SVR ENGINEERING COLLEGE

Approved by AICTE & Permanently Affiliated to JNTUA

Ayyalurmetta, Nandyal – 518503. Website: www.svrec.ac.in

Department of Electronics and Communication Engineering

ANALOG CIRCUITS LABORATORY

II B. Tech (ECE) I Semester 2021-22

STUDENT NAME	
ROLL NUMBER	
SECTION	

SVR ENGINEERING COLLEGE

Approved by AICTE & Permanently Affiliated to JNTUA

Ayyalurmetta, Nandyal – 518503. Website: www.svrec.ac.in

DEPARTMENT OF

ELECTRONICS AND COMMUNICATION ENGINEERING

CERTIFICATE

ACADEMIC YEAR: 2021-22

This is to certify that the bonafide record work done by

Mr./Ms.____________________________________ bearing

H.T.NO. ______________________ of II B. Tech I Semester in the

Analog Circuits Laboratory.

Faculty In-Charge **Head of the Department**

JAWAHARLAL NEHRU TECHNOLOGICAL UNIVERSITY ANANTAPUR
(Established by Govt. of A.P., ACT No.30 of 2008)
ANANTHAPURAMU – 515 002 (A.P) INDIA

Electronics & Communication Engineering

Course Code	ANALOG CIRCUITS LAB		L	T	P	C
20A04302P			0	0	3	1.5
Pre-requisite	Electronic Devices and Circuits lab	Semester	III			

Course Objectives:

- To review analysis & design of single stage amplifiers using BJT & MOSFETs at low and high frequencies.
- To understand the characteristics of Differential amplifiers, feedback and power amplifiers.
- To examine the response of tuned amplifiers and multivibrators
- To categorize different oscillator circuits based on the application
- To design the electronic circuits for the given specifications and for a given application.

Course Outcomes (CO):

CO1: Know about the usage of equipment/components/software tools used to conduct the experiments in analog circuits.
CO2: Conduct the experiment based on the knowledge acquired in the theory about various analog circuits using BJT/MOSFETs to find the important parameters of the circuit (viz. Voltage gain, Current gain, bandwidth, input and output impedances etc) experimentally.
CO3: Analyze the given analog circuit to find required important metrics of it theoretically.
CO4:Draw the relevant graphs between important metrics of the system from the observed measurements.
CO5: Compare the experimental results with that of theoretical ones and infer the conclusions.
CO6: Design the circuit for the given specifications.

List of Experiments:

1. Design and Analysis of Darlington pair.
2. Frequency response of CE – CC multistage Amplifier
3. Design and Analysis of Cascode Amplifier.
4. Frequency Response of Differential Amplifier
5. Design and Analysis of Series – Series feedback amplifier and find the frequency response of it.
6. Design and Analysis of Shunt – Shunt feedback amplifier and find the frequency response of it.
7. Design and Analysis of Class A power amplifier
8. Design and Analysis of Class AB amplifier
9. Design and Analysis of RC phase shift oscillator
10. Design and Analysis of LC Oscillator
11. Frequency Response of Single Tuned amplifier
12. Design and Analysis of Bistable Multivibrator
13. Design and Analysis of Monostable Multivibrator
14. Design and Analysis of Astable Multivibrator

Note: At least 12 experiments shall be performed. Both BJT and MOSFET based circuits shall be implemented.
Faculty members who are handling the laboratory shall see that students are given design specifications for a given circuit appropriately and monitor the design and analysis aspects of the circuit.

Online learning resources/Virtual labs:
https://www.vlab.co.in/

ECE DEPT VISION & MISSION PEOs and PSOs

Vision

To produce highly skilled, creative and competitive Electronics and Communication Engineers to meet the emergingneeds of the society.

Mission

- Impart core knowledge and necessary skills in Electronics and Communication Engineering throughinnovative teaching and learning.
- Inculcate critical thinking, ethics, lifelong learning and creativity needed for industry and society
- Cultivate the students with all-round competencies, for career, higher education and self-employability

I. PROGRAMME EDUCATIONAL OBJECTIVES (PEOS)

PEO1: Graduates apply their knowledge of mathematics and science to identify, analyze and solve problemsin the field of Electronics and develop sophisticated communication systems.

PEO2: Graduates embody a commitment to professional ethics, diversity and social awareness in theirprofessional career.

PEO3: Graduates exhibit a desire for life-long learning through technical training and professional activities.

II. PROGRAM SPECIFIC OUTCOMES (PSOS)

PSO1: Apply the fundamental concepts of electronics and communication engineering to design a variety of components and systems for applications including signal processing, image processing, communication, networking, embedded systems, VLSI and control system

PSO2: Select and apply cutting-edge engineering hardware and software tools to solve complex Electronics and Communication Engineering problems.

III. PROGRAMME OUTCOMES (PO'S)

1. Engineering knowledge: Apply the knowledge of mathematics, science, engineering fundamentals, and anengineering specialization to the solution of complex engineering problems.

2. Problem analysis: Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.

3. Design/development of solutions: Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety,and the cultural, societal, and environmental considerations.

4. Conduct investigations of complex problems: Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.

5. Modern tool usage: Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.

6. The engineer and society: Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.

7. Environment and sustainability: Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.

8. Ethics: Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.

9. Individual and team work: Function effectively as an individual, and as a member or leader in diverse teams, andin multidisciplinary settings.

10. Communication: Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.

11. Project management and finance: Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects andin multidisciplinary environments.

12. Life-long learning: Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.

IV. COURSE OBJECTIVES

- To review analysis & design of single stage amplifiers using BJT & MOSFETs at low andhigh frequencies.
- To understand the characteristics of Differential amplifiers, feedback and poweramplifiers.
- To examine the response of tuned amplifiers and multi-vibrators
- To categorize different oscillator circuits based on the application
- To design the electronic circuits for the given specifications and for a given application.

V. COURSE OUTCOMES

After the completion of the course students will be able to

Course Outcomes	Course Outcome statements	BTL
CO1	Know about the usage of equipment/components/software tools used to conduct the experiments in analog circuits.	L1
CO2	Conduct the experiment based on the knowledge acquired in the theory about various analog circuits using BJT/MOSFETs to findthe important parameters of the circuit (viz. Voltage gain, Current gain, bandwidth, input and output impedances etc) experimentally.	L2
CO3	Analyze the given analog circuit to find required important metrics of it theoretically.	L3
CO4	Draw the relevant graphs between important metrics of the system from the observed measurements.	L4
CO5	Compare the experimental results with that of theoretical ones and infer the conclusions and Design the circuit for the given specifications.	L5

VI.COURSE MAPPING WITH PO'S AND PEO'S

Course Title	P0 1	P0 2	P03	P0 4	P05	P 0 6	P 0 7	P 0 8	P 0 9	P0 10	P 0 11	P 0 12	P S 0 1	P S 0 2
Analog Circuits Lab	2.4	2.6	2.6	2.6	3.0	2.4	2.6	2.4	2.6	2.4	2.4	2.4	2.6	2.4

VII MAPPING OF COURSE OUTCOMES WITH PEO'S AND PO'S

Course Title	P 0 1	P 0 2	P 0 3	P 0 4	P 0 5	P 0 6	P 0 7	P 0 8	P 0 9	P 0 10	P 0 11	P 0 12	P S 0 1	P S 0 2
CO1	3	3	2	3	3	2	2	3	3	3	3	3	3	3
CO2	2	2	3	2	3	3	3	2	3	2	3	3	2	2
CO3	2	3	2	3	3	3	2	2	2	2	2	1	2	3
CO4	2	2	3	2	3	2	3	3	3	2	2	2	3	2
CO5	3	3	3	3	3	2	3	2	2	3	2	3	3	2

LABORATORY INSTRUCTIONS

1. While entering the Laboratory, the students should follow the dress code. (Wear shoes and White apron,Female Students should tie their hair back).
2. The students should bring their observation book, record, calculator, necessary stationery items and graphsheets if any for the lab classes without which the students will not be allowed for doing the experiment.
3. All the Equipment and components should be handled with utmost care. Any breakage or damage will becharged.
4. If any damage or breakage is noticed, it should be reported to the concerned in charge immediately.
5. The theoretical calculations and the updated register values should be noted down in the observation bookand should be corrected by the lab in-charge on the same day of the laboratory session.
6. Each experiment should be written in the record note book only after getting signature from the lab in-charge in the observation notebook.
7. Record book must be submitted in the successive lab session after completion of experiment.
8. 100% attendance should be maintained for the laboratory classes.

Precautions.

1. Check the connections before giving the supply.
2. Observations should be done carefully.

I N D E X

Max. Marks per each experiment : 5

------- Index Continued -------

Experiment No. : 01 **Date :**

Name of the Experiment : DARLINGTON PAIR AMPLIFIER (Using Software & Hardware)

AIM :

To obtain the frequency response curve of *Darlington pair amplifier* using software & hardware

APPARATUS :

Software :

1. System		------------	1 No.
2. Multisim software		------------	

Hardware :

1. Transistors	BC547	------------	1 No.
2. Resistors	47KΩ, 10KΩ, 1KΩ	------------	Each 1No.
3. Capacitors	0.22µF	------------	3 No.

THEORY :

Darlington Pair amplifier circuit is a connection of two **transistors** which acts as a single unit with overall current gain equal to the multiplication of the individual current gains of the transistors. Darlington pair **transistor amplifier circuit** is very popular in electronics. Clearly, it is an **Amplifier circuit**. In this article, we are going to discuss the theory and the applications of Darlington pair amplifier.

The current gain of Darlington pair amplifier is almost equal to the product between the current gains of individual transistors. If _{\beta 1}$\beta1$ and _{\beta 2}$\beta2$ be the current gains of individual transistors then overall current gain of Darlington pair amplifier = $\beta1\beta2$.

CIRCUIT DIAGRAM – SOFTWARE & HARDWARE :

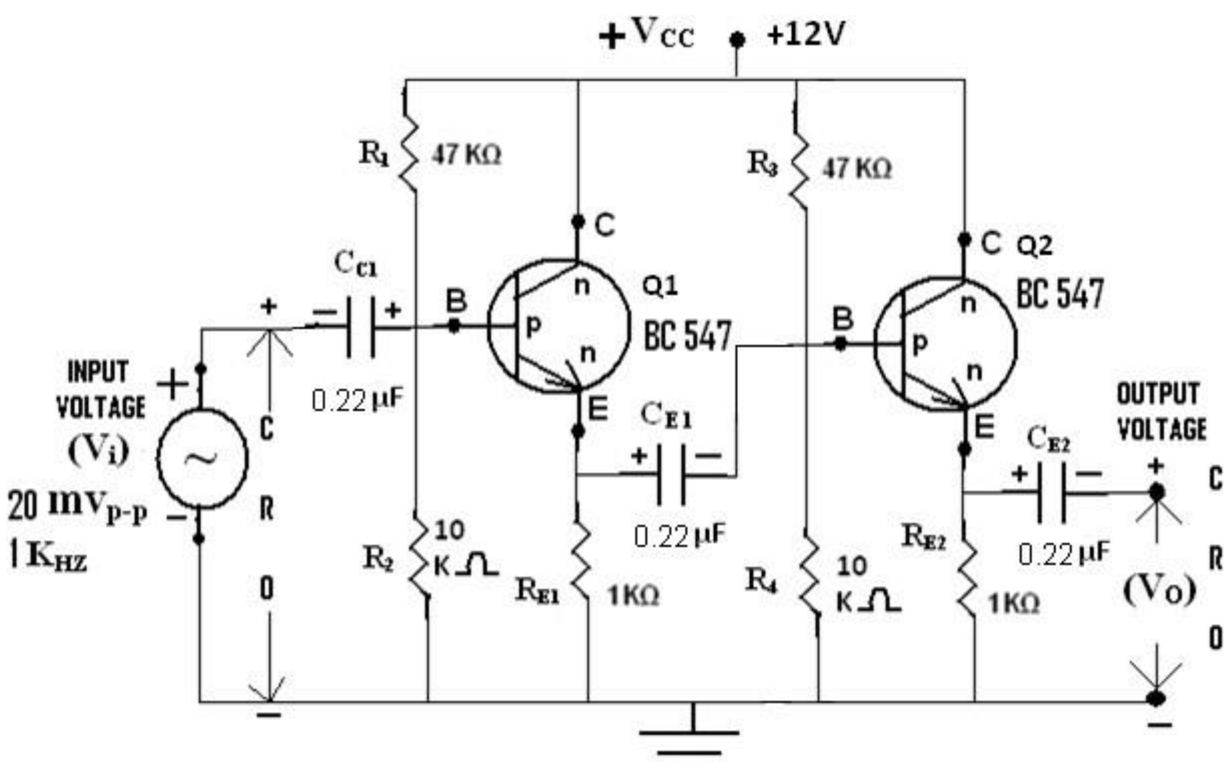


Figure: Circuit diagram of Darlington pair amplifier.

PROCEDURE – SOFTWARE :

1. We have picked up the components from the components bar as per above circuit.
2. Made the connections as per the above circuit diagram by using the components which we have pickedup.
3. Connected the CRO across the capacitor C_{E2} of second stage.
4. Set the input signal as *sine wave form which is having the* value $20mV_{P-P}$ as constant in the functiongenerator.
5. Initially set the input signal frequency value is 1KHz in the function generator.
6. To simulate the circuit clicked on *run option* through *execute button* in *tool bar.*
7. We have seen the ***sine wave*** on the ***CRO*** screen as o/p signal.
8. Calculated the *peak to peak voltage* $(V_{O(p-p)})$ and noted down in the tabular form Against the column of1KHz.
9. Stopped the simulation by clicked on *run option* through *execute button* in the *tool bar.*
10. Repeated the same procedure from points 6 to 9 for the corresponding frequency values by setting in thefunction generator for the following steps,
    20Hz, 100Hz., 200Hz., 500Hz, 1KHz, 200KHz, 400KHz,940KHz, 940KHz, 1MHz, 100MHz, 500MHz. in the function generator.
11. Finally shut down the system safely.
12. Now calculated and noted down the values of *voltage gain*(A_V) and *gain in dB* to the corresponding valuesof *output voltage*(V_O) & *input voltage*(V_i) by using the formulas given below,

$$\text{Voltage gain } (A_v) = V_o / V_i \quad \text{and} \quad \text{Gain in dB} = 20\log_{10}(A_v).$$

PROCEDURE – HARDWARE :

1. We have connected the circuit as per the circuit diagram which is shown above.
2. Initially connected the probe across the function generator as per shown in the circuit diagram to set the input signal.
3. Switched *ON* the *CRO* and *function generator*.
4. Applied the input signal as *sine wave form* of $20m_{p-p}$, 1KHz.from the function generator by observing in the CRO.
5. Later removed the probe from that place and connected it across the capacitor C_{E3} to observe the output.
6. Switched *ON* the *RPS* and kept the 12V as V_{CC}.
7. Kept the amplitude of the input signal as constant as $20mV_{p-p}$ for all frequency steps.
8. Noted down the values output voltage of output signal in terms of peak to peak voltages by varying the different frequency steps in the function generator which are given below,
   20Hz, 100Hz., 200Hz., 500Hz, 1KHz, 100KHz, 200KHz, 400KHz, 940KHz, 940KHz, 1MHz.
9. Repeated the same procedure for point 8 for corresponding frequency values.
10. Now calculated and noted down the values of *voltage gain*(A_V) and *gain in dB* to the corresponding values of *output voltage*(V_O) & *input voltage*(V_i) by using the formulas given below,

$$\text{Voltage gain } (A_v) = V_o / V_i \quad \text{and} \quad \text{Gain in dB} = 20\log_{10}(A_v).$$

TABULAR COLUMN :

	Input Voltage (V_i) = 20 mV_{P-P} (0.02V) is constant for all readings (For Software & Hardware)								
	For Software :					**For Hardware :**			
Sl.No.	**Frequ-ency In Hz/KHz.**	**Output Voltage (V_O) In mVolts.**	**Voltage gain A_V= V_o/V_i**	**Gain in dB = $20\log_{10}$ (A_V)**		**Frequ-ency In Hz/KHz.**	**Output Voltage (V_O)In mVolts.**	**Voltage Gain A_V= V_o/V_i**	**Gain in dB = $20\log_{10}$ (A_V)**
1	20 Hz.								
2	100 Hz.								
3	200 Hz.								
4	1 KHz.								
5	200KHz.								
6	400KHz.								
7	940KHz.								
8	940KHz.								
9	1 MHz.								
10	100 MHz					-------	-------	-------	-------
11	500MHz.					-------	-------	-------	-------

EXPECTED GRAPH – SOFTWARE & HARDWARE :

Note : We can't draw the graph and could not find the band width for this experiment, because there isno amplification.

CONCLUSSION :

We have formed the circuit of Darlington pair amplifier by connected two common collector amplifiers in two stages. The input impedance of two stage common collector amplifier i.e. Darlington pair amplifier is very high as compared to single stage common collector amplifier. Due to this reason only the voltage gain of Darlington pair amplifier is less than as compared to single stage common collector amplifier.

RESULT – SOFTWARE & HARDWARE :

I have obtained the voltage gain and gain in db at different frequencies of a *Darlington pair amplifier*.

VIVA VOCE QUESTIONS :

1. Applications of Darlington pair Amplifier.

2. Applications of Multi stage amplifiers?

3. Mention Advantages of Multistage Amplifiers.

4. What is Band Width?

5. What is Frequency Response?

Experiment No. : 02 **Date :**

Name of the Experiment : CE - CC MULTISTAGE AMPLIFIER

AIM :

1). To obtain the frequency response of *CE – CC multistage amplifier*.

APPARATUS :

1). Function generator(***FG***)---1 No.

2). Cathode Ray Oscilloscope(***CRO***)---1 No.

3). Regulated Power Supply (***RPS***) : (0-30)V, 1A Dual channel ----------1 No.

4). Probes ---1 No.

5). Bread board ---1 No.

6). Connecting wires :---A few Nos.

COMPONENTS :

1). Transistor BC 547 / 100A/A--- 1 No.

2). Carbon Fixed Resistors 250Ω, 2.2KΩ, 10KΩ ------------------ Each 1 No.

47KΩ, 3.3KΩ ------------------------- Each 2 No.

3). Capacitors 10µF ------------------------- 3 No.

1µF ------------------------- 1 No.

THEORY : *Note :* Student should leave half of the page to prepare theory by himself

In practical applications, the output of a single state amplifier is usually insufficient, though it is a voltage or power amplifier. Hence they are replaced by **Multi-stage transistor amplifiers**.

In Multi-stage amplifiers, the output of first stage is coupled to the input of next stage using a coupling device. These coupling devices can usually be a capacitor or a transformer. This process of joining two amplifier stages using a coupling device can be called as **Cascading**.

But in this CE-CB multistage amplifier is used to match the impedance matching.

CIRCUIT DIAGRAM :

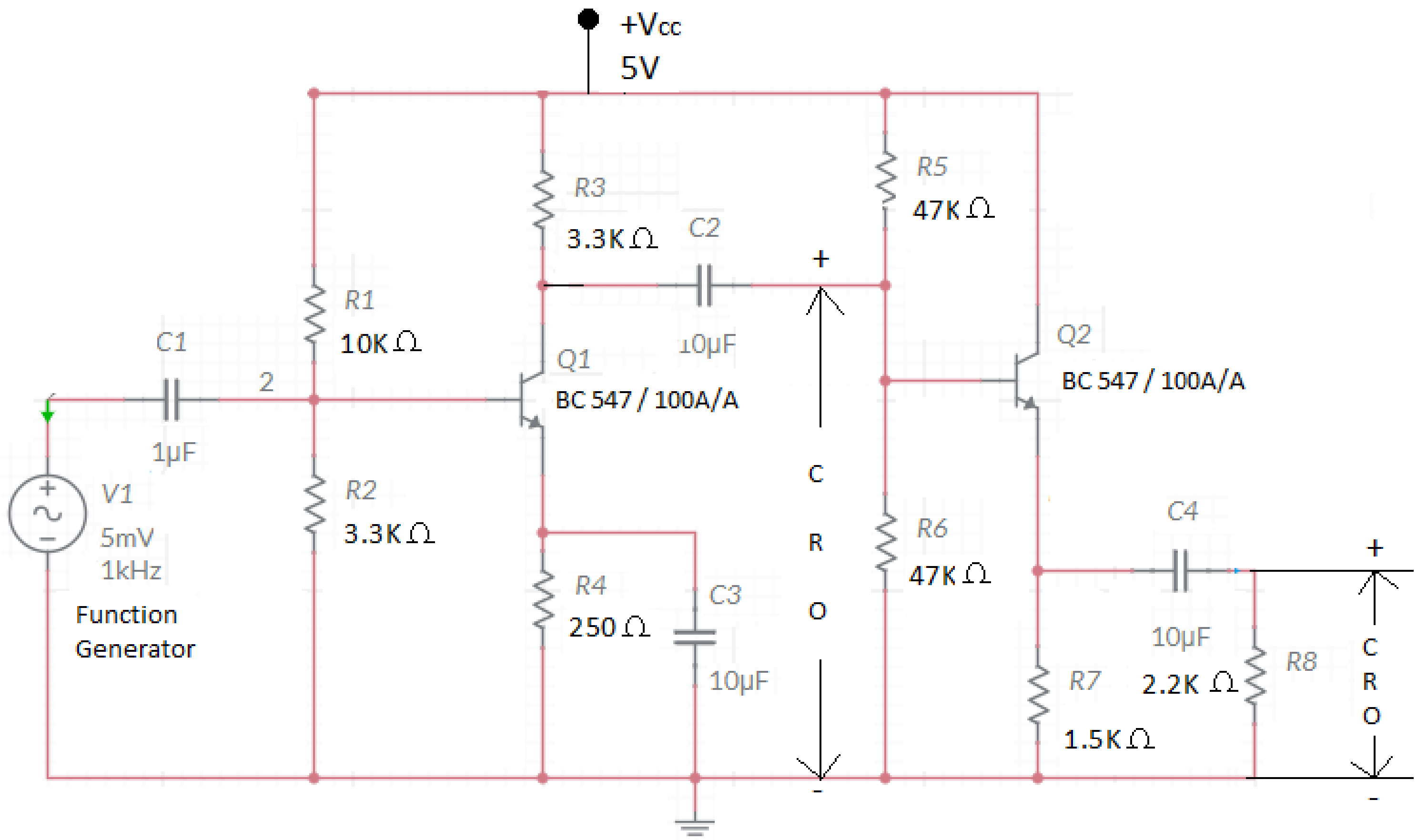


Fig : Circuit diagram of CE-CC Amplifier

PROCEDURE - HARDWARE :

1. We have connected the circuit as per the circuit diagram which is shown above.
2. Initially connected the probe across the function generator as per shown in the circuit diagram to set the input signal.
3. Switched *ON* the *CRO* and *function generator*.
4. Applied the input signal as *sine wave form* having the values of $5mV_{p-p}$, 1KHz.from the function generator by observing in the CRO.
5. Removed the probe from that place and connected it across the C2 to observe the output of CE amplifier .
6. Switched *ON* the *RPS* and kept the +5V as V_{CC}.
7. Kept the amplitude of the input signal as constant as $5mV_{p-p}$ for all frequency steps.
8. Noted down the values of output voltage in terms of peak to peak voltages by varying the different frequency steps in the function generator which are given below,
   20Hz, 100Hz, 200Hz., 500Hz, 1KHz, 100KHz, 200KHz, 400KHz, 940KHz, 940KHz, 1MHz.
9. The above readings noted in the tabular form of *CE amplifier* .
10. Disconnected the probe from C2 and reconnected it across C4 to observe the output of second stage.
11. Repeated the same procedure from the step 6 to 8 for tabular form of *CE-CC multistage Amplifier*.
12. Now calculated and noted down the values in the tabular form of *CE Amplifier* asper given below,

a). Voltage gain $(A_v) = V_o / V_i$ *and* *Gain in dB* $= 20log_{10}(A_v)$.

b). Plotted the graph between *frequency on X*- axis and *gain in dB on Y*axis.

c). Band width from the graph by using the formula- *Band width* $= f_2 - f_1$

13. Now calculated and noted down the values in the tabular form of *CE-CC multistage Amplifier* asper given below,

a). Voltage gain $(A_v) = V_o / V_i$ *and* *Gain in dB* $= 20log_{10}(A_v)$.

b). Plotted the graph between *frequency on X*- axis and *gain in dB on Y*- axis.

c). Band width from the graph by using the formula- *Band width* $= f_4 - f_3$

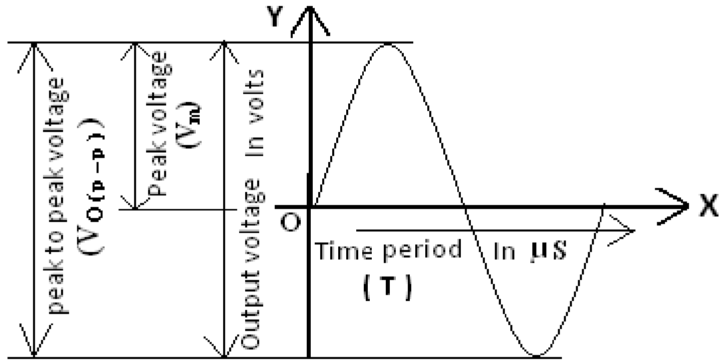
Note : The O/P of multistage amplifier is same as O/P of CE amplifier, because the voltage gain of CC amplifier is 1. As there is no amplification at O/P of multistage So, it is not possible to draw the graph and band width also.

