SVR ENGINEERING COLLEGE

Approved by AICTE & Permanently Affiliated to JNTUA

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

Department of Electronics and Communication Engineering

(15A04404) ELECTRONIC CIRCUIT ANALYSIS LABORATORY R15

II B. Tech (ECE) II Semester 2018-19

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: 2018-19

This is to certify that the bonafide record work done by

Mr./Ms.____ bearing

H.T.NO. _____________________ of II B. Tech II Semester in the

ELECTRONIC CIRCUIT ANALYSIS LABORATORY.

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

JAWAHARLAL NEHRU TECHNOLOGICAL UNIVERSITY ANANTAPUR

B.Tech –II-II Sem (15A04404) ELECTRONIC CIRCUIT ANALYSIS LABORATORY

LIST OF EXPERIMENTS Branch: ECE Regulation:R15

(Minimum 10 experiments to be done using simulation software and hardware)

1. Determination of f_T of a given transistor.
2. Voltage-Series Feedback Amplifier.
3. Current-Shunt Feedback Amplifier.
4. RC Phase Shift/Wein Bridge Oscillator.
5. Hartley/Colpitt’s Oscillator.
6. Two Stage RC Coupled Amplifier.
7. Darlington Pair Amplifier.
8. Bootstrapped Emitter follower.
9. Class A Series-fed Power Amplifier.
10. Transformer-coupled Class A Power Amplifier.
11. Class B Push-Pull Power Amplifier.
12. Complementary Symmetry Class B Push-Pull Power Amplifier.
13. Single Tuned Voltage Amplifier.
14. Double Tuned Voltage Amplifier.

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 Through innovative 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 and high frequencies.
- To understand the characteristics of Differential amplifiers, feedback and power amplifiers.
- 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	The ability to analyze and design single and multistage amplifiers at low,mid and high frequencies.	L1
CO2	Designing and analyzing the transistor at high frequencies.	L2
CO3	Determine the efficiencies of power amplifiers	L3
CO4	Determine Frequency response and design of tuned amplifiers.	L4
CO5	Able to Analyze all the circuits using simulation software and Hardware.	L5

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

Course Title	PO 1	PO 2	PO 3	PO 4	PO 5	PO 6	PO 7	PO 8	PO 9	PO 10	PO 11	PO 12	PSO 1	PSO 2
Electronic Circuit Analysis Lab	3	3	2	3	3	2	2	1	2	2	1	2	3	2

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

Course Title	PO 1	PO 2	PO 3	PO 4	PO 5	PO 6	PO 7	PO 8	PO 9	PO 10	PO 11	PO 12	PSO 1	PSO 2
CO1	3	3	3	3	3	2	1	1	1	2	1	2	3	2
CO2	2	2	2	2	2	1	2	1	2	2	1	2	2	2
CO3	3	3	3	3	3	3	2		2	2	1		3	2
CO4	2	3	2	3	2	2	2	2	2	1	2	2	2	2
CO5	3	3	2	2	3	2	1	1	1	2		1	3	1

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

PART – A

Using simulation software

(Off the Syllabus)

Experiment No. : 01 **Date :**

Name of the Experiment : VOLTAGE SERIES FEED BACK AMPLIFIER

(By using simulation software)

AIM :

To obtain the frequency response of *Voltage Series feedback Amplifier***.**

APPARATUS :

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

THEORY :

1. Amplifier means which amplifies the sinusoidal and non-sinusoidal wave forms with out change in frequency. In voltage series feedback amplifier, network is in parallel with the the output of the amplifier.
2. A fraction of the output voltage through the feedback network is applied in series with in the input voltage of the amplifier.
3. The series connections at the input, increase the input resistance. In this case the amplifier is a true voltage amplifier.
4. The common collector or emitter follower is an example of voltage series feedback amplifier. Since the voltage developed in the output is in series with the input voltage as for as the base – emitter junction is connected.

CIRCUIT DIAGRAM :

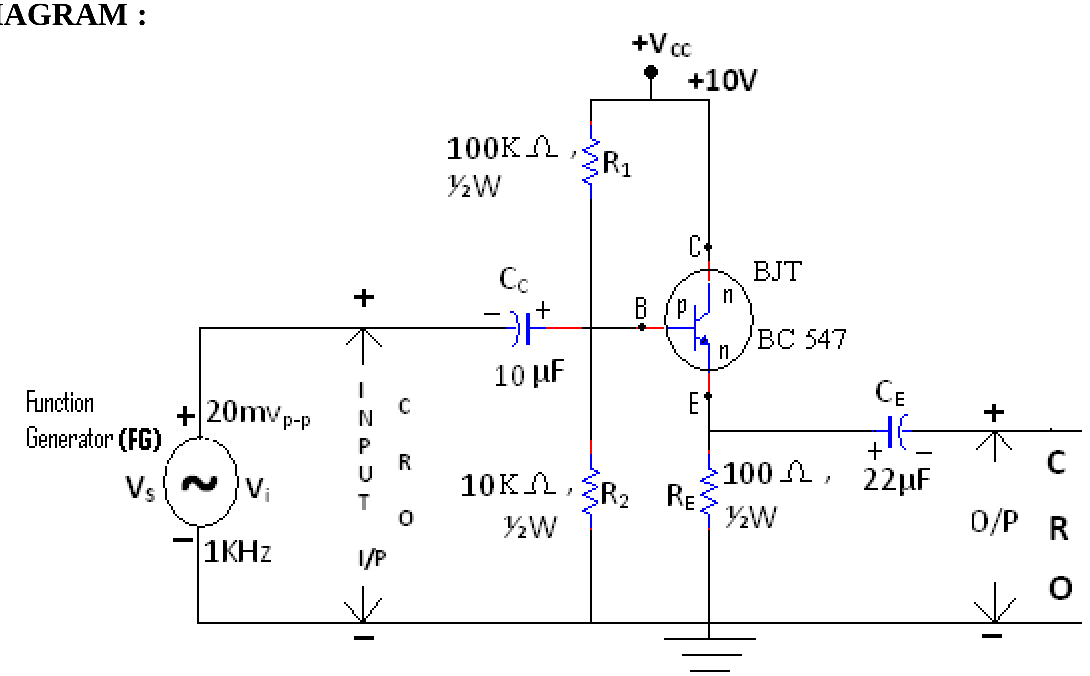


Figure: Circuit diagram of voltage series feed back amplifier.

PROCEDURE :

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 picked up.
3. Connected the CRO across the *Emitter capacitor to ground..*
4. Set the input signal as *sine wave form which is having the* value $20mV_{P-P}$ as constant in the function generator.
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 of 1KHz.
9. Stopped the simulation by clicked on *run option* through *execute button* in the *tool bar.*
10. Repeat the same procedure from points 6 to 9 for the corresponding frequency values by setting in the function generator for the following steps,
    10Hz, 500Hz, 1KHz, 100KHz, 200KHz, 400KHz,600KHz, 800KHz, 1MHz, 100MHz, 500MHz. in the function generator.
11. Observed the graph for *frequency* Vs *amplitude* through the *AC Analysis*.
12. Finally shut down the system safely.
13. We have observed that, the graph which is drawn by manually is same to the graph which is obtained from the *AC Analysis.*
14. 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,

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

15). 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.

TABULAR FORM :

Sl.No.	**InputVoltage(V_i) In milli Volts (peak to peak)**	**Frequency In Hz/KHz.**	**Output Voltage(V_O) In mVolts.**	**Voltage gain $A_V = V_o/V_i$**	**Gain in dB = $20\log_{10}(A_V)$**
01	20mV	10 Hz.			
02	20mV	500 Hz.			
03	20mV	1 KHz.			
04	20mV	100 KHz.			
05	20mV	200 KHz.			
06	20mV	400 KHz.			
07	20mV	600 KHz.			
08	20mV	800 KHz.			
09	20mV	1 MHz.			
10	20mV	100 MHz			
11	20mV	500 MHz.			

EXPECTED GRAPH :

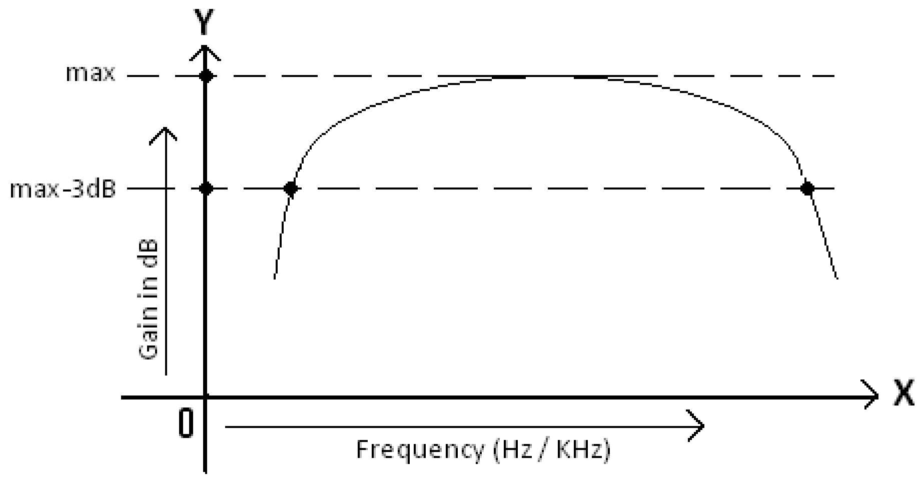


Figure: Frequency response curve of voltage series feed back amplifier.
For frequency verses gain in dB.

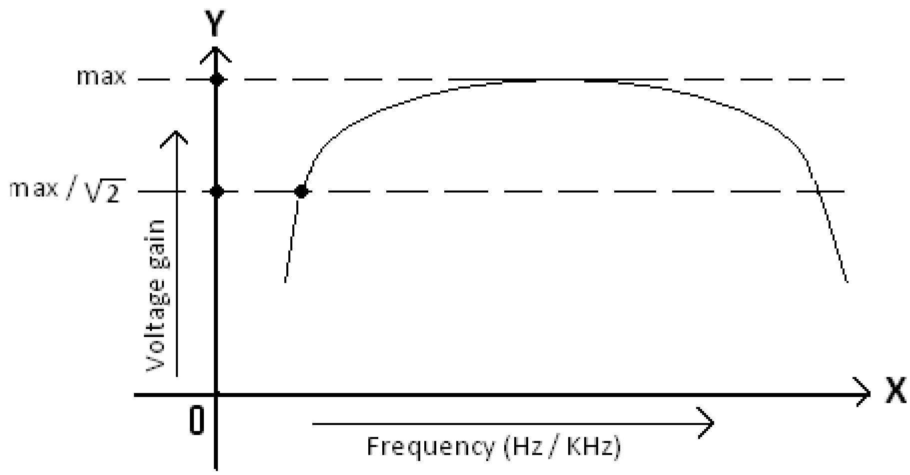


Figure: Frequency response curve of voltage series feed back amplifier.
For frequency verses voltage gain.

Note : It is not possible to find the band width because there is no amplification in this amplifier. It is just working as *buffer*.

RESULT :

We have obtained the frequency response curve *frequency verses gain in dB* and *frequency verses voltage gain of a given amplifier.*

VIVA VOICE QUESTIONS:

1. What is feedback?

2. What are the advantages of negative feedback?

3. What are the feedback topologies?

4. Example for voltage series feedback amplifier.

5. What are the CC Amplifier characteristics?

6. What are the Applications of Multi stage amplifiers?

7. Example for voltage series feedback amplifier.

8. CC Amplifier characteristics?

9. What is Band Width?

10. Explain the transistor operation with the help of four regions.

Experiment No. : **02** **Date :**
Name of the Experiment : **BOOTSTRAPPED EMITTER FOLLOWER**
(By using simulation software)

AIM :

To obtain the frequency response of *Bootstrapped Emitter Follower*

APPARATUS :

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

THEORY :

Typically Bootstrapping is technique where some part of output is used at the startup. In Bootstrap amplifier, bootstrapping is used to increase the input impedance. Due to which the loading effect on the input source also decreases. The design looks similar to the Darlington pair, having a bootstrap capacitor. Bootstrap capacitor is used to provide AC signal's positive feedback to the base of the transistor. This positive feedback help in improving the effective value of the base resistance. This increment in the base resistance also determined by the voltage gain of the amplifier circuit.

High input impedance improves the amplification of the input signal and thus required in various amplifier applications. If we have low input impedance we will get low amplification. Generally, BJT (Bipolar Junction Transistor) have low input impedance (typically 1 ohm to 50 kilo ohm). So for this, bootstrapping technique is used to increase the input impedance.

The voltage across the input impedance is calculated by using the below formula:

$$V = \{(V_{in}.Z_{in}) / (V_{in} + Z.V_{in})\}$$

Hence, according to the formula, the input impedance is proportional to the voltage across it. If the input impedance is increased the voltage across it will also increase and vice versa.

CIRCUIT DIAGRAM :

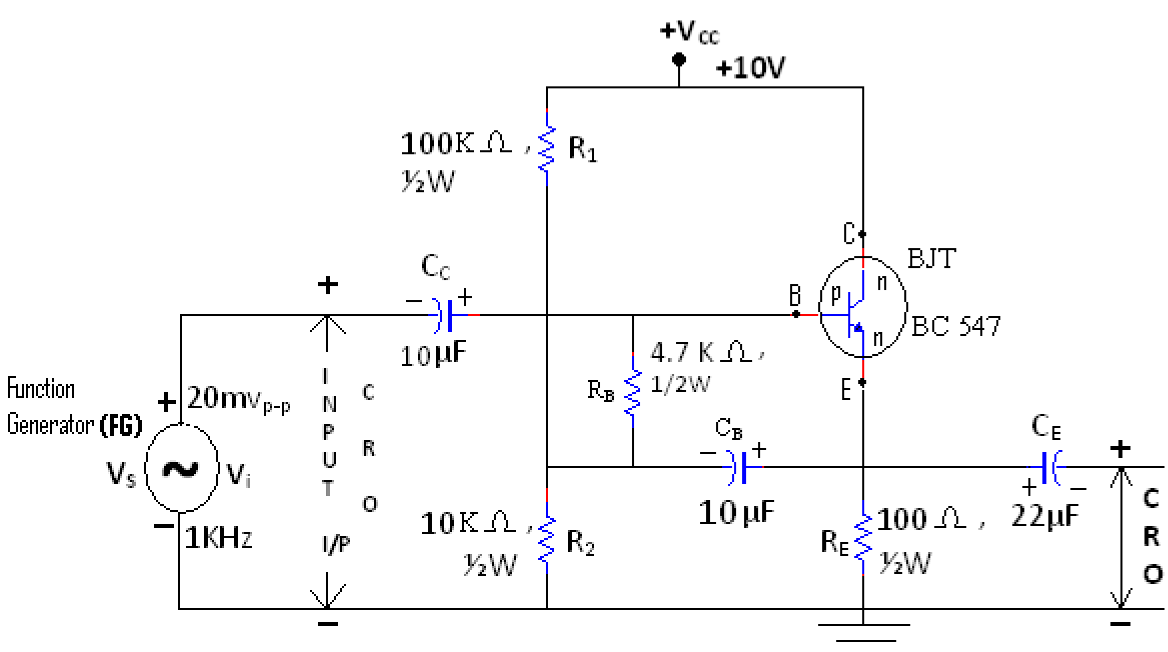


Figure: Circuit diagram of Bootstrapped Emitter Follower

PROCEDURE :

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 picked up.
3. Connected the CRO across the Emitter capacitor to ground.
4. Set the input signal as *sine wave form which is having the* value 20mV$_{P-P}$ as constant in the function generator.
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 of 1KHz.
9. Stopped the simulation by clicked on *run option* through *execute button* in the *tool bar*.
10. Repeat the same procedure from points 6 to 9 for the corresponding frequency values by setting in the function generator for the steps of 10Hz, 500Hz, 1KHz, 100KHz, 200KHz, 400KHz,600KHz, 800KHz, 1MHz, 100MHz, 500MHz. in the function generator.
11. Observed the graph for *frequency* Vs *amplitude* through the *AC Analysis*.
12. Finally shut down the system safely.
13. We have observed that, the graph which is drawn by manually is same to the graph which is obtained from the *AC Analysis*.
14. 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,

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

15). 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.

Note: Bootstrap Emitter Follower uses to increase the input impedance and to work as correct *Buffer*. For example, The voltage gain of *voltage series feedback amplifier* is 1 it means the output voltage is equal to input voltage, then we can say that it is the correct *Buffer*. Now If you observed the output of *voltage series feedback amplifier* the output voltage is less as compared to input voltage, it means buffer is incorrect. To increase the output voltage which is equal to the input voltage here we have used the *Bootstrapped Emitter Follower*.

TABULAR FORM :

Sl.No.	**InputVoltage(V_i) In milli Volts (peak to peak)**	**Frequency In Hz/KHz.**	**Output Voltage(V_O) In mVolts.**	**Voltage gain $A_V = V_o/V_i$**	**Gain in dB = $20\log_{10}(A_V)$**
01	20mV	10 Hz.			
02	20mV	500 Hz.			
03	20mV	1 KHz.			
04	20mV	100 KHz.			
05	20mV	200 KHz.			
06	20mV	400 KHz.			
07	20mV	600 KHz.			
08	20mV	800 KHz.			
09	20mV	1 MHz.			
10	20mV	100 MHz			
11	20mV	500 MHz.			

EXPECTED GRAPH :

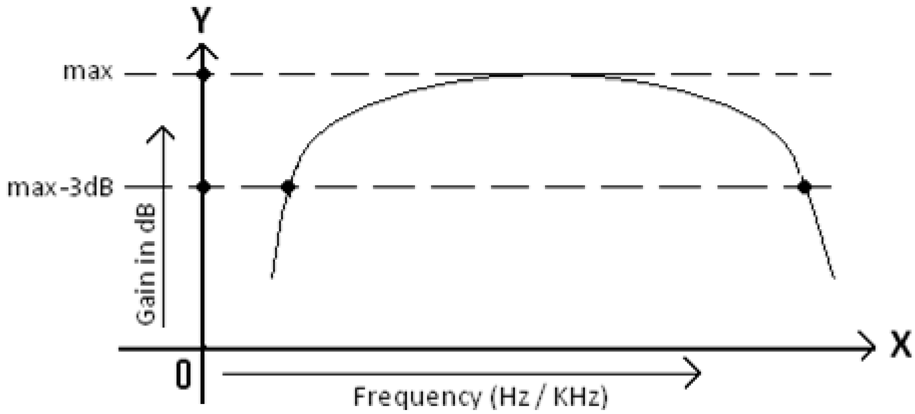


Figure: Frequency response curve of Bootstrapped Emitter Follower.
For Frequency verses gain in dB.

