the frequency response calculate the

    maximum gain A_{Vmax} =

lower cutoff frequency (f1) at A_{Vmax} - 3dB (decibel scale) value

at $A_{Vmax}/\sqrt{2}$ (linear scale) =

Higher cutoff frequency (f2) at A_{Vmax} - 3dB (decibel scale) value

at $A_{Vmax}/\sqrt{2}$ (linear scale) =

OBSERVATIONS:

From CRO:

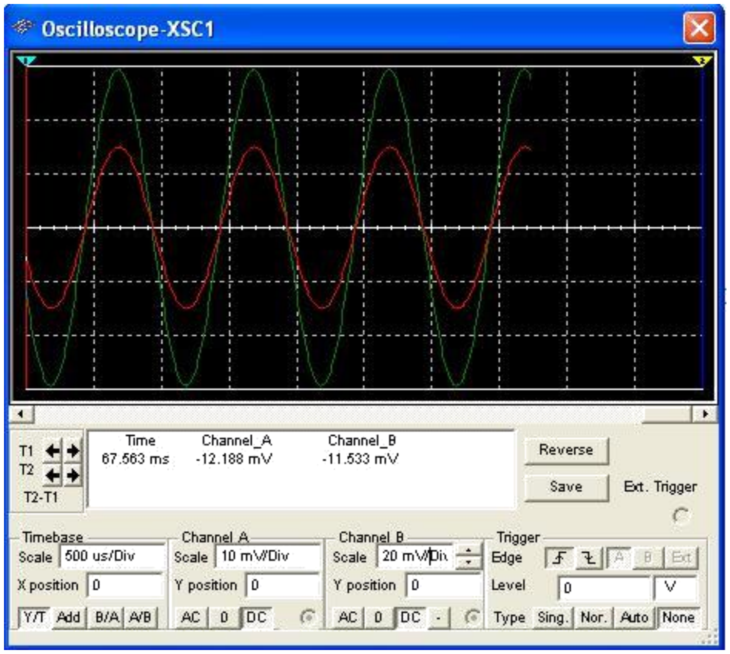


Fig: 4b Output waveform of voltage series feedback Amplifier

1. Input voltage V_i =
2. Output voltage V_0 =
3. Voltage gain $A_V = V_0/V_i$ =
4. Phase shift θ =

From Frequency response:

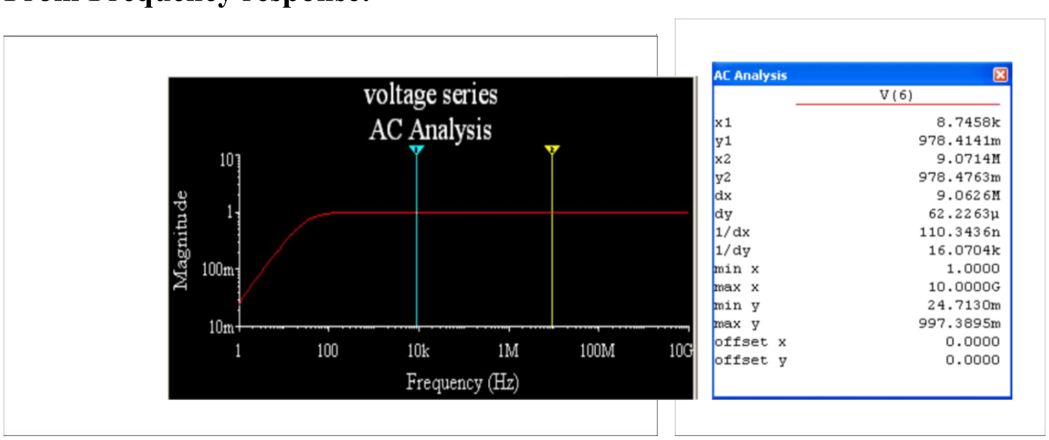


Fig:4.c Frequency response of Voltage series feedback Amplifier

RESULT: -

1. Maximum gain A_{Vmax} =
2. Lower cutoff frequency(f1) at A_{Vmax}-3dB (decibel scale) value
   at $A_{Vmax}/\sqrt{2}$ (linear scale) =
3. Higher cutoff frequency(f2) at A_{Vmax}-3dB (decibel scale)
   value at $A_{Vmax}/\sqrt{2}$ (linear scale) =

APPLICATIONS:

1. Voltage series feedback (A_f= Vo/Vs) –Voltage amplifier.
2. Voltage shunt feedback (A_f= Vo/Is) –Trans-resistance amplifier.
3. Current series feedback (A_f= Io/Vs) -Trans-conductance amplifier.
4. Current shunt feedback (A_f= Io/Is) -Current amplifier.

EXPT NO: 4

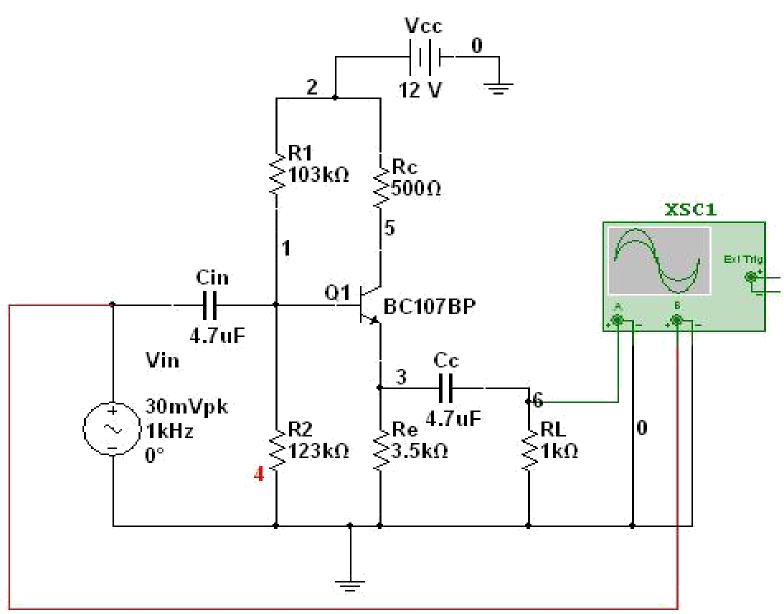
VOLTAGE SERIES FEED BACK AMPLIFIER (HARDWARE)

AIM: To study the frequency response of amplifier, calculate voltage gain and bandwidth from the response.

APPARATUS :

S.No	Device	Range/Rating	QTY
1.	(a) DC supply voltage (b) Transistor (c) Capacitors (d) Resistors	12V BC107 4.7 µ F 100KΩ ,500Ω 3.5KΩ ,1KΩ	1 1 2 1 1
2.	Signal generator	0.1Hz-1MHz	1
3.	CRO	0Hz-20MHz	1
4.	Connecting wires	5A	4

CIRCUIT DIAGRAM:



PROCEDURE: -

1. Connect the circuit diagram as shown in figure.
2. Adjust input signal amplitude in the function generator and observe an amplified voltage at the output without distortion.
3. By keeping input signal voltage, say at 50mV, vary the input signal frequency from 0 to 1MHz in steps as shown in tabular column and note the corresponding output voltages.

PRECAUTIONS:

Avoid loose connections and give proper input Voltage

TABULAR COLUMN :

Input V_i =50mV

Frequency(Hz)	Output voltage V_o	Gain $=V_o/V_i$	Gain in dB=20log(V_0/V_i)
50Hz			
100Hz			
1KHz			
10KHz			
50KHz			
100KHz			
500KHz			
1MHz			

RESULT: -

3. Frequency response of Voltage series feedback Amplifier graph is plotted.
4. Gain = _______dB (maximum).
5. Bandwidth= f_H-f_L = _________Hz.

VIVA QUESTIONS:

1. State the merits and demerits of negative feedback in amplifiers.
2. If the bypass capacitor C_E in an RC coupled amplifier becomes accidentally open circuited, what happens to the gain of the amplifier? Explain.
3. When will a negative feedback amplifier circuit be unstable?
4. What is the parameter which does not change with feedback?
5. What type of feedback has been used in an emitter follower circuit?
6. Define voltage series feedback amplifier?
7. Draw the voltage series feedback amplifier?
8. When will a negative feedback amplifier circuit be unstable?
9. What is the parameter which does not change with feedback?
10. What type of feedback has been used in an emitter follower circuit?
11. Transistor when it is working as an amplifier?
12. Why frequency response of the amplifier is drawn on semi-log scale graph?
13. If Q point is not properly selected, then what will be the effect on the output waveform?
14. What is active region?
15. Why is common base configuration used as current buffer even though it has properties of current amplifier?
16. What does the current shunt feedback amplifier amplify? And how?
17. If the bypass capacitor C_E in an RC coupled amplifier becomes accidentally open circuited,
18. What happens to the gain of the amplifier? Explain.
19. Transistor when it is working as an amplifier?
20. Why frequency response of the amplifier is drawn on semi-log scale graph?
21. If Q point is not properly selected, then what will be the effect on the output waveform?
22. What is active region?
23. What is Bandwidth of an amplifier?
24. Plot the amplitude response of BC 10 7amplifier with R_2 = 5.4 K i/p.
25. Plot frequency response of BC 107amplifier R_1 = 2.1 K, R_2 = 2.4 K with i/p.
26. Plot the frequency and amplitude response of BC 107amplifier with C_1 = 2 μF.
27. Plot the frequency response of amplifier with C_2 = 2 μF, i/p.
28. Plot the amplitude response of BC 107 amplifier with R_1 = 2.1 K.
29. Plot the amplitude response of BC 107 amplifier with R_2 = 2.4 K i/p.
30. Plot frequency response of R1 = 2.1 K, R_2 = 9.4 K with i/p.

EXERCISE PROBLEMS:

1. Plot the frequency and amplitude response of BC 107 amplifier with C_1 = 5 μF.
2. Plot the frequency response of BC 107amplifier with C_2 = 5 μF with triangular i/p.
3. Plot the amplitude response of BC 107 amplifier with R_1 = 4.1 K.
4. Plot the amplitude response of BC 107 amplifier with R_2 = 9.4 K triangular i/p.
5. Plot frequency response of BC 107 amplifier R_1 = 4.1 K, R_2 = 9.4 K with square i/p.
6. Plot the frequency and amplitude response of BC 107 amplifier with C_1 = 5 μF.

7. Plot the frequency response of amplifier with C_2 = 5 μF, triangular i/p.
8. Plot the amplitude response of BC 107 amplifier with R_1 = 4.1 K.
9. Plot the amplitude response of BC107 amplifier with R_2 = 9.4 K triangular i/p.
10. Plot frequency response of BC107 R1 = 4.1 K, R_2 = 9.4 K with square i/p.
11. Plot the frequency and amplitude response of BC107 amplifier with C_1 = 10 μF.
12. Plot the frequency response of BC 107amplifier with C_2 = 2 μF with i/p.
13. Plot the amplitude response of BC 107 amplifier with R_1 = 2.1 K.

EXPT NO: 5

RC PHASE SHIFT OSCILLATOR (Software)

PRELAB:

1. Study the different types of oscillator and their conditions.
2. Identify all the formulae you will need in this Lab.

OBJECTIVE:

1. To simulate RC phase shift oscillator in Multisim and study the transient response.
2. To determine the phase shift of RC network in the circuit.

SOFTWARE TOOL:

Multisim v 13.0

APPARATUS:

1. Regulated power supply	-	1 No.
2. Function generator	-	1 No.
3. CRO	-	1 No.
4. Transistor (BC 107 or 2N2222)	-	2 No.
5. Resistors (47 KΩ, 2.2 KΩ, 1k)	-	1 No. each
6. Resistor (10 KΩ)	-	3 Nos.
7. Capacitors (10 μF, 100 μF)	-	1No. each
(1nf,or 10nf)	-	3 No.

CIRCUIT DIAGRAM :

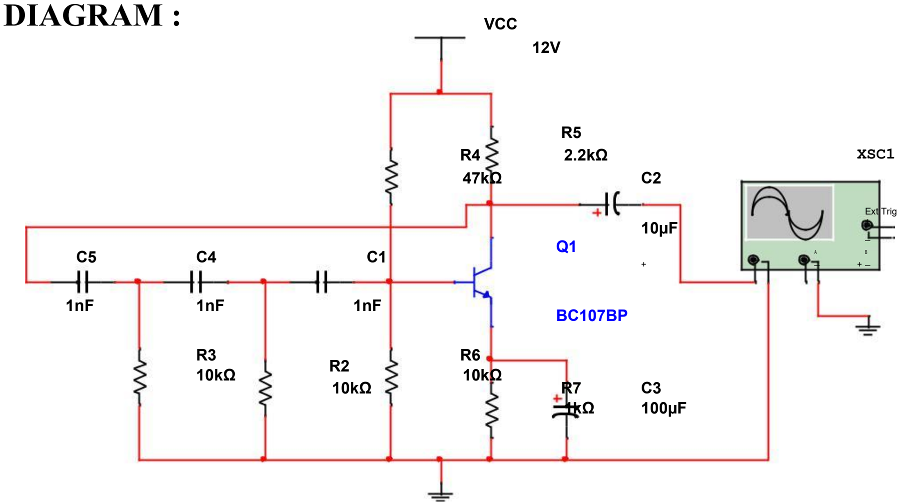


Fig: 5.a RC Phase Shift oscillator circuit diagram

THEORY:

An RC phase shift oscillator is a type of sinusoidal oscillator that generates continuous waveforms at a specific frequency determined by the components in the circuit. It consists of an amplifier, typically a transistor or an operational amplifier, and a feedback network made up of resistors and capacitors arranged in a series of three or more RC stages. Each RC stage shifts the phase of the input signal by 60 degrees, so that the total phase shift around the loop is 180 degrees. The amplifier provides an additional 180-degree phase shift, resulting in a total of 360 degrees, or zero phase shift, necessary for sustained oscillations. The frequency of oscillation is determined by the values of the resistors and capacitors in the feedback network, and it is typically chosen to be in the audio frequency range. RC phase shift oscillators are known for their simplicity and are commonly used in low-frequency applications, such as audio signal generators. However, they typically provide less frequency stability compared to other types of oscillators, like crystal oscillators.

PROCEDURE:

1. Open Multisim Software to design RC Phase shift oscillator
2. Select on New editor window and place the required component on the circuit window.
3. Make the connections using wire and check the connections and oscillator.
4. Go for simulation and using Run Key observe the output waveforms on CRO
5. Observe the Transient Response and Calculate the Frequency of the oscillator

OBSERVATIONS/GRAPHS:

TRANSIENT RESPONSE:

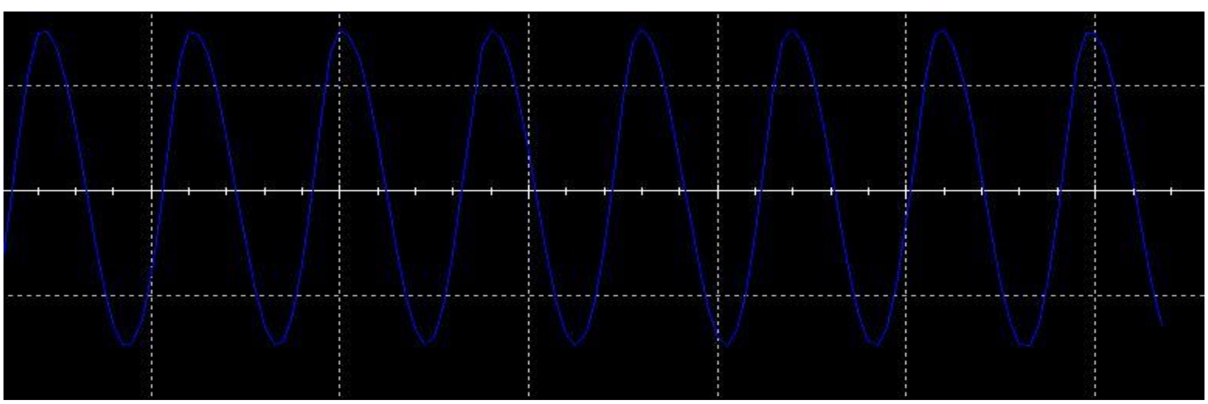


Fig: 5.b : Transient response of RC Phase Shift oscillator

RESULT:

1. For C = 0.0022 F & R=10K
   i. Theoretical frequency=
   ii. Practical frequency=

2. For C = 0.0033 F & R=10K
    i. Theoretical frequency=
    ii. Practical frequency=
3. For C = 0.01 F & R=10K
    i. Theoretical frequency=
    ii. Practical frequency=

The frequency of oscillations of RC phase shift oscillator is calculated compared the theoretical and practical values.

APPLICATIONS:

FET phase-shift oscillator is used for generating signals over a wide frequency range. The frequency may be varied from a few Hz to 200 Hz by employing one set of resistor with three capacitors ganged together to vary over a capacitance range in the 1 : 10 ratio.

EXP NO:5

RC PHASE SHIFT OSCILLATOR (Hardware)

AIM:

Find practical frequency of RC phase shift oscillator and to compare it with theoretical frequency for R=10K and C = 0.01 F, 0.0022 F & 0.0033 F respectively

COMPONENTS AND EQUIPMENTS REQUIRED:

S.No	Device	Range/ Rating	Qty
1	a) DC supply voltage b) Capacitor c) Resistor d) NPN Transistor	12V 100 F 10 F 10K ,5.6K 22K ,100K 1K , BC 107	1 1 2 2 3 3 1
2	CRO	(0-20) MHz	1
3.	BNC Connector		1
3	Connecting wires	5A	6

CIRCUIT DIAGRAM:

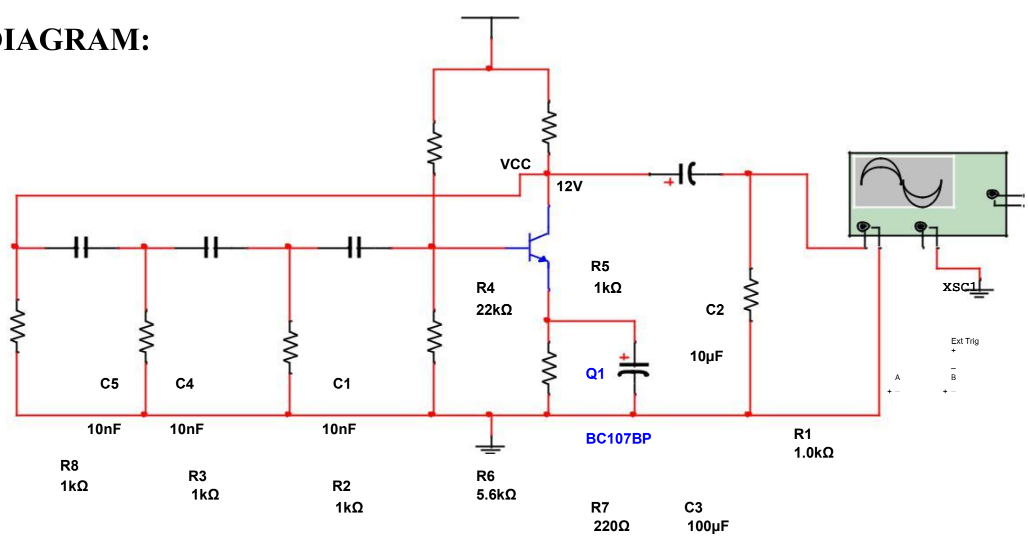


Fig: 5. c: Circuit diagram of RC Phase Shift oscillator

PROCEDURE:

1. Connect the circuit as shown in figure.
2. Connect the 0.0022 F capacitors in the circuit and observe the waveform.
3. Time period of the waveform is to be noted and frequency should be calculated by the formula f = 1/T.
4. Now fix the capacitance to 0.033 F and 0.01 F and calculate the frequency and tabulate as shown.
5. Find theoretical frequency from the formula $f = \frac{1}{2\prod RC\sqrt{6}}$ and compare theoretical and practical frequencies.