TABULAR COLUMN :

	Input Voltage (V_i) = 5 mV$_{P-P}$ (0.005V) is constant for all readings (For CE & CE Amplifiers)								
	For CE Single stage Amplifier					**For CE-CC Multistage Amplifier**			
Sl. No.	**Frequ-ency In Hz/KHz.**	**Voltage (V_O) In mVolts.**	**Voltage Gain A_V= V_o/V_i**	**Gain in dB= $20log_{10}$ (A_V)**		**Frequ-ency In Hz/KHz.**	**Output Voltage (V_O) In mVolts.**	**Voltage gain A_V=V_o/V_i**	**Gain in dB = $20log_{10}$ (A_V)**
1	20 Hz.								
2	100 Hz.								
3	200 Hz.								
4	500 Hz.								
5	1 KHz.								
6	200KHz.								
7	400KHz.								
8	600KHz.								
9	800KHz.								
10	1 MHz.								

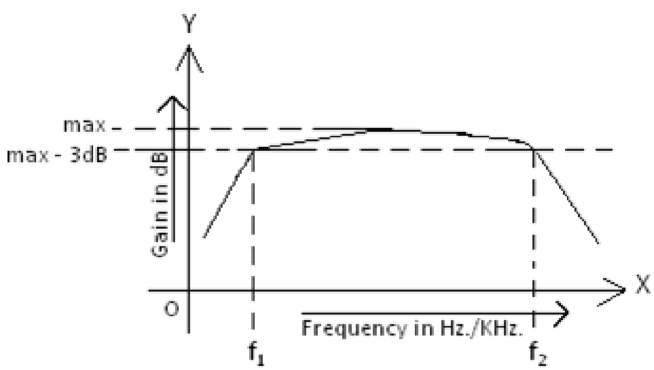
EXPECTED WAVEFORM :

I got the *Sine wave form* on the CRO as output signal for single stage as well as for CE-CC multistage amplifier which is shown below,

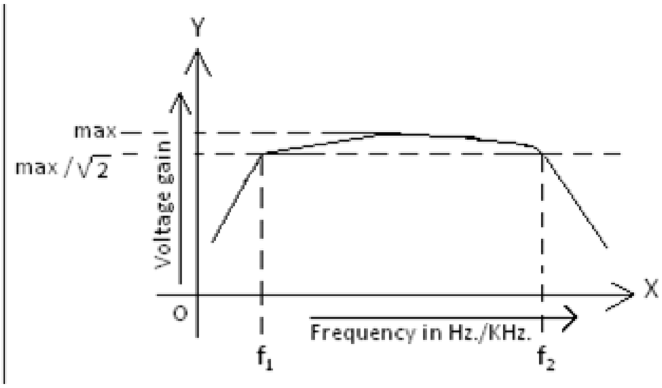


EXPECTED GRAPHS :

A). Frequency response curve for frequency vs gain in dB. of CE and CE-CC amplifier.



B). Frequency response curve for frequency vs voltage gain of CE-CC multistage amplifier.



PARAMETERS (HARDWARE) :

1). Band width of frequency response curve for frequency verses gain in dB. $= f2 - f1 =$

2). Band width of frequency response curve for frequency verses voltage gain $= f2 - f1 =$

RESULT : I have obtained the frequency response curves of CE-CC Multistage Amplifiers for frequency verses gain in dB & frequency verses voltage gain and calculated the band width of both of them.

VIVA VOCE QUESTIONS:

1. What is the need of Need for multi stage amplifiers?

2. What are the different coupling schemes?

3. Applications of CE-CC Amplifiers?

4. Mention the Characteristics of CE-CC Amplifiers.

5. What is Band Width?

Experiment No. : 03 **Date :**

Name of the Experiment : CE - CB CASCODE AMPLIFIER

AIM :

1). To obtain the frequency response of *CE – CB cascade amplifier* using Hardware and Software

2). To calculate the band width of this amplifier.

APPARATUS :

1). Function generator(***FG***) --1 No.

2). Cathode Ray Oscilloscope(***CRO***) ---1 No.

3). Regulated Power Supply (***RPS***) : (0-30)V, 1A Dual channel ----------1 No.

4). Probes --1 No.

5). Bread board---1 No.

6). Connecting wires : ---A few Nos.

COMPONENTS :

1). Transistors : BC 547 -------------------- 1No.

2). Carbon fixed Resistors 47KΩ, 40.2KΩ, 10KΩ, 6.8KΩ 4.7KΩ, 1KΩ - 1/2W - Each 1 No.

Capacitors : 0.22μF ------------------- 3 No.

33μF ------------------- 1 No.

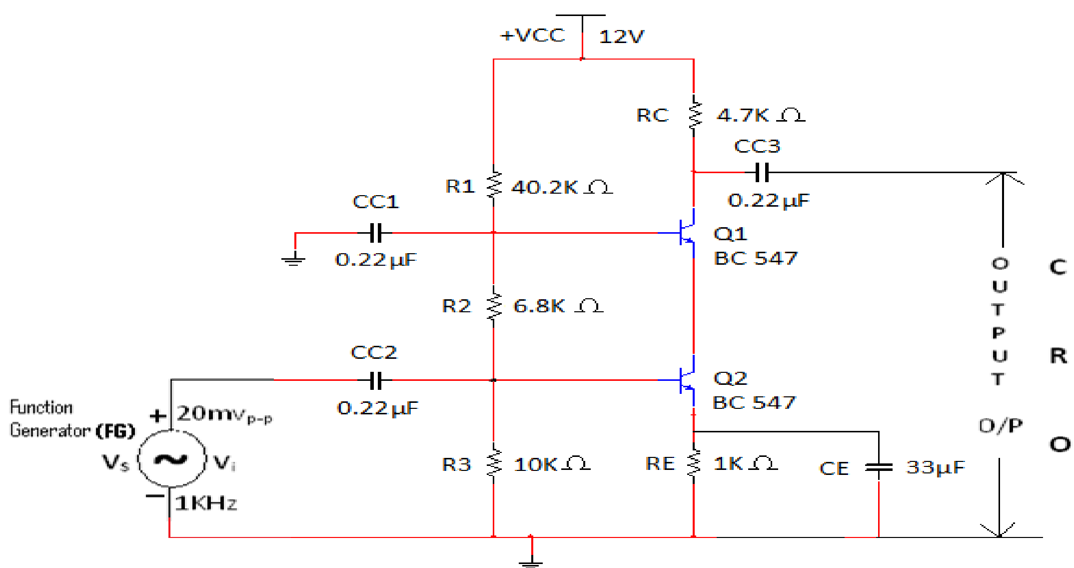
THEORY :

While the C-B (common-base) amplifier is known for wider bandwidth than the C-E (common-emitter) configuration, the low input impedance (10s of Ω) of C-B is a limitation for many applications. The solution is to precede the C-B stage by a low gain C-E stage which has moderately high input impedance (kΩs).

The stages are in a *cascode* configuration stacked in series, as opposed to cascaded for a standard amplifier chain.

The key to understanding the wide bandwidth of the cascode configuration is the *Miller effect*. The Miller effect is the multiplication of the bandwidth robbing collector-base capacitance by voltage gain Av.

CIRCUIT DIAGRAM :



PROCEDUREE :

1). Connected the circuit as per the circuit diagram.
2). Then switched ON the *function generator* and *CRO*; but don't switched ON the *RPS*.
3). Now Kept the *AC/GND/DC* switch is at *AC* position.
4). Initially kept the 1KHz. frequency by varying the frequency control in the *function generator*.
5). Now applied the peak to peak amplitude of a sine wave is of $20mV_{p-p}$ by varying the amplitude control inthe *function generator* through observing in the *CRO*.
6). Kept this input value as $20mV_{p-p}$ constant up to the completion of the experiment Otherwise the wrong output would occurred.
7). Now switched ON the *RPS* and set the 10V in it i.e. V_{CC} = 12V.
8). Varied the different frequency steps of 20Hz, 100Hz, 200Hz, 500Hz, 1KHz, 200KHz, 400KHz, 940KHz, 940KHz,1MHz. by adjusted the frequency control in the *function generator* and noted down the corresponding values of output signal i.e. peak to peak amplitude of sine wave by observing in the *CRO*.
9). Now switched OFF the *RPS, function generator* and *CRO*.
10). Then calculated the *voltage gain* $A_V = V_O/V_i$ & *gain in dB = 20log10(*A_V*)* and noted down the values in thespecified columns of the tabular column.
11). Plotted the graphs (frequency response curves) as per below,
    a). frequency on X-axis & gain in dB on Y-axis.
    b). frequency on X-axis & voltage gain on Y-axis.

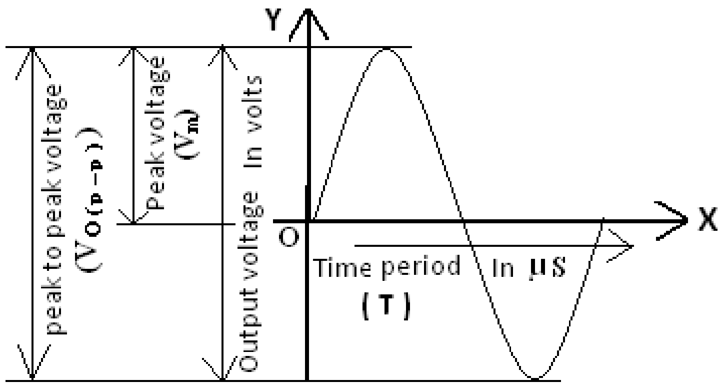
12) Calculated the *band width* from the above two (frequency response curves) graphs byusing the formula $f_2 - f_1$ which is given under the heading of *parameters*.

TABULAR COLUMNS :

	Input Voltage (V_i)= 20 mV_{P-P} (0.02V) is constant for all readings.			
Sl. No.	**Frequency In Hz/KHz.**	**Output Voltage (V_O) In Volts.**	**Voltage Gain $A_V = V_o/V_i$**	**Gain in dB = $20\log_{10}(A_V)$**
1	20 Hz.			
2	100 Hz.			
3	200 Hz.			
4	500 Hz.			
5	1 KHz.			
6	200KHz.			
7	400KHz.			
8	600KHz.			
9	800KHz.			
10	1 MHz.			

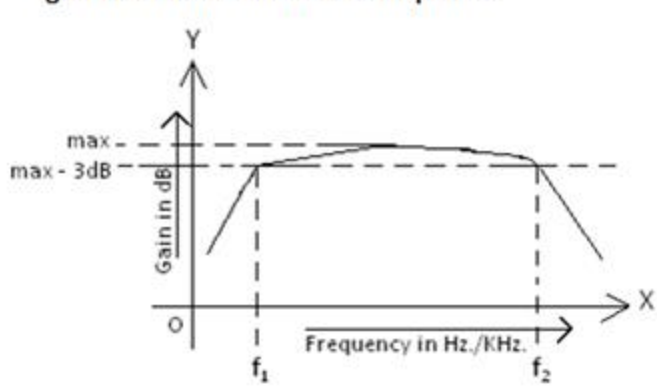
EXPECTED WAVEFORM :

I have got the *Sine wave form* on the CRO as output signal for single stage as well as for CE-CB multistage amplifier which is shown beow

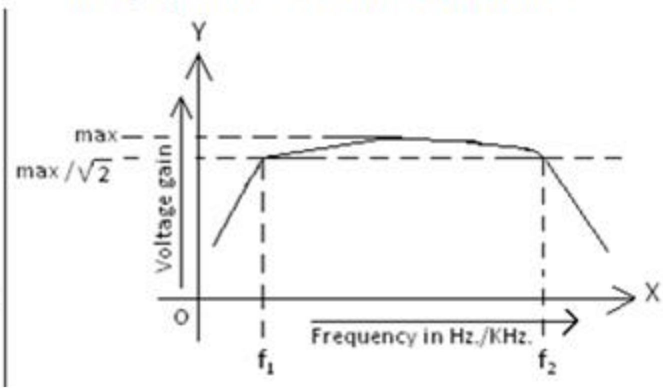


EXPECTED GRAPH :

A). Frequency response curve for frequency vs gain in dB of CE and CE- CB amplifier.



B). Frequency response curve for frequency vs voltage gain of CE- CB Multistage amplifier



PARAMETERS :

1). Band width of frequency response curve for frequency verses gain in dB. = $f_2 - f_1$ =

2). Band width of frequency response curve for frequency verses voltage gain = $f_2 - f_1$ =

RESULT :

I have obtained the frequency response curves of *CE-CB cascade Amplifier* as per follows,

1). Band width of frequency response curve for frequency verses gain in dB. =

2) Band width of frequency response curve for frequency verses voltage gain =

VIVA VOCE QUESTIONS:

1. Applications of CE-CC Amplifiers?

2. Mention the Characteristics of CE-CB Amplifiers.

3. What is Band Width?

4. What is cascade Amplifier?

5. What is voltage gain of cascade amplifier ?

Experiment No. : 04 **Date :**

Name of the Experiment : CURRENT SERIES – SERIES FEEDBACK AMPLIFIER

AIM :

i). To obtain the frequency response of *Current Series – Series feedback amplifier*

ii). To calculate the bandwidth of this amplifier.

APPARATUS :

1). Function Generator	-------- 1 No.
2). Cathode Ray Oscilloscope	-------- 1 No.
3). Regulated Power Supply	-------- 1 No.
4). Bread Board	-------- 1 No.
5). Probes	-------- 2 No.
6). Connecting wires	-------- A Few Nos.

COMPONENTS :

1). Transisitor	BC547 NPN	--------	2 No.
2). Resistors	a). 47KΩ, 10KΩ, 2 . 2KΩ, 1KΩ	--------	Each 1 No.
	b). 100KΩ, 100Ω	--------	Each 1 No.
3). Capacitors	a). 22µF	--------	3 No.
	b). 10µF	--------	1 No.
	c). 0.22µF / 10µF	--------	3 No.

THEORY :

Description :

Although while amplifying, the input signal strength can be increased whether it includes information otherwise information with some noise. This noise can be introduced in **the amplifiers** due to their strong tendency otherwise stray magnetic as well as electric fields. Thus, each high gain amplifier is liable in its output to provide noise along with the signal, which is very required. In amplifier circuits, the noise level will be significantly decreased with the help of negative feedback by introducing an output fraction within phase opposition toward the input signal.

What is a Feedback Amplifier?

The **feedback-amplifier** can be defined as an amplifier which has feedback lane that exists between o/p to input. In this type of amplifier, feedback is the limitation which calculates the sum of feedback given in the following amplifier. The feedback factor is the ratio of the feedback signal and the input signal.

Feedback Amplifier Topologies

There are four basic **amplifier topologies** for connecting the feedback signal. Both the current as well as voltage can be feedback toward the input in series otherwise in parallel.

1. Voltage Series Feedback Amplifier
2. Voltage Shunt Feedback Amplifier
3. Current Series Feedback Amplifier
4. Current Shunt Feedback Amplifier

   Out of four now we can do the experiment on

Current Shunt Feedback Amplifier

In this type of circuit, a portion of the o/p voltage is applied to the i/p voltage in shunt through the feedback circuit. The block diagram of the *current shunt feedback-amplifier* is shown below, by which it is apparent that the feedback circuit is located in shunt by means of the output as well as the input.
When the feedback circuit is allied in series through the o/p however in parallel with the input, then the o/p impedance will be increased & because of the parallel connection with the i/p, the i/p impedance will be decreased.

CIRCUIT DIAGRAMS :

Current series-Series feed back amplifier – With feed back :

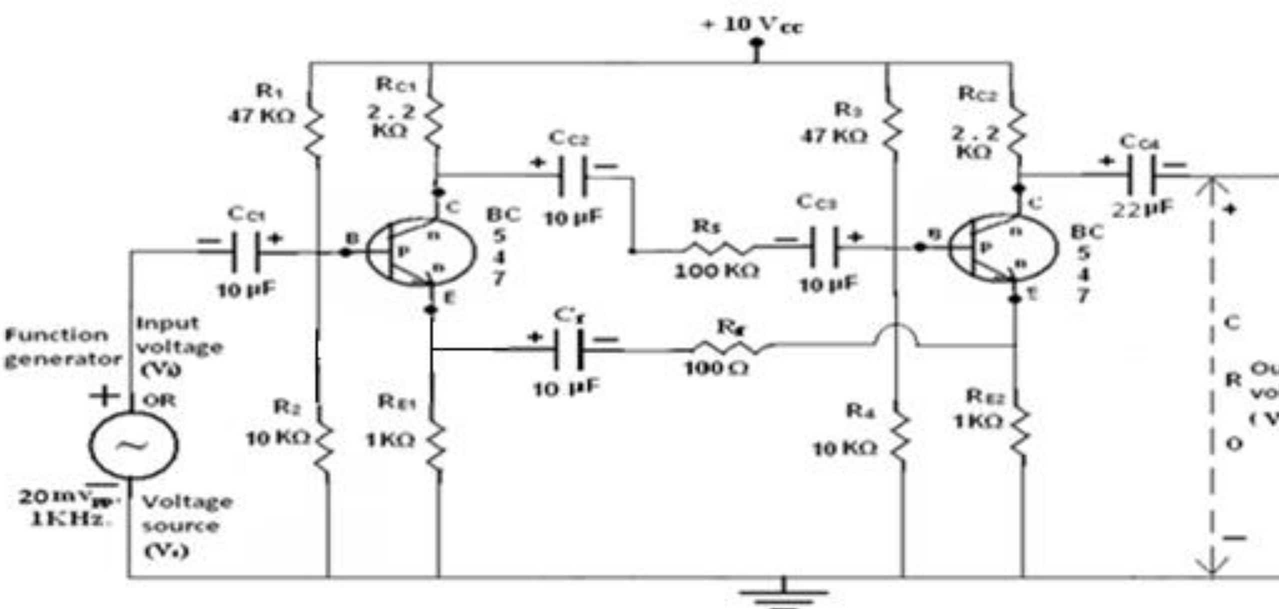


Fig : Current series - series Feed back Amplifier - With feed back.

Current series-Series feed back amplifier – Without feed back :

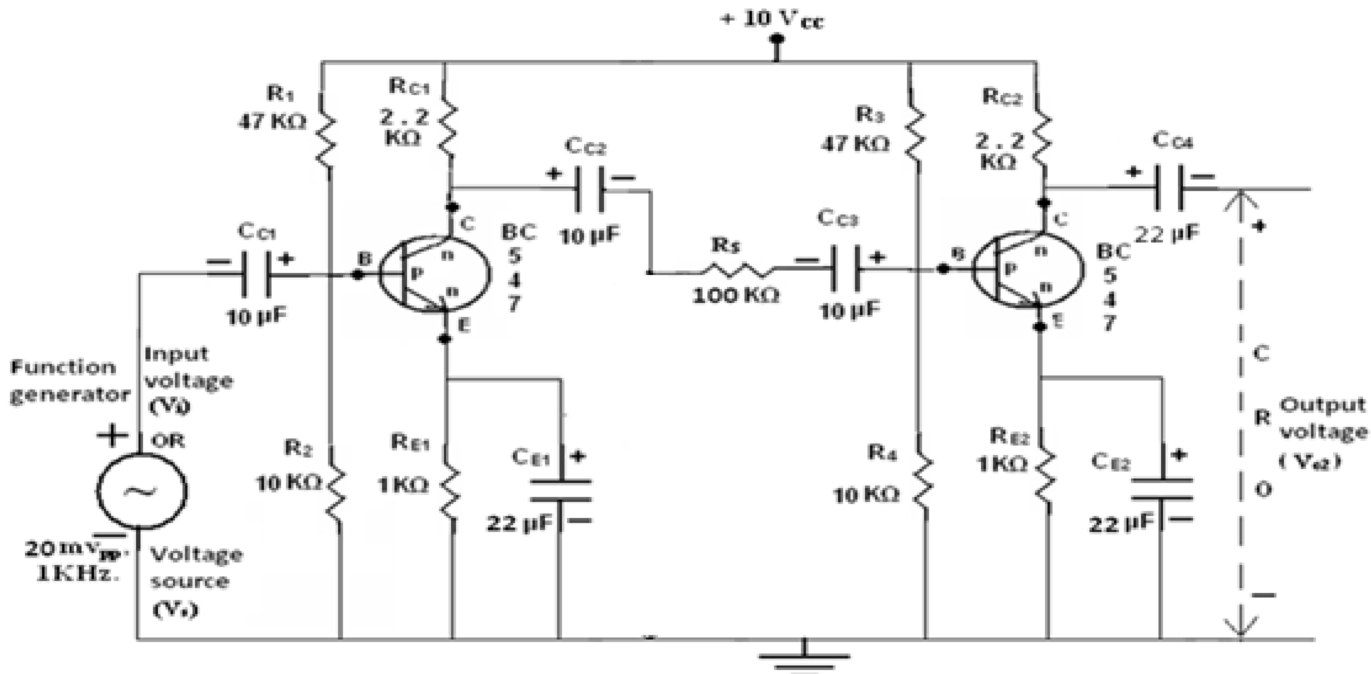


Fig : Current Series - series feed back Amplifier - Without feed back

PROCEDURE

1. Connections are made as per the circuit diagram.
2. Initially connected the *CRO* across the *Function generator.*
3. Switched **ON** the Cathode ray oscilloscope (CRO) and Function generator.
4. Applied the 20 mV_{pp} *, 1Khz sine wave* signal to the circuit from *Function generator* by observing in the *CRO*.
5. We have kept this *20* mV_{pp} input voltage (Which has been applied from function generator) as constant for all steps of frequency while taking the readings for both with fee back & without feedback .
6. Disconnected the CRO from the function generator .
7. Now Connected the CRO at output side to measure the peak to peak output voltage.
8. Applied the $+V_{CC}$ as 10V to the circuit from the *Regulated power supply (RPS).*
9. Later we have noted down the readings for output voltage in the tabular form of *with feedback.* from the CRO, by varying the different steps of frequency (i.e. 20Hz, 100Hz, 200Hz, 500Hz, 1KHz, 200KHz, 400KHz, 940KHz, 940KHz, 1MHz.) in function generator.
10. After this we removed the feed back capacitor (C_f) & resistor (R_f) from the circuit completely.
11. Now the circuit is became as the *without feed back amplifier***.**
12. Again we have noted down the readings for output voltage in the tabular form of *without feed back* .from the CRO, by varying the different steps of frequency (i.e. 20Hz, 100Hz, 200Hz, 500Hz, 1KHz, 200KHz, 400KHz, 940KHz, 940KHz, 1MHz.) in function generator.
13. I have observed that the readings of *without feed back amplifier's* output voltage is greater than the *with feed back amplifier.*
13. Finally we switched **OFF** the function generator, cathode ray oscilloscope and regulated power supply.