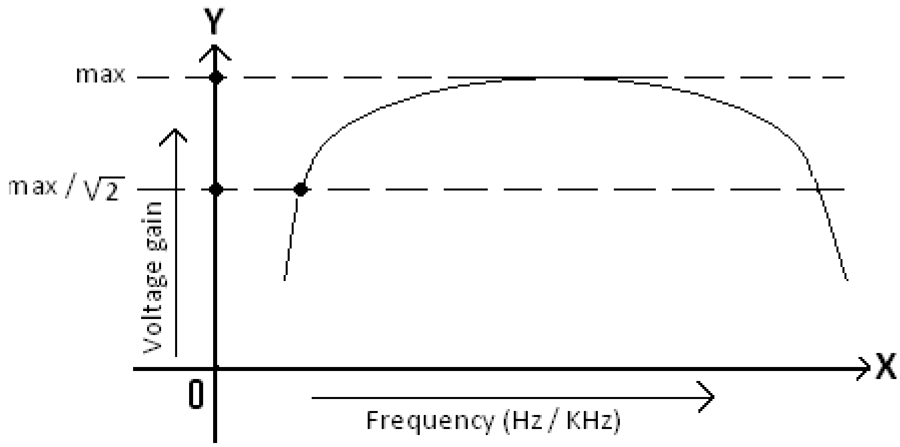


Figure: Frequency response curve of Bootstrapped Emitter Follower.
For frequency verses voltage gain

Note : It is not possible to find the band width because there is no amplification in this amplifier. It is just working as *buffer*.

RESULT :

We have obtained the gain of *Bootstrapped Emitter Follower* for different frequencies .

VIVA VOICE QUESTIONS:

1. What is Bootstrap Emitter Follower?

2. Mention the applications of Bootstrap Emitter follower?

3. Which one is Emitter follower? (CE / CB / CC)

4. What are the CC Amplifier characteristics

5. What are the Applications of Multi stage amplifiers?

Experiment No. : 03 **Date :**

Name of the Experiment : DARLINGTON PAIR AMPLIFIER

(Using Simulation software)

AIM :

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

APPARATUS :

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

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}$\beta 1$ and _{\beta 2}$\beta 2$ be the current gains of individual transistors then overall current gain of Darlington pair amplifier = $\beta 1\beta 2$.

CIRCUIT DIAGRAM :

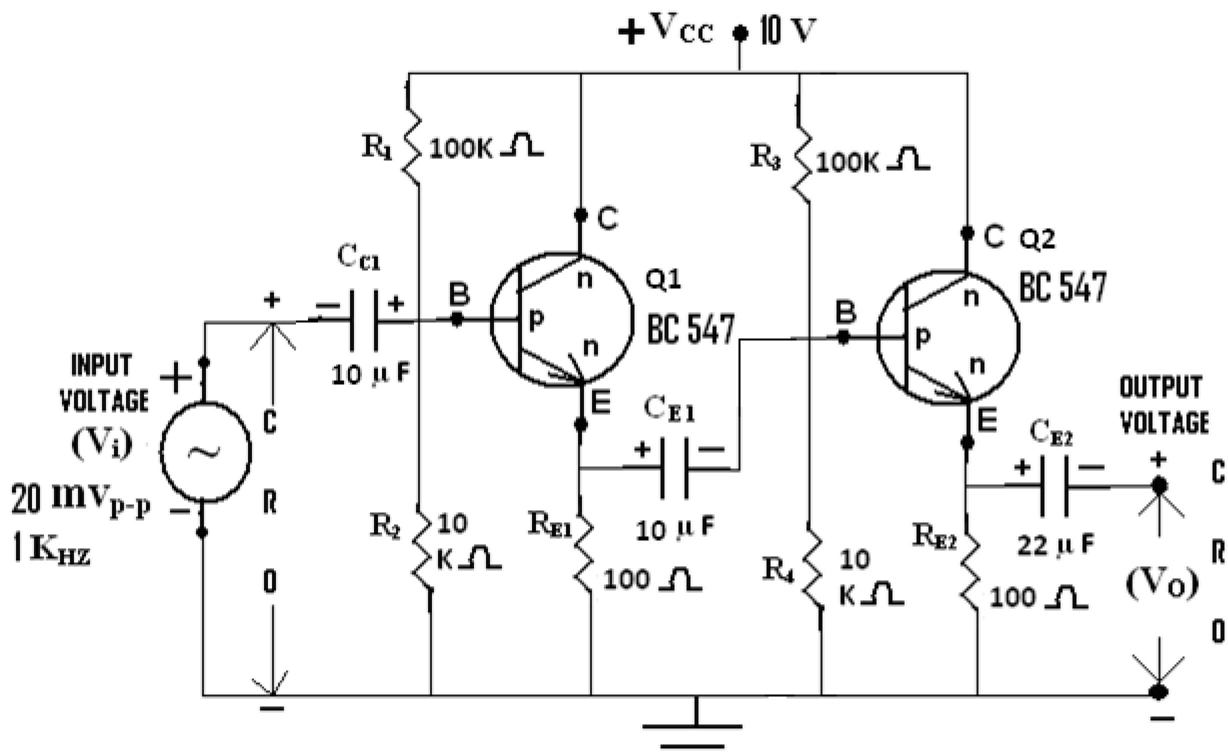


Figure: Circuit diagram of Darlington pair amplifier.

PROCEDURE :

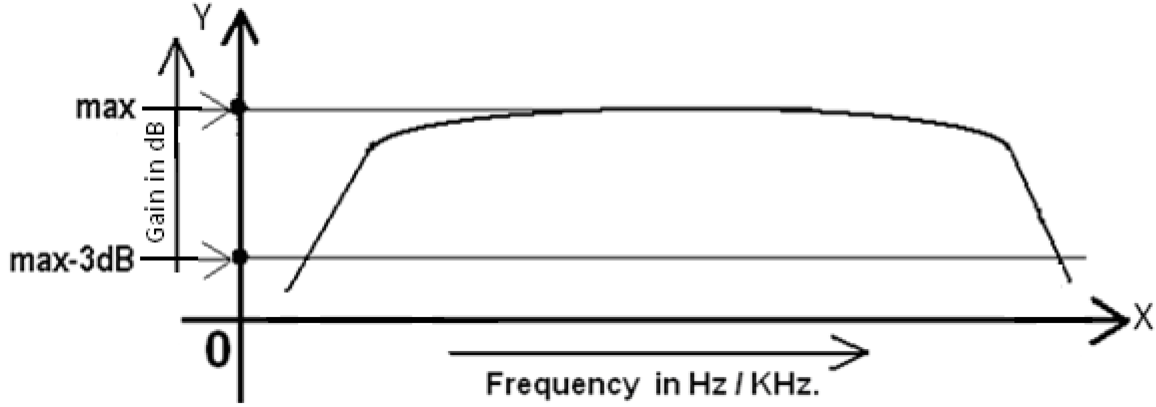
1. We have picked up the components from the components bar as per above circuit.
2. Made the connections as per the above circuit diagram by using the components which we have picked up.
3. Connected the CRO across the capacitor C_{E2} of second stage.
4. Set the input signal as *sine wave form which is having the* value $20mV_{P-P}$ as constant in the function generator.
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 of 1KHz.
9. Stopped the simulation by clicked on *run option* through *execute button* in the *tool bar.*
10. Repeat the same procedure from points 6 to 9 for the corresponding frequency values by setting in the function generator for the following steps,
    10Hz, 500Hz, 1KHz, 100KHz, 200KHz, 400KHz,600KHz, 800KHz, 1MHz, 100MHz, 500MHz. in the function generator.
11. Observed the graph for *frequency* Vs *amplitude* through the *AC Analysis*.
12. Finally shut down the system safely.
13. We have observed that, the graph which is drawn by manually is same to the graph which is obtained from the *AC Analysis.*
14. 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* $= 20\log_{10}(A_v)$.
15. Plotted the graph between *frequency on X*- axis and *gain in dB on Y*- axis.

TABULAR FORM:

Sl.No.	**InputVoltage(V_i) In milli Volts (peak to peak)**	**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	20mV	10 Hz.			
2	20mV	500 Hz.			
3	20mV	1 KHz.			
4	20mV	100 KHz.			
5	20mV	200 KHz.			
6	20mV	400 KHz.			
7	20mV	600 KHz.			
8	20mV	800 KHz.			
9	20mV	1 MHz.			
10	20mV	100 MHz			
11	20mV	500 MHz.			

EXPECTED GRAPH :

The following graph shows for frequency response curve of a *Darlington pair Amplifier*.



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 :

I have drawn the frequency response curve of a *Darlington pair amplifier*.

VIVA VOICE 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?

6. Compare CB,CE, CC configurations of a transistor

7. Explain the transistor operation with the help of four regions

8. What is cascade Amplifier?

9. Explain base width modulation of a transistor

10. Which Amplifier is having CC-CC configuration?

Experiment No. : 04 **Date :**

Name of the Experiment : TWO STAGE RC COUPLED AMPLIFIER
(Using Simulation software)

AIM :

To verify / plot the frequency response curve *and* to find the band width.of a *single stage & two stage RC coupled Amplifier* using Multisim software.

98*/74

APPARATUS :

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

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 :

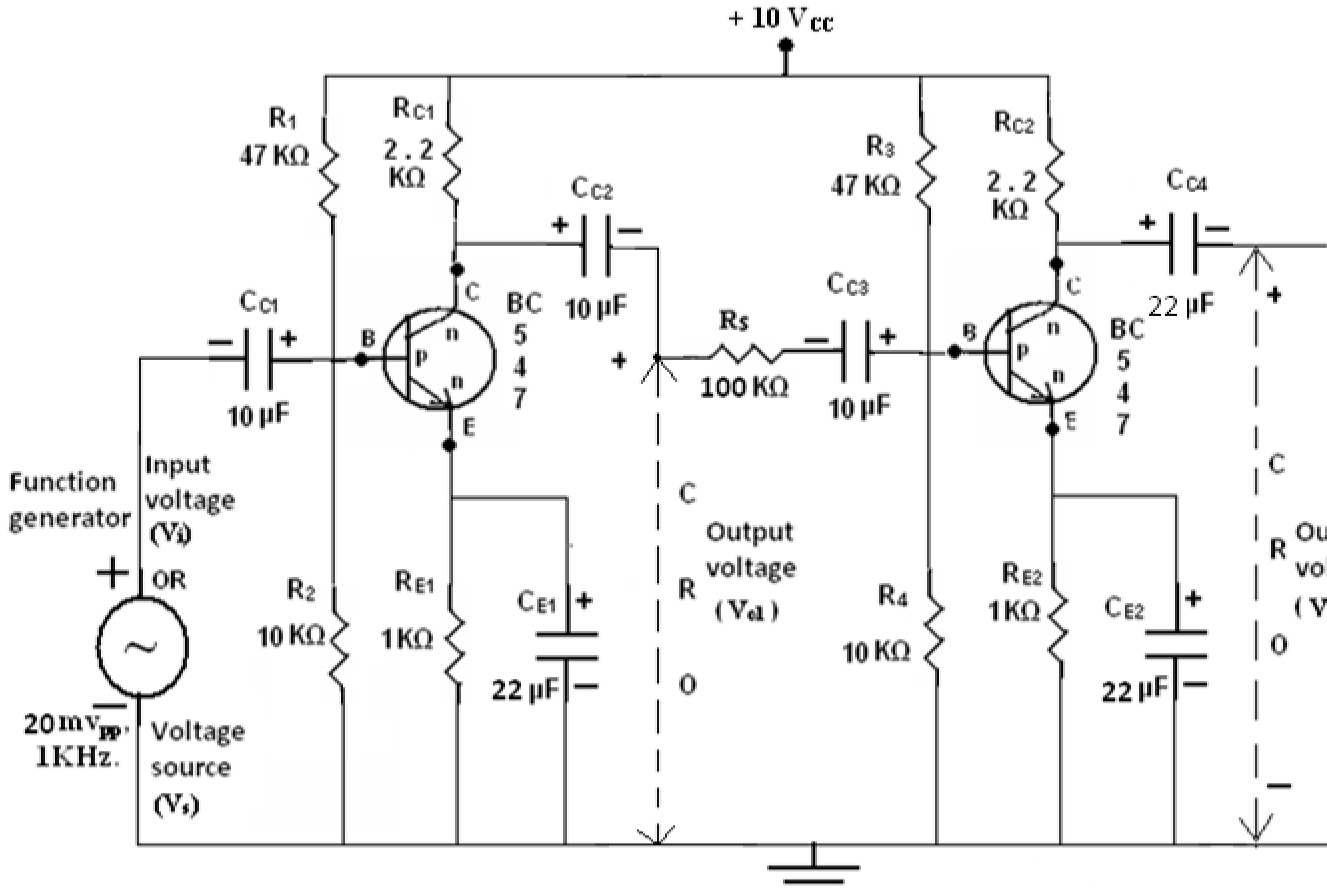


Figure: Circuit diagram of Two stage RC coupled amplifier.

PROCEDURE :

1. We have picked up the components from the tool bar as per above circuit in Multisim software.
2. Made the connections only for single stage up to capacitor C_{C2} as per the above circuit diagram by using the components which have picked up.
3. Connected the CRO across the capacitor C_{C2} .
4. Set the input signal value as 20mV$_{P-P}$, 1KHz in the function generator as constant and V_{CC} as 10V.
5. To simulate the circuit click on execute / run button in tool bar.
6. We have seen the *Sine wave* on the CRO as O/P signal.
7. Noted/ observed the readings for o/p voltage (Peak to Peak) of output signal in CRO by varying the different frequency steps (i.e. 10Hz, 500Hz, 1KHz, 100KHz, 200KHz, 400KHz,600KHz, 800KHz, 1MHz, 100MHz, 500MHz.) of the input AC signal in function generator.
8. Noted the above readings to the corresponding frequency steps in the tabular form of *Single stage RC Couple Amplifier*.
9. Stop the simulation by click on *Run button* in *tool bar*.
10. Observed the graph *frequency* Vs *amplitude* through the AC Analysis for *Single stage RC Coupled Amplifier.*
11. After this made the circuit for *Two stage RC coupled amplifier.*
12. Remove the CRO which has connected across the capacitor C_{C2} and connected it across the capacitor C_{C4} of this circuit.

13. Noted the above readings to the corresponding frequency steps in the tabular form of *Two stage RC Couple Amplifier.*

14. Stop the simulation by click on *Run button* in *tool bar*.
15. Observed the graph *frequency* Vs *amplitude* through the AC Analysis for *Two stage RC Coupled Amplifier*.
16. Shut down the system safely.
17. Calculated and noted 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 the tabular form of both *Single stage* and *Two stage RC C*oupled amplifiers.
18. Drawn the graph for which the *frequency* on X-axis and *Gain* in *dB* on Y-axis for both *RC Coupled Amplifier* circuits..
19. We have calculated the bandwidth of both *RC Coupled amplifiers* from that graph as per given formula,

    Band width for Single stage RC Coupled Amplifier (BW) = $f_2 - f_1$

    Band width for Two stage RC Coupled Amplifier (BW) = $f_4 - f_3$
20. We have observed that, the graph which is drawn by manually is same to the graph which is obtained from the *AC Analysis* for both *RC coupled amplifiers.*

TABULAR COLUMN :

A). Single stage RC coupled Amplifier :

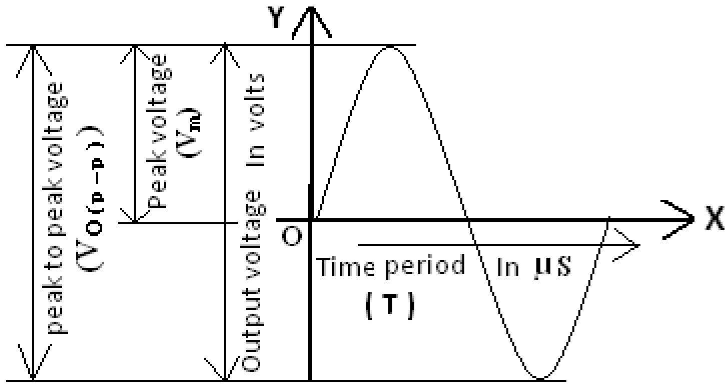
Sl. No.	**Input voltage (V_i) in mV**	**Frequency in Hz / KHz.**	**Output Voltage (V_o) in V**	**Voltage gain (A_v) = V_o / V_i**	**Gain in dB =$20\log_{10}(A_v)$**
1	20	10 Hz.			
2	20	500 Hz.			
3	20	1 KHz.			
4	20	100 KHz.			
5	20	200 KHz.			
6	20	400 KHz.			
7	20	600 KHz.			
8	20	800 KHz.			
9	20	1 MHz.			
10	20	100 MHz			
11	20	500 MHz.			

B). Two stage RC coupled Amplifier :

Sl. No.	**Input voltage (V_i) in mV**	**Frequency in Hz / KHz.**	**Output Voltage (V_o) in V**	**Voltage gain (A_v) = V_o / V_i**	**Gain in dB =$20\log_{10}(A_v)$**
1	20	10 Hz.			
2	20	500 Hz.			
3	20	1 KHz.			
4	20	100 KHz.			
5	20	200 KHz.			
6	20	400 KHz.			
7	20	600 KHz.			
8	20	800 KHz.			
9	20	1 MHz.			
10	20	100 MHz			
11	20	500 MHz.			