PRECAUTIONS: -

No loose contacts at the junctions.

TABULAR COLUMN:

S.No	C (F)	R ( )	Theoretical Frequency (KHz)	Practical Frequency (KHz)	V_o (p-p) (Volts)
1	10nf	1k			
2	1nf	1K			
3	10n	10K			

RESULT: -

1. For C = 0.0022 F & R=10K
   i. Theoretical frequency=
   ii. Practical frequency=
2. For C = 0.0033 F & R=10K
   i. Theoretical frequency=
   ii. Practical frequency=
3. For C = 0.01 F & R=10K
   i. Theoretical frequency=
   ii. Practical frequency=

VIVA QUESTIONS:

1. What are the conditions of oscillations?
2. Give the formula for frequency of oscillations?

3. What is the total phase shift produced by RC ladder network?
4. What are the types of oscillators?
5. What is the gain of RC phase shift oscillator?
6. What is the frequency of RC phase shift oscillator?
7. What is a phase shift oscillator?
8. Why RC oscillators cannot generate high frequency oscillations?
9. What are the applications of RC phase shift oscillators?
10. What phase shift does RC phase shift oscillator produce?
11. Why we need a phase shift between input and output signal?
12. How is phase angle determined in RC phase shift oscillator?
13. How can we get a maximum phase angle of 90 degrees in RC phase shift oscillator?
14. What is an Oscillator?.
15. Which feedback used in oscillators?
16. What is the output of an oscillator if transistor is ideal?
17. What are LC oscillators?
18. Why can't we use LC oscillator for low frequency oscillations?
19. How an oscillator generates oscillations without any input?
20. Classify oscillators?
21. Plot the Amplitude response of 2N3904 Oscillator with C_1 = 5 µF.
22. Plot the Amplitude response of 2N2222 Oscillator C2 = 5 µF with i/p.
23. Plot the Amplitude response of BC107 Oscillator with R_1 = 4.1 K.
24. Plot the Amplitude response of BC 547 Oscillator with R_2 = 9.4 K i/p.
25. Plot the Amplitude response of BC 548 Oscillator R1 = 4.1 K, R_2 = 9.4 K with i/p.
26. Plot the Amplitude response of BC 557 Oscillator with C_1 = 5 µF.
27. Plot the Amplitude response of BC 547 Oscillator with C_2 = 5 µF, i/p.
28. Plot the Amplitude response of 2N3904 Oscillator with R_1 = 4.1 K.
29. Plot the Amplitude response of 2N3904 Oscillator with R_2 = 9.4 K i/p.
30. Plot the Amplitude response of 2N3904 Oscillator R1 = 4.1 K, R_2 = 9.4 K with i/p.

EXP NO: 6 A

HARTLEY OSCILLATOR (SOFTWARE)

PRELAB:

Study the operation and working principle Hartley oscillator.

OBJECTIVE:

To design Hartley oscillator using Multisim software and calculate the frequency

APPARATUS:

1. Transistor – BC 107
2. Resistors – 1K, 5K,10K,100K,
3. Capacitors – 100nF(3),10nf
4. Inductor-10mH or 1mH.
5. RPS
6. CRO
7. Breadboard
8. Connecting wires and probes

SOFTWARE TOOL: Multisim V 13.0

CIRCUIT DIAGRAM:

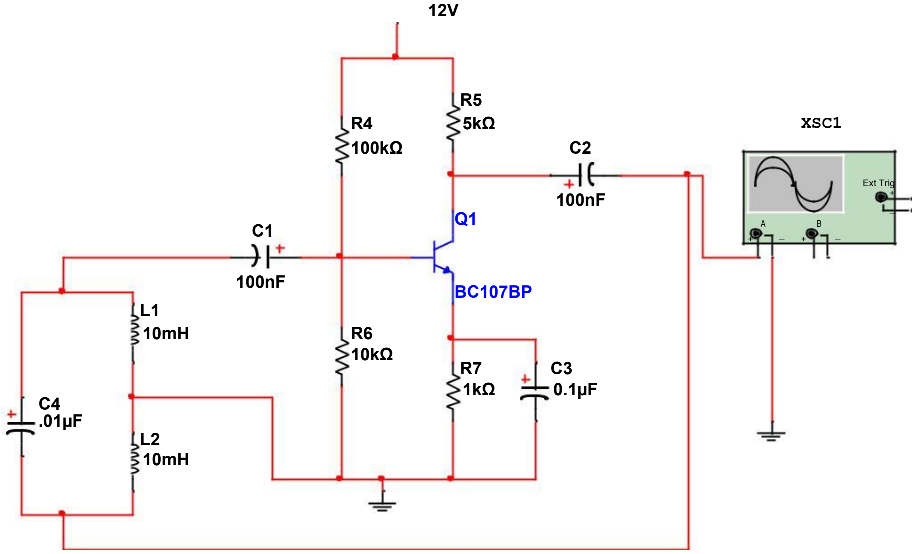


Fig:6A-a: Hartley oscillator circuit diagram

THEORY:

A Hartley oscillator is a type of LC oscillator that generates sinusoidal waveforms and is commonly used in radio frequency (RF) applications. The circuit consists of an amplifier, typically a transistor or an operational amplifier, and a tuned circuit composed of an inductor split into two sections and a capacitor. The two inductive sections are either separate inductors or a tapped inductor, with the tap point providing the necessary feedback to sustain oscillations.

PROCEDURE:

1. Open Multisim Software to design Hartley oscillator circuit
2. Select on New editor window and place the required component on the circuit window.
3. Make the connections using wire and check the connections and oscillator.
4. Go for simulation and using Run Key observe the output waveforms on CRO
5. Calculate the frequency theoretically and practically.

OBSERVATIONS/GRAPHS:

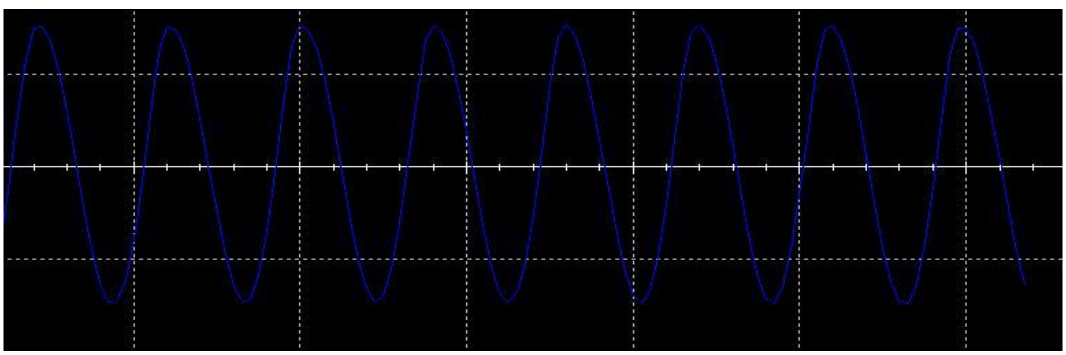


Fig:6A-b: Output waveform of Hartley oscillator

RESULT:

1. Out frequency for L1=L2=10mH, C=10nf is_________
2. Out frequency for L1=L2=10mH, C=100nf is_________
3. Out frequency for L1=L2=20mH, C=10nf is_________
4. Out frequency for L1=5, L2=10mH, C=10nf is_________

APPLICATIONS:

1. The Hartley oscillator is to produce a sine wave with the desired frequency.
2. Hartley oscillators are mainly used as radio receivers. Also note that due to its wide range of frequencies, it is the most popular oscillator.

EXP NO: 6 A

HARTLEY OSCILLATOR (HARDWARE)

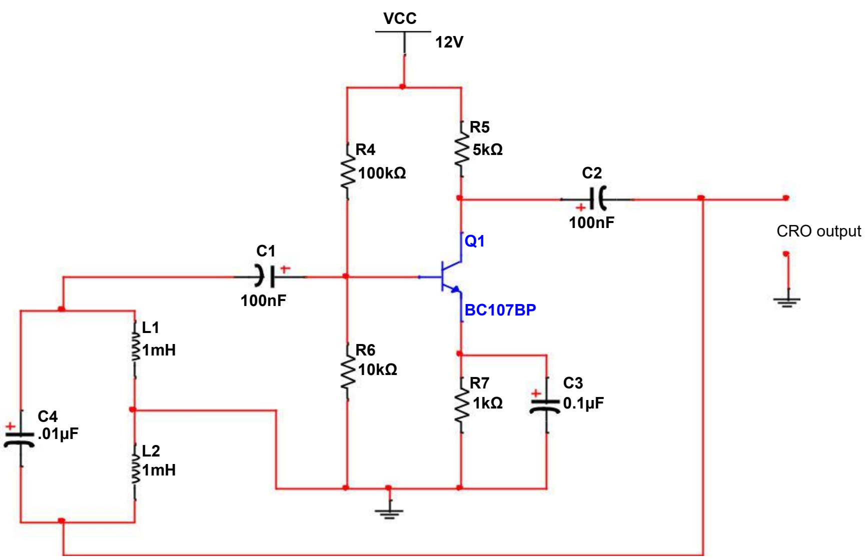
AIM:

Find practical frequency of a Hartley oscillator and to compare it with theoretical frequency for L = 10mH and C = 0.01 F, 0.033 F and 0.047 F.

COMPONENTS AND EQUIPMENTS REQUIRED:

S.No	Device	Range/Rating	Quantity
1	a) DC supply voltage b) Inductors c) Capacitor d) Resistor e) NPN Transistor	12V 5mH 0.01 F,0.022 F;0.033 F 0.047 F 1K ,10K ,47K BC 107	1 2 1 1 1 1
2	Cathode Ray Oscilloscope	(0-20) MHz	1
3.	BNC Connector		1
4	Connecting wires	5A	4

CIRCUIT DIAGRAM:



PROCEDURE:

1. Connect the circuit as shown in figure.
2. With 0.1 F capacitor and 20mH in the circuit and observe the waveform.
3. Time period of the waveform is to be noted and frequency is to be calculated by the formula f = 1/T .
4. Now fix the capacitance to 0.033 F and 0.047 F and calculate the frequency and tabulate the readings as shown.
5. Find the theoretical frequency by using below formula

$$f = \frac{1}{2\prod\sqrt{L_T C}}$$

where $L_T = L_1 + L_2$ = 5mH + 5mH = 10mH

PRECAUTIONS: No loose contacts at the junctions.

TABULATIONS:

S.No	L_T(mH)	C (F)	Theoretical frequency (KHz)	Practical frequency (KHz)	V_o (peak to peak)
1	10	0.01			
2	10	0.033			
3	10	0.047			

RESULT:

1. For C = 0.01 F, & L_T = 10 mH;
   Theoretical frequency
   = Practical frequency =
2. For C = 0.033 F, & L_T = 10 mH;
   Theoretical frequency =
   Practical frequency =
3. For C = 0.047 F, & L_{Ts} = 10 mH;
   Theoretical frequency =
   Practical frequency =

APPLICATIONS:

1. It is used for generation of sinusoidal output signals with very high frequencies.
2. The Colpitts oscillator using SAW device can be used as the different type of sensors such as temperature sensor. As the device used in this circuit is highly sensitive to perturbations, it senses directly from its surface.
3. It is frequently used for the applications in which very wide range of frequencies are involved.

VIVA QUESTIONS:

1. Give the difference between Hartley and colpitts oscillator.
2. Classification of oscillators.
3. Give an example for LC oscillator.
4. Which phenomenon is employed for colpitts oscillator?
5. Give the applications of oscillator.
6. Define barkhausen criteria
7. Which type of feedback is employed in oscillators
8. Give applications for oscillators
9. What is the condition for sustained oscillations
10. Draw an oscillator circuit with feedback network given below.
11. What is the principle behind operation of a colpits oscillator?
12. What are the advantages and disadvantages of *colpits* oscillators?
13. Mention two essential conditions for a circuit to maintain oscillations?
14. Define an oscillator?
15. Define barkhausen criteria
16. Which type of feedback is employed in oscillators
17. Give applications for oscillators
18. What is an Oscillator?
19. Which feedback used in oscillators?
20. Classify oscillators?
21. Which oscillators are AF oscillators?
22. Draw an oscillator circuit with feedback network given below.
23. What is the principle behind operation of a colpitts oscillator?
24. What are the advantages and disadvantages of colpitts oscillators?
25. Mention two essential conditions for a circuit to maintain oscillations?
26. Define an oscillator?
27. Define barkhausen criteria
28. What are RC oscillators?
29. Mention two essential conditions for a circuit to maintain oscillations?
30. Define an oscillator?

EXERCISE PROBLEMS:

1. Plot the Amplitude response of 2N3904 Oscillator with C_1 = 5 µF.
2. Plot the Amplitude response of 2N2222 Oscillator C2 = 5 µF with i/p.
3. Plot the Amplitude response of BC107 Oscillator with R_1 = 4.1 K.
4. Plot the Amplitude response of BC 547 Oscillator with R_2 = 9.4 K i/p.
5. Plot the Amplitude response of BC 548 Oscillator R1 = 4.1 K, R_2 = 9.4 K with i/p.
6. Plot the Amplitude response of BC 557 Oscillator with C_1 = 5 µF.
7. Plot the Amplitude response of BC 547 Oscillator with C_2 = 5 µF, i/p.
8. Plot the Amplitude response of 2N3904 Oscillator with R_1 = 4.1 K.
9. Plot the Amplitude response of 2N3904 Oscillator with R_2 = 9.4 K i/p.
10. Plot the Amplitude response of 2N3904 Oscillator R1 = 4.1 K, R_2 = 9.4 K with i/p.
11. Plot the Amplitude response of 2N3904 Oscillator with C_1 = 10 µF.
12. Plot the Amplitude response of CL100 Oscillator with C_2 = 2 µF with i/p.
13. Plot the Amplitude response of CL 100 Oscillator with R_1 = 2.1 K.
14. Plot the Amplitude response of CK 100 Oscillator with R_2 = 5.4 K i/p.
15. Plot the Amplitude response of 2N3904 Oscillator R1 = 2.1 K, R_2 = 2.4 K with i/p.
16. Plot the Amplitude response of 2N3904 Oscillator with C_1 = 2 µF.
17. Plot the Amplitude response of 2N3904 Oscillator with C_2 = 2 µF, i/p.
18. Plot the Amplitude response of 2N3904 Oscillator with R_1 = 2.1 K.
19. Plot the Amplitude response of SL100 Oscillator with R_2 = 2.4 K i/p.
20. Plot the Amplitude response of 2N3904 Oscillator R1 = 2.1 K, R_2 = 9.4 K with i/p.
21. Plot the Amplitude response of 2N3904 Oscillator with C_1 = 5 µF.
22. Plot the Amplitude response of 2N3904 Oscillator with C_2 = 5 µF with i/p.
23. Plot the Amplitude response of 2N3904 Oscillator with R_1 = 4.1 K.
24. Plot the Amplitude response of 2N3904 Oscillator with R_2 = 9.4 K i/p.
25. Plot the Amplitude response of 2N3904 Oscillator R1 = 4.1 K, R_2 = 9.4 K with i/p.

EXP NO: 6.B

COLPITTS OSCILLATOR (software)

PRELAB:

Study the operation and working principle Colpitts oscillator.

OBJECTIVE:

To design Colpitts oscillator using Multisim software and calculate the frequency

APPARATUS:

1. Transistor – BC 107
2. Resistors – 1K, 5K,10K,100K,
3. Capacitors – 100nF(3),10nf
4. RPS
5. CRO
6. Breadboard
7. Connecting wires and probes

SOFTWARE TOOL:

Multisim

CIRCUIT DIAGRAM:

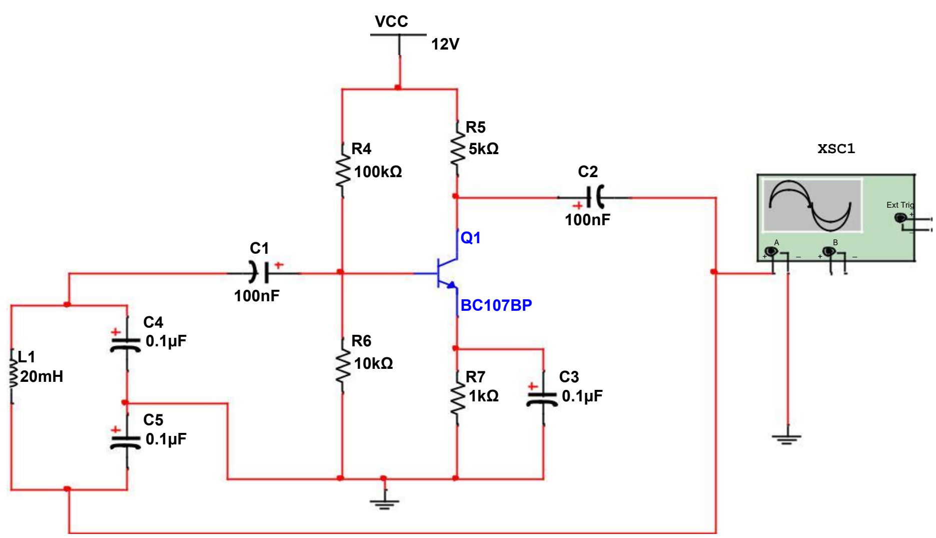


Fig:6B.a.Colpitts oscillator circuit diagram

THEORY:

A Colpitts oscillator is a type of LC oscillator that generates sinusoidal oscillations and is widely used in high-frequency applications such as RF signal generation. The key feature of the Colpitts oscillator is its use of a capacitive voltage divider as part of the feedback network. The circuit consists of an amplifier, typically a transistor or an operational amplifier, and an LC tank circuit that includes a single inductor and two capacitors connected in series, forming the capacitive voltage divider. The junction between the two capacitors is connected to the input of the amplifier to provide the necessary feedback.