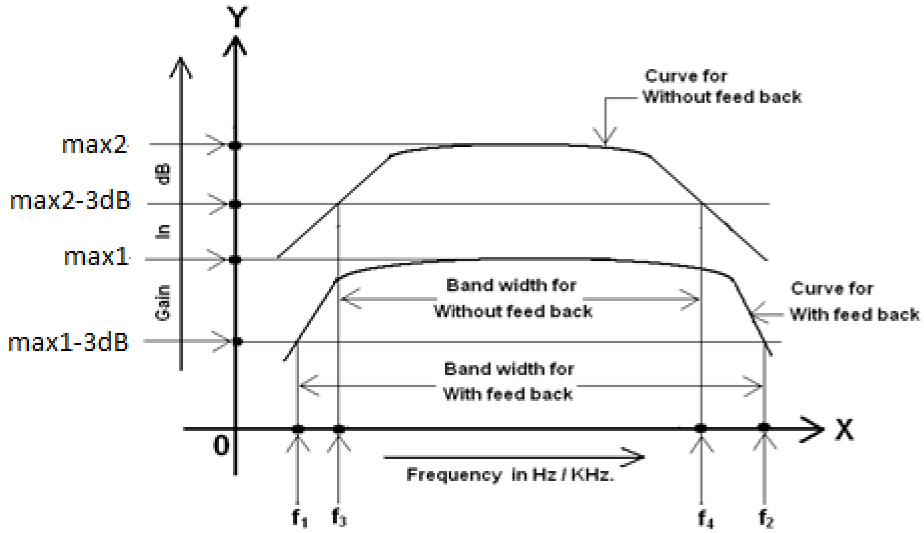
14. Calculated the Voltage gain by using the formula of $\boldsymbol{V_o / V_i}$ and Gain in dB by using the formula of $20log_{10}(A_V)$ in both tabular forms of *with feed back* and *without feed back amplifiers*.
15. Drawn the graphs of both amplifiers in single graph sheet.
16. While drawing the graph taken the *frequency* on X-axis and *Gain in dB* on Y-axis.
18. Finally calculated the *bandwidth* of both amplifiers from this graph sheet as per the following formulas,
    i). For *Current series- series feed back amplifier (With feed back*) (BW) = $f_2 - f_1$
    ii). For *Current series – series (Without feed back*) (BW) = $f_4 - f_3$
19. We have noted down that the *band width* of *with feed back amplifier* is high as compared to the *without feed back amplifier.*

TABULAR COLUMN :

	Input Voltage (V_i) = 20mV_{P-P} (0.02V) is constant for all readings (For With F/B & Without F/B)								
	For With Feed back Amplifier					**For Without Feed back Amplifier**			
Sl. No.	**Frequ-ency In Hz/KHz.**	**Voltage (V_O) In mVolts.**	**Voltage Gain A_V= V_o/V_i**	**Gain in dB= $20log_{10}$ (A_V)**		**Frequ-ency In Hz/KHz.**	**Output Voltage (V_O) In mVolts.**	**Voltage gain A_V=V_o/V_i**	**Gain in dB = $20log_{10}$ (A_V)**
1	20 Hz.								
2	100 Hz.								
3	200 Hz.								
4	500 Hz.								
5	1 KHz.								
6	200KHz.								
7	400KHz.								
8	600KHz.								
9	800KHz.								
10	1 MHz.								

EXPECTED GRAPH :

The following graph shows for *Current Series-Series feedback amplifier* for both *with feedback* and *without feedback.*



PARAMETERS :

1). Band width of frequency response curve for
frequency verses gain in dB. (With feed back) = $f_2 - f_1$ =

2). Band width of frequency response curve for
frequency verses gain in dB. (Without feed back) = $f_4 - f_3$ =

RESULT :

I have drawn the graph for frequency response of a *Current series-series feedback amplifier* for both *with feedback* and *without feedback.*

VIVA VOCE QUESTIONS:

1. What is feedback?

2. What are the advantages of negative feedback?

3. What are the feedback topologies?

4. Example for Current series feedback amplifier.

5. What are the different types of feed back amplifiers ?

Experiment No. : 05 **Date :**

Name of the Experiment : CURRENT SHUNT – SHUNT FEED BACK AMPLIFIER

AIM :

To plot the frequency response curve of a *Current shunt - shunt feedback Amplifier* for both *with feedback & without feedback.*

APPARATUS :

1. Regulated power supply (RPS)	-----	1 No.
2. Cathode Ray Oscilloscope (CRO)	-----	1 No.
3. Function generator	-----	1 No.
4. Probes	-----	1 No.
3. Bread board	-----	1 No.
4. Connecting wires	-----	A few Nos.

COMPONENTS :

1. Transistor BC 547	-----	2 No.
2. Resistors : i). 100Ω, 100KΩ	-----	Each 1 No.
ii). 1 KΩ, 2.2 KΩ, 10 KΩ, 47 KΩ	-----	Each 2 No.
3. Capacitors :		
i). 0.22 µF / 10 µF	-----	3 No.
ii). 10µF	-----	1 No.
iii). 22 µF	-----	3 No.

THEORY :

In the current shunt feedback circuit, a fraction of the output voltage is applied in series with the input voltage through the feedback circuit. This is also known as **series-driven shunt-fed** feedback i.e., a series-parallel circuit.

As the feedback circuit is connected in series with the output, the output impedance is increased and due to the parallel connection with the input, the input impedance is decreased.

CIRCUIT DIAGRAM :

Current shunt – shunt feed back amplifier – With feed back :

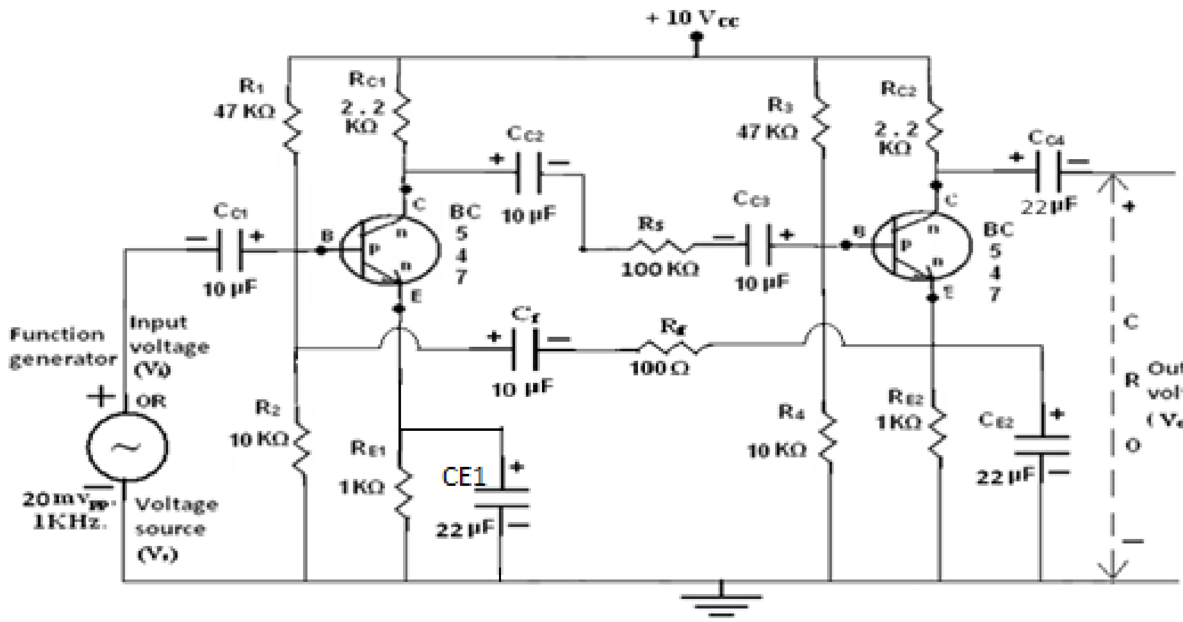


Fig : Current Shunt - Shunt Feed back Amplifier - With feed back

Current series-Series feed back amplifier – Without feed back :

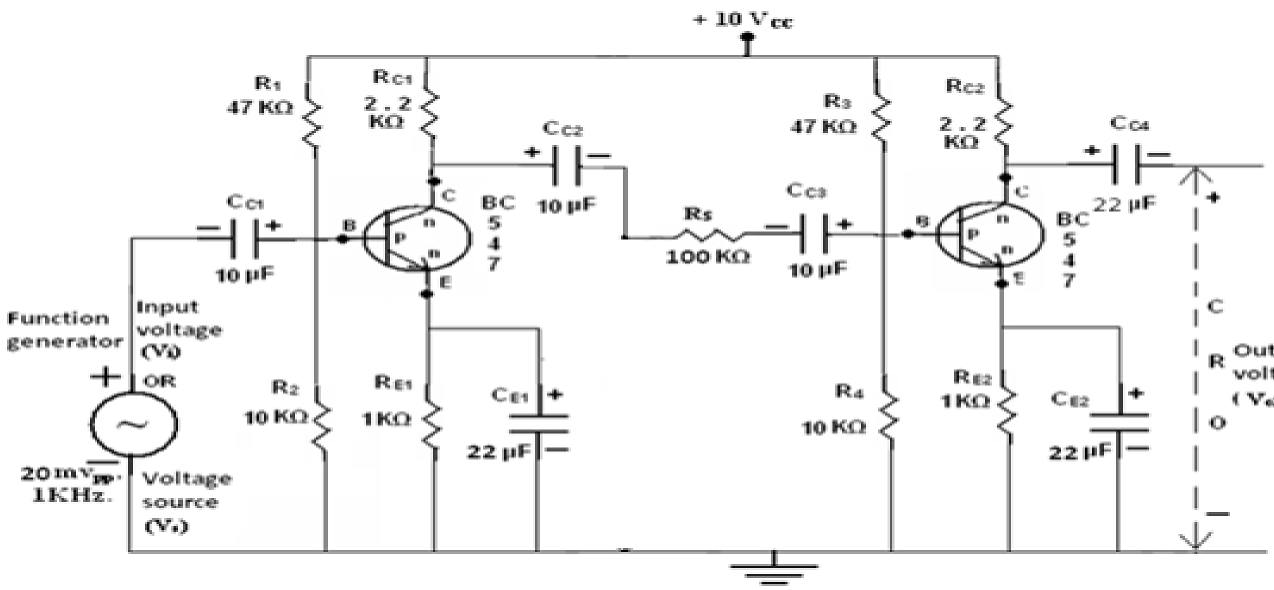


Fig : Current Series - series feed back Amplifier - Without feed back

PROCEDURE :

1. Connections are made as per the circuit diagram.
2. Initially connected the *CRO* across the *Function generator.*
3. Switched **ON** the Cathode ray oscilloscope (CRO) and Function generator.
4. Applied the 2*0 mV*$_{pp}$ *, 1Khz sine wave* signal to the circuit from *Function generator* by observing in the *CRO*.

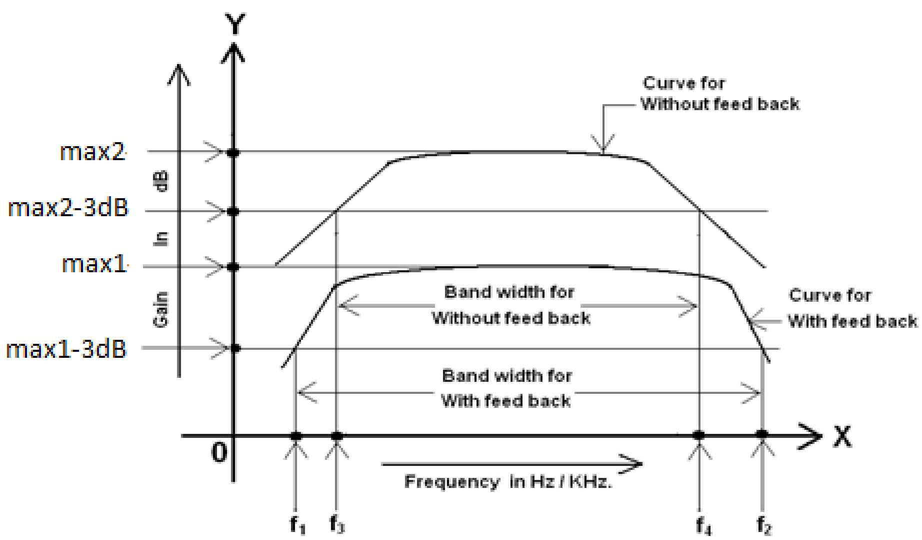
5. We have kept this *20 mV_{pp}* input voltage (Which has been applied from function generator) as constant for all steps of frequency while taking the readings for *Current shunt feedback amplifier* for both with feedback & without feedback .
6. Disconnected the CRO from the function generator .
7. Now Connected the CRO at output side to measure the peak to peak output voltage.
8. Applied the $+V_{CC}$ as 10V to the circuit from the *Regulated power supply (RPS).*
9. Later we have noted down the readings for output voltage in the tabular form of *with feedback*. from the CRO, by varying the different steps of frequency (i.e. 20Hz, 100Hz, 200Hz, 500Hz, 1KHz, 200KHz, 400KHz, 940KHz, 940KHz, 1MHz.) in the function generator.
10. After this we removed the feedback capacitor (C_f) & resistor (R_f) from the circuit completely.
11. Now the circuit is became as the *without feedback amplifier.*
12. Again we have noted down the readings for output voltage in the tabular form of *without feedback* .from the CRO, by varying the different steps of frequency (i.e. 20Hz, 100Hz, 200Hz, 500Hz, 1KHz, 200KHz, 400KHz, 940KHz, 940KHz, 1MHz.) in the function generator.
13. We have observed that the readings of *without feedback amplifier's* output voltage is greater than the *with feedback amplifier.*
14. Finally we switched **OFF** the function generator, cathode ray oscilloscope and regulated power supply.
15. Calculated the Voltage gain by using the formula of V_o / V_i and Gain in dB by using the formula of $20\log_{10}(A_V)$ in both tabular forms of *with feedback* and *without feedback amplifiers*.
16. Drawn the graphs of both amplifiers in single graph sheet.
17. While drawing the graph taken the *frequency* on X-axis and *Gain in dB* on Y-axis.
18. Finally calculated the *bandwidth* of both amplifiers from this graph sheet as per the following formulas,
    i). For *Current shunt feedback amplifier (With feedback)* (BW) = $f_2 - f_1$
    ii). For *Current shunt feedback amplifier (Without feedback)* (BW) = $f_4 - f_3$
19. I have noted down that the *band width* of *with feedback amplifier* is high as compared to the *without feedback amplifier.*

	Input Voltage (V_i) = 20mV$_{P-P}$ (0.02V) is constant for all readings (For With F/B & Without F/B)								
	For With Feed back Amplifier					**For Without Feed back Amplifier**			
Sl. No.	**Frequency In Hz/KHz.**	**Voltage (V_O) In mVolts.**	**Voltage Gain $A_V = V_o/V_i$**	**Gain in dB= $20\log_{10}(A_V)$**		**Frequency In Hz/KHz.**	**Output Voltage (V_O) In mVolts.**	**Voltage gain $A_V=V_o/V_i$**	**Gain in dB = $20\log_{10}(A_V)$**
1	20 Hz.	0	0	0		20 Hz.	0	0	0
2	100 Hz.	100	5	13.97		100 Hz.	1900	95	39.55
3	200 Hz.	200	10	20		200 Hz.	2000	100	40
4	500 Hz.	400	20	26.02		500 Hz.	2200	110	40.82
5	1 KHz.	800	40	32.04		1 KHz.	4800	240	47.60
	Continued in next page								

	Input Voltage (V_i) = 20mV_{P-P} (0.02V) is constant for all readings (For With F/B & Without F/B)								
	For With Feed back Amplifier					**For Without Feed back Amplifier**			
Sl. No.	**Frequ-ency In Hz/KHz.**	**Voltage (V_O) In mVolts.**	**Voltage Gain A_V= V_o/V_i**	**Gain in dB= $20log_{10}$ (A_V)**		**Frequ-ency In Hz/KHz.**	**Output Voltage (V_O) In mVolts.**	**Voltage gain $A_V=V_o/V_i$**	**Gain in dB = $20log_{10}$ (A_V)**
6	200KHz.	1200	60	35.56		200KHz.	1000	50	33.97
7	400KHz.	700	35	30.88		400KHz.	600	30	29.54
8	600KHz.	200	10	20		600KHz.	300	15	23.52
9	800KHz.	200	10	20		800KHz.	300	15	23.52
10	1 MHz.	200	10	20		1 MHz.	200	10	20

EXPECTED GRAPH :

The following graph shows for *Current shunt-shunt feedback amplifier* for both *with feedback* and *without feedback.*



PARAMETERS :

1). Band width of frequency response curve for frequency verses gain in dB. (With feed back) = $f_2 - f_1$ = 550KHz. – 0.700KHz. = 549.30KHz.

2). Band width of frequency response curve for frequency verses gain in dB. (Without feed back) = $f_4 - f_3$ = 60KHz. – 0.800KHz. = 59.20KHz.

RESULT : I have obtained the frequency response curve for Current shunt feed back amplifier.

VIVA VOCE QUESTIONS:

1. What is feedback?

2. What are the input and output impedances for current shunt feedback Amplifier.

3. Applications of current shunt feedback Amplifier.

4. What are the feedback topologies?

5. What is Band Width?

6. What is Frequency Response?

Experiment No. : 06 **Date :**

Name of the Experiment : CLASS A POWER AMPLIFIER

AIM :

1. To verify / plot the output signal (sine wave form) of a given *Class-A Series-fed Power Aamplifier* byusing software & hardware .
2. To calculate the conversion efficiency of a given amplifier.

APPARATUS :

Software :

1. System ------------------ 1 No.

Multisim software

Hardware :

1. Regulated power supply (RPS) ----------------------- 1 No.
2. Cathode Ray Oscilloscope (CRO) ---------------------------- 1 No.
3. Function generator --------------------------------------- 1 No.
4. Probes ------------------------- 1 No.
5. Ammeters : (0-10)mA Digital /Analog DC Type ----------- 1 No.
6. Bread board ------------------------- 1 No.
7. Connecting wires ------------------------- A few Nos.

COMPONENTS :

1. Transistor BC 547 ------------------------- 1 No.
2. Resistors : 1KΩ, 10 KΩ, 47KΩ ------------------------- Each 1 No.
3. Capacitors : 0.22 µF ------------------------- 2 No.
   33 µF -------------------------- 1 No.

THEORY :

Class A power amplifier is a type of power amplifier where the output transistor is ON full time and the output current flows for the entire cycle of the input wave form. Class A power amplifier is the simplest of all power amplifier configurations. They have high fidelity and are totally immune to crossover distortion. Even though the class A power amplifier have a handful of good feature, they are not the prime choice because of their poor efficiency. Since the active elements (transistors) are forward biased full time, some current will flow through them even though there is no input signal and this is the main reason for the inefficiency

The theoretical maximum efficiency of a Class A power amplifier is 50%. In practical scenario, with capacitive coupling and inductive loads (loud speakers), the efficiency can come down as low as 25%. This means 75% of power drawn by the amplifier from the supply line is wasted. Majority of the power wasted is lost as heat on the active elements (transistor).As a result, even a moderately powered Class A power amplifier require a large power supply and a large heatsink.