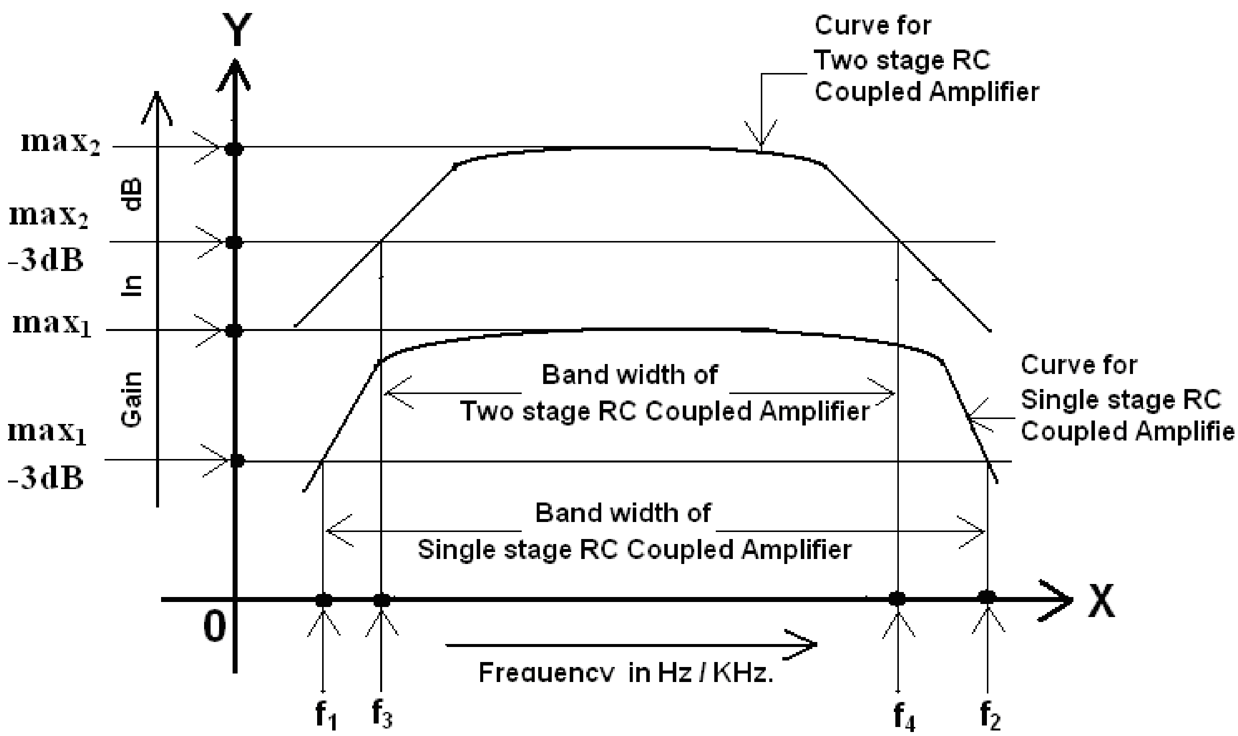
EXPECTED WAVEFORM :

I have 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 VOICE 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. Compare BJT and FET?

8. What is thermal runaway?

9. What are the advantages of using potential divider bias?

10. What is cascade Amplifier?

Experiment No. : 05 **Date :**

Name of the Experiment : CURRENT SHUNT FEEDBACK AMPLIFIER
(Using simulation software)

AIM :

1). To plot the graph for frequency response curve of a *Current shunt feedback Amplifier* for both *with feed back* and *without feedback.*

2). To find the bandwidth of *Current shunt feedback Amplifier.*

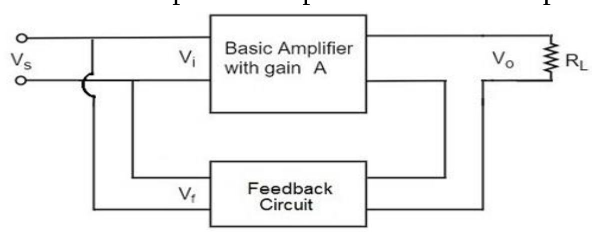
APPARATUS :

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

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.

The below figure shows the block diagram of current shunt feedback, by which it is evident that the feedback circuit is placed in series with the output but in parallel with the input.

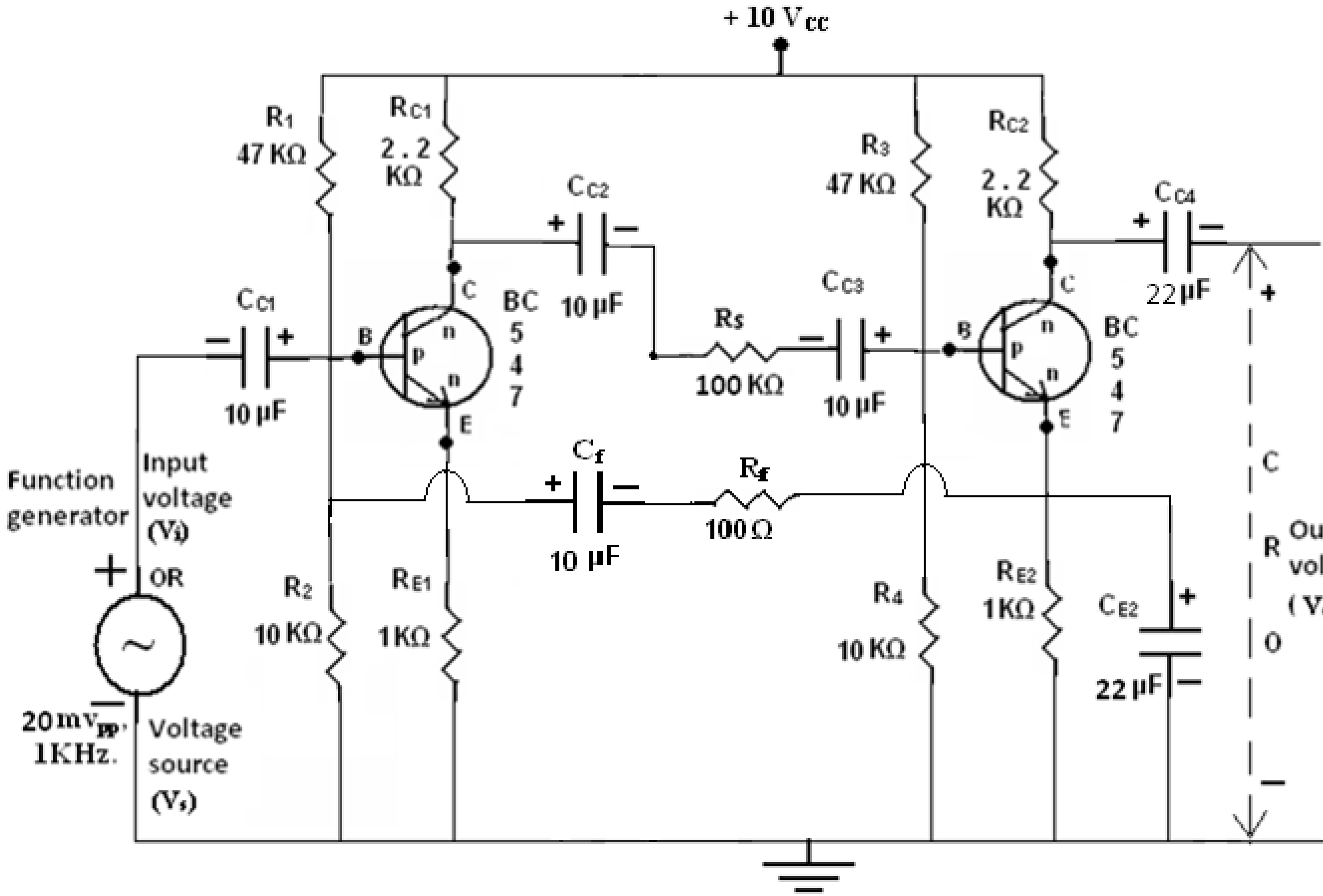


As the feedback circuit is connected in series with the output, the output impedance is increased and due to the parallel connection with the input, the input impedance is decreased.

Let us now tabulate the amplifier characteristics that get affected by different types of negative feedbacks.

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 :



PROCEDURE :

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 picked up.
3. Connected the CRO across the capacitor C_{C4}.
4. Set the input signal as *sine wave form which is having the* value $20mV_{P-P}$ as constant in the function generator.
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 column of 1 KHz. For tabular form of *with feedback* amplifier.
9. Stopped the simulation by clicked on *run option* through *execute button* in the *tool bar.*
10. Repeat the same procedure from points 6 to 9 for the corresponding frequency values by setting in the function generator for the following steps,
    10Hz, 500Hz, 1KHz, 100KHz, 200KHz, 400KHz,600KHz, 800KHz, 1MHz,100MHz, 500MHz in the function generator.
11. Observed the graph for *frequency* Vs *amplitude* through the *AC Analysis*.
12. Disconnected the C_f and R_f from the circuit and now the circuit has became as *without feedback amplifier*
13. Now taken the reading in the tabular form of *without feedback amplifier* by repeat the steps from 6 to 11
14. Finally shut down the system safely.

15. We have observed that, the graph which is drawn by manually is same to the graph which is obtained from the *AC Analysis.*
16. We have observed that the readings of *without feed back amplifier's* output voltage is greater than the *with feed back amplifier*
17. Calculated the Voltage gain by using the formula of 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*.
18. Drawn the graphs of both amplifiers in single graph sheet.
19. While drawing the graph taken the *frequency* on X-axis and *Gain in dB* on Y-axis.
20. Finally calculated the *bandwidth* of both amplifiers from this graph sheet as per the following formulas,
    i). For *Current shunt feed back amplifier (With feed back*) (BW) = $f_2 - f_1$
    ii). For *Current shunt feed back amplifier (Without feed back*) (BW) = $f_4 - f_3$
21. We have noted down that the *band width* of *with feed back amplifier* is high compared to the *without feed back amplifier.*

TABULAR COLUMN :

A). With feed back :

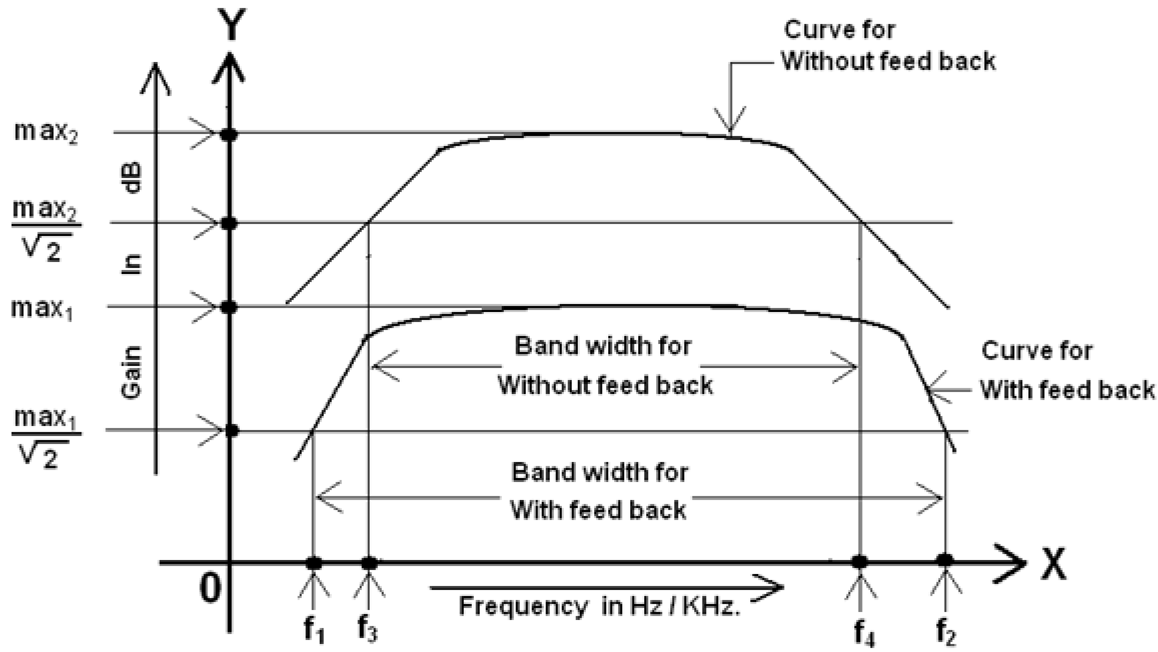
Sl. No.	Input voltage (V_i) in mV	Frequency in Hz / KHz.	Output Voltage (V_o) in mV	Voltage gain (A_v) = V_o / V_i	Gain in dB =$20log_{10}(A_v)$
1	20	10 Hz.			
2	20	500 Hz.			
3	20	1 KHz.			
4	20	100 KHz.			
5	20	200 KHz.			
6	20	400 KHz.			
7	20	600 KHz.			
8	20	800 KHz.			
9	20	1 MHz.			
10	20	100 MHz			
11	20	500 MHz.			

B). Without feed back :

Sl. No.	Input voltage (V_i) in mV	Frequency in Hz / KHz.	Output Voltage (V_o) in mV	Voltage gain (A_v) = V_o / V_i	Gain in dB =$20log_{10}(A_v)$
1	20	10 Hz.			
2	20	500 Hz.			
3	20	1 KHz.			
4	20	100 KHz.			
5	20	200 KHz.			
6	20	400 KHz.			
7	20	600 KHz.			
8	20	800 KHz.			
9	20	1 MHz.			
10	20	100 MHz			
11	20	500 MHz.			

EXPECTED GRAPH :

The following graph shows for *Current shunt feed back amplifier* for both *with feed back* and *without Feedback amplifier*.



RESULT :

We drawn the graph for frequency response of a *Current shunt feed back amplifier* for both *with feed back* and *without feed back.*

VIVA VOICE 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. CC Amplifier characteristics?

6. What is Band Width?

7. What is Frequency Response?

8. Explain the transistor operation with the help of four regions

Experiment No. : 06 **Date :**

Name of the Experiment : SINGLE TUNED VOLTAGE AMPLIFIER

(Using simulation software)

AIM :

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

APPARATUS :

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

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 :

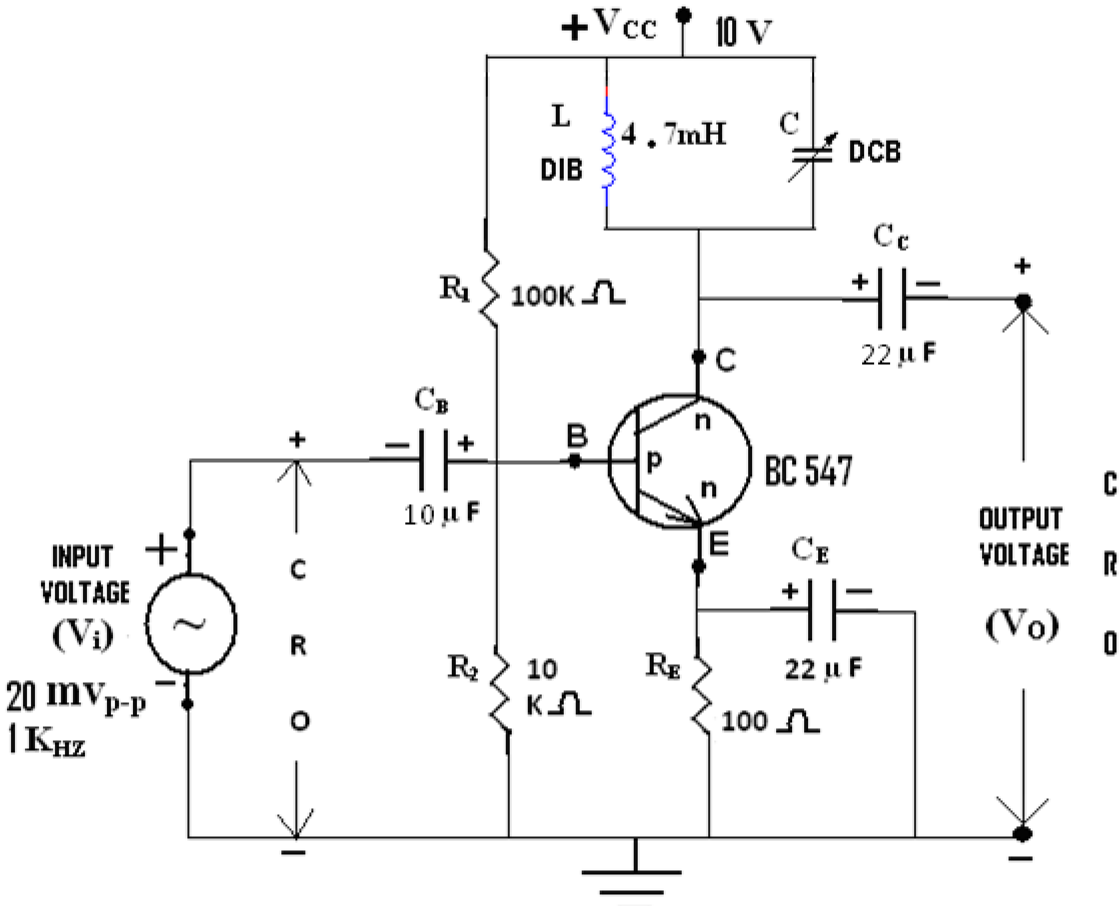


Figure: Circuit diagram of single tuned voltage amplifier.

THEORETICAL CALCULATIONS :

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

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

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

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

= 54Kpf OR 54 nF

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

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

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

= 2.16Kpf OR 2.16nF

PROCEDURE :

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 picked up.
3. Connected the CRO across the *Collector capacitor to ground..*
4. Now set the C=54Kpf and L=4.7mH to take readings in tabular form-1
5. Set the input signal as *sine wave form which is having the* value 20mV$_{P-P}$ as constant in the function Generator until the experiment would completed.
6. To simulate the circuit clicked on *run option* through *execute button* in *tool bar.*
7. We could seen the *sine wave* on the screen of *CRO* as o/p signal.
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 corresponding frequency values which are available in the tabular form-1.
10. Stopped the simulation by clicked on *run option* through *execute button* in the *tool bar.*
11. Observed the graph for *frequency* Vs *amplitude* through the *AC Analysis*.
12. Now set C=2.16Kpf and L=4.7mH to take readings in tabular form-2.
13. Repeat the same procedure from points 6 to 11 for the corresponding frequency values which are available in the tabular form
14. Finally shut down the system safely.
15. 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.
16. 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.
17. Calculated and noted the *band width* & *resonant frequency* from both frequency response curves by using the following formula,
    $$\text{Band width} = f_2 - f_1.$$
18. We have observed that, the graph which is drawn by manually is same to the graph which is obtained from the *AC Analysis.*

TABULAR FORM – 1:

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

Sl. No.	Input voltage (V_i) in mV	Frequency in Hz / KHz.	Output Voltage (V_o) in V	Voltage gain (A_v) = V_o / V_i	Gain in dB =$20\log_{10}(A_v)$
1	20	10 Hz.			
2	20	500 Hz.			
3	20	1 KHz.			
4	20	5 KHz.			
5	20				
6	20	20KHz.			
7	20	50KHz.			
8	20	100 KHz.			
9	20	200 KHz.			
10	20	500 KHz.			
11	20	1 MHz.			
12	20	100 MHz			
13	20	500 MHz.			