PROCEDURE:

1. Open Multisim Software to design Colpitts oscillator circuit
2. Select on New editor window and place the required component on the circuit window.
3. Make the connections using wire and check the connections and oscillator.
4. Go for simulation and using Run Key observe the output waveforms on CRO
5. Calculate the frequency theoritaly and practically

OBSERVATIONS/GRAPHS:

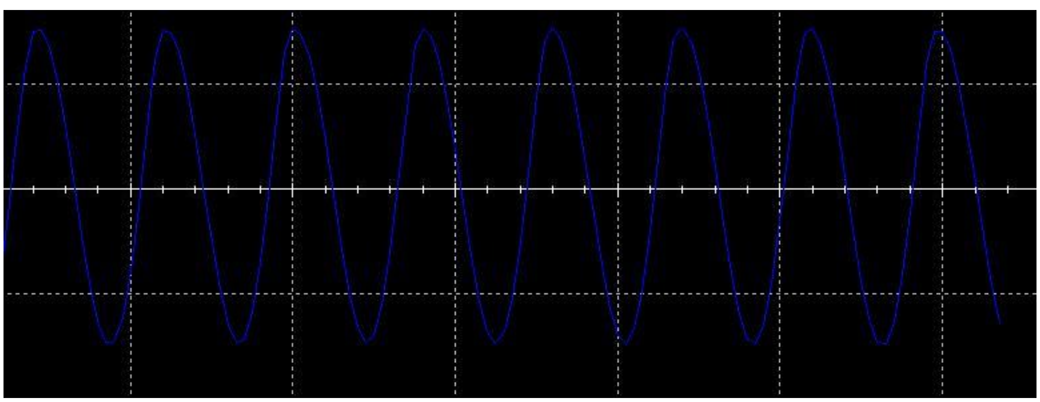


Fig:6B.b.Output waveform of Colpitts oscillator

RESULT: -

1. Out frequency for L1 =10mH, C1=C2=10nf is_________
2. Out frequency for L1 =10mH, C1=C2=100nf is_________
3. Out frequency for L1 =20mH, C1=C2=10nf is_________
4. Out frequency for L1= 10mH, C=10nf , C2=100nf _________

APPLICATIONS:

1. It is used for generation of sinusoidal output signals with very high frequencies.
2. The Colpitts oscillator using SAW device can be used as the different type of sensors such as temperature sensor. As the device used in this circuit is highly sensitive to perturbations, it senses directly from its surface.
3. It is frequently used for the applications in which very wide range of frequencies are involved.

EXP NO: 6 B

COLPITTS OSCILLATOR(HARDWARE)

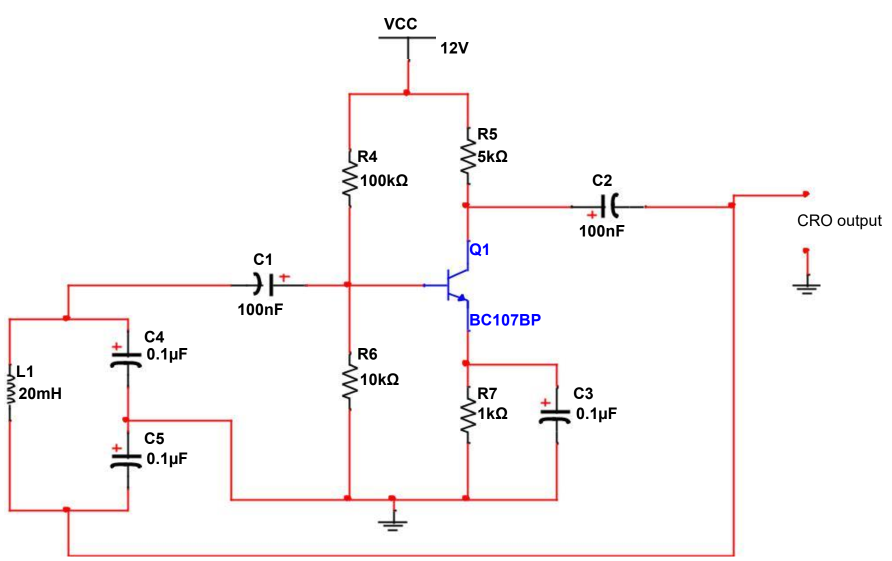
AIM:

Find practical frequency of Colpitt's oscillator and to compare it with theoretical Frequency for L= 5mH and C= 0.001 F, 0.0022 F, 0.0033 F respectively.

COMPONENTS & EQIUPMENT REQUIRED: -

S.No	Device	Range/Rating	Quantity
1	a) DC supply voltage b) Inductors c) Capacitor d) Resistor e) NPN Transistor	12V 5mH 0.01 F,0.01 F,100 F 1K ,10K ,47K BC 107	1 1 1 1 1
2	Cathode Ray Oscilloscope	(0-20) MHz	1
3.	BNC Connector		1
4	Connecting wires	5A	4

CIRCUIT DIAGRAM:



PROCEDURE:-

1. Connect the circuit as shown in the figure
2. Connect C_2= 0.001 Fin the circuit and observe the waveform.
3. Time period of the waveform is to be noted and frequency should be calculated by the formula f=1/T
4. Now, fix the capacitance to 0.002 F and then to 0.003 F and calculate the frequency and tabulate the reading as shown.
5. Find the theoretical frequency by using below formula

$$f = \frac{1}{2\Pi\sqrt{C_{eq}L}}$$

where $$C_{eq} = \frac{C_1 C_2}{C_1 + C_2}$$

PRECAUTIONS:-

1. No loose connections at the junctions.

TABULAR COLUMN:

S.NO	L(mH)	C_1 (F)	C_2 (F)	C_T (F)	Theoretical Frequency (KHz)	Practical Frequency (KHz)	Vo(V) Peak to peak
1	1mH	.1u	0.1u				
2	1mH	0.01u	0.1u				
3	1mH	0.01	0.0iu				

RESULT:

1. For C=0.01 F, 0.1uf & L= 1mH
   Theoretical frequency =
   Practical frequency =
2. For C=0.1 F, 0.1uf & L= 1mH
3. For C=0.01 F, 0.01uf & L= 5mH
   Theoretical frequency =
   Practical frequency =

Frequency of oscillations of Colpitt's oscillator is measured practically and compared with theoretical values .

APPLICATIONS:

i. The Hartley oscillator is to produce a sine wave with the desired frequency
ii. Hartley oscillators are mainly used as radio receivers. Also note that due to its wide range of frequencies, it is the most popular oscillator
iii. The Hartley oscillator is Suitable for oscillations in RF (Radio-Frequency) range, up to 30MHZ

VIVA QUESTIONS:

1. Define an oscillator?
2. Define barkhausen criteria
3. Which type of feedback is employed in oscillators
4. Give applications for oscillators
5. What is the condition for sustained oscillations
6. Draw an oscillator circuit with feedback network given below.
7. What is the principle behind operation of a *HARTLEY* oscillator?
8. What are the advantages and disadvantages of *HARTLEY* oscillators?
9. Mention two essential conditions for a circuit to maintain oscillations[
10. Define an colpits oscillator?
11. Define barkhausen criteria
12. Which type of feedback is employed in oscillators
13. Give applications for oscillators
14. Which feedback used in oscillators?
15. Classify oscillators?
16. Which oscillators are AF oscillators?
17. Draw an oscillator circuit with feedback network given below.
18. What are the applications of LC oscillator?
19. What type of feedback is used in oscillators?
20. What is the loop gain of an oscillator?
21. What is the difference between amplifier and oscillator?
22. Plot the Amplitude response of BC 547 Oscillator with C_2 = 5 µF, i/p.
23. Plot the Amplitude response of 2N3904 Oscillator with R_1 = 4.1 K.
24. Plot the Amplitude response of 2N3904 Oscillator with R_2 = 9.4 K i/p.
25. Plot the Amplitude response of 2N3904 Oscillator R1 = 4.1 K, R_2 = 9.4 K with i/p.
26. Plot the Amplitude response of 2N3904 Oscillator with C_1 = 10 µF.
27. Plot the Amplitude response of CL100 Oscillator with C_2 = 2 µF with i/p.
28. Plot the Amplitude response of CL 100 Oscillator with R_1 = 2.1 K.
29. Plot the Amplitude response of CK 100 Oscillator with R_2 = 5.4 K i/p.

30. Plot the Amplitude response of 2N3904 Oscillator R1 = 2.1 K, R_2 = 2.4 K with i/p.

EXERCISE PROBLEMS:

1. Plot the Amplitude response of 2N3904 Oscillator with C_1 = 5 μF.
2. Plot the Amplitude response of 2N2222 Oscillator C2 = 5 μF with i/p.
3. Plot the Amplitude response of BC107 Oscillator with R_1 = 4.1 K.
4. Plot the Amplitude response of BC 547 Oscillator with R_2 = 9.4 K i/p.
5. Plot the Amplitude response of BC 548 Oscillator R1 = 4.1 K, R_2 = 9.4 K with i/p.
6. Plot the Amplitude response of BC 557 Oscillator with C_1 = 5 μF.

EXP NO:7

CLASS A POWER AMPLIFIER (Transformer less) (SOFTWARE)

AIM: To calculate the efficiency of Class A power amplifier.

SOFTWARE TOOL:
Multisim V 13.0

APPARATUS REQUIRED:

1. Function generator
2. Regulated power supply (0 - 30V)
3. CRO (0-20MHz) - 1No
4. Transistor (SL - 100) - 1No.
5. Resistors (20 KΩ, 100 Ω) - 1No.
6. Capacitor (10 µF) - 1No.

CIRCUIT DIAGRAM:

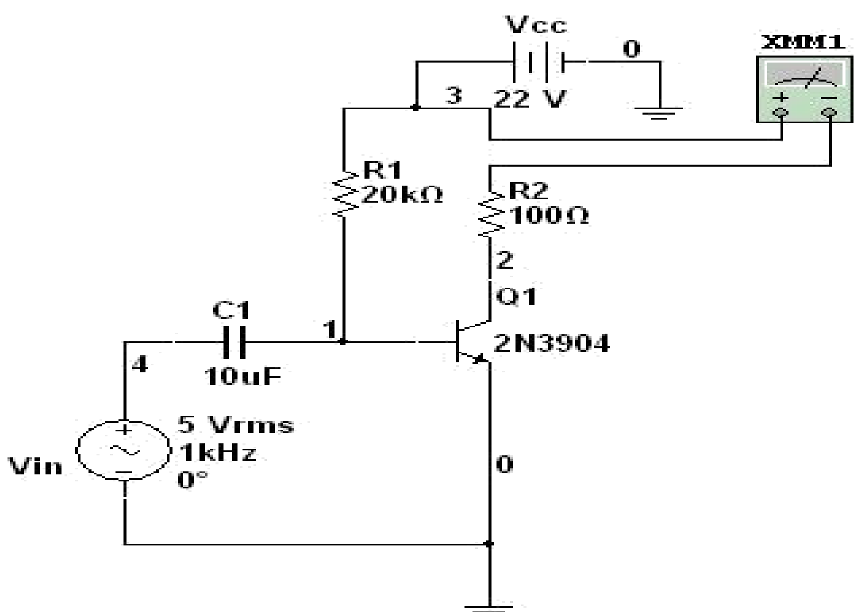


Fig 7.a: ClassA power amplifier circuit diagram

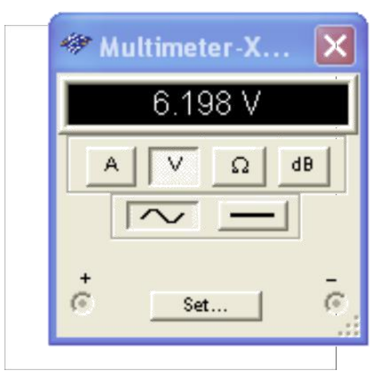
THEORY:

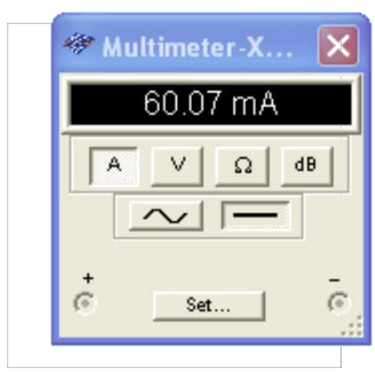
A Class A power amplifier is a type of amplifier that operates with the output transistor conducting for the entire cycle of the input signal, meaning it is always "on." This mode of operation results in high linearity, making Class A amplifiers ideal for applications where signal fidelity is crucial, such as in high-quality audio amplification. The continuous conduction ensures that the output is a faithful reproduction of the input signal, with minimal distortion. However, this comes at the cost of efficiency; Class A amplifiers are typically only 20-30% efficient, as the transistor dissipates a significant amount of power as heat even when no input signal is present. To manage the heat generated, Class A amplifiers often require large heatsinks. Despite their low efficiency, Class A amplifiers are valued in applications where the quality of the amplified signal is more important than power efficiency, providing a pure, undistorted output with excellent linearity across the entire frequency range.

PROCEDURE:

1. Connect the circuit as per the diagram.
2. Connect the function generator with sine wave of 0.3 V p-p as input at the input terminals of the circuit.
3. Note down the multi meter readings across the RL resistor. (V_{ac} and I_{dc})
4. Calculate the efficiency.

OBSERVATIONS:





From multimeter

V_{ac} = ______________ V

I_{ac} = _______________mA

Calculations:

$P_{dc} = V_{CC} \times I_{dc} =$

$P_{ac} = V_{ac}^2/R_L =$

$$efficiency \frac{P_{ac}}{P_{dc}} \times 100 =$$

EXP NO -7

CLASS A POWER AMPLIFIER (Transformer less) (Hardware)

AIM: To calculate the efficiency of Class A power amplifier.

COMPONENTS & EQUIPMENTS REQUIRED: -

S.No	Device	Range/Rating	QTY
1.	(a) DC supply voltage (b) Transistor (c) Capacitors (d) Resistors	12V CL100 100 µ F 22KΩ ,1KΩ 5.6K Ω ,100Ω,50Ω	1 1 1 1 1
2.	Signal generator	0.1Hz-1MHz	1
3.	CRO	0Hz-20MHz	1
4.	Connecting wires	5A	4

CIRCUIT DIAGRAM:

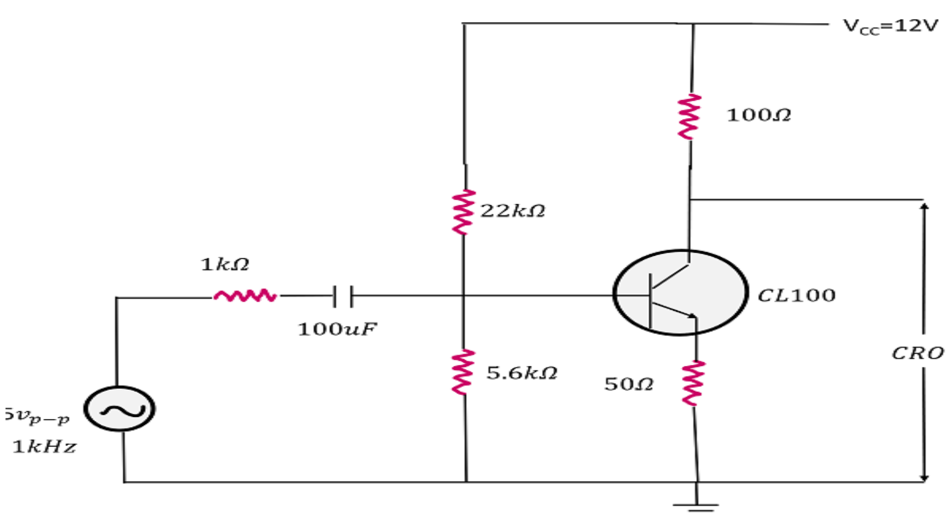


Fig: 7.B Class A power amplifier circuit diagram

PROCEDURE:

1. Connect the circuit as per the diagram.
2. Connect the function generator with sine wave of 0.3 V p-p as input at the input terminals of the circuit.
3. Note down the multi meter readings across the R_L resistor. (V_{ac} and I_{dc})
4. Calculate the efficiency.

OBSERVATIONS:

From Multimeter

V_{ac} = ________ V

I_{dc} = _________ mA

Calculations:

$P_{dc} = V_{CC} \times I_{dc} =$

$P_{ac} = V_{ac}^2/R_L =$

Efficiency $\eta = (P_{ac}/P_{dc}) * 100$

RESULT: The efficiency of Class A power amplifier is calculated.

APPLICATIONS:

a. The Class A Amplifier more suitable for outdoor musical systems, since the transistor reproduces the entire audio waveform without ever cutting off. As a result, the sound is very clear and more linear, that is, it contains much lower levels of distortion.
b. They are usually very large, heavy and they produce nearly 4-5 watts of heat energy per a watt of output. Therefore, they run very hot and need lots of ventilation. So they are not at all ideal for a car and rarely acceptable in a home.

VIVA QUESTIONS:

1. Differentiate between voltage amplifier and power amplifier?
2. Why power amplifiers are considered as large signal amplifier?
3. When does maximum power dissipation happen in this circuit?
4. What is the maximum theoretical efficiency?
5. Sketch wave form of output current with respective input signal.
6. What are the different types of class-A power amplifiers available?
7. What is the theoretical efficiency of the transformer coupled class-A power amplifier?
8. What is difference in AC, DC load line?
9. How do you locate the Q-point?
10. What are the applications of class-A power amplifier?
11. Define class A power amplifier?
12. Give the reason why class A power amplifier is called as directly coupled power amplifier?
13. What is the efficiency of class A power amplifier?
14. In class-A power amplifier, when the maximum power dissipation takes place in the transistor?
15. List out the different types of distortions?6.Define Harmonic distortion?
16. What are Class B, Class C and Class AB amplifiers and which type is used for what application?
17. Differentiate between voltage amplifier and power amplifier
18. Why power amplifiers are considered as large signal amplifier?
19. What is the theoretical efficiency of the transformer coupled class-A power amplifier?
20. Sketch wave form of output current with respective input signal.

21. Plot the frequency and amplitude response of BC 107 amplifier with C_1 = 5 μF.
22. Plot the frequency response of BC 107amplifier with C_2 = 5 μF with triangular i/p.
23. Plot the amplitude response of BC 107 amplifier with R_1 = 4.1 K.
24. Plot the amplitude response of BC 107 amplifier with R_2 = 9.4 K triangular i/p.
25. Plot frequency response of BC 107 amplifier R_1 = 4.1 K, R_2 = 9.4 K with square i/p.
26. Plot the frequency and amplitude response of BC 107 amplifier with C_1 = 5 μF.
27. Plot the frequency response of amplifier with C_2 = 5 μF, triangular i/p.
28. Plot the amplitude response of BC 107 amplifier with R_1 = 4.1 K.
29. Plot the amplitude response of BC107 amplifier with R_2 = 9.4 K triangular i/p.
30. Plot frequency response of BC107 R1 = 4.1 K, R_2 = 9.4 K with square i/p.

EXERCISE PROBLEMS:

1. Plot the frequency and amplitude response of BC107 amplifier with C_1 = 10 μF.
2. Plot the frequency response of BC 107amplifier with C_2 = 2 μF with i/p.
3. Plot the amplitude response of BC 107 amplifier with R_1 = 2.1 K.
4. Plot the amplitude response of BC 10 7amplifier with R_2 = 5.4 K i/p.
5. Plot frequency response of BC 107amplifier R_1 = 2.1 K, R_2 = 2.4 K with i/p.

EXP NO: 8

CLASS B COMPLEMENTARY SYMMETRY AMPLIFIER (SOFTWARE)

AIM: To observe the Cross over distortion of Class B complementary symmetry power amplifier.

SOFTWARE USED:

Multisim V13.0

APPARATUS REQUIRED:

1. Function generator - 1No.
2. Cathode Ray oscilloscope (CRO) - 1No.
3. Regulated power supply (0-30V) - 1No.
4. Transistor (2N3905, 2N3904) - 1No.
5. Resistor (1KΩ) - 1No.