CIRCUIT DIAGRAM :

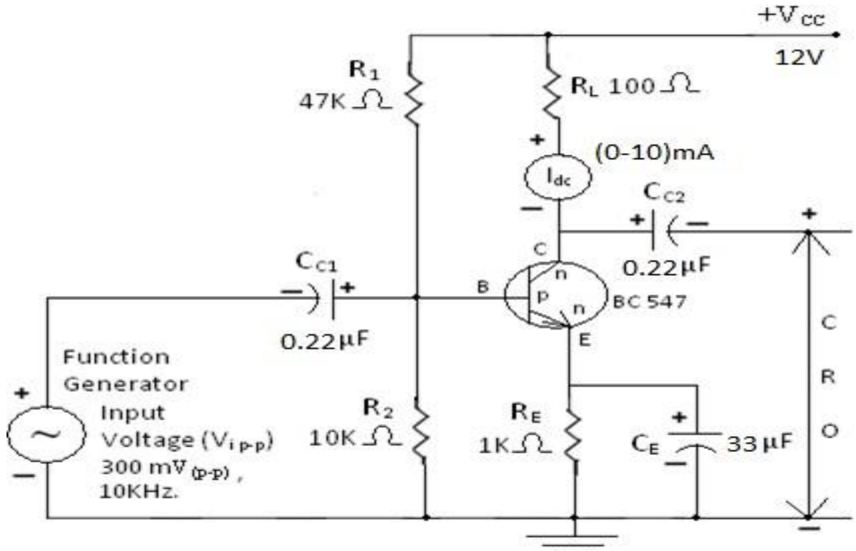


Figure : Circuit diagram of Class-A Power amplifier

PROCEDURE – SOFTWARE :

1. Picked up the components from *components bar* in multisim software as per the circuit diagram.
2. Made the connections as per the circuit diagram.
3. Set the 300 $mV_{p\text{-}p}$ (as input voltage) , *10 Khz* (as input frequency) *sine wave* signal to the circuit from the*Function generator* .
4. Noted down the *Input voltage*(V_i) , *Input frequency* against the corresponding columns of the tabular form of *practical calculations.*
5. Set the *supply voltage* 12V as V_{CC} to the circuit as shown in the circuit diagram.
6. To simulate this circuit click on *Run* button in *tool bar.*
7. Observed the *sine wave* signal in CRO and drawn this signal on the graph sheet.
8. Calculated the *output voltage* $(Vo_{p\text{-}p})$, *time period* **(*T*)**, *frequency* **(*f*)** from the graph, and noted down these values against the corresponding columns in the tabular form of practical calculations.
9. Noted down the *supply voltage* (V_{CC}) and *collector dc current* $I_{(dc)}$ at *Quiescent* condition i.e. when no signal is applied i.e. by disconnected the *function generator* from the circuit against the corresponding columns of the tabular form of *practical calculations.*
10. Stop the simulation by click on *Run* button in *tool bar*.
11 Shut down the system safely.
12. Later calculated and noted the input *dc power* $P_i(dc)$, *output ac power* $P_o(ac)$ and *% of efficiency* (η)by using the formulas which are mentioned in the corresponding columns of the tabular form of practical calculations.
13. Noted that the practical value should be less than the *Typical Max. efficiency value i.e.* ***25.4%****.*

PROCEDURE – HARDWARE :

1. Connections are made as per the circuit diagram.
2. Initially connected the *CRO* across the *Function generator.*

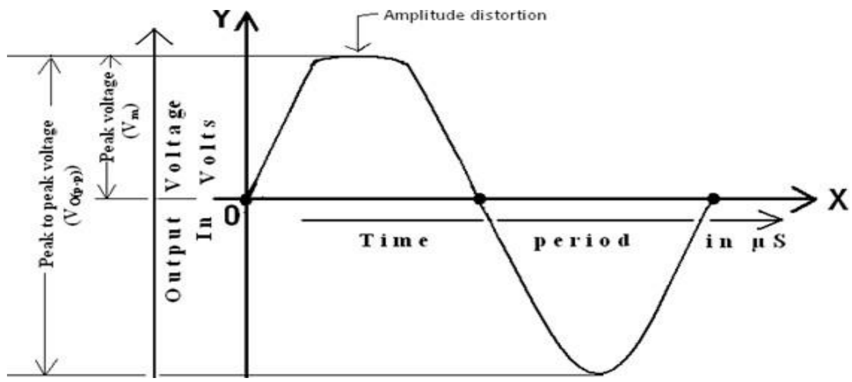
3. Switched **ON** the Cathode ray oscilloscope (CRO) and Function generator.
4. Applied the 300 mV_{p-p} , *10 Khz sine wave* signal to the circuit from the *Function generator* byobserving on the ***crt*** of the *CRO*.
5. Later connected the *CRO* across R_L i.e at output side.
6. Now switched **ON** the Regulated Power Supply (RPS) and apply the *supply voltage* 12V as V_{CC} to thecircuit as per shown in the figure.
7. Observed the *sine wave* signal on the CRT of the CRO and draw this signal on the graph sheet.
8. Now noted down the *collector dc current* $I_{(dc)}$ at *Quiescent* condition i.e. when no signal is applied and *supply voltage* (V_{CC}) by disconnected the *function generator* from the circuit against the corresponding columns in the tabular form of *practical calculations.*
9. Switched **OFF** the *function generator, RPS, CRO.*
10. Noted down the *Input voltage*(V_i) , *Input frequency* against the corresponding columns in the tabular form.
11. Calculated the *output voltage* (Vo_{p-p}) , *time period* (*T*), *frequency* (*f*) from the graph, and noted downthese values against the corresponding columns in the tabular form.
12. Later calculated the *Input dc power* $P_i(dc)$, *output ac power* $P_o(ac)$ and *% of efficiency* (η) by using The formulas which are mentioned in the corresponding columns in the tabular form.
13. Noted that The practical value should be less than the *Typical Max. efficiency value i.e.* ***25.4%.***

PRACTICAL CALCULATIONS – SOFTWARE & HARDWARE :

Sl.No.	Name of the parameter	Value for Software		Value For Hardware
01.	Input Voltage $(V_i)_{p-p}$ (In mV).	300		
02	Input frequency (In Khz.).	10		
03	Supply DC Voltage (V_{CC}) (in Volts.)	10		
04	Output voltage $V_{O(p-p)}$ (In volts.).			
05	Time period (T) for output signal (In ms)			
06	Fequency for output signal = 1/ T (In Khz.)			
07	Collector dc current (I_{dc}) (At quesient condition i.e. When no input signal is applied) (In mA.).			
08	Collector DC current when sine wave (AC) signal is applied as input signal (I_{ac})			
09	Input DC power $P_i(dc) = I_{dc} \times V_{CC}$ (In Watts).			
10	Output ac power $P_o(ac) = \frac{V_{O(P-P)}^2}{8R_L}$ (In Watts)			
11	% of efficiency (η) = [$P_O(ac)$ / P_i (dc)] ×100 =			
12	Typical Max. efficiency (η) =	25.40%		

EXPECTED WAVEFORM – SOFTWARE & HARDWARE :

The following waveform shows the output signal of *Class A Series-fed Power Amplifier* .



RESULT – SOFTWARE & HARDWARE :

I have verified / drawn the output signal and calculated the conversion efficiency of given *Class-ASeries-fed Power amplifier.*

VIVA VOCE Questions:

1. What is Power Amplifier?
2. Classifications of power Amplifiers.
3. Efficiency of class A power Amplifier.
4. What is the amplitude (Harmonic) Distortion?
5. Where is the Q point in class A power Amplifier?
6. Applications of class A power Amplifier.
7. What are the disadvantages of class A power Amplifier.

.

Experiment No. : 7 **Date :**

Name of the Experiment : SINGLE TUNED VOLTAGE AMPLIFIER

AIM :

To obtain the frequency response curve of *Single tuned voltage amplifier*.

APPARATUS :

1. Regulated power supply (RPS) ----------------------------- 1 No.
2. Cathode Ray Oscilloscope (CRO) ----------------------------- 1 No.
3. Function generator -- 1 No.
4. Decade Inductance box (DIB) ------------------------------------ 1 No.
5. Decade capacitance box (DCB) ----------------------------------- 1 No.
6. Probes --- 1 No.
7. Bread board --- 1 No.
8. Connecting wires -- A few Nos.

COMPONENTS :

1. Transistor BC 547 --- 1 No.
2. Capacitors :
   i). 10 µF -- 1 No.
   ii). 22 µF --- 2 No.
3. Resistors :
   i). 100 KΩ , 10KΩ, 100 Ω -- Each 1 No.

THEORY :

Tuned amplifiers are mainly preferred to amplify the high-frequency signals in wireless communication. The tuned amplification works based on the tuning circuit implied as load. The range of the frequencies defined for a particular amplification circuit can be fixed or dynamic based on applications. The tuning circuit present at the load consists of an inductor and capacitor. For dynamic frequencies, the values of capacitance should be varied. These amplifiers are very advantageous due to its appealing large bandwidths. The increment in bandwidth is based on the number of tuning circuits present at the load. There are three types of most frequently used tuned amplifiers they are single tuned amplifier, double-tuned amplifier and stagger tuned amplifier.

Definition: A tuned amplifier consists of a single tuning circuit at the load can be defined as a single tuned amplifier. It is a multi-stage amplifier, where each stage of this amplifier must be tuned with the same frequencies. For example, tuning a radio station. If the desired carrier wave is passed and matches the defined range of passband frequency, then the radio station is tuned otherwise it is blocked.

CIRCUIT DIAGRAM :

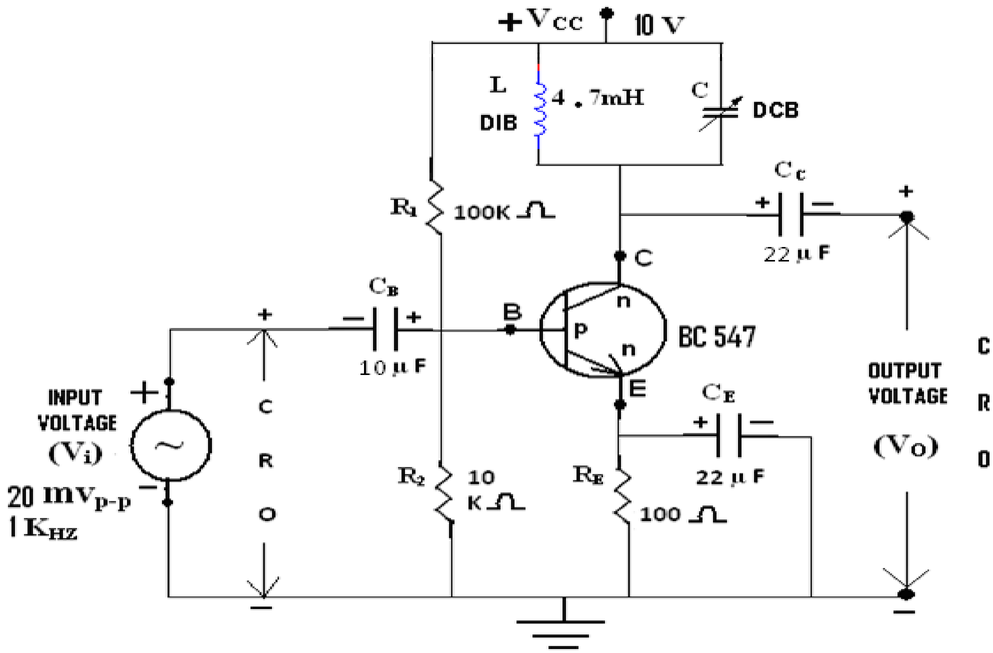


Figure: Circuit diagram of single tuned voltage amplifier.

THEORETICAL CALCULATIONS :

!). When L=4.7mH, fr = 10KHz., Then C = ?

We have $f_r = \dfrac{1}{2\pi\sqrt{LC}}$ OR $C = \dfrac{1}{\left[2\pi f_r\sqrt{L}\right]^2}$

OR $C = \dfrac{1}{4\pi^2} \times \dfrac{1}{f_r^2 L}$ OR $C = \dfrac{0.0253}{f_r^2 L}$

$C = \dfrac{0.0253}{\left[10\times10^3\right]^2 \times 4.7\times10^{-3}}$

= 54Kpf OR 54 nF

2). When L=4 . 7mH, f = 50KHz., Then C = ?

We have $C = \dfrac{0.0253}{f_r^2 L}$

$= \dfrac{0.0253}{\left[50\times10^3\right]^2 \times 4.7\times10^{-3}}$

= 2.16Kpf OR 2.16nF

PROCEDURE :

1. I have connected the circuit as per the circuit diagram which is shown above. Initially connected the CRO across the function generator as per shown in the circuit diagram to set the input signal.
2. Switched *ON* the *CRO* and *function generator*.
3. Applied the input signal as *sine wave form* having the values of $20m_{p\text{-}p}$, 1KHz.from the function generator by observing in the CRO.
4. Kept the amplitude of the input signal as constant as $20mV_{p\text{-}p}$ for all frequency steps until the experiment would completed.
5. Later removed the CRO and connected it across the capacitor C_C to observe the peak to peak output voltage.
6. Now set the C=54Kpf and L=4.7mH to take readings in tabular form-1
7. Switched *ON* the *RPS* and kept the 10V as V_{CC}.

8. Initially varied the frequency of input signal until the maximum output voltage could obtained, and noted this to the corresponding frequency value in the tabular form -1 .
9. Later calculated and noted the peak to peak output voltage to the remaining corresponding frequency values which are available in the tabular form-1.
10. Later set the C=2.16Kpf and L=4.7mH to take readings in tabular form-2
11. Repeat the same procedure from point 8 to 9.
12. Now calculated and noted down the values of *voltage gain*(A_V) and *gain in dB* to the corresponding values of *output voltage*(V_O) & *input voltage*(V_i) by using the formulas given below,

    Voltage gain $(A_v) = V_o / V_i$ *and* *Gain in dB* $= 20\log_{10}(A_v)$.

    These values has been noted in the both tabular forms.
13. Plotted the graphs for both tabular forms (frequency response curves) as per given below,
    a). frequency on X-axis & gain in dB on Y-axis.
    b). frequency on X-axis & voltage gain on Y-axis.
14. Calculated and noted the *band width* & *resonant frequency* from both frequency response curves by using the following formula,

    $$\text{Band width} = f_2 - f_1.$$

TABULAR FORMS :

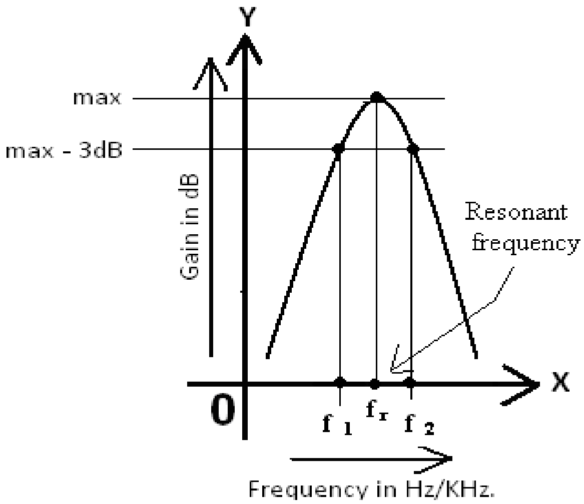
TABULAR FORM – 1: **Input Voltage (V_i) = 20mV_{P-P} (0.02V) is constant for all readings.** When f_r = 10Khz. , C = 54Kpf, L = 4.7mH						**TABULAR FORM – 2 :** **Input Voltage (V_i) = 20mV_{P-P} (0.02V) is constant for all readings.** When f_r = 50 Khz., C = 2.16 Kpf, L=4.7mH		
Sl. No	**Freque-ncy in Hz / KHz.**	**Output Voltage (V_o) in V**	**Voltage gain (A_v) = V_o /V_i**	**Gain in dB =20log₁₀(A_v)**		**Output Voltage (V_o) In V**	**Voltage Gain (A_v) = V_o /V_i**	**Gain in dB =20log₁₀(A_v)**
1	10 Hz.							
2	500 Hz.							
3	1 KHz.							
4	5 KHz.							
5								
6	20KHz.							
7	50KHz.							
8	100 KHz.							
9	200 KHz.							
10	500 KHz.							
11	1 MHz.							

EXPECTED GRAPH :

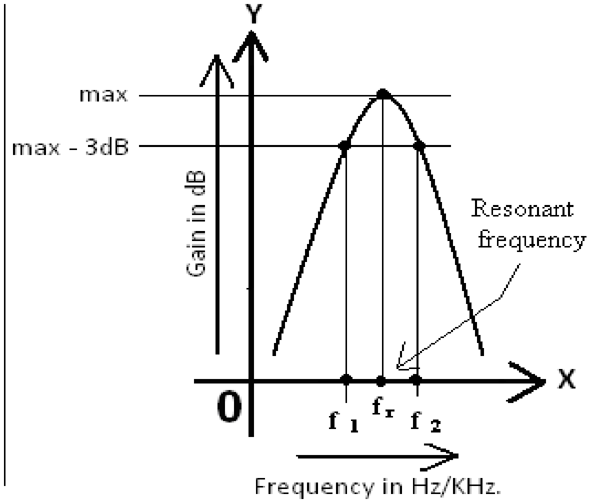
The following graphs shows the frequency response curve for *single tuned voltage* a*mplifie*

.

A). When f_r = 10 Khz. , C = 54 Kpf, L = 4.7mH



B). When f_r = 50 Khz. , C = 2.16 Kpf, L = 4.7mH



PRACTICAL CALCULATIONS :

When f_r = 10 Khz. , C = 54 Kpf, L = 4.7mH	**When f_r = 50 Khz. , C = 2.16 Kpf, L = 4.7mH**
1). Band width = $f_2 - f_1$ = 2). Resonant frequency (f_r) =	1). Band width = $f_2 - f_1$ = 2). Resonant frequency (f_r) =

TABULAR FORM -3 :

The following tabular form shows the comparison between the theoretical and practical resonant frequency values.

Sl.No.	**Inductor (L) (Note down from the theoretical calculations)**	**Capacitor (C) (Note down from the theoretical calculations)**	**Theoretical Resonant frequency (f_r) (Note down from the theoretical calculations)**	**Practical Resonant frequency (f_r) (Note down from the graph)**	**Max. voltage gain in dB at resonant frequency . (Note down from the graph)**
1.	4.7mH	54Kpf	10KHz.		
2.	4.7mH	2.16Kpf	50KHz.		

CONCLUSSION :

If I observed in the tabular form-3 the voltage gain of the output signal is maximum when the practical resonant frequency value is approximately equal to the theoretical resonant frequency value.

APPLICATIONS :

Mainly uses in the radio receivers to tuned the appropriate signal / station which is transmitted in relay station.

RESULT :

I have drawn the frequency response curve and calculated the values of band width, and resonant frequency of a *single tuned voltage amplifier*.

VIVA VOCE Questions:

1. What is single Tuned Amplifier?

2. What is Q factor?

3. What is tank circuit?

4. Mention Applications of single Tuned Amplifier.

5. What is the resonant frequency of single tuned Amplifier?

6. Difference between single tuned and double tuned Amplifier?

7. What is frequency response?

Experiment No. : 8 **Date :**

Name of the Experiment : RC PHASE SHIFT OSCILLATOR

AIM :

To draw the *sine wave form* and to calculate its frequency values of a given *RC Phase shift Oscillator.*

APPARATUS :

1. Regulated power supply (RPS) ------------------------ 1 No.
2. Cathode ray oscilloscope -------------------------------- 1 No.
3. Decade Resistance Box (DRB) ----------------------- 1 No.
4. Decade Capacitance Box (DCB) ---------------------- 3 No.
5. Bread board -- 1 No.
6. Probes -- 1 No.
7. Connecting wires -- 1 No.

COMPONENTS :

1. Resistors : 1KΩ, 4.7 KΩ, 47 KΩ, 10 KΩ ------------- Each 1 No.
   10 KΩ -------------------------------------- 3 No.
2. Capacitors : 0.047µF, 1000 µF -------------------------- 1 No.
3. Transistor : BC547 ------------------------------------- 1 No.

THEORY :

A phase shift oscillator can be defined as; it is one kind of linear oscillator which is used to generate a sine wave output. It comprises of an inverting amplifier component like operational amplifier otherwise a transistor. The output of this amplifier can be given as input with the help of the phase shifting network. This network can be built with resistors as well as capacitors in the form of a ladder network. The phase of the amplifier can be shifted to 1940 at the oscillation frequency by using a feedback network to provide a positive response. These types of oscillators are frequently used as audio oscillators on audio frequency. This article discusses an overview of RC phase shift oscillator.

RC phase-shift oscillator circuit can be built with a resistor as well as a capacitor. This circuit offers the required phase shift with the feedback signal. They have outstanding frequency strength and can give a clean sine wave for an extensive range of loads. Preferably an easy RC network can be expected to include an o/p which directs the input with 94°.

CIRCUIT DIAGRAM :

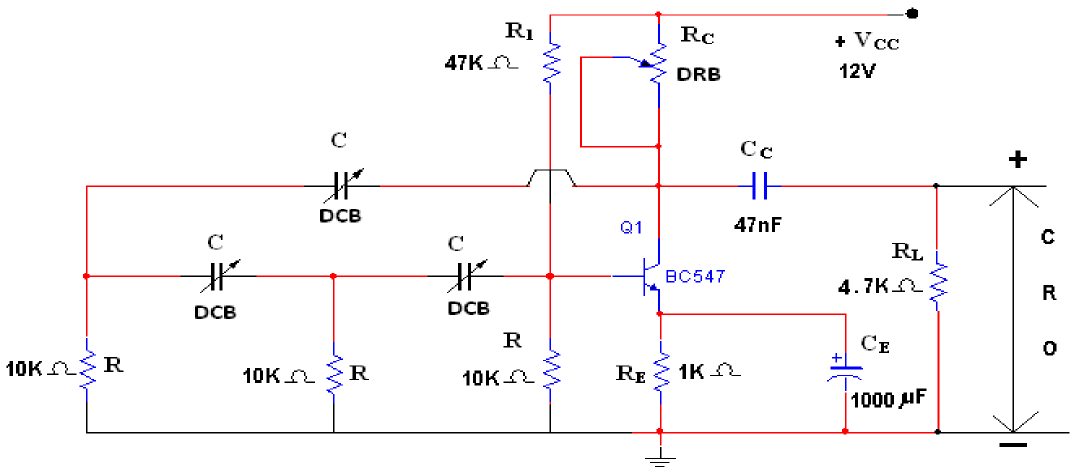


Figure: Circuit diagram of RC phase shift oscillator.

PROCEDURE :

1. Made the connections as per the circuit diagram.
2. Kept the V_{CC} value as 12V.
3. Kept the *Capacitor C* values as 1nF (0.001µF or 1Kpf) in DCB.
4. Varied the R_C (i.e. Appx. 4.3KΩ) until we get *sine wave form* which is consist the $V_{O(p-p)}$ is approximately 6V because this circuit is designed to get the output voltage as $6V_{(p-p)}$ in the *CRO*.
5. Now noted the value of R_C to the corresponding *C* value in tabular form.
6. We observed the *Sine wave form* as a output signal in the *CRO*.
7. Now calculated and noted the *theoretical frequency value* (f_O) to the corresponding *C* value in the tabular form by using the formula given below,

$$\frac{1}{2\pi RC\sqrt{6+4(R_c/R)}}$$

8. Drawn the *sine wave form* on the graph by taking the *time period* on X-axis and *amplitude($V_{O(p-p)}$)* on Y-axis.
9. Calculated the frequency and output voltage ($V_{O\,(p-p)}$) values from the graph then noted in the Columns of *practical frequency* and *output voltage* in the tabular form respectively.
10. Repeat the same procedure from points 4 to 9 for corresponding C values which are given below,
    - a). 2.2 nF (0.0022 µF or 2.2Kpf).
    - b). 3.3 nF (0.0033 µF or 3.3Kpf).
    - c). 10.0 nF (0.01 µF or 10Kpf).
11. Switch *OFF* the RPS and CRO.
12. We compared that *theoretical frequency* value (f_O) and *practical frequency* values are approximately same.