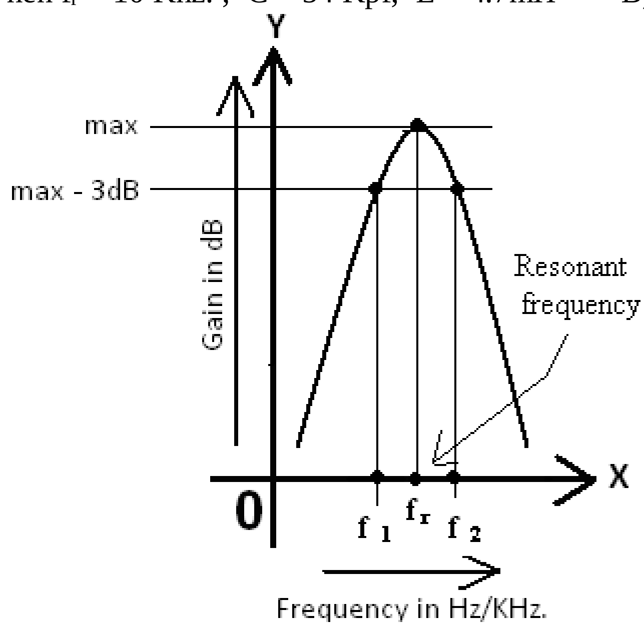
TABULAR FORM – 2 :
When f_r = 50Khz. , C = 2.16Kpf, L = 4.7mH

Sl. No.	**Input voltage (V_i) in mV**	**Frequency in Hz / KHz.**	**Output Voltage (V_o) in V**	**Voltage gain (A_v) = V_o / V_i**	**Gain in dB =$20\log_{10}(A_v)$**
1	20	10 Hz.			
2	20	500 Hz.			
3	20	1 KHz.			
4	20	20KHz.			
5	20				
6	20	60KHz.			
7	20	100 KHz.			
8	20	200 KHz.			
9	20	500 KHz.			
11	20	1 MHz.			
12	20	100 MHz.			
13	20	500 MHz.			

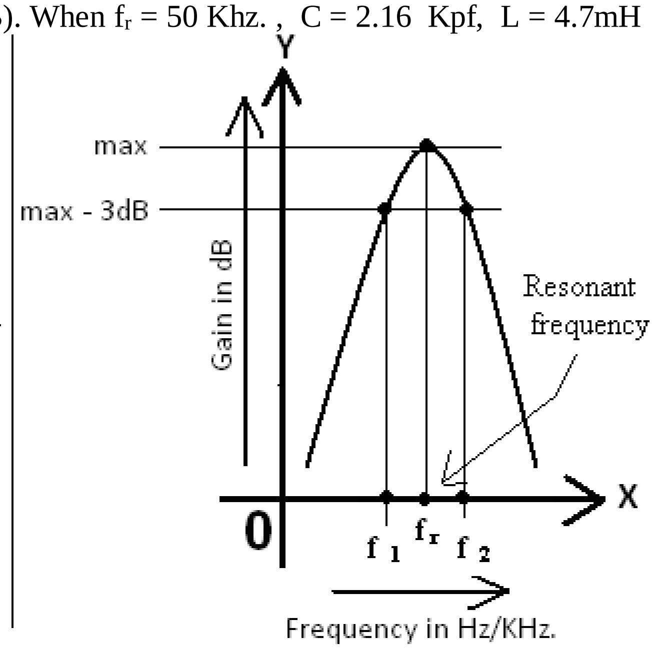
EXPECTED GRAPH :

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 VOICE 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. Tuned Amplifier is Narrow or Wide BW Amplifier?

7. Difference between single tuned and double tuned Amplifier?

8. What is stagger tuned Amplifier?

9. Effect of cascading of single tuned Amplifier on BW?

10. What is frequency response?

Experiment No. : **07** **Date :**
Name of the Experiment : **CLASS A SERIES - FED POWER AMPLIFIER (Using Simulation Software)**

AIM :

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

APPARATUS :

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

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 :

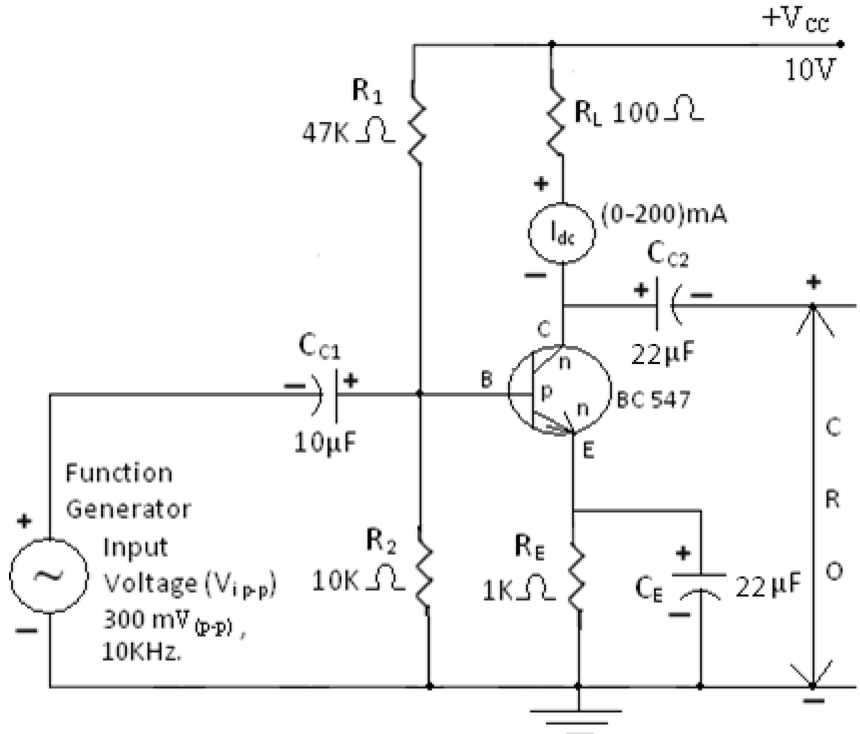


Figure: Circuit diagram of Class A Series-fed Power Amplifier

PROCEDURE :

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-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* 20V 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-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%.***

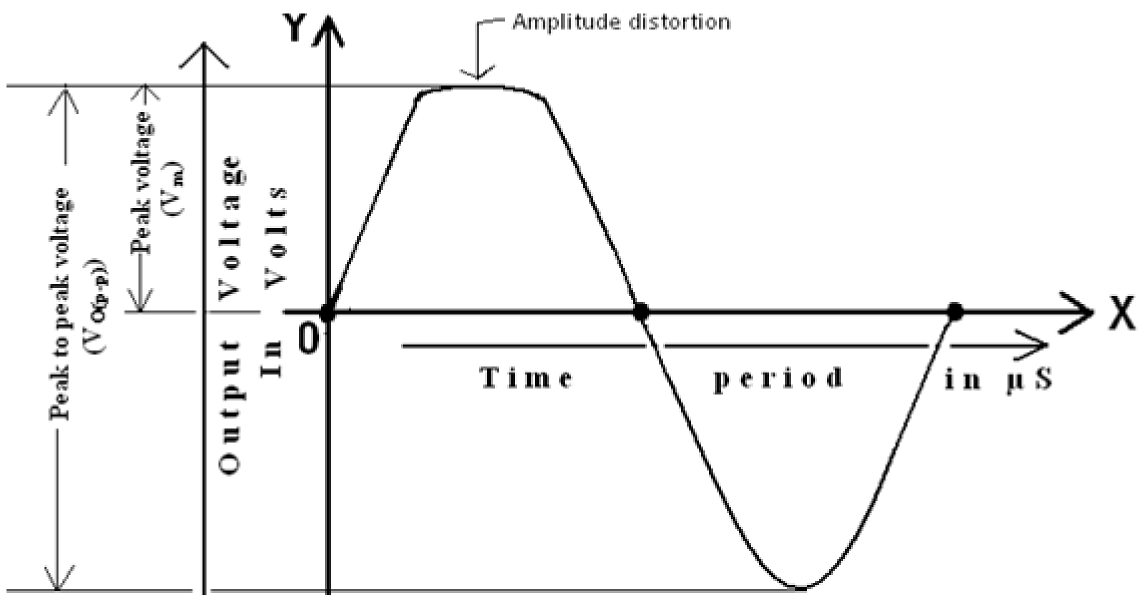
PRACTICAL CALCULATIONS :

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

Sl.No.	Name of the parameter	Value
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 $P_o\,(ac) = \dfrac{V_{O\,(p\text{-}p)}^2}{8R_L}$ (In Watts)	
11	% of efficiency (η) = [P_O(ac) / P_i (dc)] ×100 =	

EXPECTED WAVEFORM :

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



RESULT :

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

VIVA VOICE Questions:

1. What is Power Amplifier?

2. Classifications of power Amplifiers.

3. Efficiency of class A power Amplifier.

4. Difference between Direct coupled and Transformer coupled class A power Amplifier?

5. What is the amplitude (Harmonic) Distortion?

6. Where is the Q point in class A power Amplifier?

7. Applications of class A power Amplifier.

8. What are the disadvantages of class A power Amplifier.

9. Mention the conduction angle of class A power Amplifier.

10. What are the disadvantages of class A power Amplifier.

Experiment No. : 08 **Date :**

Name of the Experiment : COMPLEMENTARY SYMMETRY CLASS B PUSH-PULL POWER AMPLIFIER (Using Simulation Software)

AIM :

1. To verify / plot the output signal (sine wave form) of a given *Complementary symmetry Class-B push-pull Power Amplifier* by using *multisim* software.

2. To calculate the conversion efficiency of given *Complementary symmetry Class-B push-pull Power Amplifier* by using by using *multisim software*.

APPARATUS :

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

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 190° 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 190^o of the input waveform cycle while the other transistor amplifies the other half or remaining 190^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 190^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**

CIRCUIT DIAGRAM :

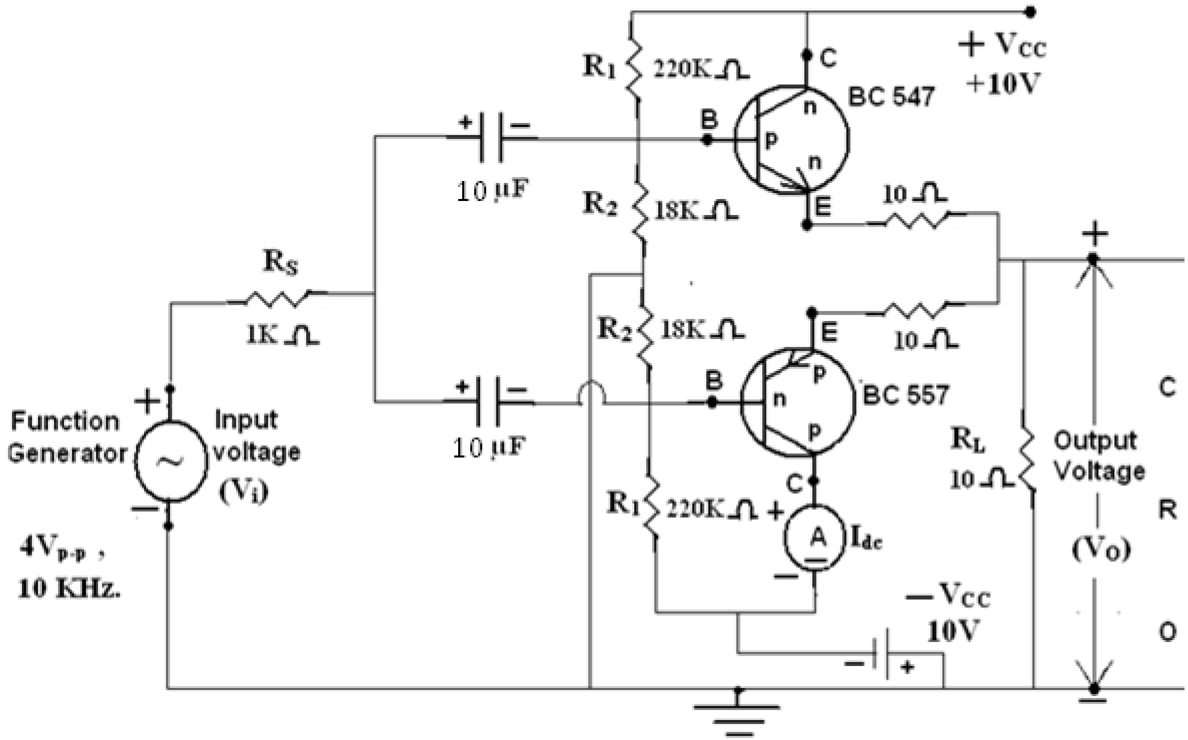


Figure: Circuit diagram of Class-B complimentary symmetry power amplifier.

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
04	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) = $\frac{V_m^2}{2R_L}$ (In mWatts) =	
12	% of efficiency (η) = $\frac{P_o(ac)}{P_i(dc)} \times 100$	
13	Typical Max. efficiency (η) =	78.50 %

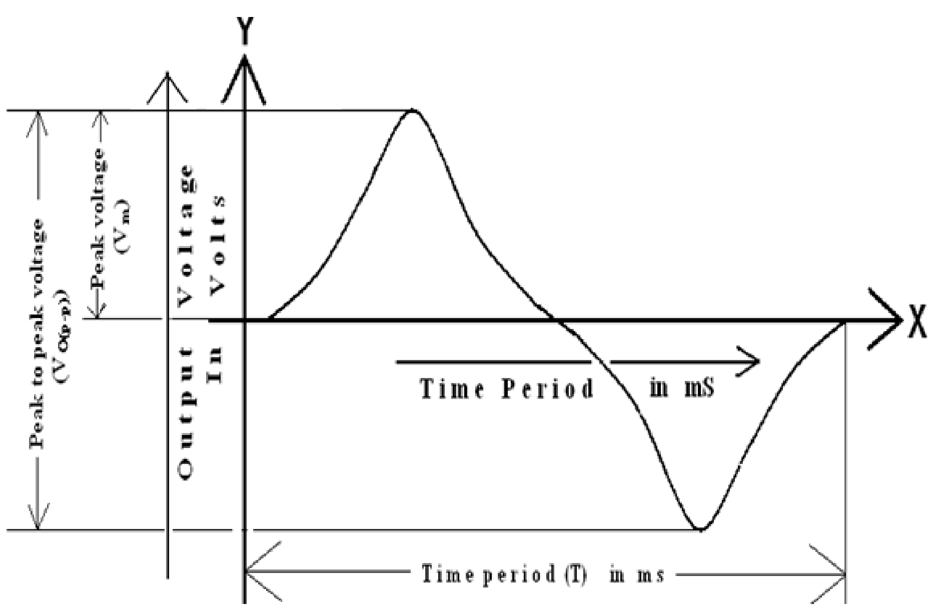
PROCEDURE :

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 4 V_{p-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* 20V as V_{CC} to the circuit as shown in the circuit..
6. To simulate this circuit click on *Run* button in *tool bar.*
7. Observed the *sine wave* signal in CRO_2 and drawn this signal on the graph sheet.
8. Calculated the *output voltage* (Vo_{p-p}) , *time period* (T), *frequency* (f) from the graph, and noted down these values against the corresponding columns of 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.* ***78.5%.***

Note: During the execution I have waited for 15 minutes of time until we get the maximum peak to peak amplitude. The max. peak to peak value is ≤ 0.84V

EXPECTED WAVEFORM :

The following waveform shows the output signal for *complementary symmetry Class B push-pull power amplifier.*



RESULT :

I have verified / drawn the output waveform and calculated the conversion efficiency of given *Complementary symmetry Class-B push-pull Power Amplifier*.

VIVA VOICE 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.

Experiment No. : 09 **Date :**

Name of the Experiment : **RC PHASE SHIFT OSCILLATOR (Using Simulation Software)**

AIM :

To verify the *sine wave form* and to calculate its frequency values of a given *RC Phase shift Oscillator* by using multisim software.

APPARATUS :

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

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 1900 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 90^o.

CIRCUIT DIAGRAM :

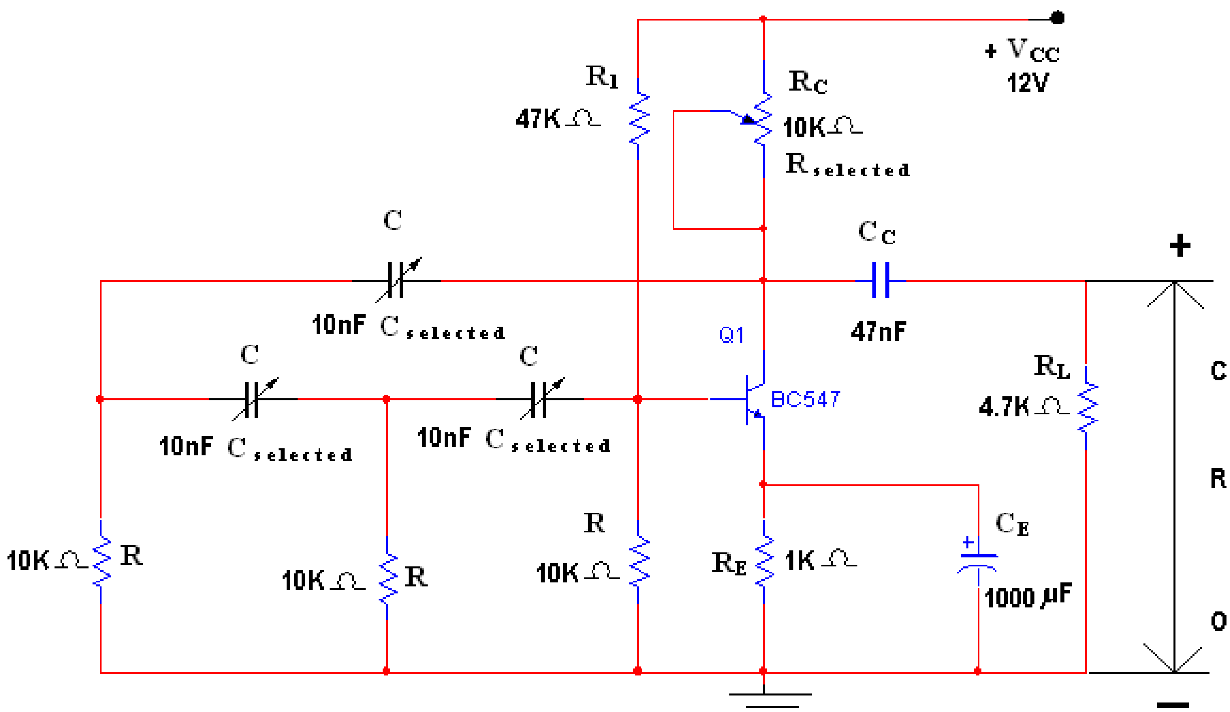


Figure: Circuit diagram of RC Phase shift oscillator.