CIRCUIT DIAGRAM:

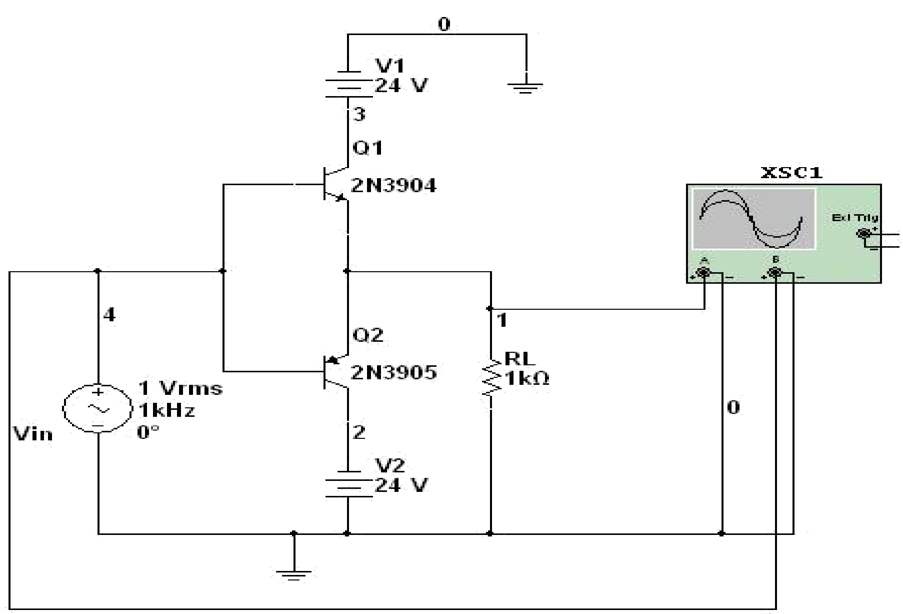


Fig: 8 a- Circuit diagram of Class B complementary symmetry power amplifier

THEORY:

A Class B complementary symmetry power amplifier is a type of amplifier designed to improve efficiency by using two complementary transistors (one NPN and one PNP) to amplify each half of the input signal. In this configuration, the NPN transistor conducts during the positive half of the input waveform, while the PNP transistor conducts during the negative half, resulting in each transistor being active for only half of the signal cycle. This push-pull arrangement reduces power dissipation, making Class B amplifiers much more efficient (up to 78.5%) compared to Class A amplifiers. However, the transition between the two transistors can introduce a small distortion known as crossover distortion, as there is a brief period when neither transistor is conducting. Despite this, Class B amplifiers are widely used in audio and RF applications where efficiency is important, and the crossover distortion can be minimized or corrected using various techniques, such as adding small biasing currents to the transistors, making them slightly conductive even when there is no input signal (Class AB operation).

PROCEDURE:

1. Switch ON the computer and open the multisim software.
2. Check whether the icons of the instruments are activated and enable.
3. Now connect the circuit using the designed values of each and every component.
4. Connect the function generator with sine wave of 30mV p-p as input at the input of terminals of the circuit.
5. Connect the Cathode Ray Oscilloscope (CRO) to the output terminals of the circuit.
6. Go to simulation button click it for simulation process.
7. Observe the cross over distortion in the CRO.

OBSERVATION:

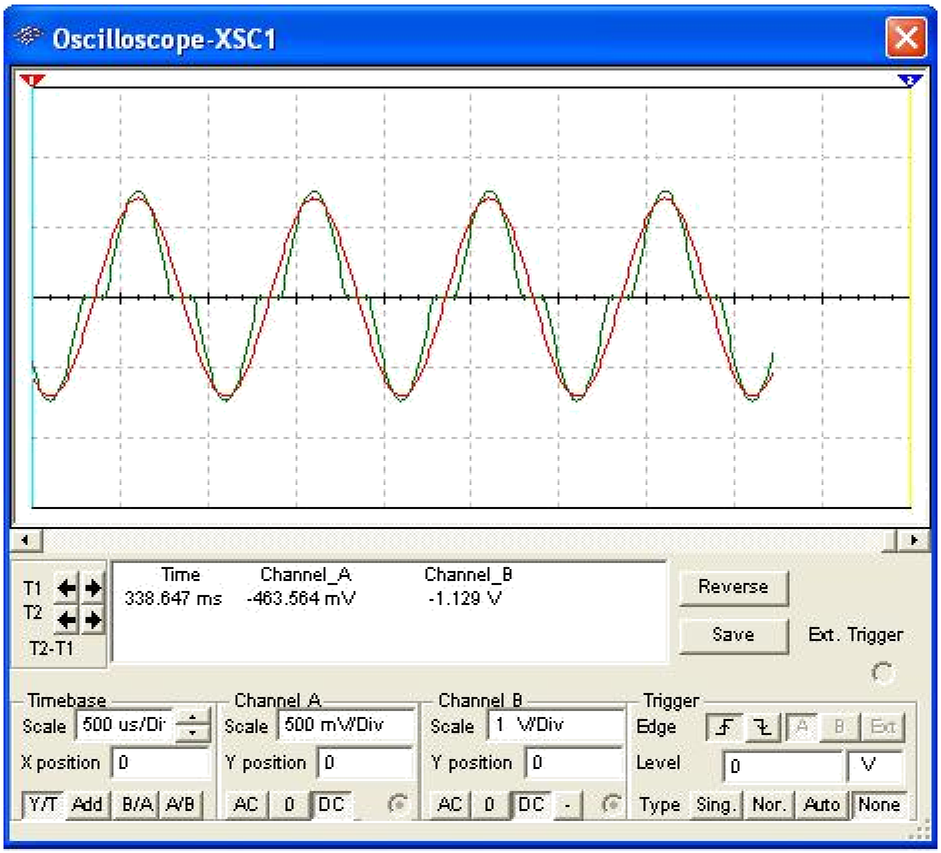


Fig: 8 b- Output waveform of Class B complementary symmetry power amplifier

EXP NO: 8

CLASS B COMPLEMENTARY SYMMETRY AMPLIFIER (HARDWARE)

AIM: To observe the Cross over distortion of Class B complementary symmetry power amplifier.

APPARATUS REQUIRED:

S.No	Device	Range/Rating	QTY
1.	(a) DC supply voltage (b) Transistor (c) Capacitors (d) Resistors	12V CL100 CK100 100 µ F 1KΩ	1 1 1 1 1
2.	Signal generator	0.1Hz-1MHz	1
3.	CRO	0Hz-20MHz	1
4.	Connecting wires	5A	4

CIRCUIT DIAGRAM:

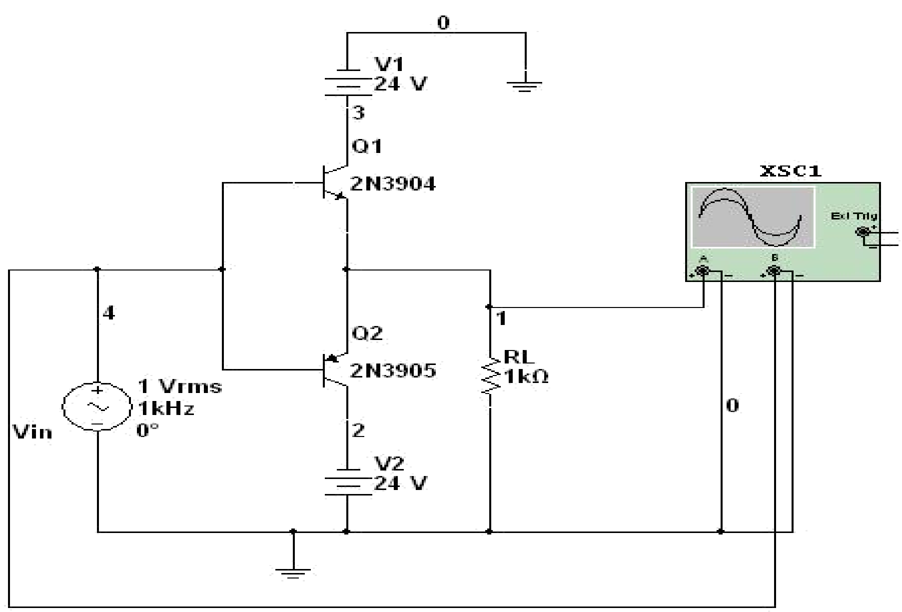


Fig: 10 c -Class B complementary symmetry power amplifier Circuit diagram

PROCEDURE: -

1. Connect the circuit diagram as shown in figure.
2. Adjust input signal amplitude in the function generator and observe an amplified voltage at the output without distortion.
3. By keeping input signal voltage, say at 50mV, vary the input signal frequency from 0 to 1MHz in steps as shown in tabular column and note the corresponding output voltages.

PRECAUTIONS:

Avoid loose connections and give proper input Voltage

TABULAR COLUMN:

Input = 50mV

Frequency(in Hz)	Output Voltage (V_o)		Gain $A_v=V_o/V_i$		Gain(in dB) $=20\log_{10}(V_o/V_i)$	
20						
40						
80						
100						
1K						
10k						
50k,100K						
1M						

RESULT: The efficiency of Class B power amplifier graph is plotted.

VIVA QUESTIONS:

1. Differentiate between voltage amplifier and power amplifier
2. Why power amplifiers are considered as large signal amplifier?
3. When does maximum power dissipation happen in this circuit?
4. What is the maximum theoretical efficiency?
5. Sketch wave form of output current with respective input signal.
6. What are the different types of class-Bpower amplifiers available?
7. What is the theoretical efficiency of the transformer coupled class-A power amplifier?
8. What is difference in AC, DC load line?
9. How do you locate the Q-point?
10. What are the applications of class-Bpower amplifier?
11. Define class –B power amplifier?
12. Give the reason why class -B power amplifier is called as directly coupled power amplifier?
13. What is the efficiency of class –B power amplifier?
14. In class-B power amplifier, when the maximum power dissipation takes place in the transistor?
15. List out the different types of distortions?
16. What are Class B, Class C and Class AB amplifiers and whichtype is used for what application?
17. Differentiate between voltage amplifier and power amplifier
18. Why power amplifiers are considered as large signal amplifier?
19. What is the theoretical efficiency of the transformer coupled class-B power amplifier?
20. Sketch wave form of output current with respective input signal.
21. How do you locate the Q-point?
22. When does maximum power dissipation happen in this circuit?
23. What are the different types of class-B power amplifiers available?
24. When does maximum power dissipation happen in this circuit?
25. What is the maximum theoretical efficiency?
26. Sketch wave form of output current with respective input signal.
27. What are the applications of class-B power amplifier?
28. Give the reason why class –B power amplifier is called as directly coupled power amplifier?
29. How do you locate the Q-point?
30. When does maximum power dissipation happen in this circuit?

EXERCISE PROBLEMS:

1. Plot the frequency and amplitude response of BC 107 amplifier with C_1 = 5 μF.
2. Plot the frequency response of BC547 amplifier with C_2 = 5 μF with triangular i/p.
3. Plot the amplitude response of BC 557 amplifier with R_1 = 4.1 K.
4. Plot the amplitude response of BC 548 amplifier with R_2 = 9.4 K triangular i/p.
5. Plot frequency response of SL 100 amplifier R_1 = 4.1 K, R_2 = 9.4 K with square i/p.
6. Plot the frequency and amplitude response of CL 100 amplifier with C_1 = 5 μF.
7. Plot the frequency response of amplifier with C_2 = 5 μF, triangular i/p.
8. Plot the amplitude response of BC 107 amplifier with R_1 = 4.1 K.
9. Plot the amplitude response of BC107 amplifier with R_2 = 9.4 K triangular i/p.
10. Plot frequency response of BC107 R1 = 4.1 K, R_2 = 9.4 K with square i/p.
11. Plot the frequency and amplitude response of BC107 amplifier with C_1 = 10 μF.
12. Plot the frequency response of BC 107amplifier with C_2 = 2 μF with i/p.
13. Plot the amplitude response of BC 107 amplifier with R_1 = 2.1 K.
14. Plot the amplitude response of BC 10 7amplifier with R_2 = 5.4 K i/p.
15. Plot frequency response of BC 107amplifier R_1 = 2.1 K, R_2 = 2.4 K with i/p.

EXP NO: 9

SINGLE TUNED AMPLIFIER (SOFTWARE)

PRELAB:

Study the operation and working principle Tuned amplifier.

OBJECTIVE:

To design single tuned amplifier using Multisim software and calculate the frequency response and bandwidth Apparatus:

1. Transistor – BC 107
2. Resistors – 2K(2), 4.7K
3. Capacitors – 10nF(2)
4. RPS
5. CRO

SOFTWARE TOOL: Multisim V 13.0

CIRCUIT DIAGRAM:

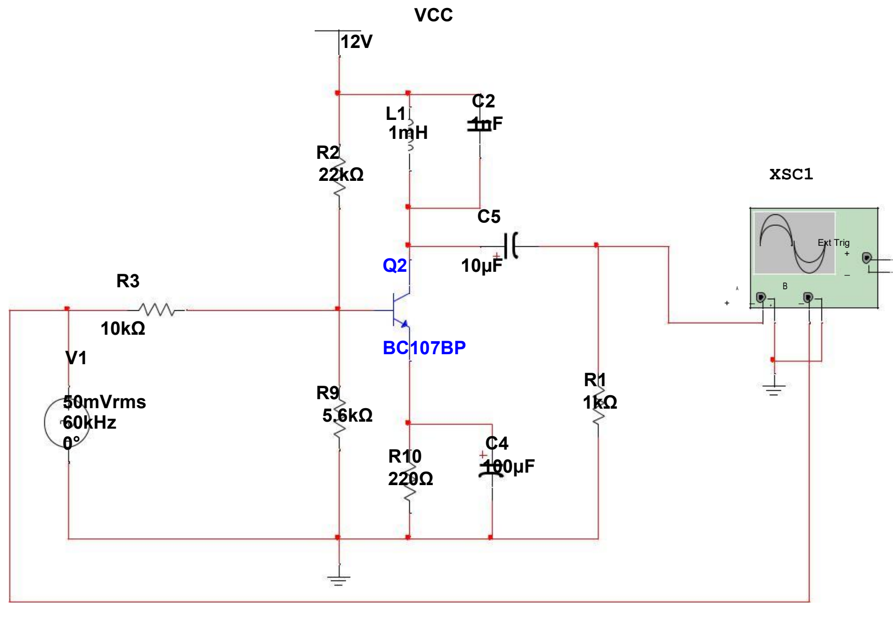


Fig: 9.a Single Tuned amplifier circuit diagram

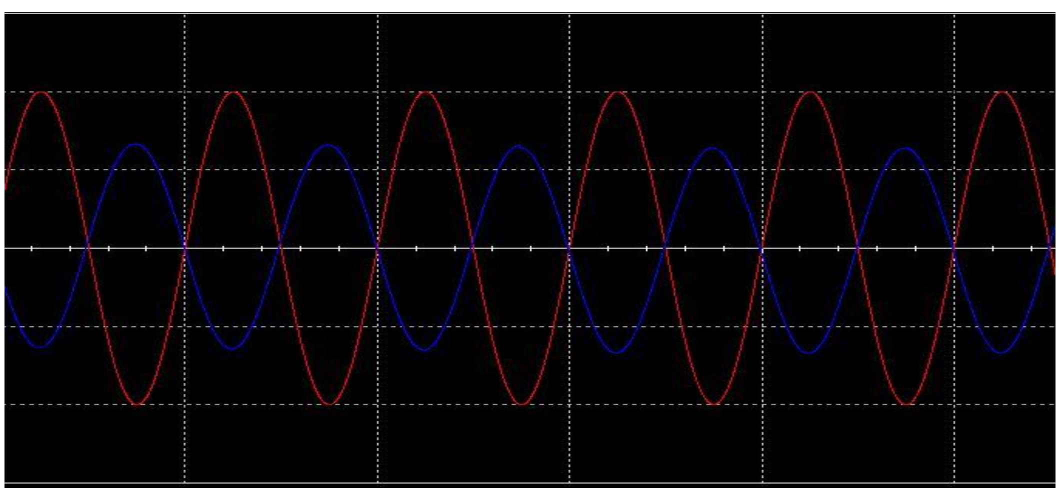
THEORY:

A single-tuned amplifier is a type of amplifier that is designed to selectively amplify a specific frequency or a narrow band of frequencies using a resonant circuit, typically consisting of an inductor (L) and a capacitor (C) in parallel. The resonant circuit, also known as a tuned circuit or LC tank, is placed either in the load or in the feedback network of the amplifier. At resonance, the impedance of the tuned circuit is at its maximum, allowing the amplifier to achieve maximum gain at the desired frequency while attenuating frequencies outside the band. Single-tuned amplifiers are commonly used in radio frequency (RF) applications, such as in radio receivers and transmitters, where they help in selecting and amplifying signals of a specific frequency while rejecting unwanted signals. However, the bandwidth of a single-tuned amplifier is relatively narrow, and its gain decreases rapidly as the frequency deviates from the resonant frequency.

PROCEDURE:

1. Open Multisim Software to design circuit
2. Select on New editor window and place the required component on the circuit window.
3. Make the connections using wire and check the connections and oscillator.
4. Go for simulation and using Run Key observe the output waveforms on CRO
5. Indicate the node names and go for AC Analysis with the output node
6. Observe the Ac Analysis and draw the magnitude response curve
7. Calculate the bandwidth of the amplifier

OBSERVATIONS/GRAPHS:



RESULT: -

1. Frequency response of single tuned Amplifier is plotted.
2. Gain = _______dB (maximum).
3. Bandwidth= f_H-f_L = _________Hz.

EXP NO:9

SINGLE TUNED VOLTAGE AMPLIFIER (Hardware)

AIM: -

Plot the frequency response of a single tuned amplifier.
Calculate gain.
Calculate bandwidth.

COMPONENTS & EQUIPMENTS REQUIRED: -

S.No	Device	Range/Rating	Qty
1.	(a) DC supply voltage	12V	1
	(b) BJT	BC107	1
	(c) Capacitors	10 F	2
		100 F	1
	(d) Resistors	220 ,22K ,1k	1
	(e) inductor (1mH)	5.6K ,10k	1
2.	Signal generator	0.1Hz-1MHz	1
3.	CRO	0Hz-20MHz	1
4.	Connecting wires	5A	4

CIRCUIT DIAGRAM:

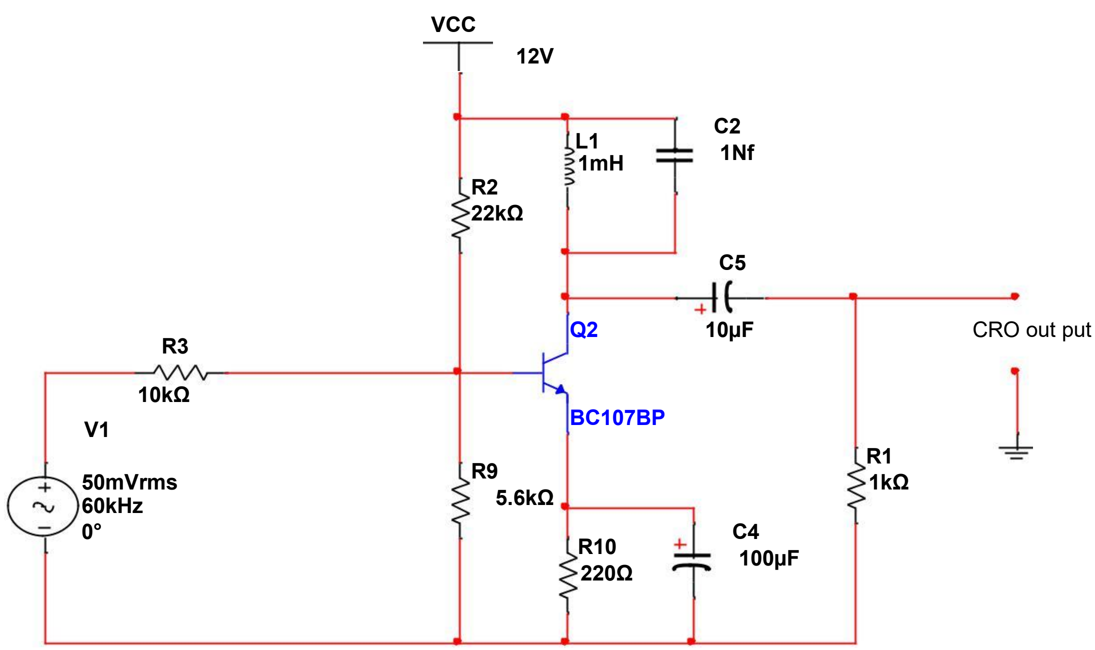


Fig: 9.c Single Tuned amplifier circuit diagram

PROCEDURE :

1. Connect the circuit diagram as shown in figure.
2. Set the input signal amplitude in the function generator and observe an amplified voltage at the output without distortion.
3. By keeping input signal voltage, say at 50mV, vary the input signal frequency from 0 to 1MHz in steps as shown in tabular column and note the corresponding output voltages.