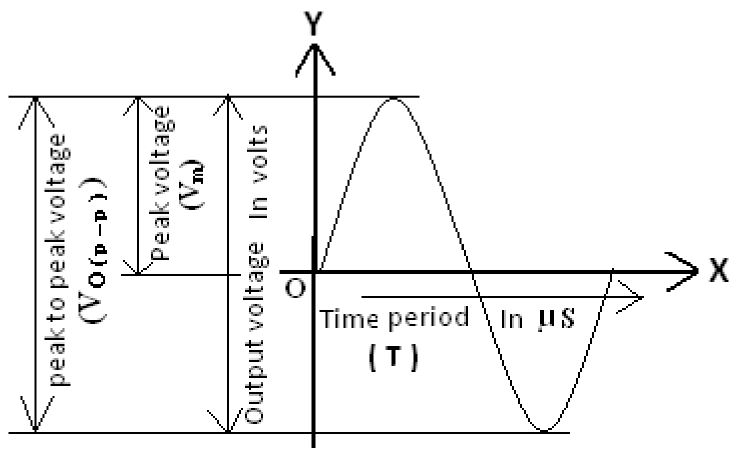
TABULAR FORM / CALCULATIONS :

Sl. No.	Reistor (R) In KΩ	Capacitor (C) In Kpf	$R_C = R_{selected} - \left[\frac{\text{Setting in \%} \times R_{selected}}{100}\right]$ In KΩ	Theoretical frequency (f_O) $\frac{1}{2\pi RC\sqrt{6+4(R_C/R)}}$ In Hz/KHz.	Practical Time Period (In μS)	Practical Fre quency In Hz/KHz.	Output Voltage ($V_{O(p-p)}$) In Volts
1	10	1					
2	10	2.2					
3	10	3.3					
4	10	10					

Note: If I observed in the tabular form, for which the C value of 10nF the peak to peak output voltage *($V_{O(p-p)}$)* is 4V approximately, but this circuit is designed to get the *($V_{O(p-p)}$)* value as 6V approximately. So for this C value we can't considered. If we observed regarding to frequency value, when the C value is increases then the frequency value is decreases.

EXPECTED WAVE FORM :

The following waveform shows the output signal for *RC phase shift Oscillator,*



RESULT :

I have drawn the output signal and calculated the frequency values of a given *RC phase shift oscillator.*

VIVA VOCE QUESTIONS:

1. What is positive feedback Amplifier?
2. State Barkhausen condition for oscillation.
3. What are the classifications of oscillators?
4. What are the types of RC oscillators?
5. What is the frequency of RC phase shift oscillator?
6. Applications of RC oscillators?
7. In RC phase shift oscillator, each RC section gives how much phase shift?
8. In AF oscillators which oscillators are used?

Experiment No. : 9	**Date :**
Name of the Experiment : LC OSCILLATOR	

AIM :

To draw the *sine wave form* and to calculate its frequency values of a given LC *Oscillator.*

APPARATUS :

1. Regulated power supply (RPS) ------------------------ 1 No.
2. Cathode ray oscilloscope -------------------------------- 1 No.
3. Decade Inductance Box (DIB) ------------------------- 1 No.
4. Decade Capacitance Box (DCB) ---------------------- 1 No.
5. Bread board --- 1 No.
6. Probes -- 1 No.
7. Connecting wires -- 1 No.

COMPONENTS :

1. Resistors : 1KΩ, 1.5 KΩ, 10 KΩ, 47 KΩ ----------- Each 1 No.
2. Capacitors : 0.1µF, 0.01µF ------------------------------ Each 1 No.
3. Transistor : BC547 ------------------------------------ 1 No.

THEORY :

Oscillator :

An amplifier that converts DC signal as input to AC signal as output like a square wave or sine wave etc. and variable frequency drive by using positive feedback is known as an oscillator. Oscillators that use capacitor C and inductor L in the circuit are termed as an LC circuit which falls into the category of a linear oscillator. There are various methods to design this LC oscillator. The most familiar oscillators among those LC oscillators are Colpitts oscillator and Hartley oscillator.

In these two types of oscillators, the Colpitts oscillator is the most used oscillator. Edwin Colpitts invented this Colpitts oscillator in 1918. Now here we can do the Colpitts oscillator experiment .

Colpitts Oscillator :

This oscillator forms a tank circuit by using two center-tapped capacitors in series with a parallel inductor and produces sinusoidal oscillations. The frequency of oscillations can be obtained by taking the values of the capacitors and inductors into consideration. In some ways, this oscillator is similar to the Hartley oscillator and in some ways, it varies.

The tank circuit connected between the collector and base of a transistor amplifier for obtaining sinusoidal output

CIRCUIT DIAGRAMS :

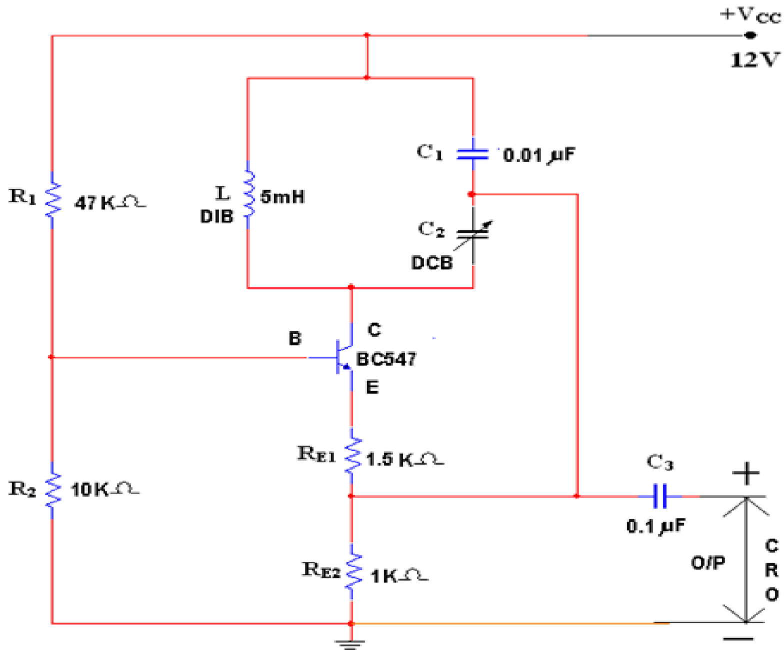


Figure: Circuit diagram of Colpitt's oscillator.

PROCEDURE :

1. Made the connections as per the circuit diagram.
2. Switched ***ON*** the *RPS* and *CRO*.
3. Set the V_{CC} value as 12V in *RPS*.
4. Set the *inductance(L)* value as 5mH in DIB .
5. Set the *Capacitor* C_2 value as 1nF (0.001µF or 1Kpf) in DCB.

6.. We observed *Sine wave form* as a output signal in the *CRO*.

7. Drawn the *sine wave form* on the graph by taking the *time period* on X-axis and *amplitude*($V_{O(p-p)}$) on Y-axis.
8. Calculated the frequency and output voltage ($V_{O\,(p-p)}$) values from the graph then noted in the columns of *practical frequency* and *output voltage* in the tabular form respectively.
9. Repeat the same procedure from points 5 to 7 for corresponding C_2 values which are given below,
   a). 2.2 nF (0.0022 µF or 2.2Kpf).
   b). 3.3 nF (0.0033 µF or 3.3Kpf).
10. Switch *OFF* the RPS and CRO.
11. Finally calculated and noted down the *theoretical frequency value* (F_O) by using the formula,

    $1 / (2\prod\sqrt{(LC_T)})$ in the tabular form.
12. I compared that *theoretical frequency* value (F_O) and *practical frequency* values are approximately same.

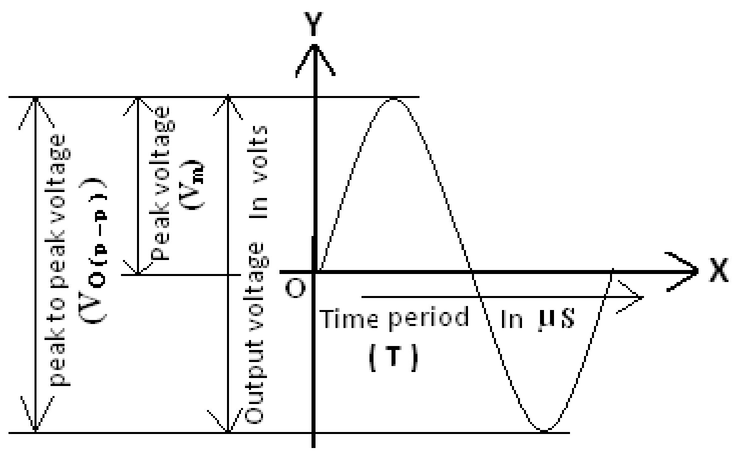
TABULAR FORM / CALCULATIONS :

Colpitt's oscillator :

Sl No.	Capa Citor (C_1)	Capa Citor (C_2)	Indu-ctor. (L) In mH	Total Capaci-tance (C_T) $= \frac{C_1 C_2}{C_1+C_2}$ In nF	Theoretical Frequency (f_o) = $\frac{1}{2\pi\sqrt{LC_T}}$ In KHz	Practical Time Period. In μS	Pract-Ical frequency In KHz.	Output voltage ($V_{O\ p-p}$) In Volts.
1.	10Kpf	1Kpf	5					
2.	10Kpf	2.2Kpf	5					
3.	10Kpf	3.3Kpf	5					

EXPECTED WAVEFORM :

The following waveform shows the output signal for Colpitt's Oscillator,



RESULT :

I have drawn the output signal and calculated the frequency values of a given Colpitt's oscillator.

VIVA VOCE Questions:

1. What is positive feedback Amplifier?
2. What are the conditions for oscillations?
3. What are the classifications of oscillators?
4. What are the types of LC oscillators?
5. What is the frequency of Hartley oscillator?
6. What is the frequency of Colpitts oscillator?
7. Applications of LC oscillators?
8. In Colpitts oscillator, feedback circuit consists of how many Inductors and capacitors?
9. In RF oscillators which oscillators are used?

Experiment No. : 10	**Date :**
Name of the Experiment : BISTABLE MULTIVIBRATOR	

AIM :

To conduct and verify the Bi stable multi vibrator and draw the waveforms.

APPARATUS :

1. Regulated power supply (RPS) ------------------------ 1 No.
2. Cathode ray oscilloscope -------------------------------- 1 No.
3. Function Generator ------------------------ 1 No.
4. Bread board --- 1 No.
5. Probes -- 1 No.
6. Connecting wires --- 1 No.

COMPONENTS :

1. Resistors : 694 Ω, 1KΩ, 10KΩ, 100 KΩ ----------- 2 No.
2. Capacitors : 0.1µF / 100nF ----------------------------- 2 No.
   0.33 / 330nF ------------------------------------- 1 No.
3. Transistor : BC547 -- 2 No.
4. Diodes 1N4007 ------------------------------------- 3 Nos.

THEORY :

Multivibrator :

Multivibrator is an electronic circuit which will work as two stage amplifier operating in both stable and astable mode.In the multivibrator the output of first stage is given to the second stage and the second stage output is again feed back to the first stage by this the cutoff state will become saturate and saturate

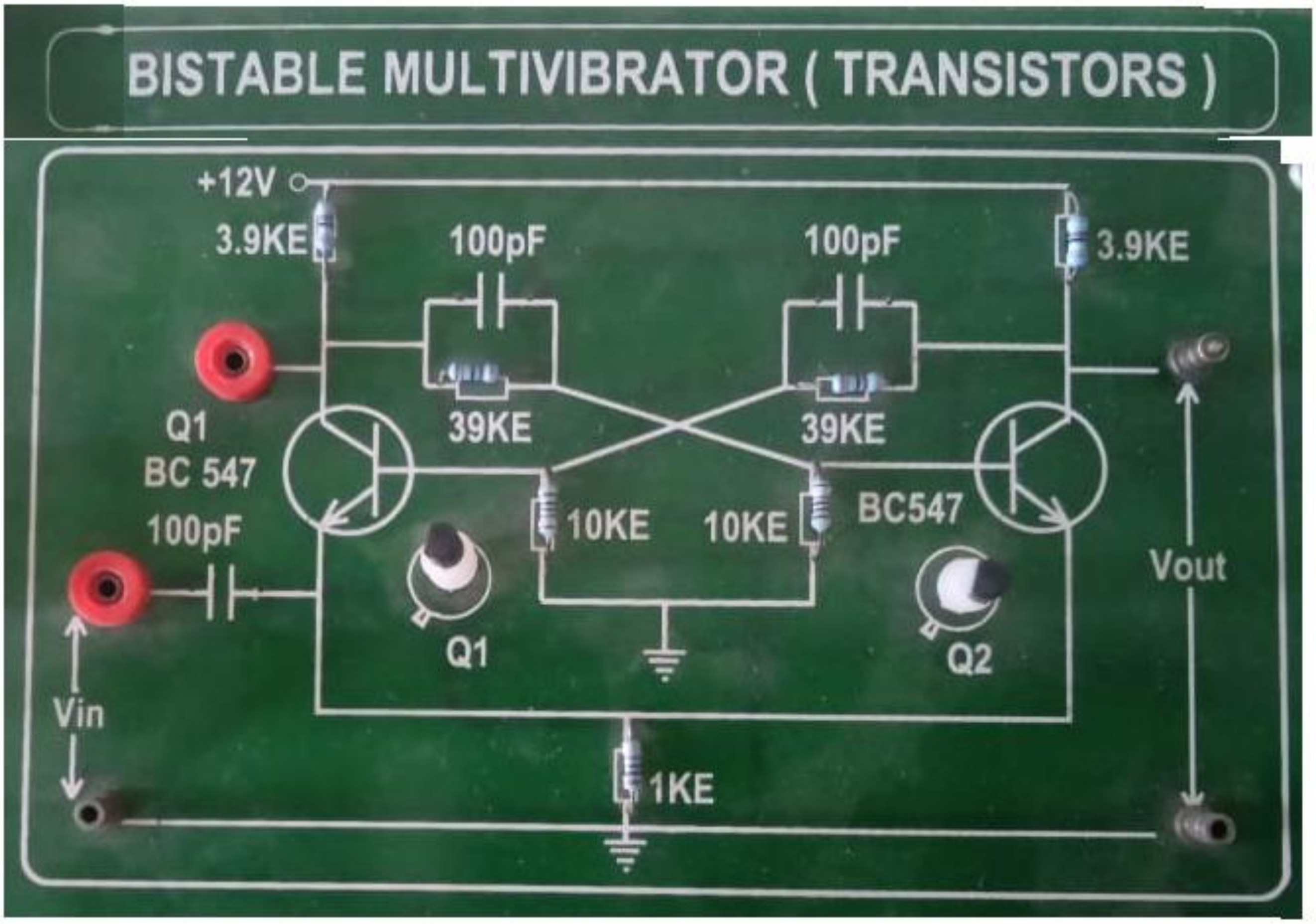
Bistable Multivibrator :

Bistable multivibrator, in which the circuit is stable in either state. It can be flipped from one state to the other by an external trigger pulse. This circuit is also known as a flip-flop. It can store one bit of information, and is widely used in digital logic and computer memory.

In other words a multivibrator which has both the state stable is called a bistable multivibrator. It is also called flip-flop, trigger circuit or binary.

When the input is below a different (lower) chosen threshold the output is low, and when the input is between the two levels the output retains its value. This dual threshold action is called hysteresis and implies that the Schmitt trigger possesses memory and can act as a bistable multivibrator (latch or flip-flop).

CIRCUIT DIAGRAMS :



Design Procedure:

$$R_C = \frac{V_{CC} - V_{CESAT}}{I_{Cmax}} = \frac{15 - 0.3}{15 \times 10^{-3}} = 1K\Omega$$

$$V_{B1} - \frac{-V_{BB}}{R1 + R2}R1 + \frac{-V_{CEsat}}{R1 + R2}R2$$

$-1.2 = (-15R_1 + 0.2R_2) / (R_1 + R_2)$; given $R1 = 10K\Omega$

$R2 = 100K\Omega$

$F_{max} = (R_1 + R_2)/2C\,R_1 R_2$ $R_1 = 10K\Omega$, $R_2 = 100K\Omega$ and $C = 0.1\mu F$

$= (10 + 100) X 10^3 / (2 X 0.3 X 10^{-6} X 10 X 100 X 10^6) = 55KHz$

PROCEDURE :

1. I Made the connections as per the circuit diagram
2. Applied the trigger pulse of 1KHz, $5V_{P-P}$ from function generator
3. Obtained waveforms at different points such as V_{B1}, V_{B2}, V_{C1} and V_{C2}
4. Traced the waveform at collector and base of each transistor with the help of dual trace CRO.
5. Noted the time relation waveforms.

EXPECTED WAVEFORM :

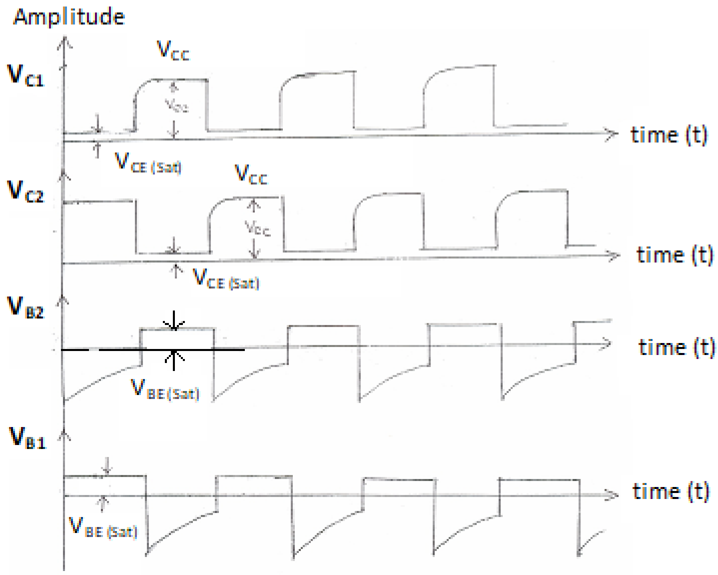


Fig : Waveforms of Bistable multivibrator

RESULT :

I have conducted and verified the Bistable multivibrator.

VIVA VOCE Questions:

1. What is Multi-vibrator?

2. Mention the Applications of Bi-stable Multi-vibrator.

3. Schmitt Trigger is basically which Multi- vibrator?

4. Bi-stable Multi-vibrator is having how many stable states?

5. What are the types of Multi-vibrators?

6. Mention advantages & disadvantages of Bi-stable Multi-vibrator.

7. Bi-stable Multi vibrator can be used for storing information. (True or False).

8. Compare Mono-stable, Bi-stable and Astable multi-vibrators.

9. Which Multi vibrator is Triggered oscillator?

10. Bi-stable circuit is also known as________________________. (Latch / Gate / Flip-Flop/ Bi directional circuit)

Experiment No. : 11 **Date :**

Name of the Experiment : MONO STABLE MULTIVIBRATOR

AIM :

To conduct and verify the Mono stable multi vibrator and draw the waveforms.

APPARATUS :

1. Regulated power supply (RPS) ------------------------ 1 No.
2. Cathode ray oscilloscope --------------------------------- 1 No.
3. Function Generator ------------------------ 1 No.
4. Bread board -- 1 No.
5. Probes -- 1 No.
6. Connecting wires -- 1 No.

COMPONENTS :

1. Resistors : 694 Ω, 1KΩ, 10 KΩ, 100 KΩ --------- Each 2 No
2. Capacitors : 0.1μF / 100nF --------------------------- 2 No.
   0.33 / 330nF ----------------------------- 1 No.
3. Transistor : BC547 ------------------------------------ 2 No.
4. Diodes 1N4007 ------------------------------ 3 Nos.

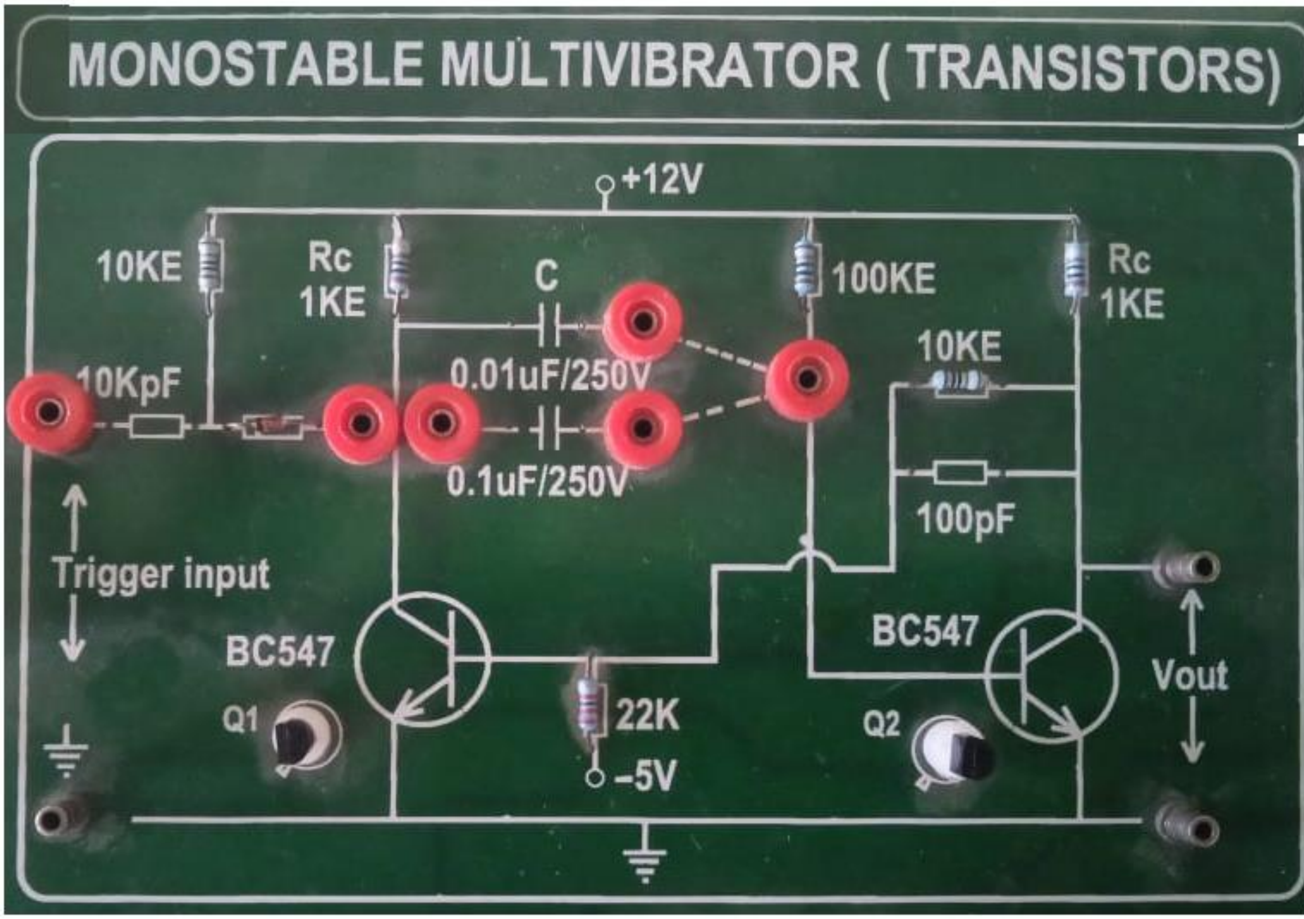
THEORY :

A monostable multivibrator, also called a one shot or a monoflop, is a sequential logic electronic circuit that generates an output pulse. When triggered, a pulse of pre-defined duration is produced. The circuit then returns to its stable state and produces no more output until triggered again.