PROCEDURE :

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 V_{CC} value as 12V.
4. Initially set the *Capacitor C* values as 1nF (0.001µF or 1Kpf) by varied the *capacitor C* Value in % by using the following formula,

$$\text{Setting in \%} = \frac{C_{required} \times 100}{C_{selected}}$$

5. To start the simulation clicked on *Run button* .
6. Varied the R_C value until we get *sine wave form* which is consist the $V_{O(p\text{-}p)}$ is approximately 6V because this circuit is designed to get the output voltage as $6V_{(p\text{-}p)}$ in the *CRO*.
7. . We observed *Sine wave form* as a output signal in the *CRO*.
8. Drawn the *sine wave form* on the graph by taking the *time period* on X-axis and *Amplitude* ($V_{O(p\text{-}p)}$) on Y-axis.
9. Calculated and noted the *collector resistor* (R_C) and *theoretical frequency* (f_o) value for corresponding *capacitor C* values in the tabular form by using the formulas which are given below,

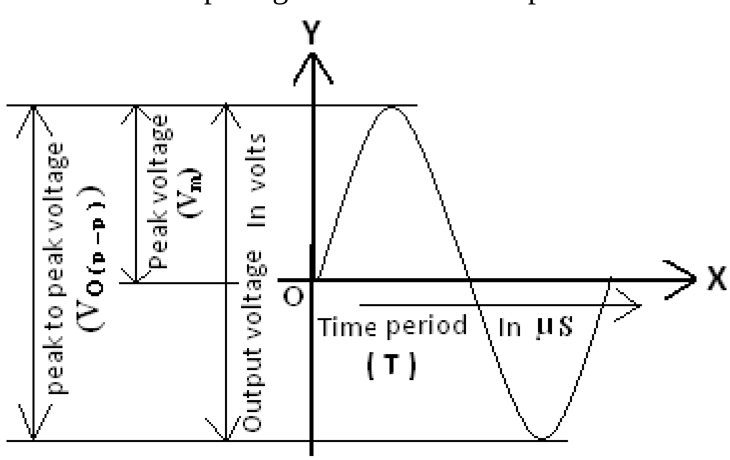
$$R_C = R_{Selected} - \left[\frac{\text{Setting in \%} \times R_{Selected}}{100}\right]$$

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

10. Calculated the frequency and output voltage ($V_{O\,(p\text{-}p)}$) values from the graph then noted in the Columns of *practical frequency* and *output voltage* for corresponding *capacitor C* valuesin the tabular form respectively.
11. Stopped the simulation by click on *Run option* through *Execute button.*
12. Repeat the same procedure from points 4 to 11 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).
13. Shut down the system safely.
14. We compared that *theoretical frequency* value (f_O) and *practical frequency* value are same approximately.

EXPECTED WAVEFORM :

The following waveform shows the output signal for different capacitor values of *RC phase shift Oscillator,*



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)	Pract-Ical 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 we 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.

RESULT :

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

VIVA VOICE 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. : 10 **Date :**

Name of the Experiment : COLPITTS OSCILLATOR

(By using Simulation Software)

AIM :

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

APPARATUS :

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

THEORY :

The basic configuration of the **Colpitts Oscillator** resembles that of the *Hartley Oscillator* but the difference this time is that the centre tapping of the tank sub-circuit is now made at the junction of a "capacitive voltage divider" network instead of a tapped autotransformer type inductor as in the Hartley oscillator.

the resonant frequency of the LC tank circuit and is given as:

$$f_r = \frac{1}{2\pi\sqrt{L C_T}}$$

where C_T is the capacitance of C1 and C2 connected in series and is given as:

$$\frac{1}{C_T} = \frac{1}{C_1} + \frac{1}{C_2} \quad \text{or} \quad C_T = \frac{C_1 \times C_2}{C_1 + C_2}$$

The configuration of the transistor amplifier is of a *Common Emitter Amplifier* with the output signal 180^o out of phase with regards to the input signal. The additional 180^o phase shift require for oscillation is achieved by the fact that the two capacitors are connected together in series but in parallel with the inductive coil resulting in overall phase shift of the circuit being zero or 360^o.

CIRCUIT DIAGRAM :

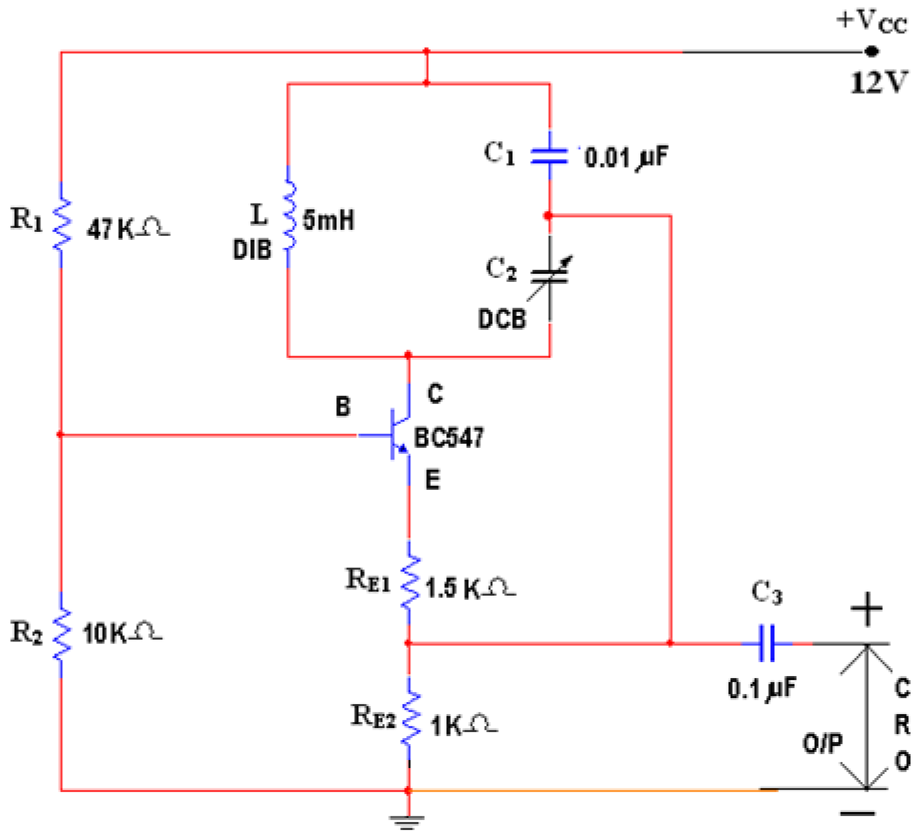


Figure: Circuit diagram of Colpitt's oscillator.

PROCEDURE :

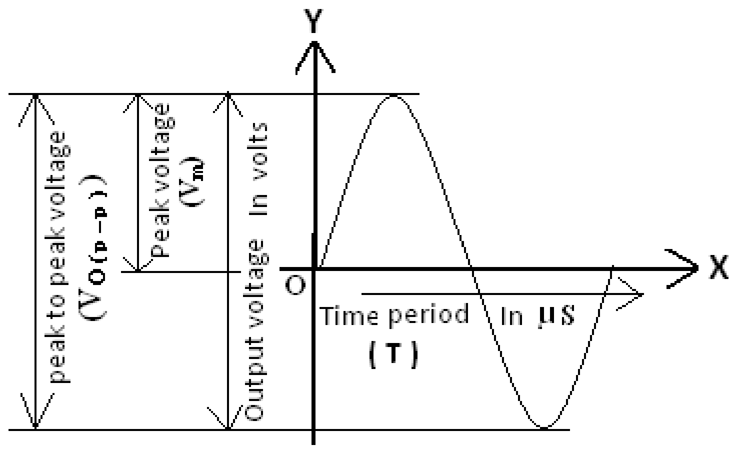
1. First calculated the theoretical frequency for all capacitor C_2 values by using the formula which is available in the tabular form.
2. Picked up the components from *components bar* in multisim software as per the circuit diagram.
3. Made the connections as per the circuit diagram.
4. Set the V_{CC} value as 12V.
5. Set the *inductance(L)* value as 5mH in DIB and *Capacitor* C_2 value as 1nF (0.001µF or 1Kpf) in DCB.
6. To start the simulation clicked on *Run button* .
7. We observed *Sine wave form* as a output signal in the *CRO*.
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 time period and output voltage ($V_{O\,(p-p)}$) values from the graph then noted in the Columns of *practical time period* and *output voltage* for corresponding *capacitor* C_2 values in the tabular form respectively.
10. Stopped the simulation by click on *Run option* through *Execute button.*
11. Repeat the same procedure from points 5 to 11 for corresponding C_2 values which are given below,
    a). 2.2 nF/2.2Kpf. b). 3.3 nF/3.3Kpf. c). 10.0 nF/10Kpf.
12. Shut down the system safely.
13. Now calculated the practical frequency by using formula 1/T and noted it in corresponding columns of C_2
14. We compared that *theoretical frequency* value (f_O) and *practical frequency* value are same approximately.

TABULAR FORM / CALCULATIONS :

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	Pract-ical – 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 *Colpitts Oscillator*



RESULT :

I have drawn the output signal and calculated the frequency values of a given *Colpitts Oscillator.*

VIVA VOICE Questions:

1. What are LC oscillators?
2. What is the frequency of Colpitts oscillator?
3. What is the condition for sustained oscillation in Colpitts oscillator?
4. Applications of LC oscillators?
5. In Colpitts oscillator, feedback circuit consists of how many Inductors and capacitors?
6. Which type of feedback is used for Colpitts oscillator?
7. What is Q in Colpitts oscillator?
8. What is the advantage of Colpitts oscillator?
9. How does Colpitts oscillator calculate frequency?
10. What are the advantages and disadvantages of LC oscillator?

PART – B

Using Hardware

(Off the Syllabus)

Experiment No. : 1 **Date :**

Name of the Experiment : VOLTAGE SERIES FEED BACK AMPLIFIER

(By using Hardware)

AIM :

i). To obtain the frequency response of *Voltage Series feedback Amplifier* by using hardware.

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	---------1 No.
2). Resistors	a). 100 Ω	-------- 1 No.
	b). 10 K Ω	-------- 1 No.
	c). 100 K Ω	-------- 1 No.
3). Capacitors	a). 10 µF	-------- 1 No.
	a). 22 µF	-------- 1 No.

THEORY :

Note : Theory had given in *Software section-A* of this same experiment. That same theory can apply here also.

CIRCUIT DIAGRAM :

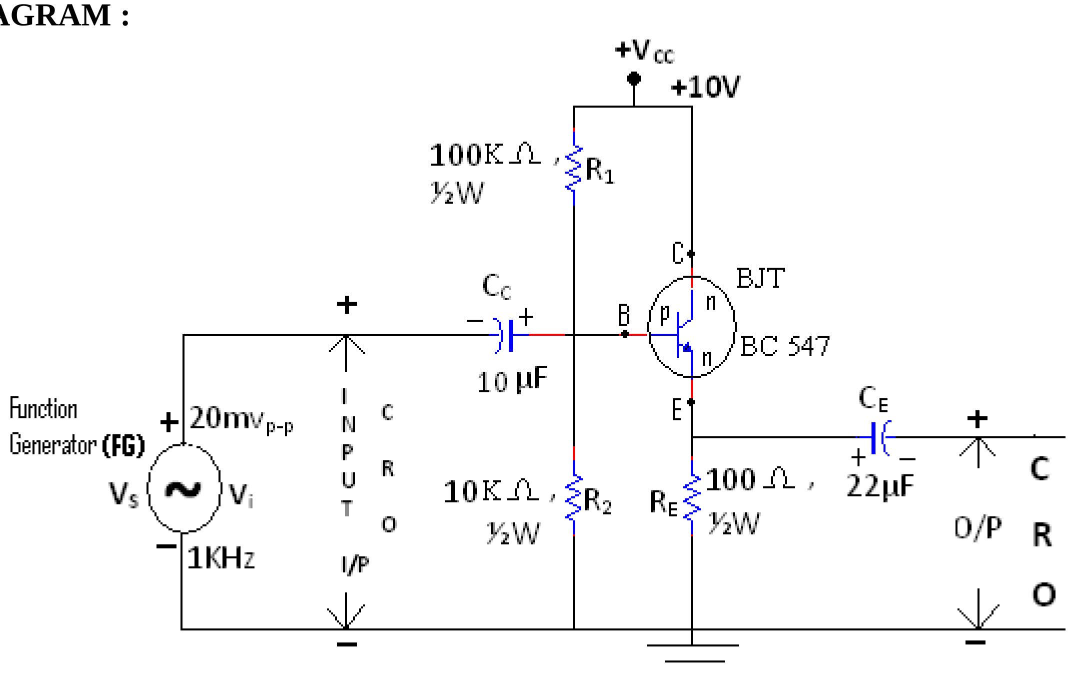


Figure: Circuit diagram of voltage series feed back amplifier.

PROCEDURE :

1). Connected the circuit as per the circuit diagram.

2). Removed the probe of *CRO* from output (O/P) side and connected it at input (I/P) side to set the input signal i.e. sine wave having the value of $20mV_{p-p}$&1KHz.

3). Then switched ON the *function generator* and *CRO*; but don't switched ON the *RPS*.

4). Now Kept the *AC/GND/DC* switch is at *AC* position.

5). Now applied the input signal i.e. sine wave by pressing the sine wave function key in the *function generator*.

6). Initially kept the 1KHz. frequency by varying the frequency control in the *function generator*.

7). Now applied the peak to peak amplitude of a sine wave is of $20mV_{p-p}$ by varying the amplitude control in the *function generator* through observing in the *CRO*.

8). Kept this value of input signal as constant up to the completion of the experiment Otherwise the wrong output would occurred.

9) Then removed the probe of *CRO* from the input side and connected it across the output side.

10). Now switched ON the *RPS* and set the 10V in it i.e. V_{CC} = 10V.

11). Varied the different frequency steps of 10Hz, 500Hz, 1KHz, 100KHz, 200KHz, 400KHz, 600KHz, 800KHz,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 (voltage) of sine wave by observing in the *CRO*.

12). Now switched OFF the *RPS*, *function generator* and *CRO*.

13). Then calculated the *voltage gain* $A_V = V_O/V_i$ & *gain in dB* $= 20\log10(A_V)$ and noted down the values in the specified columns of the tabular column.

14). 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.

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

Notes:

1. Amplifier means which amplifies the sinusoidal and non-sinusoidal wave forms with out change in frequency. In voltage series feedback amplifier, network is in parallel with the the output of the amplifier.
2. A fraction of the output voltage through the feedback network is applied in series with in the input voltage of the amplifier.
3. The series connections at the input, increase the input resistance. In this case the amplifier is a true voltage amplifier.
4. The common collector or emitter follower is an example of voltage series feedback amplifier. Since the voltage developed in the output is in series with the input voltage as for as the base – emitter junction is connected.

TABULAR FORM :

Sl.No.	InputVoltage(V_i) In milli Volts (peak to peak)	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	20mV	10 Hz.			
2	20mV	500 Hz.			
3	20mV	1 KHz.			
4	20mV	100 KHz.			
5	20mV	200 KHz.			
6	20mV	400 KHz.			
7	20mV	600 KHz.			
8	20mV	800 KHz.			
9	20mV	1 MHz.			

EXPECTED GRAPH :

A). Frequency response curve for *frequency* verses *gain in dB*.

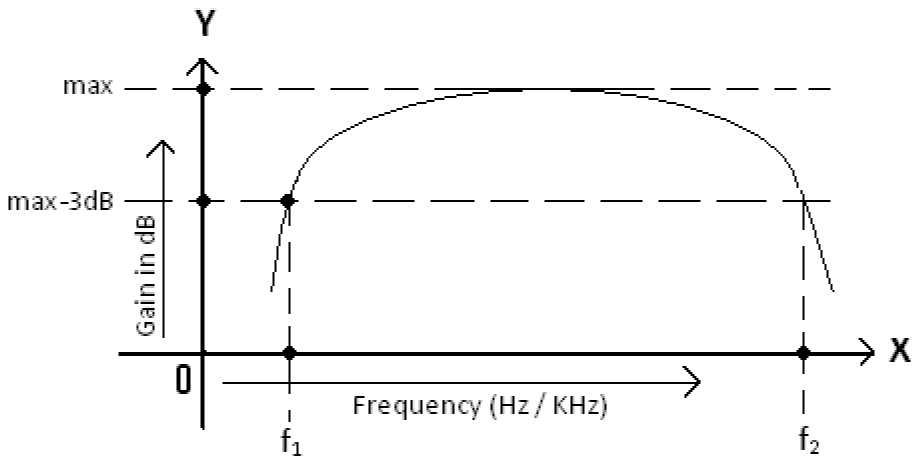


Figure: Frequency response curve of voltage series feed back amplifier For frequency verses gain in dB.

B). Frequency response curve for *frequency* verses *voltage gain*.

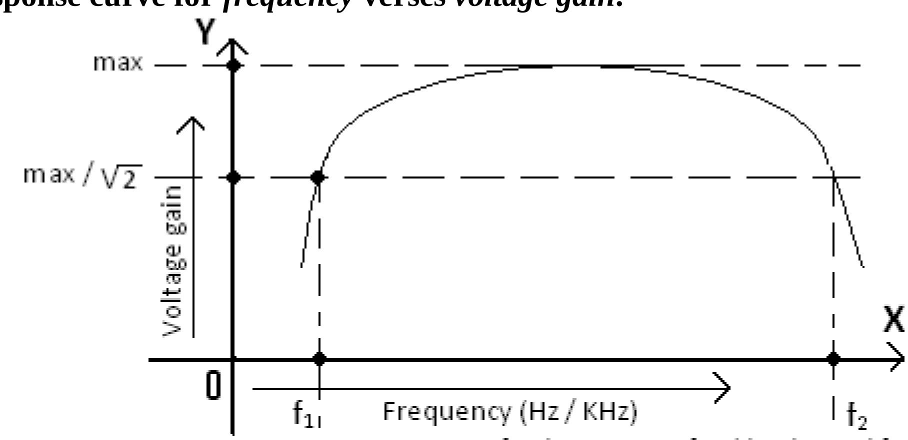


Figure: Frequency response curve of voltage series feed back amplifier For frequency verses voltage gain.

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 :

We have obtained the frequency response curves of *Voltage series feed back Amplifier* for frequency verses gain in dB & frequency verses voltage gain and calculated the band width of both of them. The band width values are given below,

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 VOICE Questions:

Note : The questions which are avail in *Software section-A* under the same experiment are applicable here also.

Experiment No. : 2 **Date :**

Name of the Experiment : BOOTSTRAPPED EMITTER FOLLOWER

(By using Hardware)

AIM :

To obtain the frequency response of *Bootstrapped Emitter Follower***.**

APPARATUS :

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

COMPONENTS :

1). Transisitor	a). BC547 NPN	-------	1 No.
2). Resistors	a).100 Ω	-------	1 No.
	b). 4.7K Ω	-------	1 No.
	b).10 K Ω	-------	1 No.
	c).100 K Ω	-------	1 No.
3). Capacitors	a).10µF	-------	2 No.
	b).22 µF	-------	1 No.