PRECAUTIONS:

1. Avoid loose connections and give proper input Voltage

TABULAR COLUMN:

Input = 50mV

Frequency (in Hz)	Output Voltage (V_o)	Gain $A_v=V_o/V_i$	Gain (in dB) = $20\log_{10}(V_o/V_i)$
20			
40			
80			
100			
500			
1000			
5000			
10K			

RESULT:

1. Frequency response of single Tuned voltage amplifier is plotted.
2. Gain = _______dB (maximum).
3. Bandwidth= f_H-f_L = _________Hz.

VIVA QUESTIONS:

1. What is a tuned amplifier?
2. Define Q-factor?
3. What is selectivity?
4. Is tuned amplifier a hallow band or wide band amplifier?
5. give the applications for tuned amplifier?
6. Is tuned amplifier a hallow band or wide band amplifier?
7. What is the type of capacitor used in RC coupled amplifier for a) coupling two phases b)by pass emitter
8. What is signal source used for experiment of an RC coupled amplifier and how much maximum voltage it could give
9. What is the pin configuration on bread board used in the lab
10. How do you determine AC power output in class A amplifier i.e., do you measure current or voltage and how
11. How much current do you pass through reference zener in series regulated power supply experiment
12. In shunt regulator how is the value of resistor between base and emitter of shunt transistor determined
13. How do you determine Q of oil used in tuned amplifier experiment
14. What is a tuned amplifier
15. Define Q-factor
16. What is selectivity?
17. Is tuned amplifier a hallow band or wide band amplifier
18. give the applications for tuned amplifier
19. Is tuned amplifier a hallow band or wide band amplifier
20. What is the type of capacitor used in RC coupled amplifier for a) coupling two phases b)by pass emitter
21. What is signal source used for experiment of an RC coupled amplifier and how much maximum voltage it could give
22. What is the pin configuration on bread board used in the lab
23. How do you determine AC power output in class A amplifier i.e., do you measure current or voltage and how
24. How much current do you pass through reference zener in series regulated power supply experiment
25. In shunt regulator how is the value of resistor between base and emitter of shunt transistor determined
26. How do you determine Q of oil used in tuned amplifier experiment
27. What is the type of capacitor used in RC coupled amplifier for a) coupling two phases b)by pass emitter
28. What is signal source used for experiment of an RC coupled amplifier and how voltage it could give?
29. What is the pin configuration on bread board used in the lab
30. How do you determine AC power output in class A amplifier i.e., do you measure current or voltage and how?

EXERCISE PROBLEMS:

1. Plot the frequency response of BC 107 amplifier with C_1 = 5 µF.
2. Plot the frequency response of BC 547 amplifier with C_2 = 5 µF with i/p.
3. Plot the frequency response of BC 2N2222 amplifier with R_1 = 4.1 K.
4. Plot the frequency response of BC 107 amplifier with R_2 = 9.4 K i/p.
5. Plot frequency response of BC 107 amplifier R_1 = 4.1 K, R_2 = 9.4 K with i/p.
6. Plot the frequency and amplitude response of BC 107 amplifier with C_1 = 5 µF.
7. Plot the frequency response of amplifier with C_2 = 5 µF, i/p.
8. Plot the frequency response of BC 107 amplifier with R_1 = 4.1 K.
9. Plot the frequency response of BC107 amplifier with R_2 = 9.4 K i/p.
10. Plot frequency response of BC107 R1 = 4.1 K, R_2 = 9.4 K with i/p.
11. Plot the frequency and amplitude response of BC107 amplifier with C_1 = 10 µF.
12. Plot the frequency response of BC 107amplifier with C_2 = 2 µF with i/p.
13. Plot the frequency response of BC 107 amplifier with R_1 = 2.1 K.
14. Plot the frequency response of BC 10 7amplifier with R_2 = 5.4 K i/p.
15. Plot frequency response of BC 107amplifier R_1 = 2.1 K, R_2 = 2.4 K with i/p.
16. Plot the frequency and amplitude response of BC 107amplifier with C_1 = 2 µF.
17. Plot the frequency response of amplifier with C_2 = 2 µF, i/p.
18. Plot the frequency response of BC 107 amplifier with R_1 = 2.1 K.
19. Plot the frequency response of BC 107 amplifier with R_2 = 2.4 K i/p.
20. Plot frequency response of R1 = 2.1 K, R_2 = 9.4 K with i/p.
21. Plot the frequency response of BC 107 amplifier with C_1 = 5 µF.
22. Plot the frequency response of BC 107 amplifier with C_2 = 5 µF with i/p.
23. Plot the frequency response of BC 107 amplifier with R_1 = 4.1 K.
24. Plot the frequency response BC 107 amplifier with R_2 = 9.4 K i/p.
25. Plot frequency response of BC 107amplifier R_1 = 4.1 K, R_2 = 9.4 K with i/p.
26. Plot the frequency and amplitude response of BC 107 amplifier with C_1 = 5 µF.
27. Plot the frequency response of amplifier with C_2 = 5 µF, i/p.
28. Plot the frequency response of BC107 amplifier with R_1 = 4.1 K.
29. Plot the frequency response of BC 107 amplifier with R_2 = 9.4 K i/p.
30. .Plot frequency response of R_1 = 4.1 K, R_2 = 9.4 K with i/p

APPLICATIONS:

(a) Intermediate frequency (IF) amplifier in a super heterodyne receiver;

(b) very narrow-band IF amplifier in a spectrum analyzer;

(c) IF amplifier in a satellite transponder;

(d) RF amplifiers in receivers;

EXPT NO: 10

MONOSTABLE MULTIVIBRATOR (SOFTWARE)

AIM: To observe the waveforms of Monostable Multivibrator at base and collector of the transistors and find the Gate Width

APPARATUS REQUIRED

1. Transistor (BC107B) - 2No's.
2. Resistor R_1 = 10kΩ, R = 12.5kΩ
3. Capacitor C = 100nF
4. CRO probes & CRO
5. Bread Board Trainer
6. Connecting wires.

SOFTWARE TOOL: Multisim 13.0

CIRCUIT DIAGRAM :

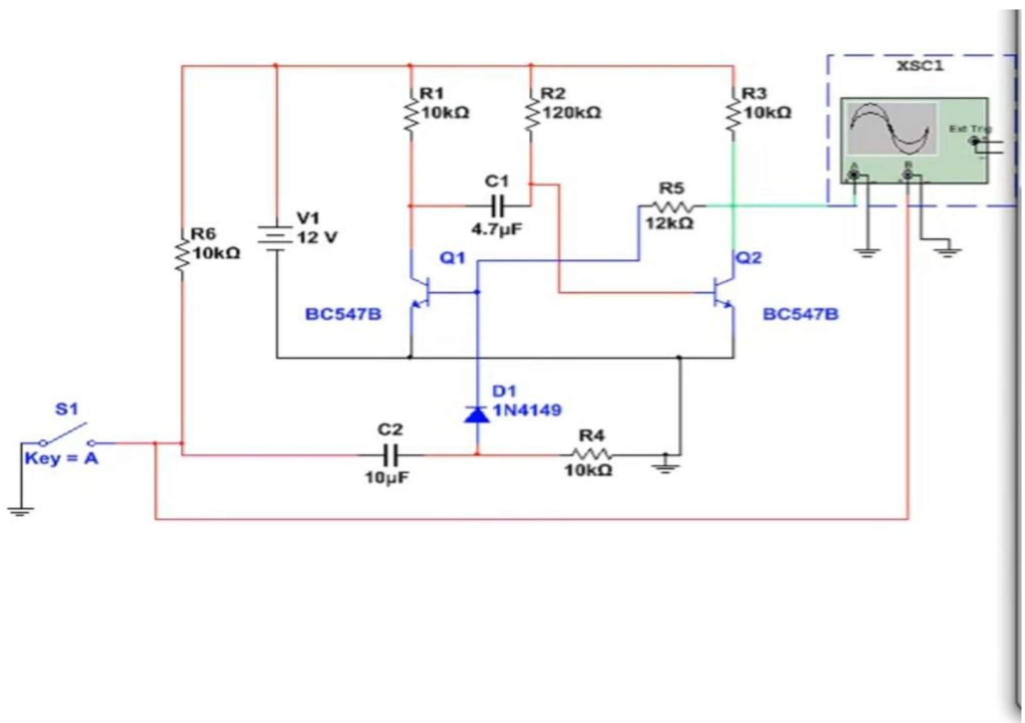


Fig 10.a: Circuit diagram of monostable multivibrator

THEORY:

A monostable multivibrator, also known as a one-shot pulse generator, is a type of electronic circuit that generates a single output pulse of fixed duration in response to a triggering input pulse. The circuit has one stable state and one quasi-stable state. When the circuit is in its stable state, it remains there indefinitely until a trigger pulse is applied. Upon receiving the trigger, the circuit temporarily switches to its quasi-stable state, where it generates a pulse with a duration determined by external components, typically a resistor and a capacitor. After this period, the circuit automatically returns to its stable state. The pulse width is governed by the RC time constant, allowing for precise timing applications. Monostable multivibrators are widely used in timing applications, pulse width modulation, and digital signal processing where a specific time delay or single pulse generation is required.

PROCEDURE:

1. Open Multisim Software to design Monostable mutivibrator circuit
2. Select on New editor window and place the required component Monostable mutivibrator on the circuit window.
3. Make the connections using wire and check the connections and oscillator.
4. Go for simulation and using Run Key observe the output waveforms on CRO
5. Indicate the node names and go for AC Analysis with the output node

6.Observe the output waveform .

OBSERVATIONS/GRAPHS:

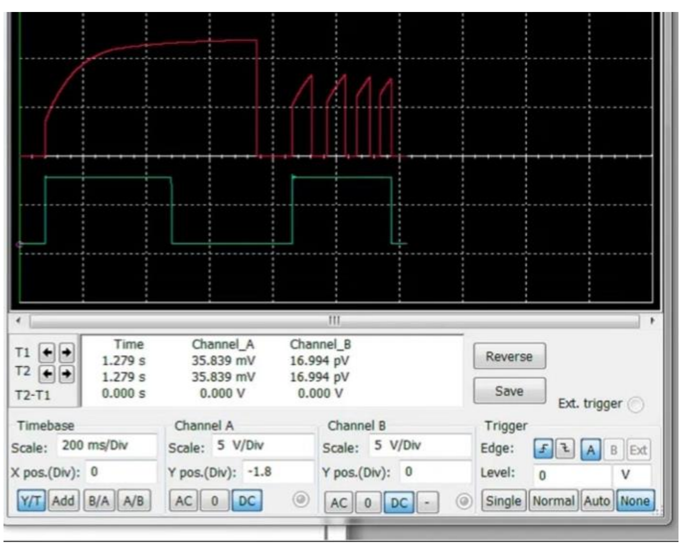


Fig:10b:output waveform of monostable multivibrator

RESULT: -

Output waveform of monostable multivibrator is plotted.

EXP NO: 10

MONOSTABLE MULTIVIBRATOR (HARDWARE)

AIM: To observe the waveforms of Monostable Multivibrator at base and collector of the transistors and find the Gate Width.

APPARATUS REQUIRED:

7. Transistor (BC107B) - 2No's.
8. Resistor R_1 = 10kΩ, R = 12.5kΩ
9. Capacitor C = 100nF
10. CRO probes & CRO
11. Bread Board Trainer
12. Connecting wires.

CIRCUIT DIAGRAM:

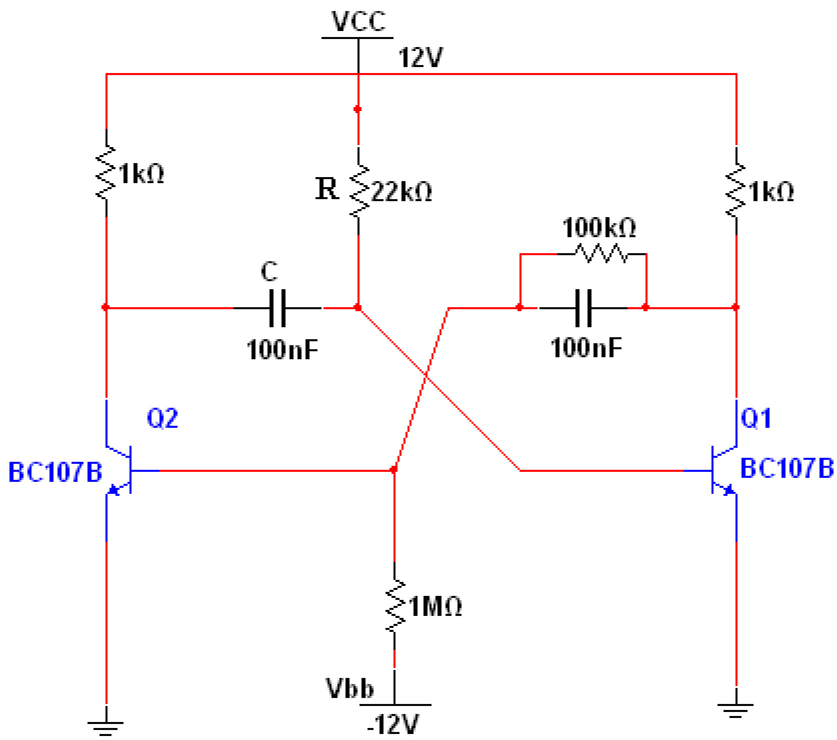


Fig: 10 a– Monostable multivibrator Circuit diagram

TRIGGER CIRCUIT:

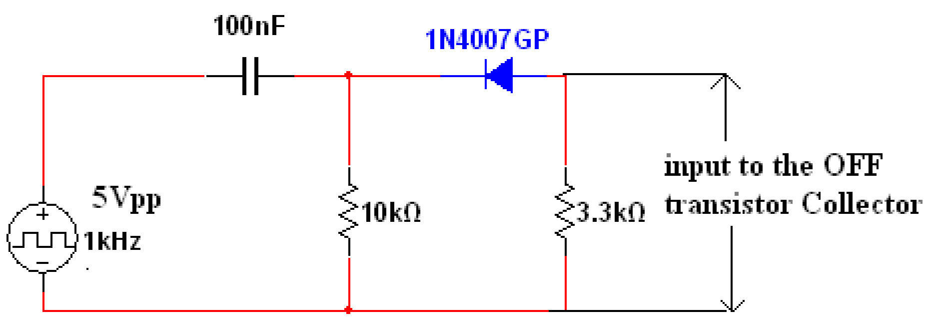


Fig: 10 b– Trigger Circuit diagram

PROCEDURE:

1. Connect the circuit as per circuit diagram.
2. Assume Q_1 – OFF, Q_2 – ON
3. Measure V_{B1}, Vc_1, V_{B2}, Vc_2.
4. Measure quasi stable state voltage and current Q_1 – ON & Q_2 – OFF
5. Connect the base of ON transistor and find out the Vc_1, Vc_2, V_{B1}, V_{B2},
6. Connect the base of OFF transistor and observe the waveform.

CALCULATIONS:

- Gate Width T = 0.69RC

EXPECTED WAVEFORMS:

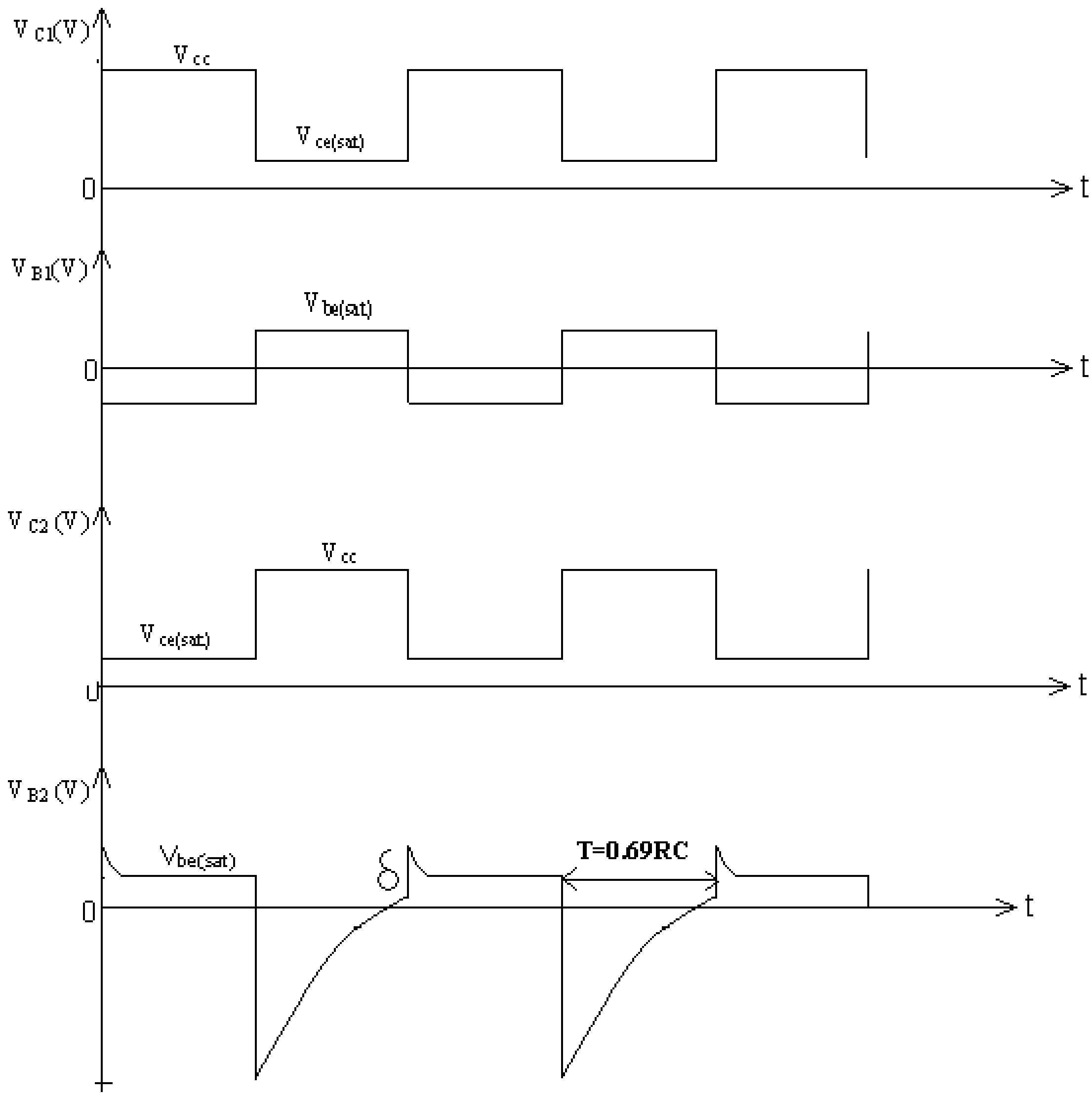


Fig: 10 c– Output waveforms of monostable multivibrator

RESULT: Monostable multivibrator wave forms are plotted.