Monostable Multivibrators have only ONE stable state (hence their name: “Mono”), and produce a single output pulse when it is triggered externally. Monostable Multivibrators only return back to their first original and stable state after a period of time determined by the time constant of the RC coupled circuit.

Multivibrator is an electronic circuit which will work as two stage amplifier operating in both stable and astable mode.In the multivibrator the output of first stage is given to the second stage and the second stage output is again feed back to the first stage by this the cutoff state will become saturate and saturate.

CIRCUIT DIAGRAMS :



DESIGN PROCEDURE:

To design a monostable multivibrator for the Pulse width of 0.3 mSec.

Let I_{Cmax} = 15mA, V_{CC} = 15V, V_{BB} = 15V, R_1 = 10K%u2126.

$T = 0.69RC$

Choose C = 10nf(0.01µF) $T = 0.69\ RC$

$0.3 \times 10^{-3} Sec = 0.69 \times R \times 10 \times 10^{-9}$

$R = 43.47$ Kohms ≈ 47Kohms

$R_C = (V_{CC} - V_{CESAT}) / I_{CMAX} = (15 - 0.3) / 15 \times 10^{-3}$

$= 1$ Kohms

Minimum requirement of $|V_{B1}| \leq 0.1$

For more margin, given $V_{B1} = -1.185$

Theoretical calculations : $T_{ON} = 0.69\ RC$

R= 47K%u2126 and C = 10nF or 0.01µF

Note : Normally Monostable Multivibrator generates single pulse output whenever a trigger is given. To observe this output storage oscilloscope is required.

PROCEDURE :

1. I made the connections as per the circuit diagram.
2. Selected the triggering pulse such that the frequency is less than 1/T

3. Applied the triggering input to the circuit and to the CRO's channel and connected the CRO channel-2 to the collector and base of the Transistor Q1 & Q2.
4. Adjusted the triggering pulse frequency to get stable pulse on the CRO and now measure the pulse width.
5. Obtained wave forms at different points like V_{B1}, V_{B2}, V_{C1} and V_{C2} and plotted the graph.

EXPECTED WAVEFORM :

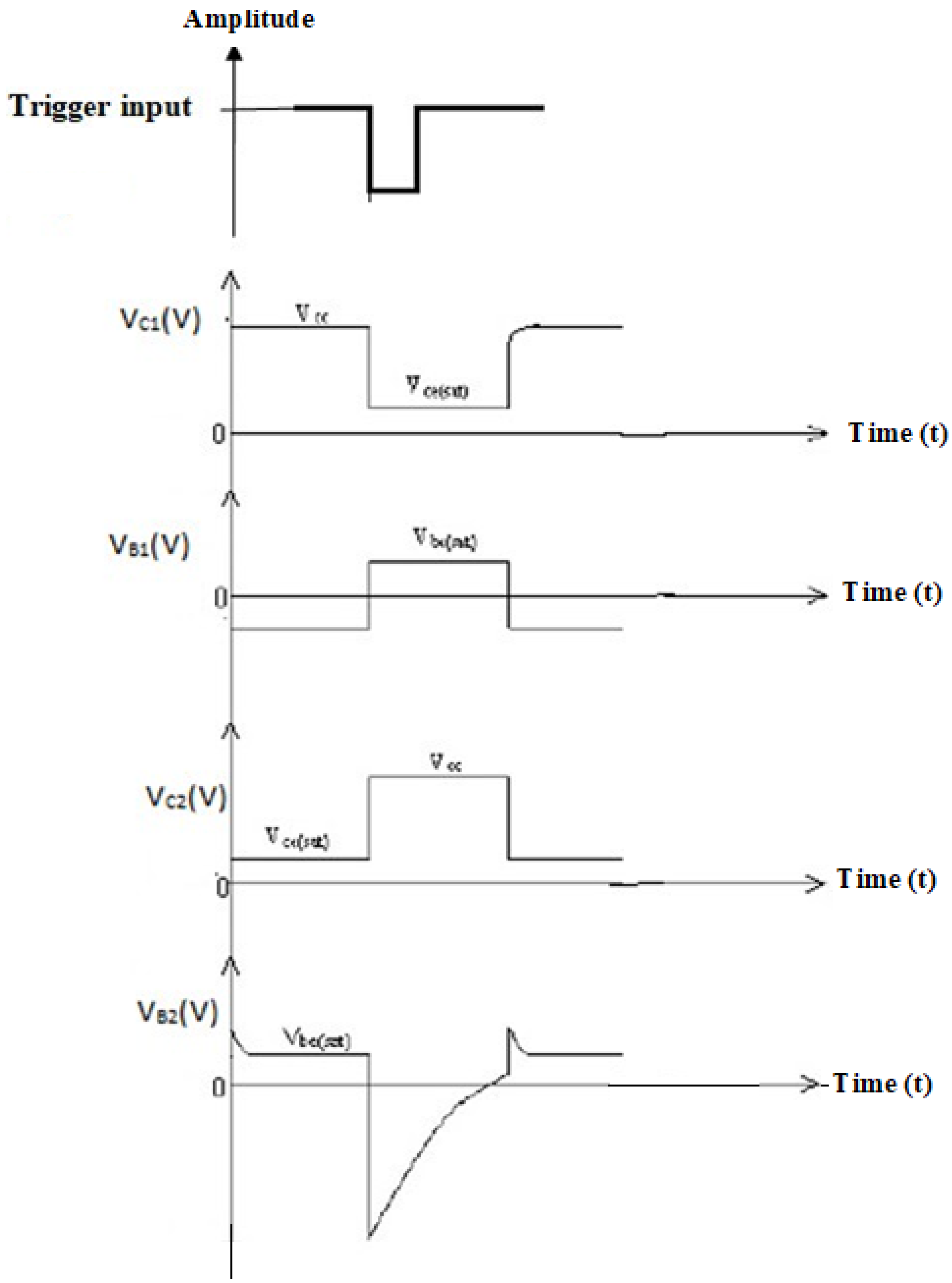


Fig : Waveforms of Monostable Multivibrator

RESULT :

I have conducted and verified the Mono stable Multi vibrator

VIVA VOCE Questions:

1. What is Multi-vibrator?

2. Mention the Applications of Mono stable Multi-vibrator.

3. Why mono-stable multi-vibrator is also called as delay circuit?

4. Mono-stable Multi-vibrator is having how many stable states?

5. What are the types of Multi-vibrators?

6. Which one is one shot Multi- vibrator?

7. What is Quasi stable?

8. Circuit which consists of a quasi -stable state is called__________________________Multi vibrator.

9. Mono stable Multi-vibrator is also referred as_________________________.

Experiment No. : 12 **Date :**

Name of the Experiment : ASTABLE MULTIVIBRATOR

AIM :

To conduct and verify the Astable multi vibrator and draw the waveforms.

APPARATUS :

1. Regulated power supply (RPS) ------------------------- 1 No.
2. Cathode ray oscilloscope --------------------------------- 1 No.
3. Function Generator ------------------------ 1 No.
4. Bread board -- 1 No.
5. Probes ---1 No.
6. Connecting wires -- 1 No.

COMPONENTS :

1. Resistors : 1KΩ, 10 KΩ, 100 KΩ ---------------------------- Each 2 No.
2. Capacitors : 0.1µF / 100nF ---------------------------- 2 No.
3. Transistor : BC547 ------------------------------------- 2 No.

THEORY :

Astable Multivibrator is a two stage switching circuit in which the output of the first stage is fed to the input of the second stage and vice versa. The outputs of both the stages are complementary. This free running multivibrator generates square wave without any external triggering pulse.

It is also called free-running relaxation oscillator. It has no stable state but only two quasi-stable states between which it keeps oscillating continuously of its own accord without any external excitation. When one transistor is in ON state and other remains in OFF state.

As a timing oscillator or clock of a computer system. It is also used for a flashing lights, switching and power supply circuits.

Advantages :

1. They work consistently and are not influenced by any outside forces or events.
2. They are inexpensive.
3. They are simple in design.
4. They can remain functional for an extraordinary length of time.

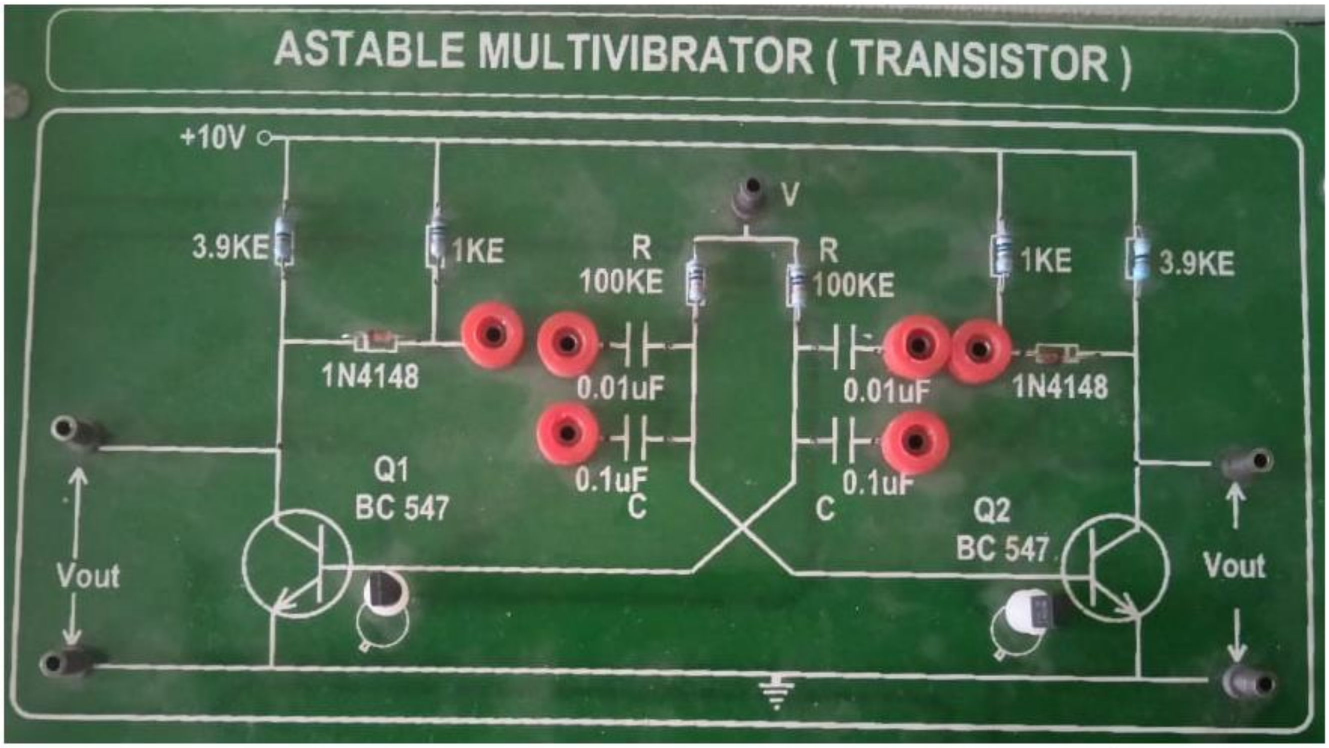
Disadvantages :

They do not transfer the entire output signal to the input due to several reasons like:

1. There is resistance within the circuit.
2. Absence of a completely closed loop at the output terminals.
3. One capacitor or transistor has a tendency to absorb energy at a slightly different rate than the other.
4. Even though the amplifier restores the lost energy when it amplifies the signal, the signal is too small.

https://vikramlearning.com/jntuh/notes/electronic-circuits-and-pulse-circuits-lab/astable-multi-vibrator/277

CIRCUIT DIAGRAMS :



Design Procedure:

The period T is given by

$T = T_1 + T_2 = 0.69 (R_1C_1 + R_2C_2)$

For symmetrical circuit, with $R_1 = R_2 = R$ & $C_1 = C_2 = C$

$T = 1.38 RC$

Let $V_{CC} = 12V$; $h_{fe} = 51$ (for BC107), $V_{BESat} = 0.7V$; $V_{CESat} = 0.3V$ Let $C = 0.1\mu F$ & T = 1mSec.

$10^{-3} = 1.38 \times R \times 0.1 \times 10^{-6}$

$R = 7.24K\Omega$ (Practically choose $10K\Omega$) i.e., R1 and R2 resistors.

Let I_{Cmax}=10mA

$R_C = \frac{V_{cc} - V_{cesat}}{I_{cmax}} = \frac{12-0.3}{0.01} = 1.17K\Omega$ ($1K\Omega$ is selected for Rc1 and Rc2)

Theoretical calculations : F = 1/ T = (1/1.38RC)

$R = 10K\Omega$ $C = 0.1\mu F$

PROCEDURE :

1. I have made the connections as per the circuit diagram.
2. Observed the Base Voltage and Collector Voltages of Q1 & Q2 on CRO in DC mode and measured the frequency (f = 1/T).
3. Traced the waveforms at collector and base as each transistor with the help of dual trace CRO and plot the waveforms.
4. Verified the practical output frequency with theoretical values f = 1/T, where T = 1.38RC

EXPECTED WAVEFORM :

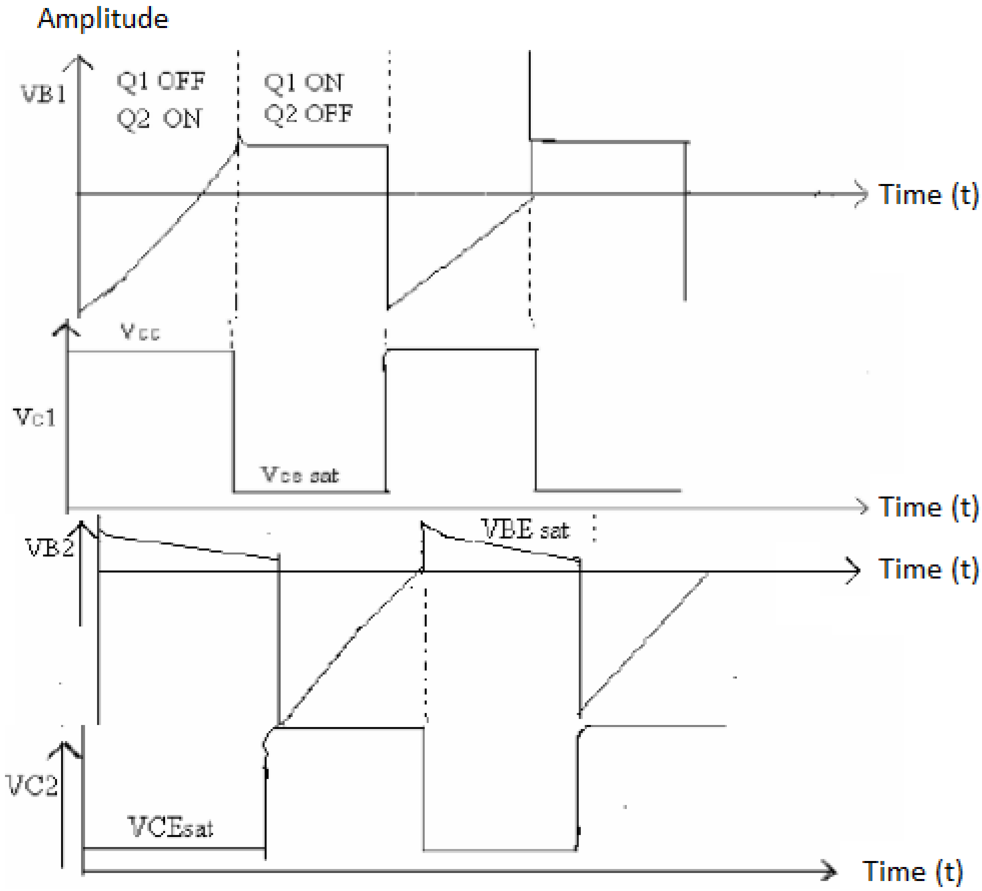


Fig : Wafeforms of Astable Multivibrator

RESULT :

I have conducted and verified the Astable Multivibrator.

VIVA VOCE Questions:

11. What is Multi-vibrator?

12. What are the types of Multi-vibrators?

13. What is Astable Multi-vibrator?

14. Mention the Applications of Astable Multi-vibrator.

15. Astable Multi-vibrator is having how many stable states?

16. Which one is Square wave oscillator?

17. Which of the multi-vibrator used in Relaxation oscillators?

18. Compare Mono-stable, Bi-stable and Astable multi-vibrators.

19. What is Quasi stable?

20. Free running Multi vibrator generates Square wave. (True or False)

Experiment No. : 13 **Date :**

Name of the Experiment : TWO STAGE RC COUPLED AMPLIFIER

AIM :

i). To obtain the frequency response of *Two stage RC Coupled Amplifier.*

ii). To calculate the bandwidth of this amplifier.

APPARATUS :

1). Function Generator ----- 1 No.
2). Cathode Ray Oscilloscope ----- 1 No.
3). Regulated Power Supply ----- 1 No.
4). Bread Board ----- 1 No.
5). Probes ----- 2 No.
6). Connecting wires ----- A Few Nos.

COMPONENTS :

1). Transisitor	a). BC547 NPN	-----	2No.
2). Resistors	a). 1K Ω, 2.2 K Ω, 10 K Ω, 47 K Ω	------	Each 2 No.
	b). 100 K Ω	-----	1 No.
3). Capacitors	10 µF, 22 µF	-----	Each 3 No.

THEORY :

RC coupling is the most widely used method of coupling in multistage amplifiers. ... In this case the resistance R is the resistor connected at the collector terminal and the capacitor C is connected in between the amplifiers. It is also called a blocking capacitor, since it will block DC voltage.

In RC coupling, a capacitor is used as the coupling device. The capacitor connects the output of one stage to the input of the next stage in order to pass the a.c. signal on while blocking the d.c. bias voltages.

In this way overall voltage gain can be increased, when number of amplifier stage is used in succession, it is called a multistage amplifier. The load of first amplifier is the input resistance of the second amplifier. Thus overall gain is reduced. The output of one stage is connected to the input of next stage through the coupling capacitor.It increases the overall gain of the amplifier and decreases the overall bandwidth of the amplifier.

The applications are : Optical Fiber Communications. Public address systems as pre-amplifiers. Controllers. Radio or TV Receivers as small signal amplifiers.

CIRCUIT DIAGRAM :

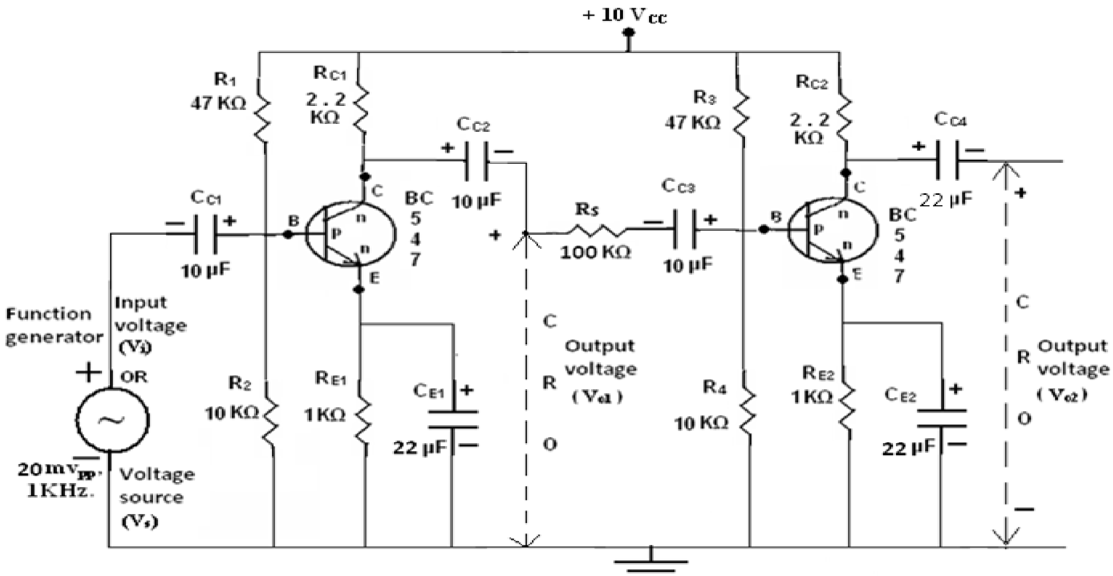


Figure: Circuit diagram of Two stage RC coupled amplifier.