THEORY :

Note : Theory had given in *Software section-A* of this same experiment. That same theory can apply here also.

CIRCUIT DIAGRAM :

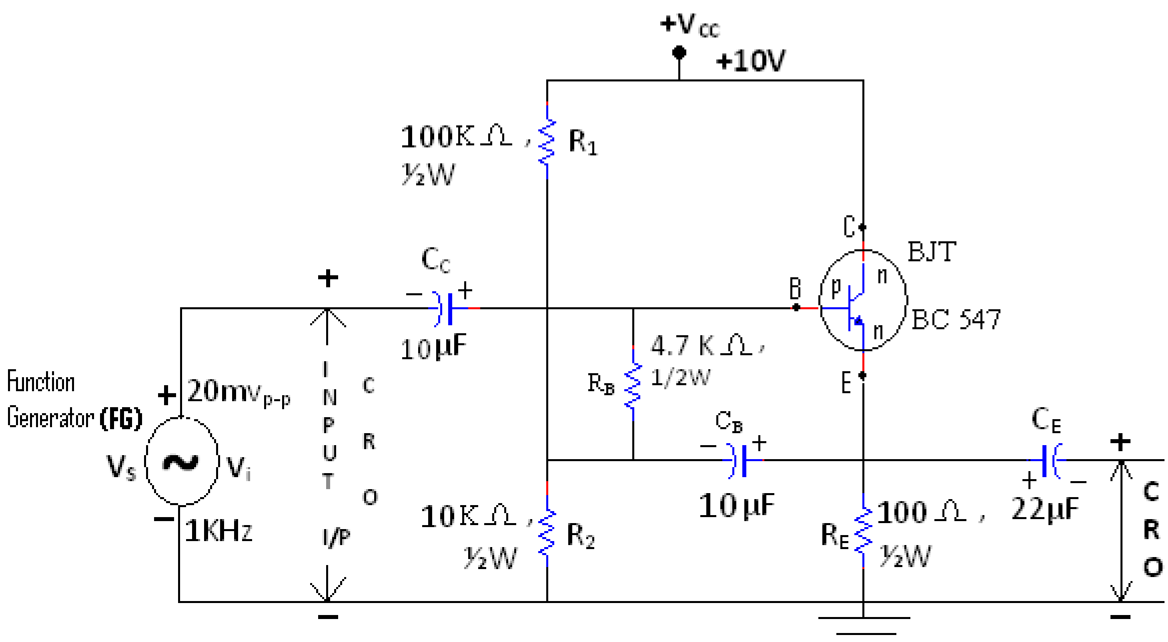


Figure: Circuit diagram of Bootstrapped Emitter Follower

PROCEDURE :

1). Connected the circuit as per the circuit diagram.

2). Removed the probe of *CRO* from output (O/P) side and connected it at input (I/P) side to set the input signal i.e. sine wave having the value of 20mV_{p-p}&1KHz.

3). Then switched ON the *function generator* and *CRO*; but don't switched ON the *RPS*.

4). Now Kept the *AC/GND/DC* switch is at *AC* position.

5). Now applied the input signal i.e. sine wave by pressing the sine wave function key in the *function generator*.

6). Initially kept the 1KHz. frequency by varying the frequency control in the *function generator*.

7). Now applied the peak to peak amplitude of a sine wave is of 20mV_{p-p} by varying the amplitude control in the *function generator* through observing in the *CRO*.

8). Kept this value of input signal as constant up to the completion of the experiment Otherwise the wrong output would occurred.

9) Then removed the probe of *CRO* from the input side and connected it across the output side.

10) Now switched ON the *RPS* and set the 10V in it i.e. V_{CC} = 10V.

11). Varied the different frequency steps of 10Hz, 500Hz, 1KHz, 100KHz, 200KHz, 400KHz, 600KHz, 800KHz, 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*.

12). Now switched OFF the *RPS*, *function generator* and *CRO*.

13). Then calculated the *voltage gain* $A_V = V_O/V_i$ & *gain in dB* = $20log10(A_V)$ and noted down the values in the specified columns of the tabular column.

14). 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.

Note: Bootstrap Emitter Follower uses to increase the input impedance and to work as correct *Buffer*. For example, The voltage gain of this a*mplifier* is 1 it means the output voltage is equal to input voltage, then we can say that it is the correct *Buffer*. Now If you observed the output of *voltage series feedback amplifier* the max. output voltage is less as compared to input voltage, it means buffer is incorrect. To increase the output voltage which is equal to the input voltage here we have used the *Bootstrapped Emitter Follower*. It is not possible to find out the band width because there is no amplification.

TABULAR FORM :

Sl.No.	InputVoltage($V_{i\ P-P}$) In mVolts.	Frequency In Hz/KHz.	OutputVoltage($V_{O\ P-P}$) In mVolts.	Voltage gain $A_V = V_o/V_i$	Gain in dB = $20log_{10}(A_V)$
01	20mV	10 Hz.			
02	20mV	500 Hz.			
03	20mV	1 KHz.			
04	20mV	100 KHz.			
05	20mV	200 KHz.			
06	20mV	400 KHz.			
07	20mV	600 KHz.			
08	20mV	800 KHz.			
09	20mV	1 MHz.			

EXPECTED GRAPH :

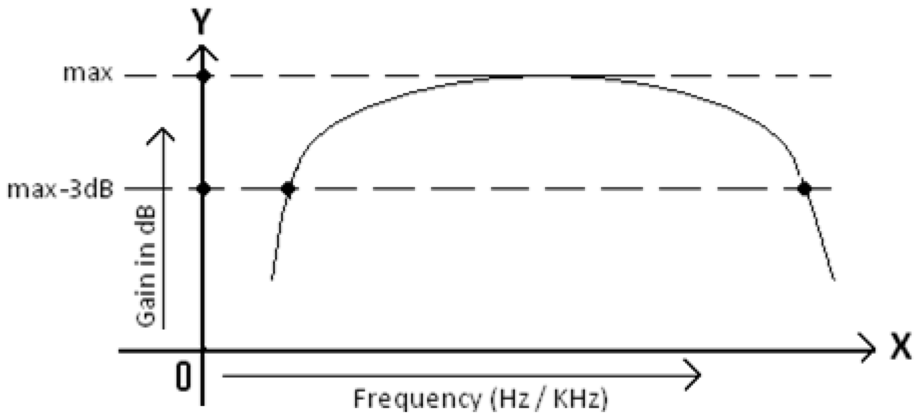


Figure: Frequency response curve of Bootstrapped Emitter Follower.
For Frequency verses gain in dB.

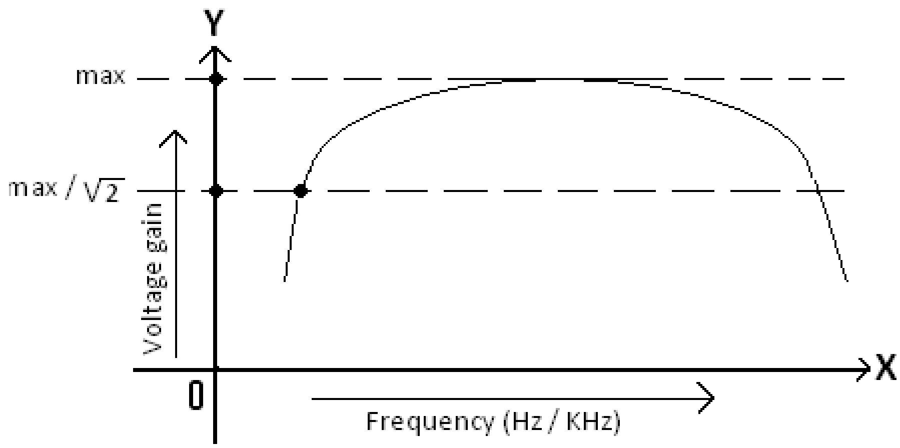


Figure: Frequency response curve of Bootstrapped Emitter Follower.
For frequency verses voltage gain

RESULT :

We have obtained the gain for different frequencies of *Bootstrapped Emitter Follower.*

VIVA VOICE Questions:

Note : The questions which are avail in *Software section-A* under the same experiment are applicable here also.

Experiment No. : 3 **Date :**

Name of the Experiment : DARLINGTON PAIR AMPLIFIER
(Using hardware)

AIM :

To obtain the frequency response curve for *Darlington pair amplifier* using hardware.

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. Bread board --- 1 No.
6. Connecting wires --- A few Nos.

COMPONENTS :

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

THEORY :

Note : Theory had given in *Software section-A* of this same experiment. That same theory can apply here also.

CIRCUIT DIAGRAM :

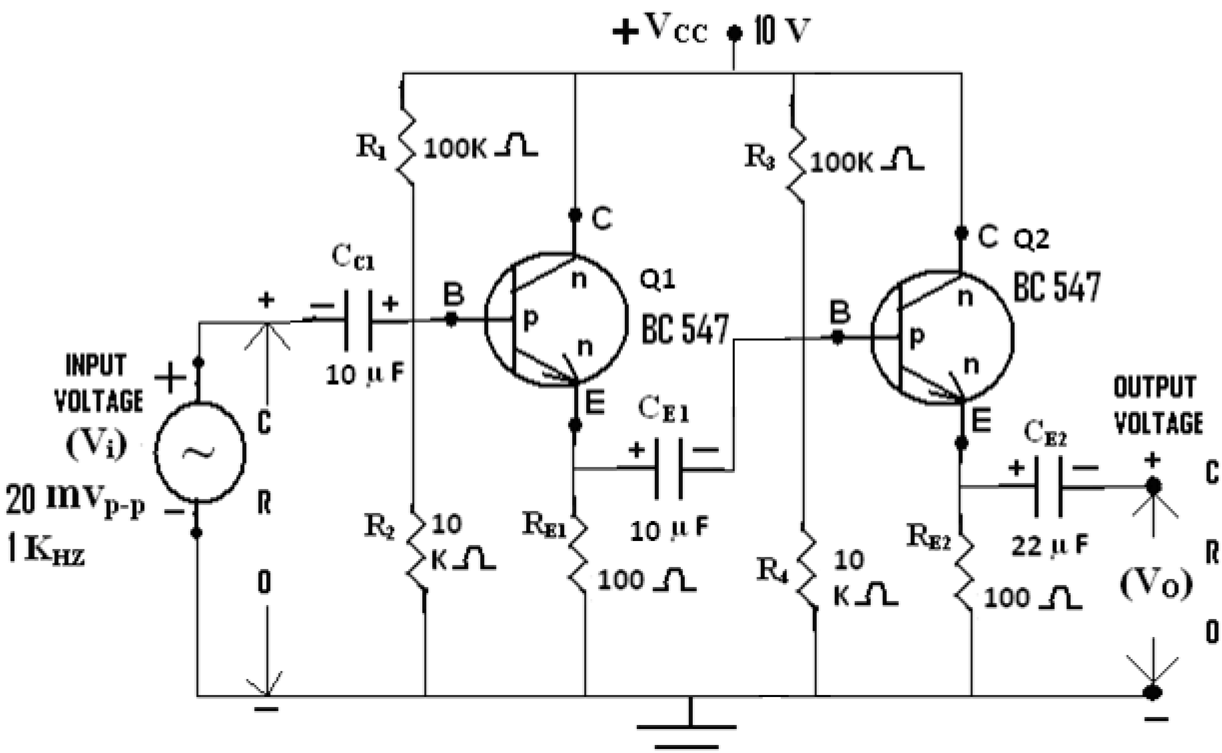


Figure: Circuit diagram of Darlington pair amplifier.

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.
3. Switched *ON* the *CRO* and *function generator*.
4. 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.
5. Later removed the probe from that place and connected it across the capacitor C_{C2} to observe the output.
6. Switched *ON* the *RPS* and kept the 10V 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, 10Hz, 500Hz, 1KHz, 100KHz, 200KHz, 400KHz, 600KHz, 800KHz, 1MHz.
9. Repeat 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,

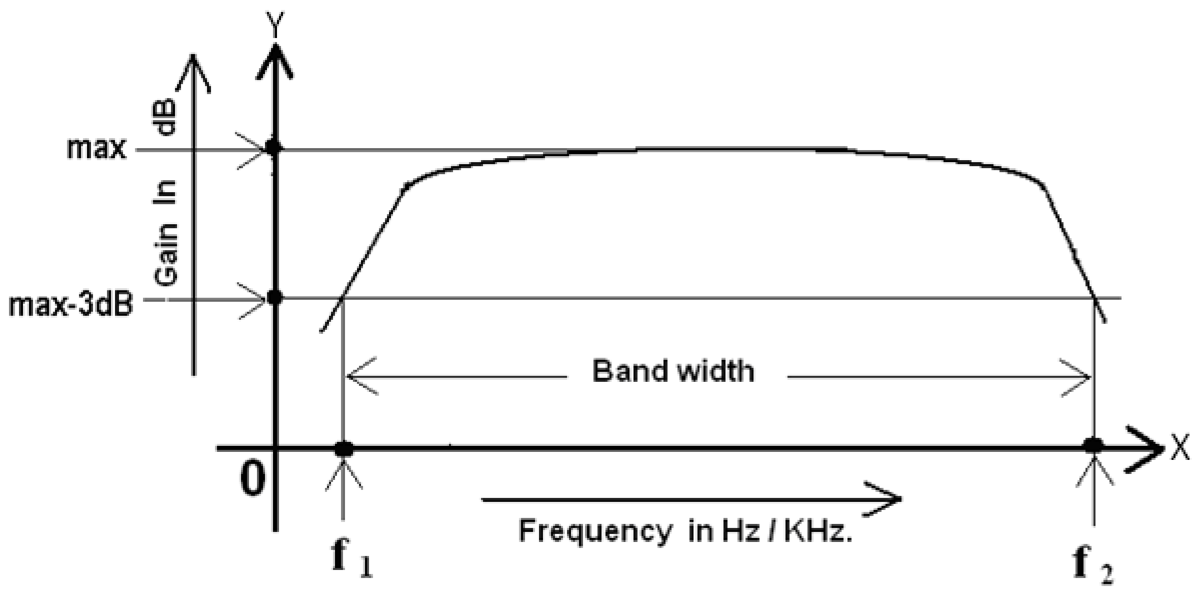
    $$\text{Voltage gain } (A_v) = V_o / V_i \quad \text{and} \quad \text{Gain in dB} = 20\log_{10}(A_v).$$
11. Plotted the graph between *frequency on X*- axis and *gain in dB on Y*- axis.

TABULAR FORM:

Sl. No.	Input voltage (V_i) in mV	Frequency in Hz / KHz.	Output Voltage (V_o) in mV	Voltage gain (A_v) = V_o / V_i	Gain in dB =$20\log_{10}(A_v)$
1	20	10 Hz.			
2	20	500 Hz.			
3	20	1KHz.			
4	20	100 KHz.			
5	20	200 KHz.			
6	20	400 KHz.			
7	20	600 KHz.			
8	20	800 KHz.			
9	20	1 MHz.			

EXPECTED GRAPH :

The following graph shows for frequency response curve of a *Darlington pair Amplifier*.



Note : . It is not possible to find out the band width because there is no 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 :

I have drawn the frequency response curve and calculated the band width of a *Darlington pair amplifier*.

VIVA VOICE Questions :

Note : The questions which are avail in *Software section-A* under the same experiment are applicable here also.

Experiment No. : **04** **Date :**

Name of the Experiment : **TWO STAGE RC COUPLED AMPLIFIER**
(Using Hardware)

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 No.
	b). 2.2 K Ω	-----	2 No.
	c). 10 K Ω	-----	2 No.
	d). 47 K Ω	-----	2 No.
	e). 100 K Ω	-----	1 No.
3). Capacitors	a). 10 µF	-----	3 No.
	b). 22 µF	-----	3 No.

THEORY :

Note : Theory had given in *Software section-A* of this same experiment. That same theory can apply here also.

CIRCUIT DIAGRAM :

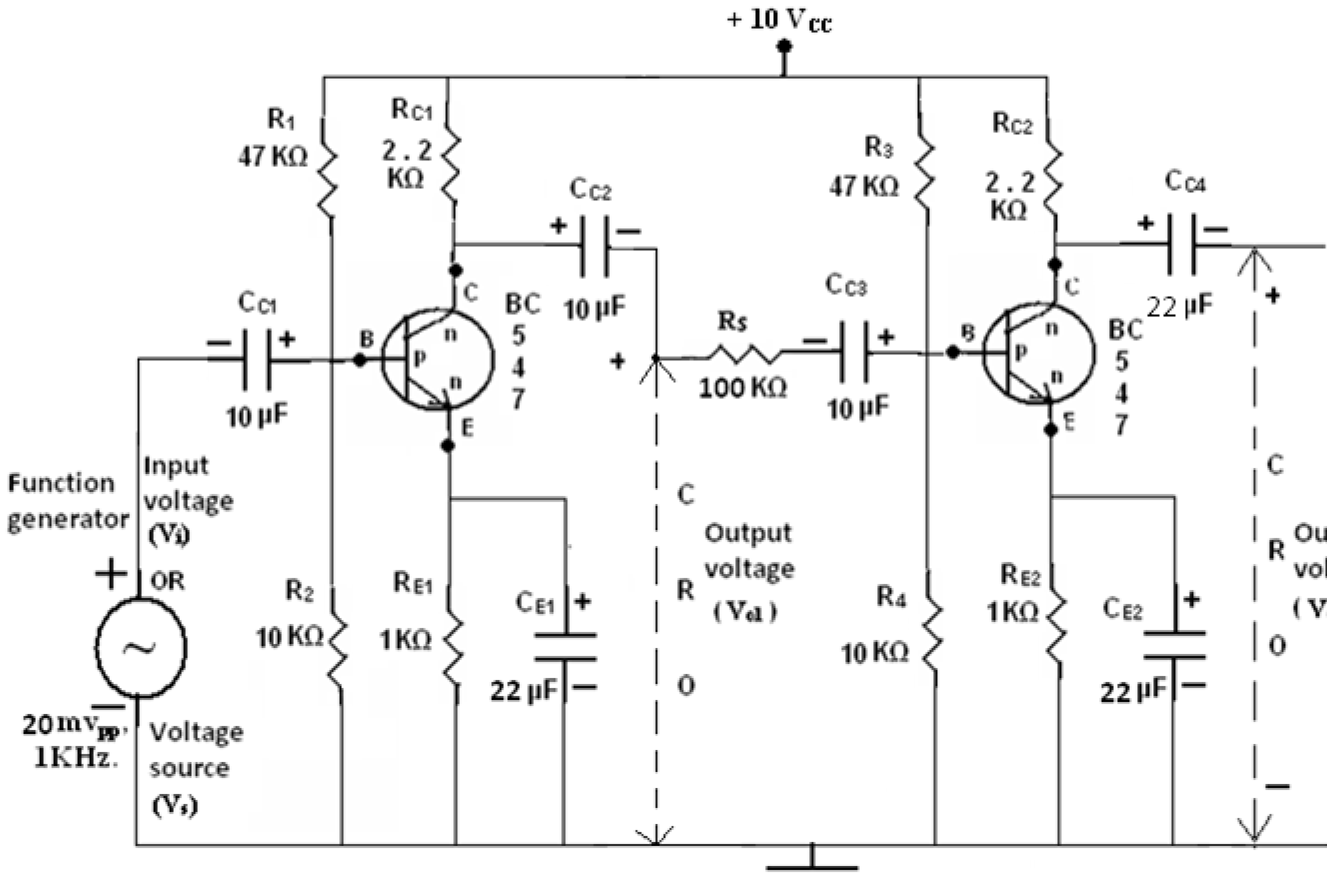


Figure: Circuit diagram of Two stage RC coupled amplifier.