VIVA 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.
4. What are applications of Monostable Multivibrator?
5. Why is a Monostable Multivibrator called a gating circuit?
6. Explain the waveform of VB1?
7. Describe the operation of the capacitor C3 in the circuit?
8. Why is the time period T also called Delay time?
9. Justify, Why Monostable Multivibrator is called one-shot circuit?
10. Why is the –ve voltage given at the base of Q1 transistor?
11. What is the no of quasi & stable states of Monostable Multivibrator?
12. What is a multivibrator? What is a quasi state?
13. What are applications of Monostable Multivibrator?
14. The monostable multivibrator is also called __, ___, __, ___ or ___?
15. A Monostable Multivibrator generates __ waveform?
16. Why is the time period T also called Delay time?
17. Justify, Why Monostable Multivibrator is called one-shot circuit?
18. In monostable multivibrator, the coupling elements are __?
19. What is a switching circuit?
20. What are the other names of monostable multivibrator?
21. What are the applications of monostable multivibrator? Design a Monostable multivibrator with Vcc=10V, R_1=2K, R_2=20K, Rc_1=Rc_2 =2K, C=0.1μF, For the pulse width of 1μsec
22. Design a One-shot multivibrator with Vcc=12V, R_1=2K, R_2=20K, Rc_1=Rc_2 =2K, C=0.1μF at Vin=4V
23. Design a Monostable multivibrator with Vcc=10V, R_1=5K, R_2=20K, Rc_1=Rc_2 =2K, C=0.1μF at Vin=8V

24. Design a Monostable multivibrator with Vcc=10V, R_1=5K, R_2=25K, Rc_1=Rc_2 =2K, C=10µF, For the pulse width of 1µsec
25. Design a One-shot multivibrator with Vcc=10V, R_1=3K, R_2=25K, Rc_1=Rc_2 =2K, C=10µF at Vin=5V
26. Design a Monostable with BFW10 and R_1=2K, R_2=20K, Rc_1=Rc_2 =2K, C=0.1µF at Vin=8V with triggering circuit having the values of R_T=1k, c_T=1µF
27. Design Monostable with BFW10 and R_1=5K, R_2=20K, Rc_1=Rc_2 =2K, C=0.1µF at Vin=8V
28. Design a One-shot multivibrator with Vcc=10V, R_1=2K, R_2=15K, Rc_1=Rc_2 =4K, C=0.1µF at Vin=10V
29. Design a Monostable multivibrator with Vcc=10V, R_1=2K, R_2=20K, Rc_1=Rc_2 =2K, C=0.1µF at Vin=8V with commutating capacitors C_1=C_2=1µF
30. Monostable multivibrator is also vcalled as __________.

REAL TIME APPLICATIONS OF MONOSTABLE MULTIVIBRATORS :

1. Monostable vibrators are used in analog systems to control an output signal frequency.
2. Synchronize the line and frame rate of television broadcasts.
3. Even moderate the tunes of different octaves with electronic organs.
4. Used to hold output voltages in its unstable state for a certain period of time.
5. The monostable multivibrator is used as delay and timing circuits.
6. It is also used for temporary memories.
7. It is often used to trigger another pulse generator.
8. It is used for regenerating old and worn out pulses.

EXP NO: 11

BISTABLE MULTIVIBRATOR(SOFTWARE)

AIM:

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

APPARATUS REQUIRED :

1. Multisim software version 13.0
2. Operating system windows XP

CIRCUIT DIAGRAM:

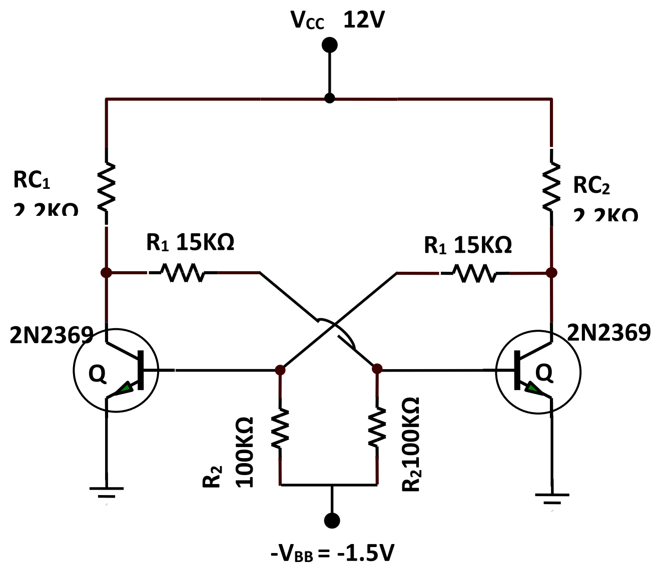


Fig 11.a circuit diagram of Bistable multivibrator

THEORY:

A bistable multivibrator, also known as a flip-flop, is an electronic circuit that has two stable states and can store a single bit of binary data. Unlike monostable or astable multivibrators, the bistable multivibrator remains in one of its two stable states indefinitely until an external input or trigger causes it to switch to the other state. It operates as a memory element or a digital storage device in various applications. The bistable multivibrator is typically implemented using two cross-coupled logic gates or transistors, where each state represents a different binary value (0 or 1). The transition between states is controlled by input signals, such as clock pulses or control inputs, which determine when the output should switch states. Common examples include the SR (Set-Reset) flip-flop, D (Data) flip-flop, and JK flip-flop.

PROCEDURE:

1. Open Multisim Software to design Bistable multivibrator circuit
2. Select on New editor window and place the required component Bistable multivibrator on the circuit window.
3. Make the connections using wire and check the connections and oscillator.
4. Go for simulation and using Run Key observe the output waveforms on CRO
5. Indicate the node names and go for AC Analysis with the output node
6. Observe the output waveform .

OBSERVATIONS/GRAPHS:

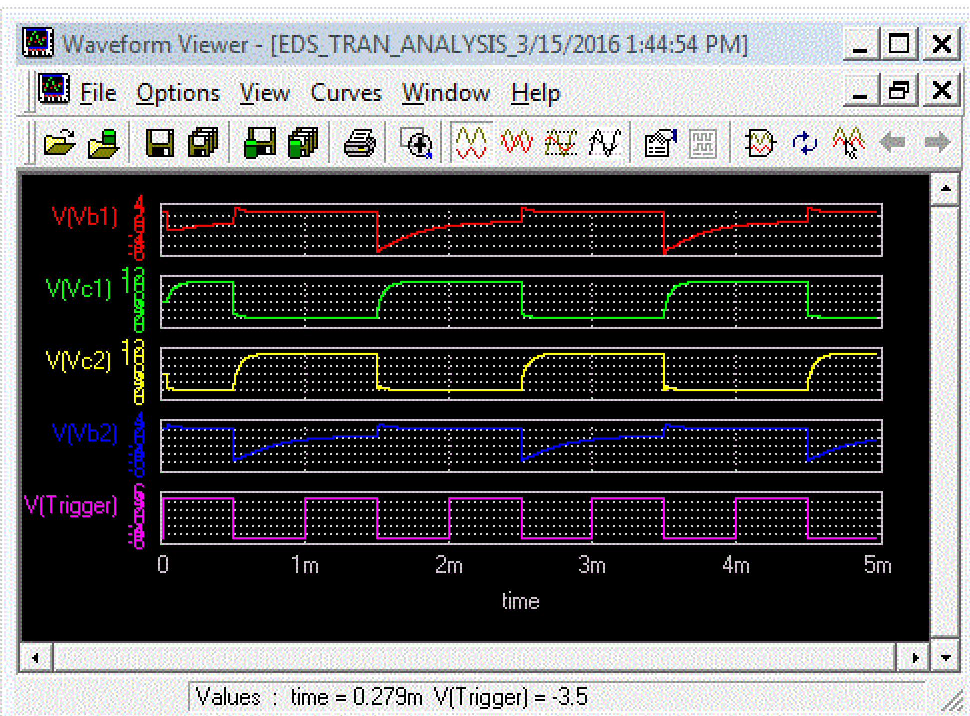


Fig 11.b. Output waveforms of Bistable multivibrator

RESULT: -

Output waveform of Bistable multivibrator is plotted.

EXP NO: 11

BISTABLE MULTIVIBRATOR(HARDWARE)

AIM:

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

COMPONENTS & EQUIPMENTS REQUIRED :

S.No	Device	Range/Rating	Qty
1.	(a) DC supply voltage (b) BJT (c) Resistors	12V 2N2369 2.2K Ω ,15KΩ ,100KΩ	1 2 Each 2NO
2.	Signal generator	0.1Hz-1MHz	1
3.	CRO	0Hz-20MHz	1
4.	Connecting wires	5A	4

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.1. The transistor Q_1 and Q_2 are n-p-n transistors. They are coupled to each other as shown in figure 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 dropI_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:

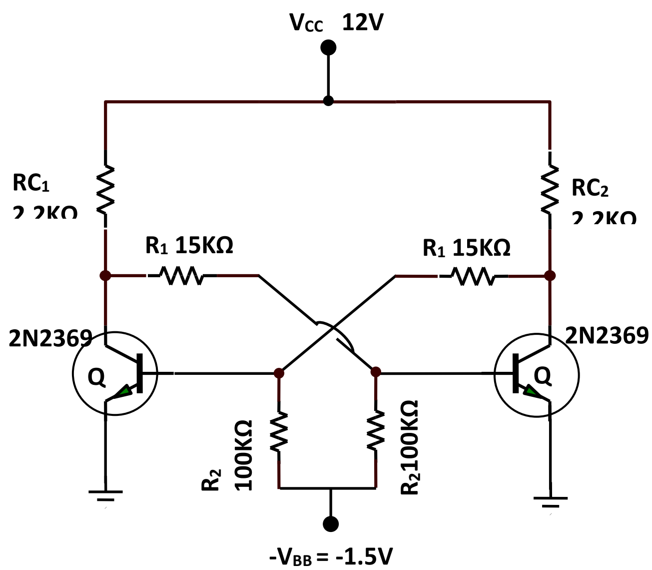


Fig 11.a circuit diagram of Bistable multivibrator

PROCEDURE:

1. Connect the circuit as shown in figure 2.
2. Observe the waveforms at V_{BE1}, V_{BE2}, V_{CE1}, V_{CE2} using CRO &multimeter
3. Observe the voltages at V_{C1} and V_{C2}.
4. If $V_{C1}= V_{CE(Sat)}$ and $V_{C2}=V_{CC}$(Approximately) then Q1 is 'ON' and Q2 is 'OFF' Otherwise $V_{C1}= V_{CC}$ (Approximately) and $V_{C2}= V_{CE(Sat)}$) then Q1 is'OFF' and Q2 is'ON'.
5. Observe which transistor is in ON state and which transistor is in OFF state. and observe the voltages V_{C1}, V_{C2}, V_{B1}, and V_{B2}.
6. Apply –ve triggering at the base of the ON transistor and observe the voltages V_{C1}, V_{C2}, V_{B1}, and V_{B2}.

OBSERVATIONS:

Before Triggering : When Q1 is 'ON' and Q2 is 'OFF'

	V_{BE1}	V_{BE2}	V_{CE1}	V_{CE2}
Stable state Voltages				

After Triggering: When Q1 is'OFF' and Q2 is'ON'.

	V_{BE1}	V_{BE2}	V_{CE1}	V_{CE2}
Stable state Voltages				

RESULTS:

Conclusion can be made on which transistor is 'ON' and which transistor is 'OFF' before triggering and after triggering.

DESIGN PROBLEMS:

1. Design a Bistable multivibrator with Vcc=10V, R_1=2K, R_2=20K, Rc_1=Rc_2 =2K, C=0.1μF at Vin=8V
2. Design a Fixed Bias Binary multivibrator with Vcc=12V, R_1=2K, R_2=20K, Rc_1=Rc_2 =2K, C=0.1μF at Vin=4V
3. Design a Bistable multivibrator with Vcc=10V, R_1=5K, R_2=20K, Rc_1=Rc_2 =2K, C=0.1μF at Vin=8V
4. Design a Bistable multivibrator with Vcc=10V, R_1=2K, R_2=15K, Rc_1=Rc_2 =4K, C=0.1μF at Vin=10V
5. Design a Fixed Bias Binary multivibrator with Vcc=10V, R_1=3K, R_2=25K, Rc_1=Rc_2 =2K, C=10μF at Vin=5V
6. Design a flip flop or Binary with BFW10 and R_1=2K, R_2=20K, Rc_1=Rc_2 =2K, C=0.1μF at Vin=8V
7. Design a flip flop or Binary with BFW10 and R_1=5K, R_2=20K, Rc_1=Rc_2 =2K, C=0.1μF at Vin=8V
8. Design a Bistable multivibrator with Vcc=10V, R_1=2K, R_2=15K, Rc_1=Rc_2 =4K, C=0.1μF at Vin=10V
9. Design a Bistable multivibrator with Vcc=10V, R_1=2K, R_2=20K, Rc_1=Rc_2 =2K, C=0.1μF at Vin=8V with commutating capacitors C_1=C_2=1μF

10. Design a Bistable multivibrator with Vcc=10V, R_1=2K, R_2=20K, Rc_1=Rc_2 =2K, C=10µF at Vin=8V with commutating capacitors C_1=C_2=0.1µF

VIVA 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.
4. What are the applications of a Bitable multivibrator?
5. Describe the operation of commutating capacitors?
6. Why is a Binary also called a flip-flop?
7. Mention the name of different kinds of triggering used in the circuit shown?
8. What are the disadvantages of direct coupled Binary?
9. How many types of unsymmetrical triggering are there?
10. What are catching diodes?
11. Which triggering is used in binary counting circuits?
12. What are the applications of a Bitable Multivibrator?
13. Describe the operation of commutating capacitors?
14. Commutating capacitors are also called as __ or ?
15. What is the meaning of a stable state in a multi-vibrator?
16. Mention the names of different kinds of triggering used in the circuit shown?
17. What are the disadvantages of direct coupled Binary?
18. The diodes used in a bistable multivibrator to maintain a constant output swing are called diodes?
19. The interval during which conduction transfers from one transistor to another is called the?
20. What are the coupling elements of a Bi-stable Multivibrator?
21. What are the other names of Bistable Multivibrator?
22. What is multivibrator?
23. Given the applications of bistable multivibrator?
24. Explain the multivibrator?
25. What are the other names of Bistable Multivibrator?
26. Describe the operation of commutating capacitors?
27. What is the meaning of a stable state in a multi-vibrator?

28. Mention the names of different kinds of triggering used in the circuit shown?
29. What are the disadvantages of direct coupled Binary?
30. The diodes used in a bistable multivibrator to maintain a constant output swing are called __ diodes?

APPLICATIONS OF BISTABLE MULTIVIBRATORS :

1. The bistable multivibrator or Flip Flop is of great importance in digital operation in computers, digital communications.
2. It is also used for reversing to the supply to a given circuit or change supply to two circuit at regular intervals.

EXP NO: 12

ASTABLE MULTIVIBRATOR (SOFTWARE)

AIM :

To design and simulate an Astable Multivibrator to generate clock pulse for a give frequency and obtain the wave forms and test its performance.

COMPONENTS REQUIRED:

1.	Resistors	1KΩ,	2 No.s
		4.7KΩ	2 No.s
2.	Capacitors	0.1 μf	2 No.s
3.	Transistors	2N2369	2 No.s

SOFTWARE TOOL :

MULTISIM 13.0

CIRCUIT DIAGRAM :

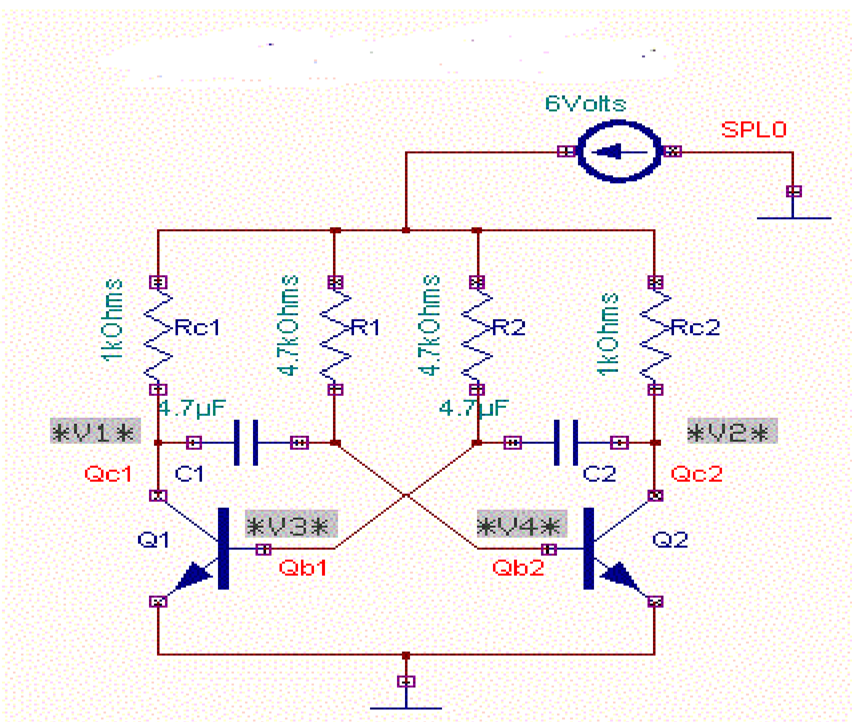


Fig12.a :circuit diagram of Astable multivibrator

THEORY:

An astable multivibrator is a type of electronic oscillator circuit that continuously switches between its two unstable states, generating a square wave output with a defined frequency and duty cycle. Unlike monostable and bistable multivibrators, which require external triggering, the astable multivibrator oscillates autonomously, producing a periodic waveform without the need for an external input. The circuit typically consists of an amplifier (such as a transistor or operational amplifier) and a feedback network that includes resistors and capacitors.

PROCEDURE:

1. Open Multisim Software to design Astable multivibrator circuit
2. Select on New editor window and place the required component Astable multivibrator on the circuit window.
3. Make the connections using wire and check the connections and oscillator.
4. Go for simulation and using Run Key observe the output waveforms on CRO
5. Indicate the node names and go for AC Analysis with the output node
7. Observe the output waveform .