PROCEDURE :

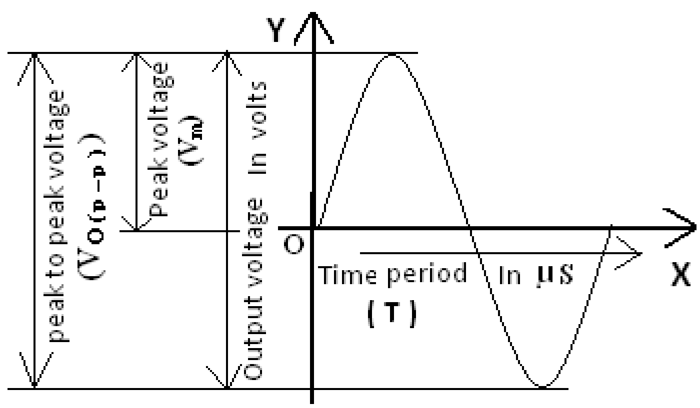
1. We have connected the circuit as per the circuit diagram which is shown above.
2. Initially connected the probe across the function generator as per shown in the circuit diagram to set the input signal.
1. Switched *ON* the *CRO* and *function generator*.
2. Applied the input signal as *sine wave form* having the values of $20m_{p-p}$, 1KHz.from the function generator by observing in the CRO.
3. Removed the probe from that place and connected it across the C_{C2} to observe the output of single stage.
4. Switched *ON* the *RPS* and kept the 10V as V_{CC}.
5. Kept the amplitude of the input signal as constant as $20mV_{p-p}$ for all frequency steps.
6. Noted down the values of output voltage in terms of peak to peak voltages by varying the different frequency steps in the function generator which are given below,
   20Hz, 100Hz, 200Hz, 500Hz, 1KHz, 200KHz, 400KHz, 940KHz, 940KHz, 1MHz.
7. The above readings noted in the tabular form of *single stage RC coupled amplifier*.
8. Disconnect the probe from C_{C2} and reconnected it across C_{C4} to observe the output of second stage.
9. Repeated the same procedure as per point 8 for tabular form of *Two stage RC Coupled Amplifier*.
10. Now calculated and noted down the values in the tabular form of *single stage RC Coupled Amplifier* as per given below,
    a). *Voltage gain* $(A_v) = V_o / V_i$ *and* *Gain in dB* $= 20\log_{10}(A_v)$.
    b). Plotted the graph between *frequency on X*- axis and *gain in dB on Y*- axis.
    c). Band width from the graph by using the formula- *Band width* $= f_2 - f_1$
11. Now calculated and noted down the values in the tabular form of *Two stage RC Coupled Amplifier* as per given below,
    a). *Voltage gain* $(A_v) = V_o / V_i$ *and* *Gain in dB* $= 20\log_{10}(A_v)$.
    b). Plotted the graph between *frequency on X*- axis and *gain in dB on Y*- axis.
    c). Band width from the graph by using the formula- *Band width* $= f_4 - f_3$

TABULAR COLUMN :

	Input Voltage (V_i) = 20mV_{P-P} (0.02V) is constant for all readings (For Single stage & Two stage RC coupled Amplifier)								
	For Single stage RC coupled Amplifier					For Two stage RC coupled Amplifier			
Sl. No.	Frequ-ency In Hz/KHz.	Voltage (V_O) In mVolts.	Voltage Gain $A_V = V_o/V_i$	Gain in dB= $20\log_{10}(A_V)$		Frequ-ency In Hz/KHz.	Output Voltage (V_O) In mVolts.	Voltage gain $A_V=V_o/V_i$	Gain in dB = $20\log_{10}(A_V)$
1	20 Hz.								
2	100 Hz.								
3	200 Hz.								
4	500 Hz.								
5	1 KHz.								
6	200KHz.								
7	400KHz.								
8	940KHz.								
9	940KHz.								
10	1 MHz.								

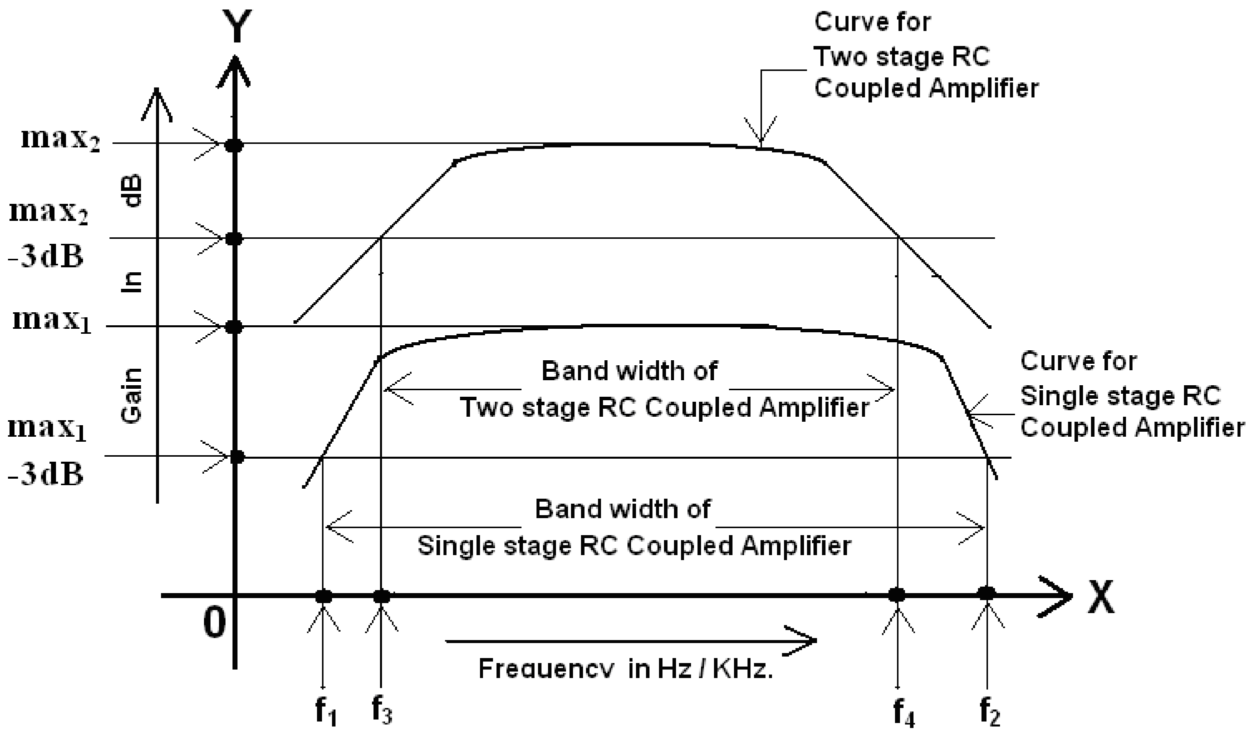
EXPECTED WAVEFORM :

I got the *Sine wave form* on the CRO as output signal for both *RC Coupled Amplifiers* which is shown below,



EXPECTED GRAPH :

The following graph shows the frequency response curves of both *Single stage & Two stage RC coupled Amplifiers.*



CALCULATIONS :

1). Band width "*single stage RC coupled amplifier* = $f_2 - f_1$

=

2). Band width "*two stage RC coupled amplifier* = $f_4 - f_3$

=

CONCLUSION :

1. I have observed that
   a). The bandwidth of *Two stage RC coupled amplifier* is less as compared to *Single stage RC coupled amplifier* and
   b). The gain of *Two stage RC coupled amplifier* is more as compared to *Single stage RC coupled amplifier*

RESULT :

I verified / drawn the frequency response curve and found the bandwidth values of a *single stage & two stage RC coupled amplifiers*. The band width values are,

1). Band width of *single stage RC coupled amplifier* =

2). Band width of *two stage RC coupled amplifier* =

VIVA VOCE QUESTIONS:

1. Need for multi stage amplifier?

2. What are the different coupling schemes?

3. Applications of Multi stage amplifiers?

4. Mention Advantages of Multistage Amplifiers.

5. What is Band Width?

6. What is cascade Amplifier?

7. What is the gain in Two stage RC coupled amplifier ?

Experiment No. : 14 **Date :**

Name of the Experiment : COMPLEMENTARY SYMMETRY CLASS B PUSH – PUSH POWER AMPLIFIER

AIM :

1. To draw the output signal (sine wave form) on the graph of a given *complementary symmetry push-pull class-Bpush-pull power amplifier*.
2. To study the operation of this *amplifier*.
3. To calculate the conversion efficiency of a given *power amplifier*.

APPARATUS :

1. Regulated power supply (RPS) ------------------------ 1 No.
2. Cathode Ray Oscilloscope (CRO) ------------------------ 1 No.
3. Function generator -- 1 No.
4. Probes --- 1 No.
5. **Ammeters :**
   i). (0-10) mA Digital / Analog DC Type --------- 1 No.
3. Bread board -- 1 No.
4. Connecting wires -- A few Nos.

COMPONENTS :

1. Transistors : BC 547, BC 557 -------------------------------- Each 1No.
2. Resistors :
   i). 220 KΩ, 18 KΩ -- Each 2 No.
   ii). 1 KΩ -- 1 No.
   iii). 10Ω -- 3 No.
3. Capacitors : 10 µF -------------------------------------- 2 No.

THEORY :

Class B amplifier is a type of power amplifier where the active device (transistor) conducts only for one half cycle of the input signal. That means the conduction angle is 194° for a Class B amplifier. Since the active device is switched off for half the input cycle, the active device dissipates less power and hence the efficiency is improved. Theoretical maximum efficiency of Class B power amplifier is 78.5%.

Class-B or Push-pull amplifiers use two “complementary” or matching transistors, one being an NPN-type and the other being a PNP-type with both power transistors receiving the same input signal together that is equal in magnitude, but in opposite phase to each other. This results in one transistor only amplifying one half or 194^{o} of the input waveform cycle while the other transistor amplifies the other half or remaining 194^{o} of the input waveform cycle with the resulting “two-halves” being put back together again at the output terminal. Then the conduction angle for this type of amplifier circuit is only 194^{o} or 50% of the input signal. This pushing and pulling effect of the alternating half cycles by the transistors gives this type of circuit its amusing “push-pull” name, but are more generally known as the **Class B Amplifier**

The basic class B amplifiers are used in two complementary transistors which are FET and bipolar. Power amplifiers are used in broadcast transmitters, wireless transmitters and high audio systems. Bipolar transistors are used for these applications. Power output and efficiency are mostly considered in power amplifications.

Advantages	Disadvantages
Very low standing bias current. Negligible power consumption without signal.	Creates Crossover distortion.
Can be used for much more powerful outputs than class A	Supply current changes with signal, stabilised supply may be needed.
More efficient than Class A.	More distortion than Class A.

CIRCUIT DIAGRAM :

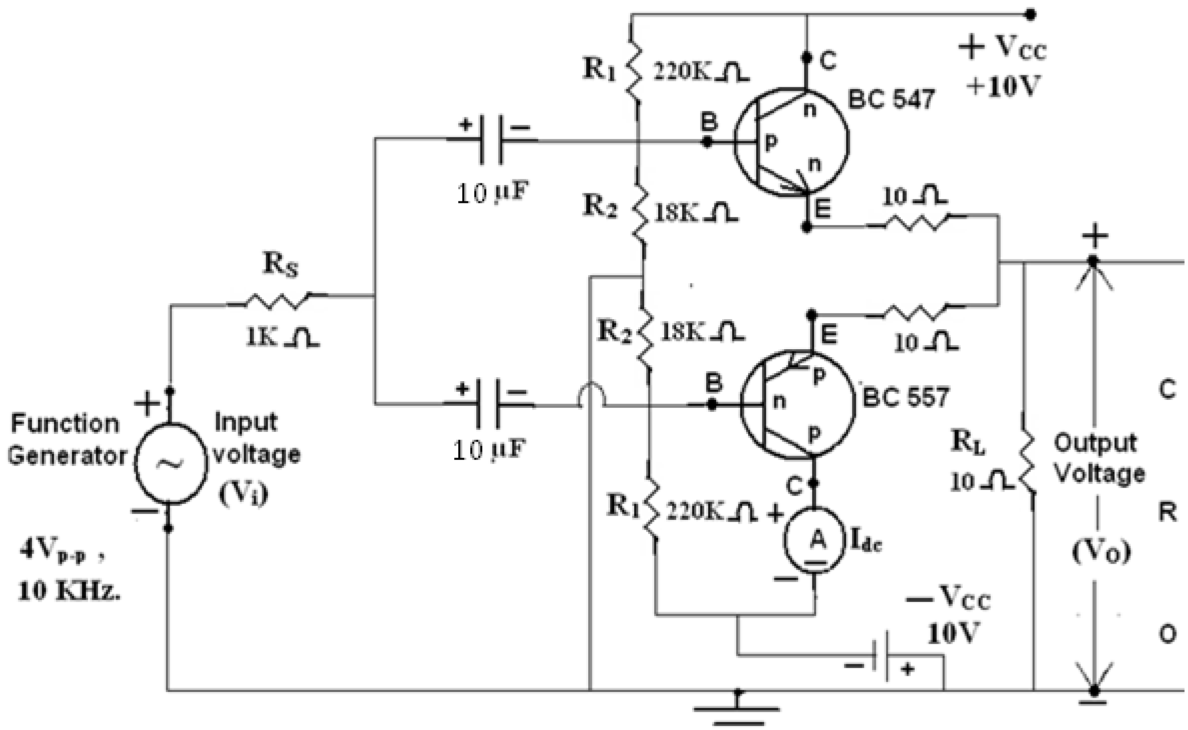


Figure: Circuit diagram of Class-B complimentary symmetry power amplifier.

PROCEDURE :

1. Connections are made as per the circuit diagram.
2. Initially connected the *CRO* across the *Function generator*.
3. Switched **ON** the Cathode ray oscilloscope (CRO) and Function generator.
4. Applied the $4V_{p\text{-}p}$, *10 KHz sine wave* signal to the circuit from the *Function generator* by observing on the ***crt*** of the *CRO*.
5. Later connected the *CRO* across $\boldsymbol{R_L}$ i.e. at output side.
6. Now switched **ON** the Regulated Power Supply (RPS) and apply the *supply voltage* +10V from one channel (+V_{CC}) and -10V from another (-V_{CC}) to the circuit as per shown in the figure.

7. Observed the *sine wave* signal on the CRT of the CRO and draw this signal on the graph sheet.
8. Now noted down the *collector dc current* ***$I_{(dc)}$*** at *Quiescent* condition i.e. when no signal is applied by disconnected the *function generator* from the circuit and *supply voltage* **(*V_{CC}*)** against the corresponding columns of the tabular form of *practical calculations.*
9. Noted down the *Input voltage**(V_i)*** , *Input frequency* against the corresponding columns of the tabular form of *practical calculations.*
10. Switched **OFF** the *function generator, RPS, CRO.*
11. Calculated the peak to peak *voltage* **(*Vo_{p-p}*)** , *peak voltage* ***(V_m),*** *time period* ***(T)***, *frequency* ***(f)*** from the graph, and noted down these values against the corresponding columns of the tabular form of practical calculations.
12. Later calculated the *Input dc power* ***P_i(dc),*** *output ac power* ***P_o(ac)*** and *% of efficiency* ***(η)*** by using the formulas which are mentioned in the corresponding columns of the tabular form of practical calculations.
13. Noted that The practical value should be less than the *Typical Max. efficiency value i.e.* ***78.5%****.*

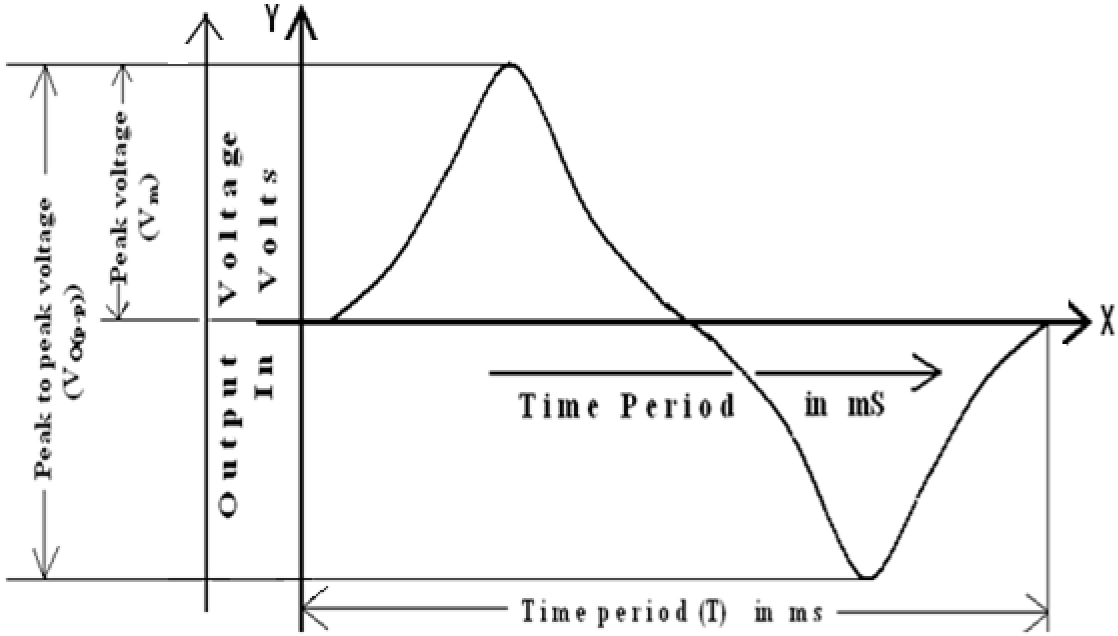
PRACTICAL CALCULATIONS :

The practical calculations for the parameters are shown in the following tabular form,

SlNo.	Name of the parameter	Value
01.	Input peak to peak voltage (V_i) (In Volts).	4
02	Input frequency (In Khz.).	10
03	Positive supply DC Voltage ($+V_{CC}$) (in Volts.)	10
	Negative supply DC Voltage ($-V_{CC}$) (in Volts.)	10
04	Peak to peak voltage of output $V_{O(p-p)}$ (In volts.).	
05	Peak voltage of output (V_m) = $V_{O(p-p)} / 2$ (In volts).	
06	Time period (T) for output signal (In ms)	
07	Fequency for output signal = 1/ T (In Khz.)	
08	Collector dc current (I_{dc}) (At quesient condition i.e. When no input signal is applied) (In mA.).	
09	Collector DC current when sine wave (AC) signal is applied as input signal (I_{ac})	
10	Input DC power $P_i(dc) = I_{dc} \times V_{CC}$ (In mWatts).	
11	Output ac power $P_o\ (ac) = \dfrac{V_m^2}{2R_L}$ (In mWatts) =	
12	% of efficiency $(\eta) = \dfrac{P_0(ac)}{P_i(dc)} \times 100$	
13	Typical Max. efficiency (η) =	78.50 %

EXPECTED GRAPH :

The following graph shows for ***Class B complementary symmetry power amplifier.***



RESULT :

I have drawn the graph for output signal and calculated the conversion efficiency of given *complementary symmetry Class-B push-pull power amplifier*.

VIVA VOCE Questions:

1. What is Power Amplifier?
2. Classifications of power Amplifiers.
3. Efficiency of class B power Amplifier.
4. Difference between Transformer coupled and Complementary symmetry class B power Amplifier ?
5. What is the Crossover Distortion?
6. Where is the Q point in class B power Amplifier?
7. Applications of class B power Amplifier.
8. What are the disadvantages of class B power Amplifier.
9. Mention the conduction angle of class B power Amplifier.
10. What are the disadvantages of class B power Amplifier.

A. DATA SHEETS

PN JUNCTION DIODE :

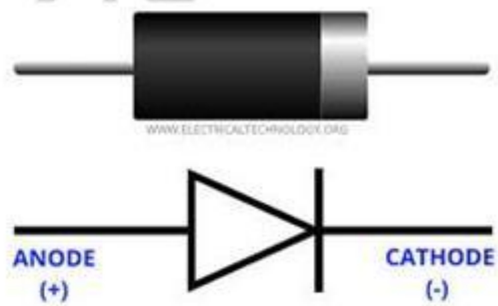
1N4001 - 1N4007 1.0A

Features

- Diffused Junction
- High Current Capability and Low Forward Voltage Drop
- Surge Overload Rating to 30A Peak
- Low Reverse Leakage Current
- Lead Free Finish, RoHS Compliant (Note 3)

Mechanical Data

- Case: DO-41
- Case Material: Molded Plastic. UL Flammability Classification Rating 94V-0
- Moisture Sensitivity: Level 1 per J-STD-020D
- Terminals: Finish - Bright Tin. Plated Leads Solderable per MIL-STD-202, Method 208
- Polarity: Cathode Band
- Ordering Information: See Page 2
- Marking: Type Number
- Weight: 0.30 grams (Approximate)



Maximum Ratings and Electrical Characteristics (@T_A = +25°C unless otherwise specified.) Single phase, half wave,

For capacitive load, derate current by 20%.

Characteristic	Symbol	1N4001	1N4002	1N4003	1N4004	1N4005	1N4006	1N4007	Unit
Peak Repetitive Reverse Voltage Working Peak Reverse Voltage DC Blocking Voltage	V_{RRM} V_{RWM} V_R	50	100	200	400	940	940	1000	V
RMS Reverse Voltage	$V_{R(RMS)}$	35	70	140	294	420	594	700	V
Average Rectified Output Current (Note 1) @ T_A =+75°C	I_O	1.0							A
Non-Repetitive Peak Forward Surge Current 8.3ms Single Half Sine-Wave Superimposed on Rated Load	I_{FSM}	30							A
Forward Voltage @ I_F = 1.0A	V_{FM}	1.0							V
Peak Reverse Current @T_A = +25°C at Rated DC Blocking Voltage @ T_A = +100°C	I_{RM}	5.0 50							µA
Typical Junction Capacitance (Note 2)	C_j	15				8			pF
Typical Thermal Resistance Junction to Ambient	$R_{\theta JA}$	100							K/W
Maximum DC Blocking Voltage Temperature	T_A	+150							°C
Operating and Storage Temperature Range	T_J, T_{STG}	-65 to +150							°C

ZENER DIODE :

TOSHIBA 1Z6.2~1Z390,1Z6.8A~1Z30A

TOSHIBA ZENER DIODE SILICON DIFFUSED JUNCTION TYPE

1Z6.2~1Z390,1Z6.8A~1Z30A

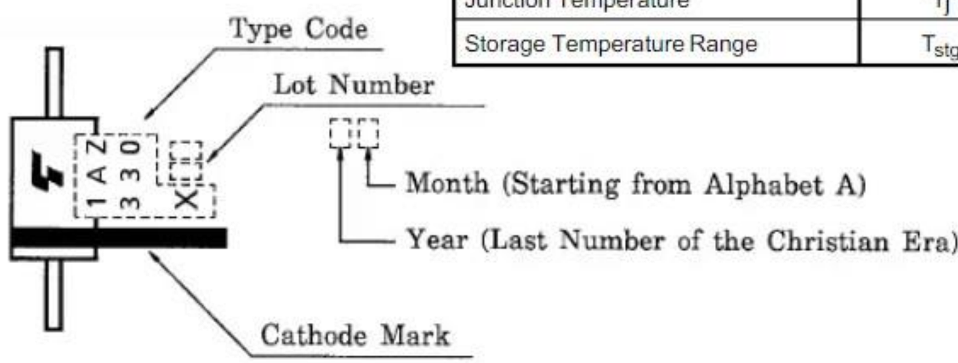
CONSTANT VOLTAGE REGULATION
TRANSIENT SUPPRESSORS

- Average Power Dissipation : P = 1W
- Peak Reverse Power Dissipation : P_{RSM} = 200W at t_w = 200μs
- Zener Voltage : V_Z = 6.2 ~ 390V
- Tolerance of Zener Voltage
  1Z6.2 Series : ±10%
  1Z6.8A Series : ±5%
- Plastic Mold Package

MAXIMUM RATINGS (Ta=25°C)

CHARACTERISTIC	SYMBOL	RATING	UNIT
Power Dissipation	P	1	W
Junction Temperature	T_j	−40~150	°C
Storage Temperature Range	T_{stg}	−40~150	°C

MARK



Color : Silver

DATA SHEET OF BJT :

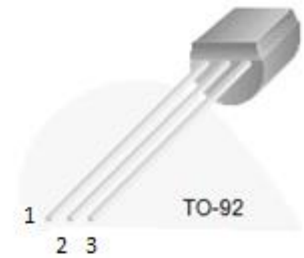
BIPOLAR JUNCTION TRANSISTORS (BJT) :

FAIRCHILD

BC546 / BC547 / BC548 / BC549 / BC550
NPN Epitaxial Silicon Transistor

Features

- Switching and Amplifier
- High-Voltage: BC546, V_{CEO} = 65 V
- Low-Noise: BC549, BC550
- Complement to BC556, BC557, BC558, BC559, and BC560



1. Collector 2. Base 3. Emitter

Absolute Maximum Ratings

Stresses exceeding the absolute maximum ratings may damage the device. The device may not function or be operable above the recommended operating conditions and stressing the parts to these levels is not recommended. In addition, extended exposure to stresses above the recommended operating conditions may affect device reliability. The absolute maximum ratings are stress ratings only. Values are at T_A = 25°C unless otherwise noted.