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.
3. Switched *ON* the *CRO* and *function generator*.
4. 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.
5. Removed the probe from that place and connected it across the C_{C2} to observe the output of single stage.
6. Switched *ON* the *RPS* and kept the 10V 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 of output voltage in terms of peak to peak voltages by varying the different frequency steps in the function generator which are given below,
   10Hz, 500Hz, 1KHz, 100KHz, 200KHz, 400KHz, 600KHz, 800KHz, 1MHz.
9. The above readings noted in the tabular form of *single stage RC coupled amplifier.*
10. Disconnect the probe from C_{C2} and reconnected it across C_{C4} to observe the output of second stage.
11. Repeat the same procedure as per point 8 for tabular form of *Two stage RC Coupled Amplifier.*

12. 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* $= f_2 - f_1$
13. 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 :

A). Single stage RC coupled Amplifier :

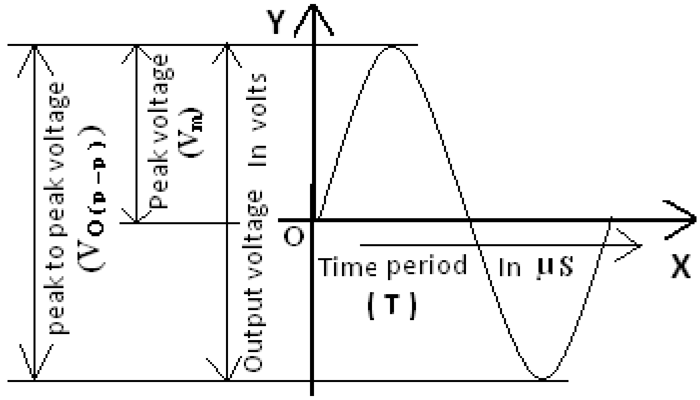
Sl. No.	**Input voltage (V_i) in mV**	**Frequency in Hz / KHz.**	**Output Voltage (V_o) in V**	**Voltage gain (A_v) = V_o / V_i**	**Gain in dB =20log$_{10}$(A_v)**
1	20	10 Hz.			
2	20	500 Hz.			
3	20	1 KHz.			
4	20	100 KHz.			
5	20	200 KHz.			
6	20	400 KHz.			
7	20	600 KHz.			
8	20	800 KHz.			
9	20	1 MHz.			

B). Two stage RC coupled Amplifier :

Sl. No.	**Input voltage (V_i) in mV**	**Frequency in Hz / KHz.**	**Output Voltage (V_o) in V**	**Voltage gain (A_v) = V_o / V_i**	**Gain in dB =20log$_{10}$(A_v)**
1	20	10 Hz.			
2	20	500 Hz.			
3	20	1 KHz.			
4	20	100 KHz.			
5	20	200 KHz.			
6	20	400 KHz.			
7	20	600 KHz.			
8	20	800 KHz.			
9	20	1 MHz.			

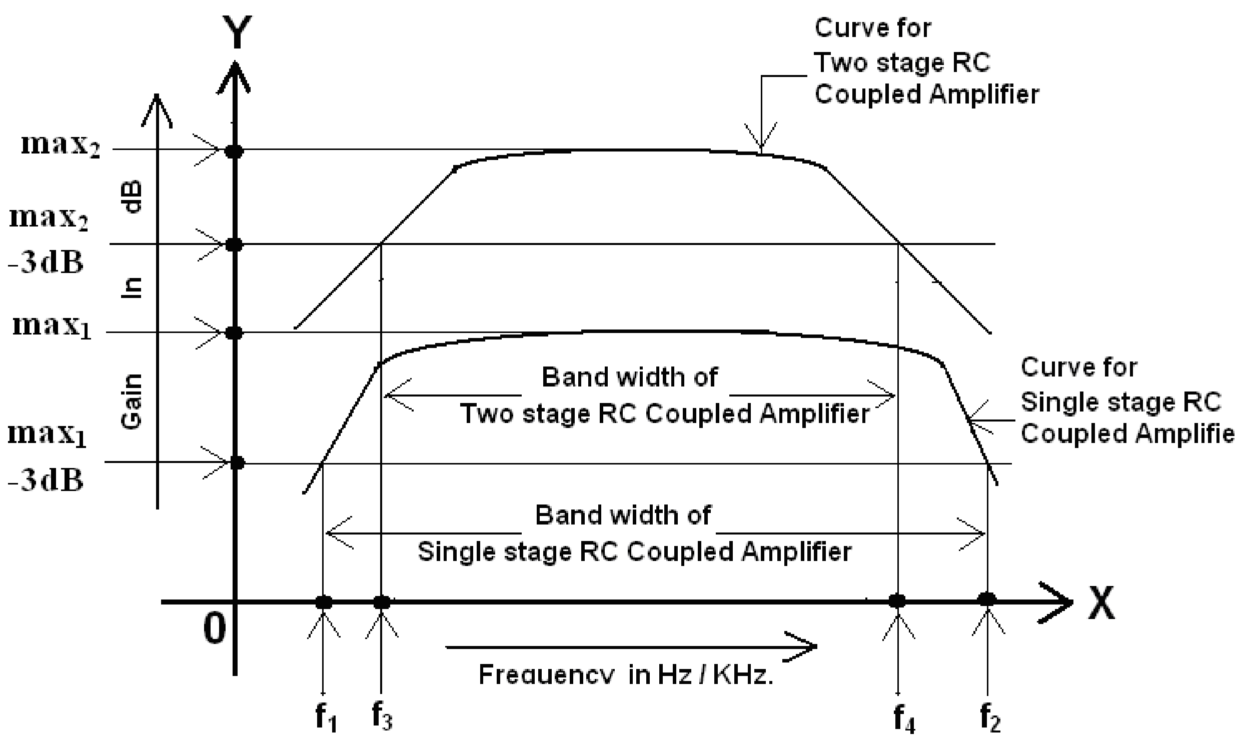
EXPECTED WAVEFORM :

I have 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 VOICE Questions :

Note : The questions which are avail in *Software section-A* under the same experiment are applicable here also.

Experiment No. : **05** **Date :**

Name of the Experiment : CURRENT SHUNT FEED BACK AMPLIFIER

(By using Hardware)

AIM :

1. To plot the frequency response curve of a *Current shunt feed back Amplifier* for both *with feed back & without feed back*.

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.

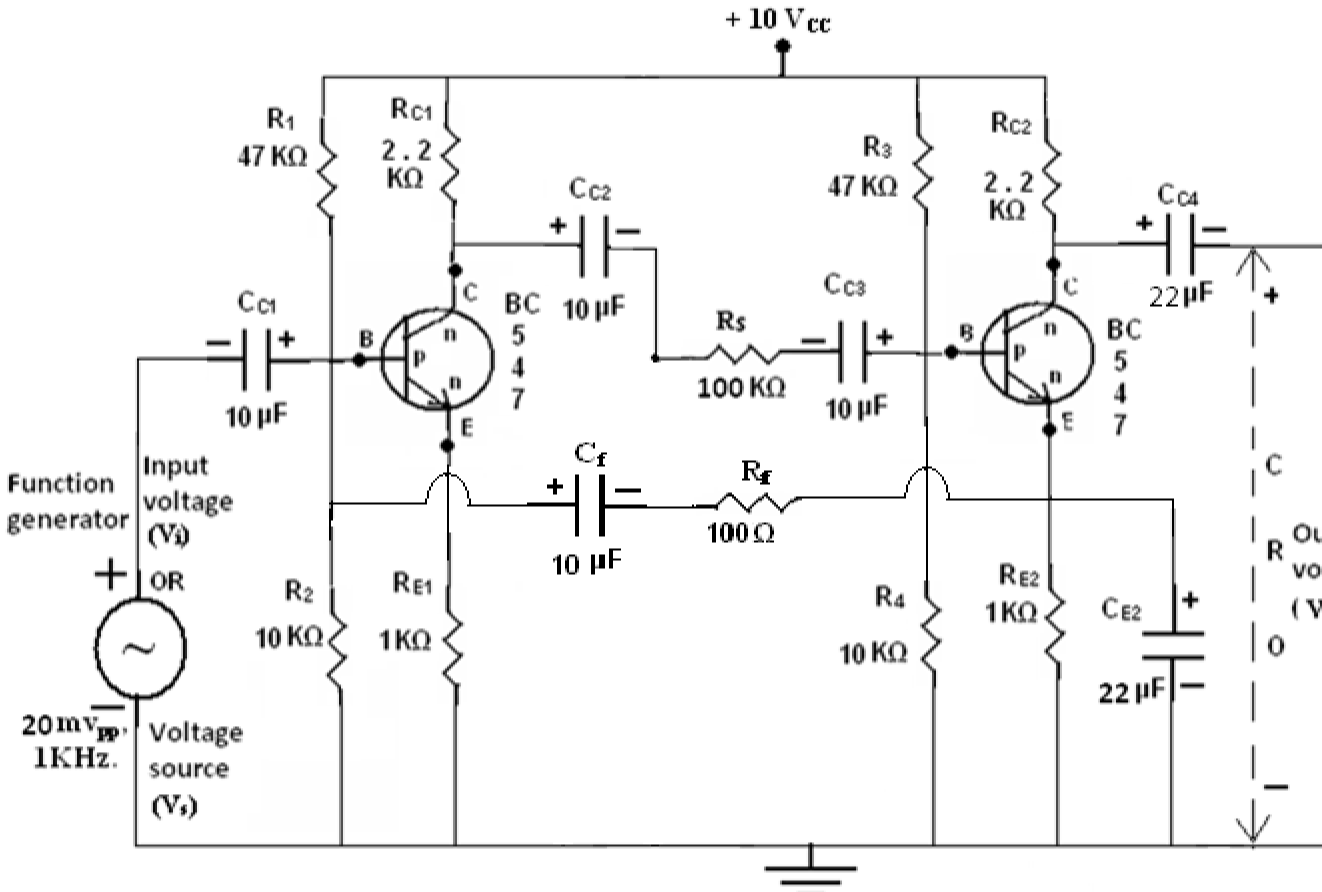
COMPONENTS :

1. Transistor BC 547	-----	2 No.
2. Resistors :		
i). 100Ω	-----	1 No.
ii). 1 KΩ	-----	2 No.
iii). 2 . 2 KΩ	-----	2 No.
iv). 10 KΩ	-----	2 No.
v). 47 KΩ	-----	2 No.
vi). 100 KΩ	-----	1 No.
3. Capacitors :		
i). 10 µF	-----	4 No.
ii). 22 µF	-----	2 No.

THEORY :

Note : Theory had given in *Software section-A* of this same experiment. That same theory can apply here also.

CIRCUIT DIAGRAM :



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 feed back amplifier* for both with feed back & without feed back .
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 feed back*. from the CRO, by varying the different steps of frequency (i.e. 10Hz, 500Hz., 1KHz, 100KHz, 200KHz, 400KHz, 600KHz, 800KHz, 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. 10Hz, 500Hz, 1KHz, 100KHz, 200KHz, 400KHz, 600KHz, 800KHz, 1MHz.) in function generator.

13. We have observed that the readings of *without feed back amplifier's* output voltage is greater than the *with feed back 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 $20log_{10}(A_V)$ in both tabular forms of *with feed back* and *without feed back 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 feed back amplifier (With feed back)* (BW) = $f_2 - f_1$
    ii). For *Current shunt feed back amplifier (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 :

A). With feed back :

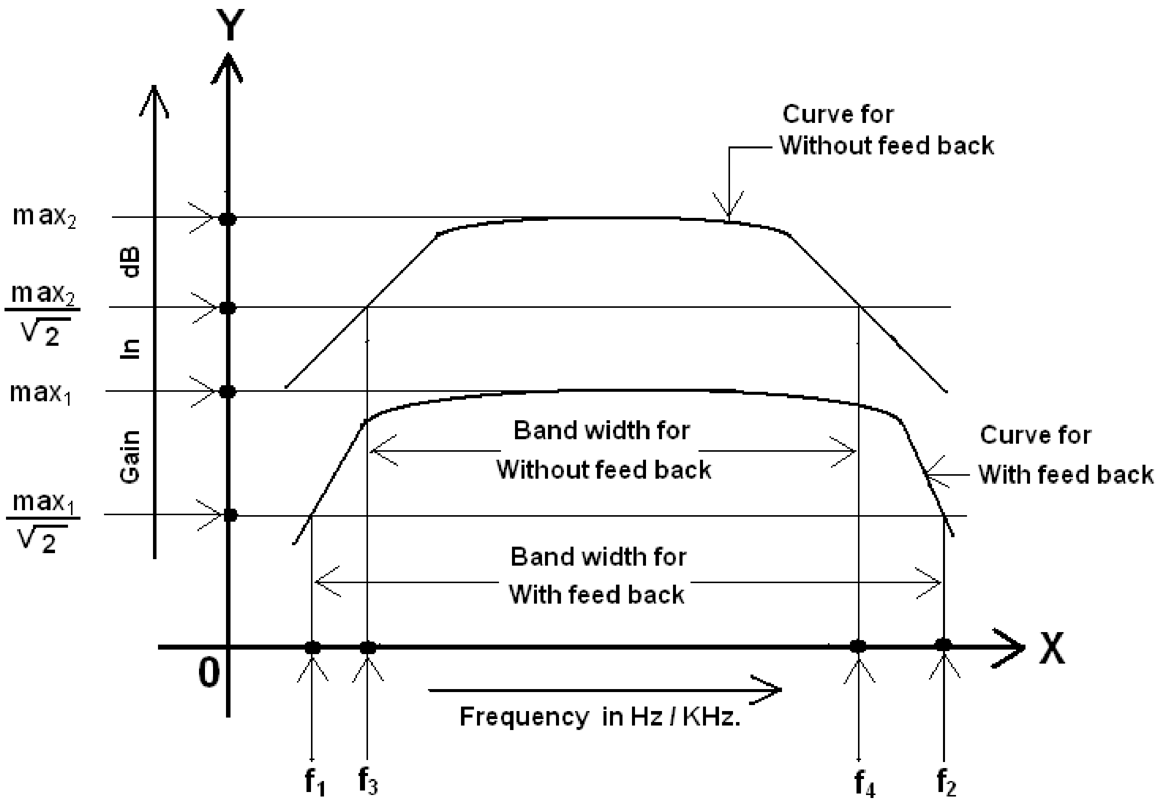
Sl. No.	Input voltage (V_i) in mV	Frequency in Hz / KHz.	Output Voltage (V_o) in mV	Voltage gain (A_v) = V_o / V_i	Gain in dB =$20log_{10}(A_v)$
1	20	10 Hz.			
2	20	500 Hz.			
3	20	1 KHz.			
4	20	100 KHz.			
5	20	200 KHz.			
6	20	400 KHz.			
7	20	600 KHz.			
8	20	800 KHz.			
9	20	1 MHz.			

B). Without feed back :

Sl. No.	Input voltage (V_i) in mV	Frequency in Hz / KHz.	Output Voltage (V_o) in mV	Voltage gain (A_v) = V_o / V_i	Gain in dB =$20log_{10}(A_v)$
1	20	10 Hz.			
2	20	500 Hz.			
3	20	1 KHz.			
4	20	100 KHz.			
5	20	200 KHz.			
6	20	400 KHz.			
7	20	600 KHz.			
8	20	800 KHz.			
9	20	1 MHz.			

EXPECTED GRAPH :

The following graph shows for ***Current shunt feed back amplifier*** for both ***with feed back*** and ***without feed back.***



RESULT :

We drawn the graph for frequency response of a *Current shunt feed back amplifier* for both *with feed back* and *without feed back.*

VIVA VOICE Questions :

Note : The questions which are avail in *Software section-A* under the same experiment are applicable here also.

Experiment No. : 6 **Date :**

Name of the Experiment : SINGLE TUNED VOLTAGE AMPLIFIER

(Using hardware)

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. Dicade Inductance box (DIB) ------------------------------------- 1 No.
5. Dicade 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 -- 1 No.
ii). 10 KΩ -- 1 No.
iii). 100 Ω -- 1 No

THEORY :

Note : Theory had given in *Software section-A* of this same experiment. That same theory can apply here also.