OBSERVATION RESULTS:

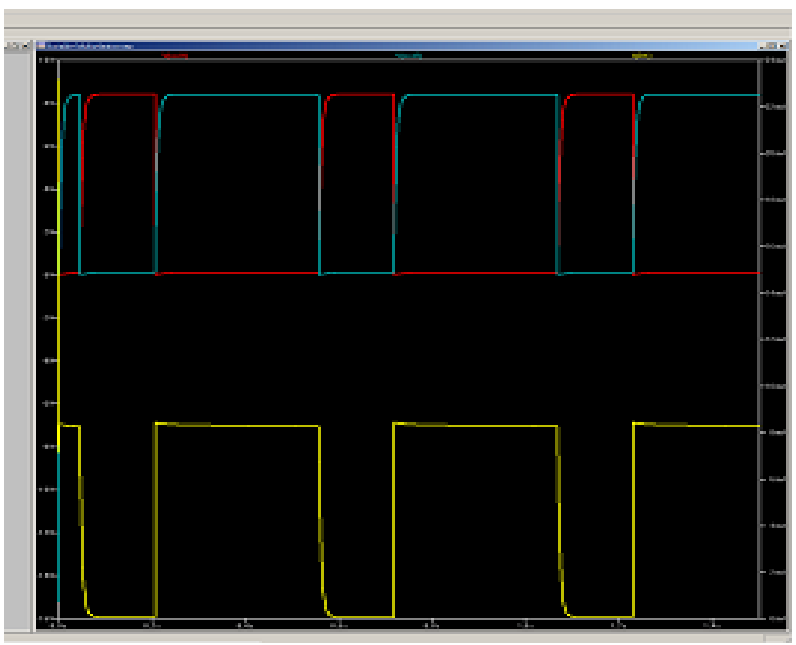


Fig12.b : output waveform of Astable multivibrator

RESULTS :

Output Waveforms of Astable multivibrator is plotted.

EXP NO: 12

ASTABLE MULTIVIBRATOR (HARDWARE)

AIM :

To design an Astable Multivibrator to generate clock pulse for a give frequency and obtain the wave forms and test its performance.

COMPONENTS & EQUIPMENTS REQUIRED: -

S.No	Device	Range/Rating	QTY
1.	(a) DC supply voltage	12V	1
	(b) Transistor	BC107	1
	(c) Capacitors	10 µ F	2
	(d) Resistors	15KΩ	2
		4.7KΩ	2
2.	Signal generator	0.1Hz-1MHz	1
3.	CRO	0Hz-20MHz	1
4.	Connecting wires	5A	4

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.

CIRCUIT DIAGRAM :

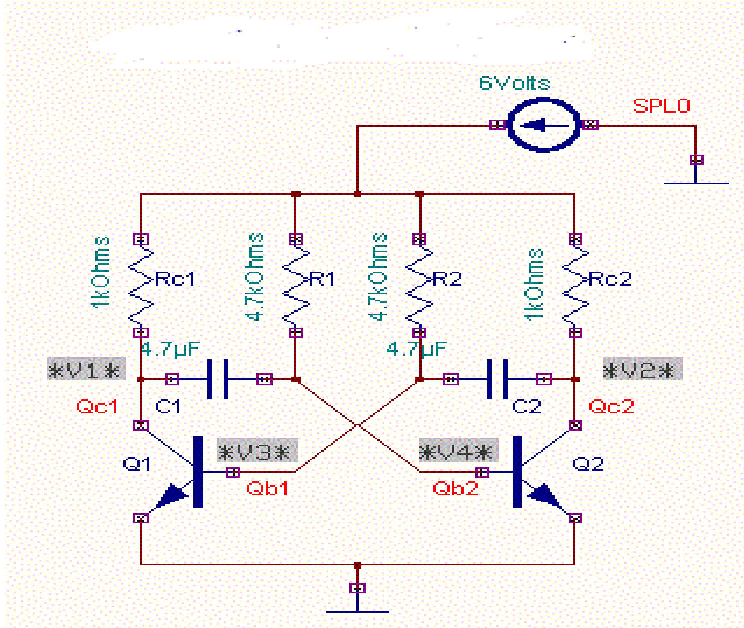


Fig 12.a circuit diagram of Astable multivibrator

THEORY :

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

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

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

OBSERVATION TABLE:

S.No	Theoretical Time Period ($T(T_{ON} + T_{OFF})$) = 1.38RC Calculation	Practical Time Period($T = T_{ON} + T_{OFF}$) Calculation
1		

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

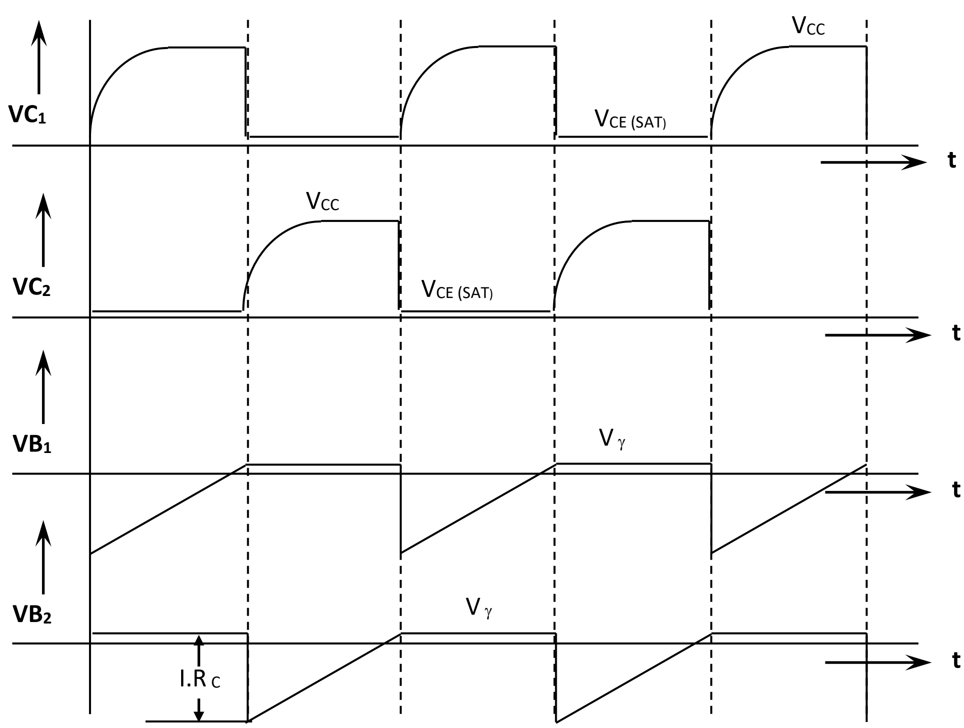


Fig 12.b:Expected waveform of Astable multivibrator

OBSERVATIONS:

T_{ON} (Theoritical) = 0.69RC =
T_{OFF} (Theoritical) = 0.69RC =
$T(T_{ON} + T_{OFF})$ (Theoritical) = 1.38RC =
T_{ON} (practical) =
T_{OFF} (practical) =
$T(T_{ON} + T_{OFF})$ (practical) =

RESULT:

Conclusion can be made on time period of the output waveforms of the Astable multivibrator theoretically and practically and output waveforms of the multi vibrator are identical or not WHEN compared with the theoretical wave forms

DESIGN PROBLEMS:

1. Design a Astable multivibrator with Vcc=10V, R_1=2K, R_2=20K, Rc_1=Rc_2 =2K, , C_1=C_2=0.1µF, For the time period of 1µsec
2. .Design a Square wave generator withVcc=10V, R_1=5K, R_2=15K, Rc_1=Rc_2 =5K, C_1=C_2=10 µF, For the time period of 5µsec
3. Design a Astable multivibrator with Vcc=10V, R_1=2K, R_2=10K, Rc_1=Rc_2 =2K, C_1=C_2=0.1µF, For the time period of 5µsec
4. Design a Square wave generator withVcc=10V, R_1=2K, R_2=20K, Rc_1=Rc_2 =2K, C_1=C_2=4.7 µF, For the time period of 1µsec
5. .Design a Astable multivibrator with Vcc=10V, R_1=2K, R_2=20K, Rc_1=Rc_2 =2K, , C_1=C_2=0.1µF, For the time period of 1µsec
6. Design a Square wave generator withVcc=10V, R_1=2K, R_2=20K, Rc_1=Rc_2 =2K, C_1=C_2=4.7 µF, For the time period of 1µsec
7. .Design a Astable multivibrator with Vcc=10V, R_1=2K, R_2=20K, Rc_1=Rc_2 =2K, , C_1=C_2=0.1µF, For the time period of 1µsec
8. .Design a Square wave generator withVcc=10V, R_1=5K, R_2=15K, Rc_1=Rc_2 =5K, C_1=C_2=10 µF, For the time period of 5µsec
9. Design a Astable multivibrator with Vcc=10V, R_1=2K, R_2=10K, Rc_1=Rc_2 =2K, C_1=C_2=0.1µF, For the time period of 5µsec

10.Design a Square wave generator withVcc=10V, R_1=2K, R_2=20K, Rc_1=Rc_2 =2K, C_1=C_2=4.7 µF, For the time period of 1µsec.

VIVA QUESTIONS:

1. What is a switching circuit?
2. Justify that the Astable Multivibrator is a two stage RC coupled Amplifier using negative feedback.
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?
5. Is it possible to change time period of the waveform with out changing R& C? Support your answer?
6. Collector waveforms are observed with rounded edges. Explain?
7. Explain charging and discharging of capacitors in an Astable Multivibrator?
8. How can an Astable multivibrator be used as VCO?
9. Why do you get overshoots in the Base waveforms?
10. What are the applications of Astable Multivibrator?
11. How can Astable multivibrator be used as a voltage to frequency converter?
12. What is the formula for frequency of oscillations?
13. What are the other names of Astable multivibrator?
14. What are the other names of Astable multivibrator?
15. The smaller allowable interval between two triggers is called the __ of the flip-flop?
16. Explain charging and discharging of capacitors in an Astable Multivibrator?
17. How can an Astable multivibrator be used as VCO?
18. What are symmetrical triggering and unsymmetrical triggering?
19. What are the applications of Astable Multivibrator?
20. Which multivibrator has two quasi-stable states? What is duty cycle?
21. What is the formula for frequency of oscillations?
22. An astable multivibrator is used as a ___ generator?
23. Design R and C for a frequency of 2KHz of a symmetric Astable oscillator?
24. Define astable state?
25. Define quasi stable state?
26. Explain about the the astable multivibrator?

27. Explain the wave form the astable multivibrator?
28. Describe the operation of commutating capacitors?
29. What is multivibrator? explain the principle on which it works? why is it called a bonary?
30. What are the applications of a astable multivibrator?

REAL TIME APPLICATIONS :

1. Used in amateur radio equipment to receive and transmit radio signals.
2. Used in Morse code generators, timers, and systems that require a square wave, including television broadcasts and analog circuits.
3. Involve in radio gears to transmit and receive radio signals and also in time, morse code generators and some systems which require a square wave like analog integrated circuits and TV broadcasts.
4. The astable or free running multivibrator is used as a square wave frequency generator
5. As a timing oscillator or clock of a computer system.
6. It is also used for flashing lights, switching and power supply circuits.
7. Amateur radio equipment to receive and transmit radio signals.
8. Used in Morse code generators.
9. Timers.
10. Systems that require a square wave, including television broadcasts and analog circuits.

EXP NO: 13
SCHMITT TRIGGER USING TRANSISTOR(SOFTWARE)

AIM:

To design and simulate a Schmitt trigger and to observe the waveforms for a given UTP & LTP Values and test its performance.

COMPONENTS REQUIRED:

1. Resistors 5KΩ,3.3KΩ,27KΩ,14KΩ 25KΩ,2KΩ
2. Transistors 2N2369 – 2
3. Capacitors - 10µF

SOFTWARE TOOL :

Multisim 13.0

CIRCUIT DIAGRAM :

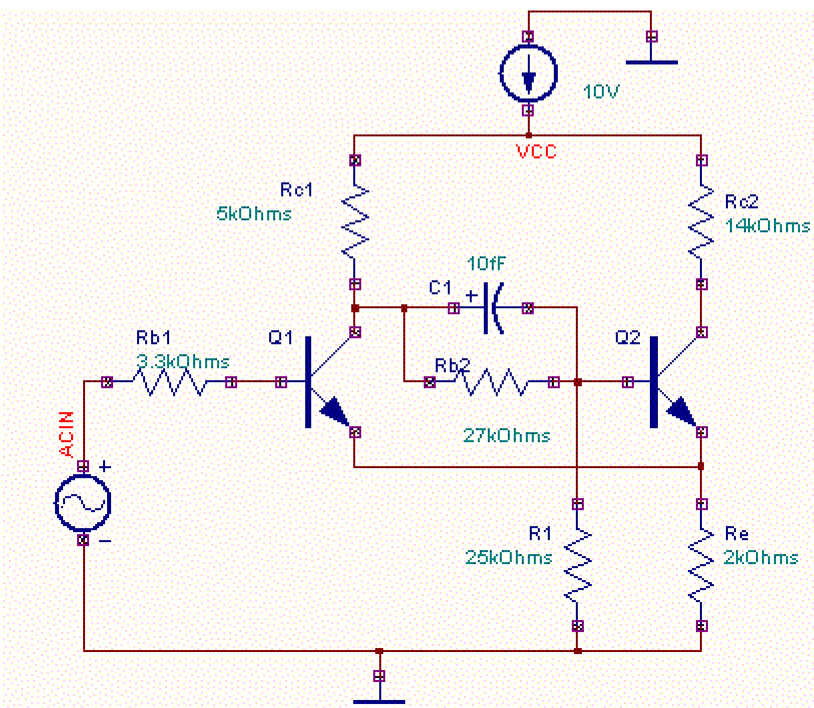


Fig 13.a:Circuit Diagram of Schmitt trigger

THEORY:

A Schmitt trigger is a type of electronic comparator with hysteresis, designed to provide a stable output when the input signal is noisy or slow-changing. It features two distinct threshold voltage levels: a higher threshold for transitioning from a low to a high output state and a lower threshold for transitioning back to a low state. This hysteresis ensures that the output switches states only when the input signal crosses these predefined thresholds, thus reducing the impact of noise and preventing false triggering due to small fluctuations around the threshold levels. The circuit typically uses positive feedback to achieve these thresholds, creating a more robust and stable switching behavior. Schmitt triggers are commonly implemented using operational amplifiers, logic gates, or dedicated ICs.

PROCEDURE:

1. Open Multisim Software to design Astable multivibrator circuit
2. Select on New editor window and place the required component Astable multivibrator on the circuit window.
3. Make the connections using wire and check the connections and oscillator.
4. Go for simulation and using Run Key observe the output waveforms on CRO
5. Indicate the node names and go for AC Analysis with the output node
6. Observe the output waveform .

OBSERVATION RESULTS:

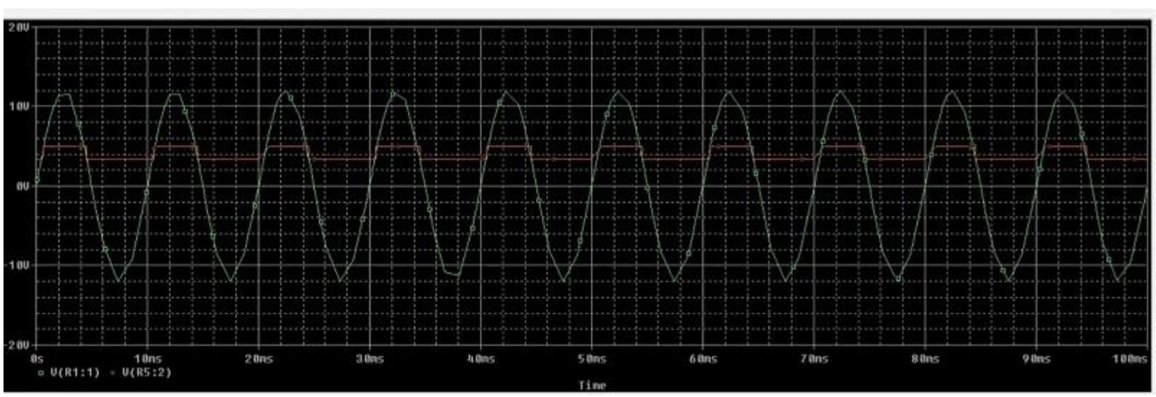


Fig 13.b:Output waveform of Schmitt trigger circuit

RESULTS:

Output waveform of Schmitt trigger is plotted.

EXP NO: 13

SCHMITT TRIGGER USING TRANSISTOR (HARDWARE)

AIM:
To design a Schmitt trigger and to observe the waveforms for a given UTP & LTP Values and test its performance.

COMPONENTS & EQUIPMENTS REQUIRED: -

S.No	Device	Range/Rating	QTY
1.	(a) DC supply voltage (b) Transistor (c) Capacitors (d) Resistors	12V BC107 10 µF 5KΩ ,14KΩ 3.3KΩ ,27KΩ, 25KΩ,2KΩ	1 1 1 1 1
2.	Signal generator	0.1Hz-1MHz	1
3.	CRO	0Hz-20MHz	1
4.	Connecting wires	5A	4

CIRCUIT DIAGRAM:

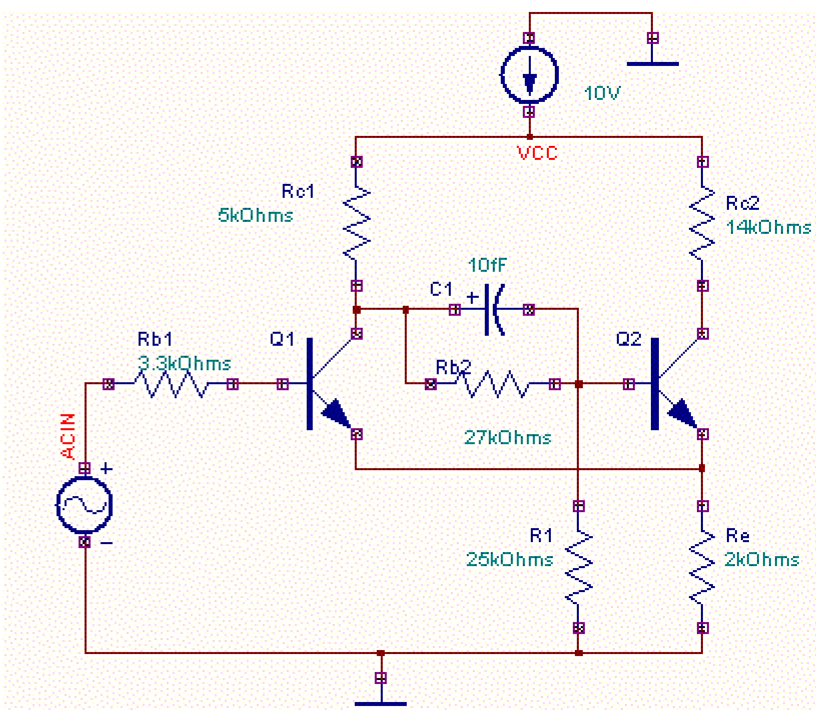


Fig 13.a: circuit diagram of Schmitt trigger using transistor

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_2continues 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:

Given UTP=5V,LTP=3V

$I_{C2} = 5mA$

$(Rc_2 + R_E) = V_{CC} / I_{C2}$

$U.T.P = V_{E2} = 5V$

$V_{E2} = (R_E \times V_{CC}) / (Rc_2+R_E)$

$I_2 = 0.1 \times I_{C2}$

$L.T.P = V_{E1} = 3V$

$R_2 = E_{R2i} / I_2 = V_{E1} / I_2 = L.T.P / I_2$

$Rc_1 = \{(R_E \times V_{CC}) / V_{E1}\} – R_E$

$I_{B2} = I_{C2} / hfe(min)$

$(V_{CC} - V_{E2}) / (R_1+R_{c1}))) = (V_{E2}/R_2)+I_{B2}$

$R_B = (h_{fe} \times R_E) / 10$

Find R_1, R_2, R_E, $Rc_1$and Rc_2 from the above equations

OBSERVATION TABLE:

	Theoritical Calculation			Practical Calculation
S.No	Upper threshold Point(U.T.P) $= + Vsat(+V_{CC}) * \frac{R2}{R1+R2}$	Lower threshold Point(L.T.P) $= - Vsat(-V_{CC}) * \frac{R2}{R1+R2}$	Hysistersis(V_H)= U.T.P - L.T.P	Hysistersis(V_H)= U.T.P - L.T.P
1				

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:

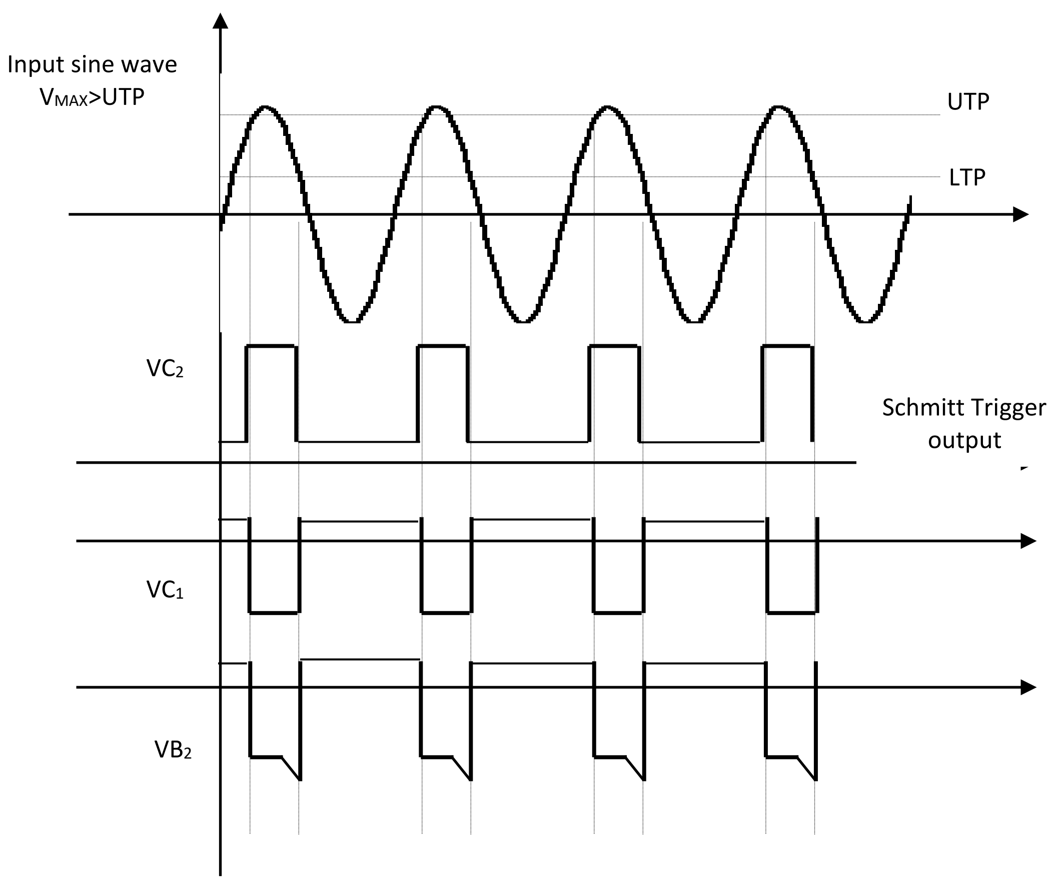


Fig 13.d: expected waveform of Schmitt trigger circuit

RESULTS :

Conclusion can be made on desined and practical values of U.T.P and L.T.P. and also made on output waveform of Schmitt trigger for the given sinusoidal input.

VIVA QUESTIONS:

1. Explain how a Schmitt trigger acts as a comparator?
2. Derive its expressions for UTP & LTP.
3. What are the applications of Schmitt Trigger?
4. Define hysteresis action?
5. Why is Schmitt Trigger called a squaring circuit?
6. What is UTP?
7. What is LTP?
8. What is the difference between a Binary and Schmitt Trigger?
9. Other names of Schmitt trigger.
10. Difference between comparator and Schmitt trigger.
11. Difference between Schmitt trigger and Multivibrator.
12. For good Schmitt trigger the hysteresis value always?
13. Determine UTP and LTP of a Schmitt trigger having the values , $R_1=R_2=18K$, $Rc_1=Rc_2=2.7K$, $R_E=1.5K$
14. Design a Schmitt Trigger using npn transistor for Vcc=12 V, UTP=3.5V, LTP=2.5V ,hfe=50,Ic2=2mA
15. Determine UTP and LTP of a Schmitt trigger having the values , $R_1=R_2=10K$, $Rc_1=Rc_2=3.3K$, $R_E=4.7K$
16. Design a Schmitt Trigger using npn transistor for Vcc=10 V, UTP=5.5V, LTP=2.5V ,hfe=50,Ic2=3mA
17. Design a Schmitt trigger for the Hysteresis of 3V
18. Determine UTP and LTP of a Schmitt trigger having the values , $R_1=R_2=18K$, $Rc_1=Rc_2=2.7K$, $R_E=1.5K$
19. Design a Schmitt Trigger using npn transistor for Vcc=12 V, UTP=5V, LTP=4.5V ,hfe=50,Ic2=1.5mA
20. Determine UTP and LTP of a Schmitt trigger having the values , $R_1=R_2=15K$, $Rc_1=Rc_2=3K$, $R_E=4K$
21. Design a Schmitt Trigger using npn transistor for Vcc=10 V, UTP=5.5V, LTP=2.5V ,hfe=50,Ic2=3mA
22. Design a Schmitt trigger for the Hysteresis of 7V
23. Determine UTP and LTP of a Schmitt trigger having the values , $R_1=R_2=20K$, $Rc_1=Rc_2=4.7K$, $R_E=1.5K$
24. Design a Schmitt Trigger using npn transistor for Vcc=12 V, UTP=5.5V, LTP=3.5V ,hfe=70,Ic2=5mA

25. Determine UTP and LTP of a Schmitt trigger having the values , $R_1=R_2=12K$, $Rc_1=Rc_2=3.3K$, $R_E=5.7K$
26. Design a Schmitt Trigger using npn transistor for Vcc=12 V, UTP=5.5V, LTP=2.5V ,hfe=50,Ic2=5mA
27. Design a Schmitt trigger for the Hysteresis of 5V
28. Determine UTP and LTP of a Schmitt trigger having the values , $R_1=R_2=20K$, $Rc_1=Rc_2=2.7K$, $R_E=3.5K$
29. Design a Schmitt Trigger using npn transistor for Vcc=12 V, UTP=7V, LTP=5.5V ,hfe=50,Ic2=3.5mA
30. Determine UTP and LTP of a Schmitt trigger having the values , $R_1=R_2=20K$, $Rc_1=Rc_2=3K$, $R_E=7K$

REAL TIME APPLICATIONS OF SCHMITT TRIGGER:

1. Squaring Circuit.
2. Sine-To-Square Comparator.
3. Amplitude comparator.
4. As Flip Flops.
5. A Comparator Circuit Which Converts Any Arbitrary Signal(Slope!=1) To Square.

EXP NO: 14

MILLER SWEEP CIRCUIT

AIM: To observe the characteristics of boot strap sweep generator

APPARATUS REQUIRED:

S.No	Device	Range/Rating	QTY
1.	(a) DC supply voltage (b) Transistor (c) Capacitors (d) Resistors	12V BC107 10µ F 1KΩ 22KΩ ,2KΩ	1 1 1 1 2 1
2.	Signal generator	0.1Hz-1MHz	1
3.	CRO	0Hz-20MHz	1
4.	Connecting wires	5A	4

CIRCUIT DIAGRAM:

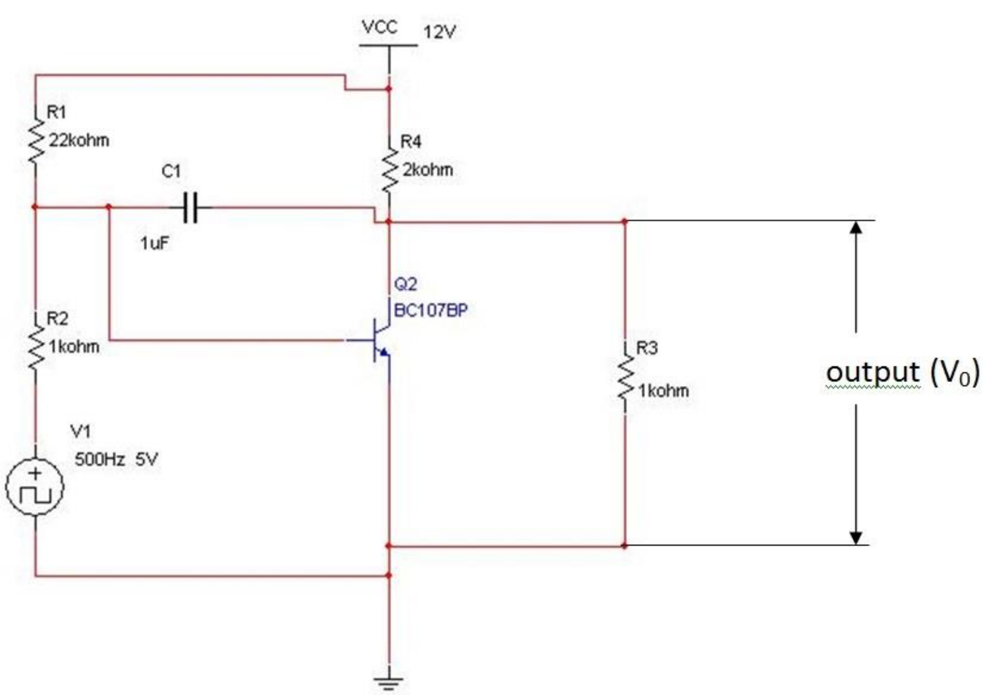


Fig: 14 a– circuit diagram of Miller sweep circuit

THEORY:

The Miller sweep circuit is an electronic circuit used to analyze the frequency response and characteristics of amplifiers, particularly to measure their bandwidth and gain-frequency relationship. It typically involves an amplifier with a capacitive feedback network that includes a variable capacitor or a capacitor in series with a resistor, which creates a sweeping effect. By varying the capacitance, the circuit effectively sweeps through a range of frequencies, allowing the observation of how the amplifier's gain changes with frequency. This sweeping process can help identify the -3 dB bandwidth of the amplifier, assess its stability, and evaluate its high-frequency performance.

PROCEDURE:

1. Connect the circuit as per the circuit diagram
2. Generate a control square wave amplitude vc of 5v pp at 500 Hz Frequency and apply into the circuit.
3. Observe the output wave forms

EXPECTED WAVE FORMS:

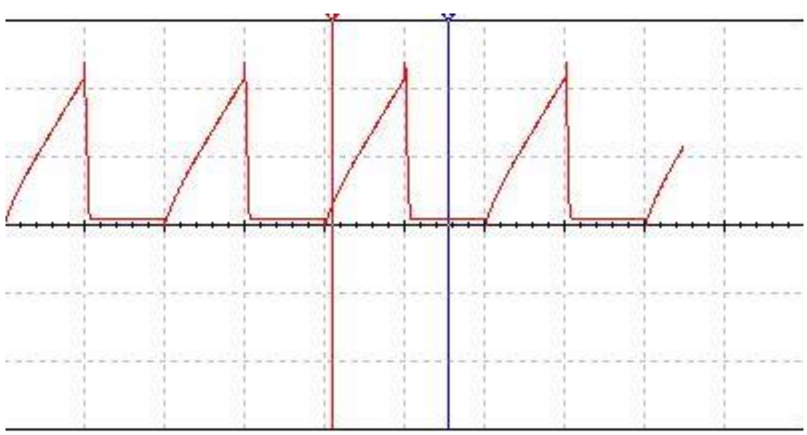


Fig: 14 b– Output waveform of Miller sweep circuit

Inference:

Conclusions can be made on sweep time Ts and retrace time T_R and sweep voltage V_I of the sweep waveform theoretically and practically and also made on if the output waveform of the Bootstrap are identical with the theoretical wave forms or not.

RESULT:

Miller sweep circuit wave forms are plotted.

VIVA QUESTIONS:

1. What is a Miller Sweep Circuit?

2. Explain the principle behind the Miller Sweep Circuit.

3. What is the role of the Miller effect in this circuit?

4. Describe the components typically used in a Miller Sweep Circuit.

5. How does the capacitor influence the sweep operation in the circuit?

6. What type of waveform is typically generated by the Miller Sweep Circuit?

7. Why is the Miller Sweep Circuit used in oscilloscopes?

8. How do you calculate the sweep rate in a Miller Sweep Circuit?

9. What is the significance of the time constant in the circuit?

10. Explain the working of the transistor in the Miller Sweep Circuit.

11. How does the biasing of the transistor affect the circuit operation?

12. What are the typical applications of the Miller Sweep Circuit?

13. Can the sweep linearity be improved in a Miller Sweep Circuit? If so, how?

14. What is the role of the feedback loop in the Miller Sweep Circuit?

15. How does changing the capacitor value affect the sweep duration?

16. What happens to the sweep signal if the resistor value is increased?

17. Why is it important to have a stable DC operating point in the circuit?

18. How does temperature affect the performance of the Miller Sweep Circuit?

19. Explain the difference between a linear and non-linear sweep in this circuit.

20. What precautions should be taken when setting up the Miller Sweep Circuit?

21. How would you troubleshoot if the circuit fails to generate the desired sweep?

22. What is the impact of parasitic capacitances on the circuit operation?

23. How does the Miller effect contribute to the circuit's high-frequency behavior?

24. What is the role of the load resistor in the Miller Sweep Circuit?

25. Why is the circuit called a "sweep" circuit?

26. How does the Miller Sweep Circuit compare to other types of sweep circuits?

27. What are the limitations of the Miller Sweep Circuit?

28. In what scenarios would you prefer a Miller Sweep Circuit over other types?

29. How does the Miller Sweep Circuit achieve frequency modulation?

30. What are the safety considerations when working with a Miller Sweep Circuit?

EXP NO: 15

BOOTSTRAP TIME BASE GENERATOR

AIM:

To verify the output characteristics of Bootstrap sweep generator

APPARATUS:

S.No	Device	Range/Rating	QTY
1.	(a) DC supply voltage (b) Transistor (c) Capacitors (d) Resistors	12V BC107 1µF 10µ F 220KΩ,14KΩ, 10KΩ	1 1 2 1 1 1
2.	Signal generator	0.1Hz-1MHz	1
3.	CRO	0Hz-20MHz	1
4.	Connecting wires	5A	4

CIRCUIT DIAGRAM:

Fig 15.a : Circuit diagram of Bootstrap time base generator

THEORY:

A bootstrap time base generator is a type of electronic circuit used to generate a time base or timing signal, often employed in oscilloscopes and other applications requiring precise timing intervals. The circuit utilizes a bootstrap capacitor in conjunction with a timing resistor and an active device, such as a transistor or operational amplifier. The bootstrap capacitor is charged and discharged to create a periodic timing waveform. The key advantage of the bootstrap approach is its ability to provide a stable and accurate time base signal with minimal drift and variation, as the bootstrap capacitor enhances the linearity of the timing interval. This method also improves the performance of the timing circuit by reducing the impact of variations in component values and power supply fluctuations. By adjusting the resistor and capacitor values, the frequency and duty cycle of the generated time base signal can be precisely controlled, making it ideal for applications requiring reliable and consistent timing signals.

PROCEDURE:

1. Connect the circuit diagram as shown in the figure .(1) Apply +15V (Vcc) & -15V($-V_{EE}$) DC as power supply to the circuit.
2. Apply 1kHz symmetrical square wave signal with the help of signal generator.
3. Observe the input and output wave forms of CRO and plot the waveforms.
4. Calculate the sweep & retrace time from the CRO .
5. Clarify the values with the theoretical calculation

THEORITICAL CALCULATION:

Ts = Rc1.Cs

$$Tr = \left(\frac{CsVs}{Vcc}\right) \;/\; \left[\frac{hfe}{R_B} - \frac{1}{Rc1}\right]$$

$$\text{Whose } Vs = \frac{VccTg}{Rc1Cs}$$

Practical Calulation:

RESULT:

The wave forms shown in figure are observed.

VIVA QUESTIONS:

1. Explain the basic principle involved in Bootstrap Sweep generator.
2. Mention the type of feedback employed in Bootstrap Sweep generator.
3. Mention the characteristics of the amplifier used in Bootstrap sweep generator.
4. What is meant by Bootstrap sweep generator?
5. How much is the amplifier gain for Bootstrap sweep generator.
6. The most important application of time base generator is_______________

7. Write the sweep time of a UJT expression.
8. Write the methods of generating time base wave forms.
9. Which oscillator generates the non-sinusoidal wave forms
10. Write the expression for sweep speed error.
11. Define (a) Voltage time base generator, (b) current time base generator (c) linear time base generator.
12. What is the relation between the slope error, displacement error and transmission error?
13. What are the various methods of generating time base wave-form?
14. Which amplifier is used in Boot-strap time base generator?
15. Which amplifier is used in Boot-strap time base generator?
16. Which type of sweep does a bootstrap time-base generator produce?
17. What is the gain of the amplifier used in Bootstrap time base generator?
18. What is retrace time? Write the formula for the same for Bootstrap time base generator.
19. What is the formula for sweep amplitude in Bootstrap time base generator?
20. To have less flatness time of sweep signal, then the gate signal time has to be __.
21. A Bootstrap sweep circuit employs __ type of feedback.
22. Which amplifier is used in Boot-strap time base generator?
23. Which type of sweep does a bootstrap time-base generator produce?
24. Design a Bootstrap Sweep generator for Vcc=12V,hfe=20,Rc1=11.2K, Rb=150K,C_s=0.8µF, C_b=8.8µF and Re=5.8K.
25. Find a reasonable value for Rb, where Rb is the collector to base resistor for a 20V sweep in 560µsec. Assume hfe=50, Rc1=5K, Tg=500msec
26. Design a Bootstrap Sweep generator for Vcc=12V,hfe=20,Rc1=11.2K, Rb=150K,C_s=0.8µF, C_b=5.8µF and Re=5.8K.
27. Find a reasonable value for Rb, where Rb is the collector to base resistor for a 20V sweep in 400µsec. Assume hfe=50, Rc1=5K, Tg=20msec
28. Design a Bootstrap Sweep generator for Vcc=12V,hfe=20,Rc1=11.2K, Rb=150K,C_s=0.8µF, C_b=8µF and Re=5.8K.
29. Find a reasonable value for Rb, where Rb is the collector to base resistor for a 20V sweep in 550µsec. Assume hfe=50, Rc1=5K, Tg=560msec
30. Design a Bootstrap Sweep generator for Vcc=12V,hfe=20,Rc1=11.2K, Rb=150K,C_s=0.8µF, C_b=6.8µF and Re=5.8K.