Symbol	Parameter		Value	Unit
V_{CBO}	Collector-Base Voltage	BC546	80	V
		BC547 / BC550	50	
		BC548 / BC549	30	
V_{CEO}	Collector-Emitter Voltage	BC546	65	V
		BC547 / BC550	45	
		BC548 / BC549	30	
V_{EBO}	Emitter-Base Voltage	BC546 / BC547	6	V
		BC548 / BC549 / BC550	5	
I_C	Collector Current (DC)		100	mA
P_C	Collector Power Dissipation		500	mW
T_J	Junction Temperature		150	°C
T_{STG}	Storage Temperature Range		-65 to +150	°C

Electrical Characteristics

Values are at T_A = 25°C unless otherwise noted.

Symbol	Parameter	Conditions	Min.	Typ.	Max.	Unit
I_{CBO}	Collector Cut-Off Current	V_{CB} = 30 V, I_E = 0			15	nA
h_{FE}	DC Current Gain	V_{CE} = 5 V, I_C = 2 mA	110		800	
$V_{CE}(sat)$	Collector-Emitter Saturation Voltage	I_C = 10 mA, I_B = 0.5 mA		90	250	mV
		I_C = 100 mA, I_B = 5 mA		250	600	
$V_{BE}(sat)$	Base-Emitter Saturation Voltage	I_C = 10 mA, I_B = 0.5 mA		700		mV
		I_C = 100 mA, I_B = 5 mA		900		
$V_{BE}(on)$	Base-Emitter On Voltage	V_{CE} = 5 V, I_C = 2 mA	580	660	700	mV
		V_{CE} = 5 V, I_C = 10 mA			720	
f_T	Current Gain Bandwidth Product	V_{CE} = 5 V, I_C = 10 mA, f = 100 MHz		300		MHz
C_{ob}	Output Capacitance	V_{CB} = 10 V, I_E = 0, f = 1 MHz		3.5	6.0	pF
C_{ib}	Input Capacitance	V_{EB} = 0.5 V, I_C = 0, f = 1 MHz		9		pF
NF	Noise Figure: BC546 / BC547 / BC548	V_{CE} = 5 V, I_C = 200 μA, f = 1 kHz, R_G = 2 kʌ		2.0	10.0	dB
	Noise Figure: BC549 / BC550			1.2	4.0	
	Noise Figure: BC549	V_{CE} = 5 V, I_C = 200 μA, R_G = 2 kʌ, f = 30 to 15000 MHz		1.4	4.0	
	Noise Figure: BC550			1.4	3.0	

h_{FE} Classification

Classification	A	B	C
h_{FE}	110 ~ 220	200 ~ 450	420 ~ 800

DATA SHEET OF UNIJUNCTION TRANSISTOR (UJT) :

2N2646
2N2647

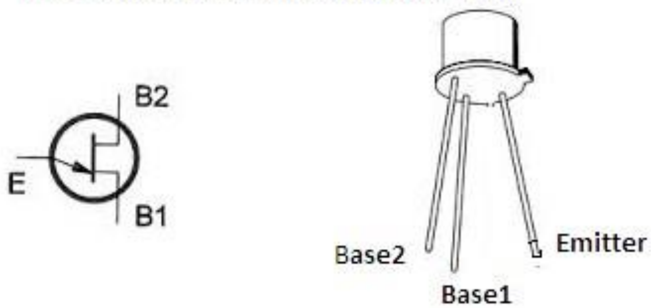
SILICON
PN UNIJUNCTION TRANSISTORS

Central Semiconductor Corp.

www.centralsemi.com

DESCRIPTION:
The CENTRAL SEMICONDUCTOR 2N2646 and 2N2647 devices are silicon PN Unijunction Transistors designed for general purpose industrial applications.

UJT Symbol & Terminal Identification



(a). Symbol **(b). Terminal Identification**

MAXIMUM RATINGS: (T_A=25°C)	**SYMBOL**		**UNITS**
Emitter Reverse Voltage	V_{B2E}	30	V
Interbase Voltage	V_{B2B1}	35	V
RMS Emitter Current	I_e	50	mA
Peak Emitter Current (Duty Cycle ≤1%, PRR≤10pps)	i_e	2.0	A
RMS Power Dissipation	P_D	300	mW
Operating and Storage Junction Temperature	T_J, T_{stg}	-65 to +150	°C

ELECTRICAL CHARACTERISTICS: (T_A=25°C unless otherwise noted)

		2N2646		2N2647		
SYMBOL	**TEST CONDITIONS**	**MIN**	**MAX**	**MIN**	**MAX**	**UNITS**
η	V_{B2B1}=10V	0.56	0.75	0.68	0.82	
R_{BB}	V_{B2B1}=3.0V	4.7	9.1	4.7	9.1	kΩ
I_{EB2O}	V_{B2E}=30V	-	12	-	0.2	µA
I_V	V_{B2B1}=20V, R_{B2}=100Ω	4.0	-	8.0	18	mA
I_P	V_{B2B1}=25V	-	5.0	-	2.0	µA
V_{OB1}	V_1=20V	3.0	-	6.0	-	V

DATA SHEETS OF FET :

NXP Semiconductors Product specification

N-channel silicon field-effect transistors BF245A; BF245B; BF245C

FEATURES

- Interchangeability of drain and source connections
- Frequencies up to 700 MHz.

APPLICATIONS

- LF, HF and DC amplifiers.

DESCRIPTION

General purpose N-channel symmetrical junction field-effect transistors in a plastic TO-92 variant package.

CAUTION
The device is supplied in an antistatic package. The gate-source input must be protected against static discharge during transport or handling.

PINNING

PIN	SYMBOL	DESCRIPTION
1	d	drain
2	s	source
3	g	gate

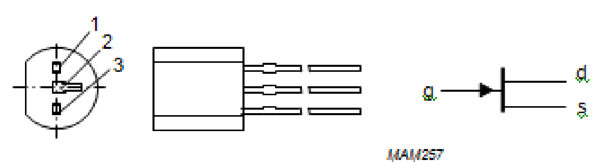


Fig.1 Simplified outline (TO-92 variant) and symbol.

QUICK REFERENCE DATA

SYMBOL	PARAMETER	CONDITIONS	MIN.	TYP.	MAX.	UNIT
V_{DS}	drain-source voltage		–	–	±30	V
V_{GSoff}	gate-source cut-off voltage	I_D = 10 nA; V_{DS} = 15 V	–0.25	–	–8	V
V_{GSO}	gate-source voltage	open drain	–	–	–30	V
I_{DSS}	drain current	V_{DS} = 15 V; V_{GS} = 0				
	BF245A		2	–	6.5	mA
	BF245B		6	–	15	mA
	BF245C		12	–	25	mA
P_{tot}	total power dissipation	T_{amb} = 75 °C	–	–	300	mW
$\|y_{fs}\|$	forward transfer admittance	V_{DS} = 15 V; V_{GS} = 0; f = 1 kHz; T_{amb} = 25 °C	3	–	6.5	mS
C_{rs}	reverse transfer capacitance	V_{DS} = 20 V; V_{GS} = –1 V; f = 1 MHz; T_{amb} = 25 °C	–	1.1	–	pF

LIMITING VALUES

In accordance with the Absolute Maximum Rating System (IEC 134).

SYMBOL	PARAMETER	CONDITIONS	MIN.	MAX.	UNIT
V_{DS}	drain-source voltage		–	±30	V
V_{GDO}	gate-drain voltage	open source	–	–30	V
V_{GSO}	gate-source voltage	open drain	–	–30	V
I_D	drain current		–	25	mA
I_G	gate current		–	10	mA
P_{tot}	total power dissipation	up to T_{amb} = 75 °C;	–	300	mW
		up to T_{amb} = 90 °C; note 1	–	300	mW
T_{stg}	storage temperature		–65	+150	°C
T_j	operating junction temperature		–	150	°C

Note

1. Device mounted on a printed-circuit board, minimum lead length 3 mm, mounting pad for drain lead minimum 10 mm × 10 mm.

THERMAL CHARACTERISTICS

SYMBOL	PARAMETER	CONDITIONS	VALUE	UNIT
$R_{th\ j-a}$	thermal resistance from junction to ambient	in free air	250	K/W
	thermal resistance from junction to ambient		200	K/W

STATIC CHARACTERISTICS

T_j = 25 °C; unless otherwise specified.

SYMBOL	PARAMETER	CONDITIONS	MIN.	MAX.	UNIT
$V_{(BR)GSS}$	gate-source breakdown voltage	I_G = –1 μA; V_{DS} = 0	–30	–	V
V_{GSoff}	gate-source cut-off voltage	I_D = 10 nA; V_{DS} = 15 V	–0.25	–8.0	V
V_{GS}	gate-source voltage	I_D = 200 μA; V_{DS} = 15 V			
	BF245A		–0.4	–2.2	V
	BF245B		–1.6	–3.8	V
	BF245C		–3.2	–7.5	V
I_{DSS}	drain current	V_{DS} = 15 V; V_{GS} = 0; note 1			
	BF245A		2	6.5	mA
	BF245B		6	15	mA
	BF245C		12	25	mA
I_{GSS}	gate cut-off current	V_{GS} = –20 V; V_{DS} = 0	–	–5	nA
		V_{GS} = –20 V; V_{DS} = 0; T_j = 125 °C	–	–0.5	μA

Note

1. Measured under pulse conditions: t_p = 300 μs; $\delta \leq 0.02$.

DYNAMIC CHARACTERISTICS

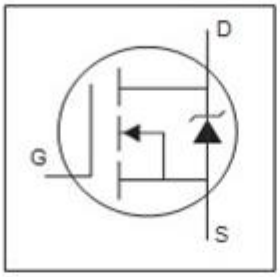
Common source; T_{amb} = 25 °C; unless otherwise specified.

SYMBOL	PARAMETER	CONDITIONS	MIN.	TYP.	MAX.	UNIT
C_{is}	input capacitance	V_{DS} = 20 V; V_{GS} = −1 V; f = 1 MHz	−	4	−	pF
C_{rs}	reverse transfer capacitance	V_{DS} = 20 V; V_{GS} = −1 V; f = 1 MHz	−	1.1	−	pF
C_{os}	output capacitance	V_{DS} = 20 V; V_{GS} = −1 V; f = 1 MHz	−	1.6	−	pF
g_{is}	input conductance	V_{DS} = 15 V; V_{GS} = 0; f = 200 MHz	−	250	−	µS
g_{os}	output conductance	V_{DS} = 15 V; V_{GS} = 0; f = 200 MHz	−	40	−	µS
$\|y_{fs}\|$	forward transfer admittance	V_{DS} = 15 V; V_{GS} = 0; f = 1 kHz	3	−	6.5	mS
		V_{DS} = 15 V; V_{GS} = 0; f = 200 MHz	−	6	−	mS
$\|y_{rs}\|$	reverse transfer admittance	V_{DS} = 15 V; V_{GS} = 0; f = 200 MHz	−	1.4	−	mS
$\|y_{os}\|$	output admittance	V_{DS} = 15 V; V_{GS} = 0; f = 1 kHz	−	25	−	µS
f_{gfs}	cut-off frequency	V_{DS} = 15 V; V_{GS} = 0; g_{fs} = 0.7 of its value at 1 kHz	−	700	−	MHz
F	noise figure	V_{DS} = 15 V; V_{GS} = 0; f = 100 MHz; R_G = 1 kΩ (common source); input tuned to minimum noise	−	1.5	−	dB

DATA SHEET OF MOSFET IRFZ 44N

IRFZ44NPbF

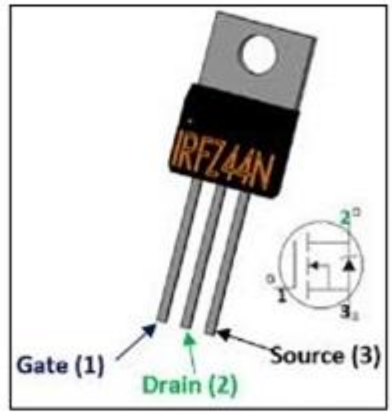
HEXFET® Power MOSFET



V_{DSS} = 55V

$R_{DS(on)}$ = 17.5mΩ

I_D = 49A



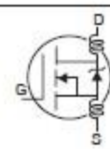
Absolute Maximum Ratings

	Parameter	Max.	Units
I_D @ T_C = 25°C	Continuous Drain Current, V_{GS} @ 10V	49	A
I_D @ T_C = 100°C	Continuous Drain Current, V_{GS} @ 10V	35	
I_{DM}	Pulsed Drain Current ①	160	
P_D @T_C = 25°C	Power Dissipation	94	W
	Linear Derating Factor	0.63	W/°C
V_{GS}	Gate-to-Source Voltage	± 20	V
I_{AR}	Avalanche Current①	25	A
E_{AR}	Repetitive Avalanche Energy①	9.4	mJ
dv/dt	Peak Diode Recovery dv/dt ③	5.0	V/ns
T_J T_{STG}	Operating Junction and Storage Temperature Range	-55 to + 175	°C
	Soldering Temperature, for 10 seconds	300 (1.6mm from case)	
	Mounting torque, 6-32 or M3 srew	10 lbf•in (1.1N•m)	

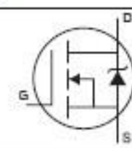
Thermal Resistance

	Parameter	Typ.	Max.	Units
$R_{\theta JC}$	Junction-to-Case	——	1.5	°C/W
$R_{\theta CS}$	Case-to-Sink, Flat, Greased Surface	0.50	——	
$R_{\theta JA}$	Junction-to-Ambient	——	62	

Electrical Characteristics @ $T_J = 25°C$ (unless otherwise specified)

	Parameter	Min.	Typ.	Max.	Units	Conditions
$V_{(BR)DSS}$	Drain-to-Source Breakdown Voltage	55	——	——	V	$V_{GS} = 0V$, $I_D = 250\mu A$
$\Delta V_{(BR)DSS}/\Delta T_J$	Breakdown Voltage Temp. Coefficient	——	0.058	——	V/°C	Reference to 25°C, $I_D = 1mA$
$R_{DS(on)}$	Static Drain-to-Source On-Resistance	——	——	17.5	mΩ	$V_{GS} = 10V$, $I_D = 25A$ ④
$V_{GS(th)}$	Gate Threshold Voltage	2.0	——	4.0	V	$V_{DS} = V_{GS}$, $I_D = 250\mu A$
g_{fs}	Forward Transconductance	19	——	——	S	$V_{DS} = 25V$, $I_D = 25A$④
I_{DSS}	Drain-to-Source Leakage Current	——	——	25	μA	$V_{DS} = 55V$, $V_{GS} = 0V$
		——	——	250		$V_{DS} = 44V$, $V_{GS} = 0V$, $T_J = 150°C$
I_{GSS}	Gate-to-Source Forward Leakage	——	——	100	nA	$V_{GS} = 20V$
	Gate-to-Source Reverse Leakage	——	——	-100		$V_{GS} = -20V$
Q_g	Total Gate Charge	——	——	63	nC	$I_D = 25A$
Q_{gs}	Gate-to-Source Charge	——	——	14		$V_{DS} = 44V$
Q_{gd}	Gate-to-Drain ("Miller") Charge	——	——	23		$V_{GS} = 10V$, See Fig. 6 and 13
$t_{d(on)}$	Turn-On Delay Time	——	12	——	ns	$V_{DD} = 28V$
t_r	Rise Time	——	60	——		$I_D = 25A$
$t_{d(off)}$	Turn-Off Delay Time	——	44	——		$R_G = 12\Omega$
t_f	Fall Time	——	45	——		$V_{GS} = 10V$, See Fig. 10 ④
L_D	Internal Drain Inductance	——	4.5	——	nH	Between lead, 6mm (0.25in.) from package and center of die contact 
L_S	Internal Source Inductance	——	7.5	——		
C_{iss}	Input Capacitance	——	1470	——	pF	$V_{GS} = 0V$
C_{oss}	Output Capacitance	——	360	——		$V_{DS} = 25V$
C_{rss}	Reverse Transfer Capacitance	——	88	——		$f = 1.0MHz$, See Fig. 5
E_{AS}	Single Pulse Avalanche Energy②	——	530⑤	150⑥	mJ	$I_{AS} = 25A$, $L = 0.47mH$

Source-Drain Ratings and Characteristics

	Parameter	Min.	Typ.	Max.	Units	Conditions
I_S	Continuous Source Current (Body Diode)	——	——	49	A	MOSFET symbol showing the integral reverse p-n junction diode. 
I_{SM}	Pulsed Source Current (Body Diode)①	——	——	160		
V_{SD}	Diode Forward Voltage	——	——	1.3	V	$T_J = 25°C$, $I_S = 25A$, $V_{GS} = 0V$ ④
t_{rr}	Reverse Recovery Time	——	63	95	ns	$T_J = 25°C$, $I_F = 25A$
Q_{rr}	Reverse Recovery Charge	——	170	260	nC	di/dt = 100A/μs ④
t_{on}	Forward Turn-On Time	Intrinsic turn-on time is negligible (turn-on is dominated by L_S+L_D)				

Notes:

① Repetitive rating; pulse width limited by max. junction temperature. (See fig. 11)

② Starting $T_J = 25°C$, $L = 0.48mH$ $R_G = 25\Omega$, $I_{AS} = 25A$. (See Figure 12)

③ $I_{SD} \le 25A$, di/dt ≤ 230A/μs, $V_{DD} \le V_{(BR)DSS}$, $T_J \le 175°C$

④ Pulse width ≤ 400μs; duty cycle ≤ 2%.

⑤ This is a typical value at device destruction and represents operation outside rated limits.

⑥ This is a calculated value limited to $T_J = 175°C$.

B. RULES TO BE FOLLOWED WHILE OPERATING THE REGULATED POWER SUPPLY(RPS):

The flowing rules should be followed before switch ON the Regulated Power Supply,

1. Initially Keep the *voltage Course & Voltage fine controls* of RPS at minimum position. Later (After switch ON the RPS) can vary these controls slowly to get the required voltage.
2. Always keep the Current Limit control at maximum position, Otherwise the display can shows the constant voltage instead of varying.

Trouble shooting while operating the *rps* :

The following trouble shooting can done while operating the RPS,

During connecting the RPS to the circuit and varying the Voltage Course & Voltage Fine Controls, If it displays the voltage as constant or above 30V then it can said that either the circuit is shorted OR the Current Limit control is not kept at maximum position. This problem can solve to prevent the circuit from shorted and by keeping the Current Limit control at maximum.

RULES TO OPERATE THE CRO:

The following rules should be follows before operate the CRO.

1. Keep the following controls at middle position or vary until the electron beam is generated.
   a) INTENSITY b) FOCUS c) ◄► (Horizontal position)
   (Horizontal position common for both channels)
   d) ▲▼ Vertical Position (Vertical position individual per each channel) e) LEVEL (Trigger Level)
1. Keep the following controls at maximum position.
   (a). VARIABLE controls of VOLTS/DIV switch in both channels.
   (b). SWP.VAR (Sweep Variation)
2. Keep the following switches at releasing mode.
   a) ×10 MAG b) TRIG.ALT c) SLOPE d) ALT/CHOPe) CH2 INV
3. Initially should keep the **TIME/DIV** control at 1mS position, later can change this switch depending upon our requirement , i.e. if we can't get the signal clearly on the CRT, then we can vary this switch until to get the signal.
4. Set the channel selector control **MODE** at the appropriate position i.e. if we want to seethe signal in channel1, set this control at CH1, in channel2 set at CH2, in both channels set at DUAL. To add the signals (algebraically sum or difference) available in both channels set atADD.
5. **AC/GND/DC**: Before setting the signals on CRT, first we should keep the electron beam on referenceline. To set this beam on reference line, keep this control at GND positio and then vary vertical position control until to get the beam on the reference line. After that to see the applied signals, keep this control at AC or DC positions.
6. Always keep the **TRIGGER MODE** control at AUTO position.
7. Keep the **SOURCE** control at approximate channel. It means if MODE control is selectedto CH1, then the SORCE control should select to CH1. If MODE control at CH2, set the SOURCE control at CH2. If MODE control at DUAL or ADD, set the SOURCE control either at CH1 or CH2.

Rules for how to write the observation and records:

1. Make the top & right margins in each page of right side.
2. In top margin make the headings as Experiment No., date and name of the experiment.
3. Circuit diagrams, tabular columns, expected graphs and parameters/calculations should write on leftside page (even No. page) .
4. Aim, apparatus, components, theory, procedure, applications, conclusion and result should write onright side page (Odd No. Page) .
5. Headings should underline with any other ink except red, orange and green.
6. The every new experiment should start with right side page.
7. leave the half of the page under the heading of *theory*.

C. SYLLABUS

JAWAHARLAL NEHRU TECHNOLOGICAL UNIVERSITY ANANTAPUR

B.Tech –II-III Sem 20A04302P ANALOG CIRCUITS LAB

LIST OF EXPERIMENTS **Branch : For ECE only** **R20**

1. Design and Analysis of Darlington pair.
2. Frequency response of CE – CC multistage Amplifier
3. Design and Analysis of Cascode Amplifier.
4. Frequency Response of Differential Amplifier
5. Design and Analysis of Series – Series feedback amplifier and find the frequency response ofit.
6. Design and Analysis of Shunt – Shunt feedback amplifier and find the frequency response ofit.
7. Design and Analysis of Class A power amplifier
8. Design and Analysis of Class AB amplifier
9. Design and Analysis of RC phase shift oscillator
10. Design and Analysis of LC Oscillator
11. Frequency Response of Single Tuned amplifier
12. Design and Analysis of Bistable Multivibrator
13. Design and Analysis of Monostable Multivibrator
14. Design and Analysis of Astable Multivibrator

Note :

1. At least 12 experiments shall be performed.
2. Both BJT and MOSFET based circuits shall be implemented.
3. Faculty members who are handling the laboratory shall see that students are given design specificationsfor a given circuit appropriately and monitor the design and analysis aspects of the circuit.
4. Know about the usage of equipment/components/software tools used to conduct the Experiments in analog circuit.