CIRCUIT DIAGRAM :

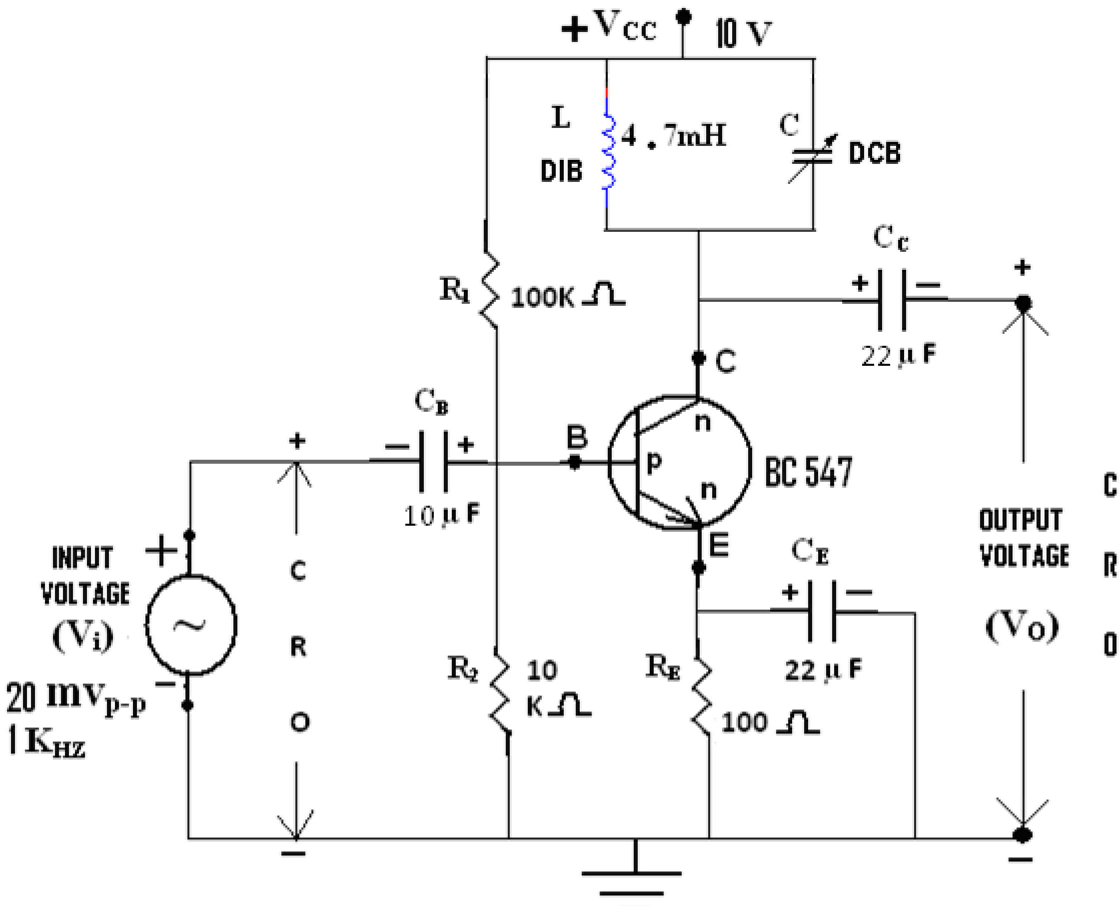


Figure: Circuit diagram of single tuned voltage amplifier.

THEORETICAL CALCULATIONS :

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

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

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

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

= 54Kpf OR 54 nF

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

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

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

= 2.16Kpf OR 2.16nF

PROCEDURE :

1. We 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,

$$Band\ width = f_2 - f_1.$$

TABULAR FORM – 1:

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

Sl. No.	**Input voltage (V_i) in mV**	**Frequency in Hz / KHz.**	**Output Voltage (V_o) in V**	**Voltage gain (A_v) = V_o / V_i**	**Gain in dB =20log$_{10}$(A_v)**
1	20	10 Hz.			
2	20	500 Hz.			
3	20	1 KHz.			
4	20	5 KHz.			
5	20				
6	20	20KHz.			
7	20	50KHz.			
8	20	100 KHz.			
9	20	200 KHz.			
10	20	500 KHz.			
11	20	1 MHz.			

TABULAR FORM – 2 :

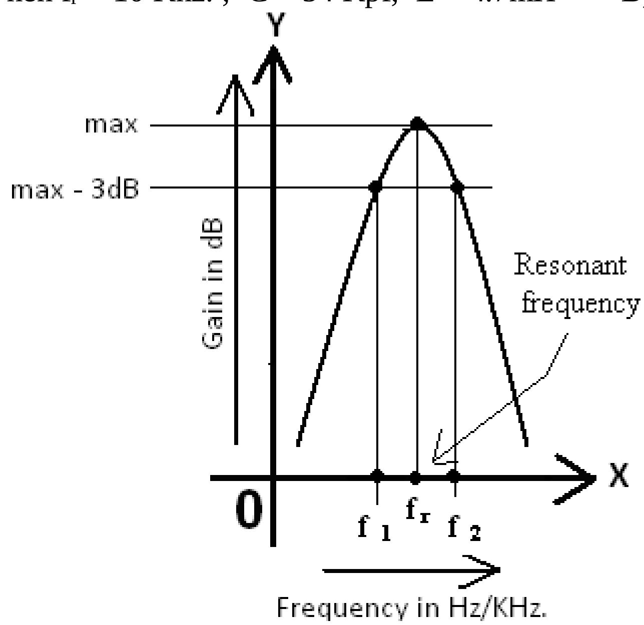
When f_r = 50 Khz. , C = 2.16 Kpf, L = 4.7mH

Sl. No.	Input voltage (V_i) in mV	Frequency in Hz / KHz.	Output Voltage (V_o) in V	Voltage gain (A_v) = V_o / V_i	Gain in dB =20$\log_{10}$(A_v)
1	20	10 Hz.			
2	20	500 Hz.			
3	20	1 KHz.			
4	20	20KHz.			
5	20				
6	20	60KHz.			
7	20	100 KHz.			
8	20	200 KHz.			
9	20	500 KHz.			
10	20	1 MHz.			

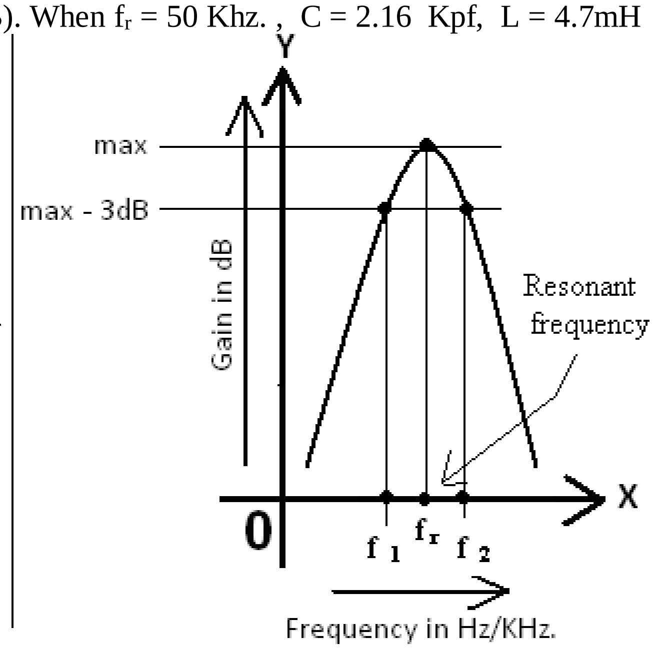
EXPECTED GRAPH :

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 VOICE Questions :

Note : The questions which are avail in *Software section-A* under the same experiment are applicable here also.

Experiment No. : 7 **Date :**

Name of the Experiment : CLASS A SERIES - FED POWER AMPLIFIER Using Hardware

AIM :

1. To draw the output signal (sine wave form) on the graph of a given *Class-A series-fed power amplifier.*
2. To calculate the conversion efficiency of given *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. Transistor BC 547 -------------------------------------- 1 No.

2. Resistors :

ii). 1KΩ -- 1 No.

iii). 10 KΩ -- 1 No.

iv). 47KΩ -- 1 No.

3. Capacitors :

i). 10 µF -- 1 No.

i i). 22 µF -- 2 No.

THEORY :

Note : Theory had given in *Software section-A* of this same experiment. That same theory can apply here also.

CIRCUIT DIAGRAM :

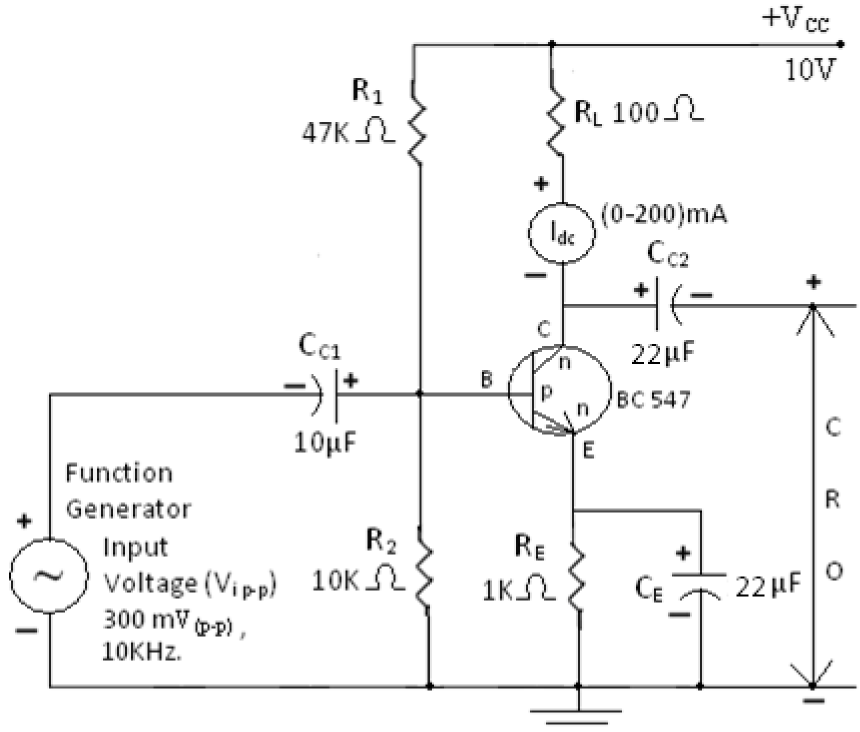


Figure: Circuit diagram of Class A Series-fed Power Amplifier

PRACTICAL CALCULATIONS :

The practical calculations are shown in the following tabular form,

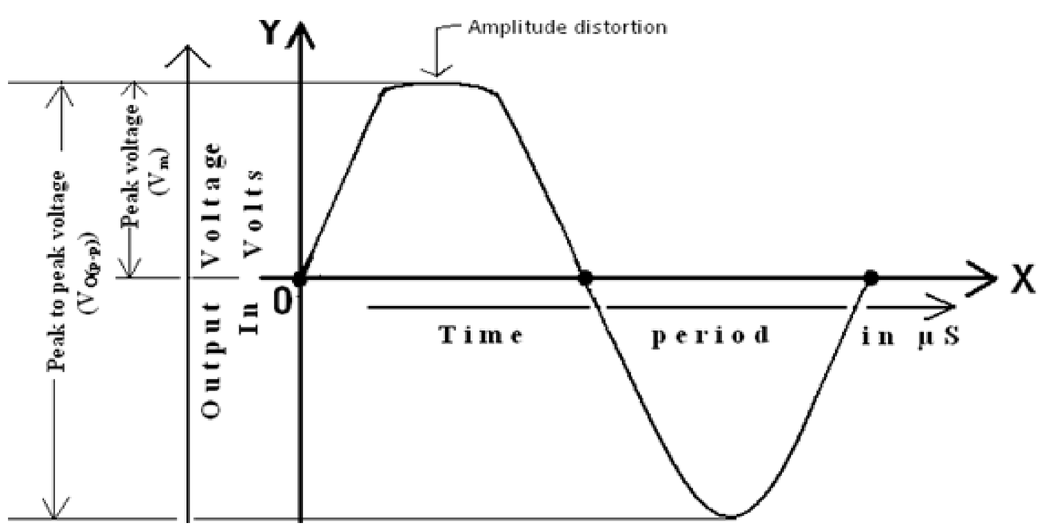
Sl.No.	Name of the parameter	Value
01.	Input Voltage $(V_i)_{p\text{-}p}$ (In mV).	300
02	Input frequency (In Khz.).	10
03	Supply DC Voltage (V_{CC}) (in Volts.)	10
04	Output voltage $V_{O(p\text{-}p)}$ (In volts.).	
05	Time period (T) for output signal (In ms)	
06	Fequency for output signal = 1/ T (In 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 $P_o\,(ac) = \dfrac{V_{O(P\text{-}P)}^2}{8R_L}$ (In Watts)	
11	% of efficiency (η) = [$P_O(ac) / P_i\,(dc)$] ×100 =	
12	Typical Max. efficiency (η) =	25.40%

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 300 mV_{p-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 R_L i.e at output side.
6. Now switched **ON** the Regulated Power Supply (RPS) and apply the *supply voltage* 10V as 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 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 down these 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%.***

EXPECTED GRAPH :

The following graph shows for ***Class A power amplifier.***



RESULT :

I have drawn the graph for output signal and calculated the conversion efficiency of a given ***Class-A series-fed power amplifier.***

VIVA VOICE Questions :

Note : The questions which are avail in *Software section-A* under the same experiment are applicable here also.

Experiment No. : 8 **Date :**

Name of the Experiment : COMPLEMENTARY SYMMETRY CLASS B PUSH – PUSH POWER AMPLIFIER **(Using hardware)**

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**
   i). BC 547 --- 1 No.
   ii). BC 557 --- 1 No.

2. Resistors :

i). 220 KΩ -- 2 No.
ii). 18 KΩ --- 2 No.
iii). 1 KΩ --- 1 No.
iv). 10Ω --- 3 No.

3. Capacitors :

i). 10 μF -- 2 No.

THEORY :

Note : Theory had given in *Software section-A* of this same experiment. That same theory can apply here also.

CIRCUIT DIAGRAM :

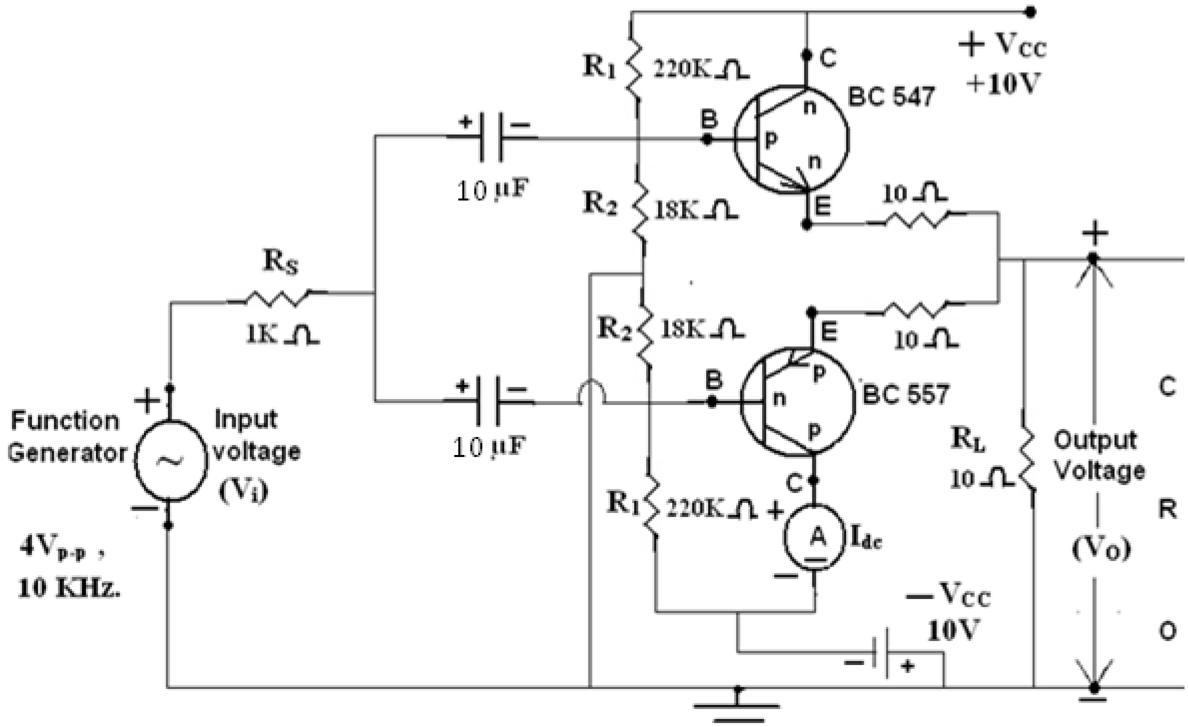


Figure: Circuit diagram of Class-B complimentary symmetry power amplifier.

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 4$V_{p\text{-}p}$ *, 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 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* ($Vo_{p\text{-}p}$) *, 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%****.*

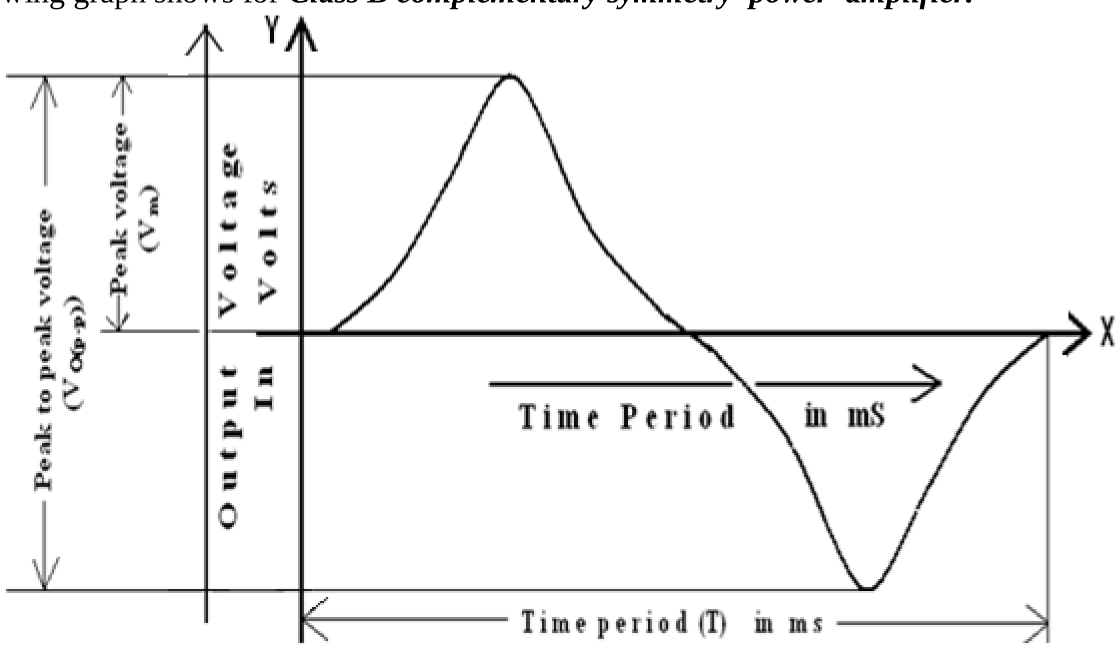
PRACTICAL CALCULATIONS :

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 (η) = $\dfrac{P_0(ac)}{P_i(dc)} \times 100$	
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 VOICE Questions :

Note : The questions which are avail in *Software section-A* under the same experiment are applicable here also.

Experiment No. : 9 **Date :**

Name of the Experiment : RC PHASE SHIFT OSCILLATOR

(By using hardware)

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

THEORY :

Note : Theory had given in *Software section-A* of this same experiment. That same theory can apply here also.

CIRCUIT DIAGRAM :

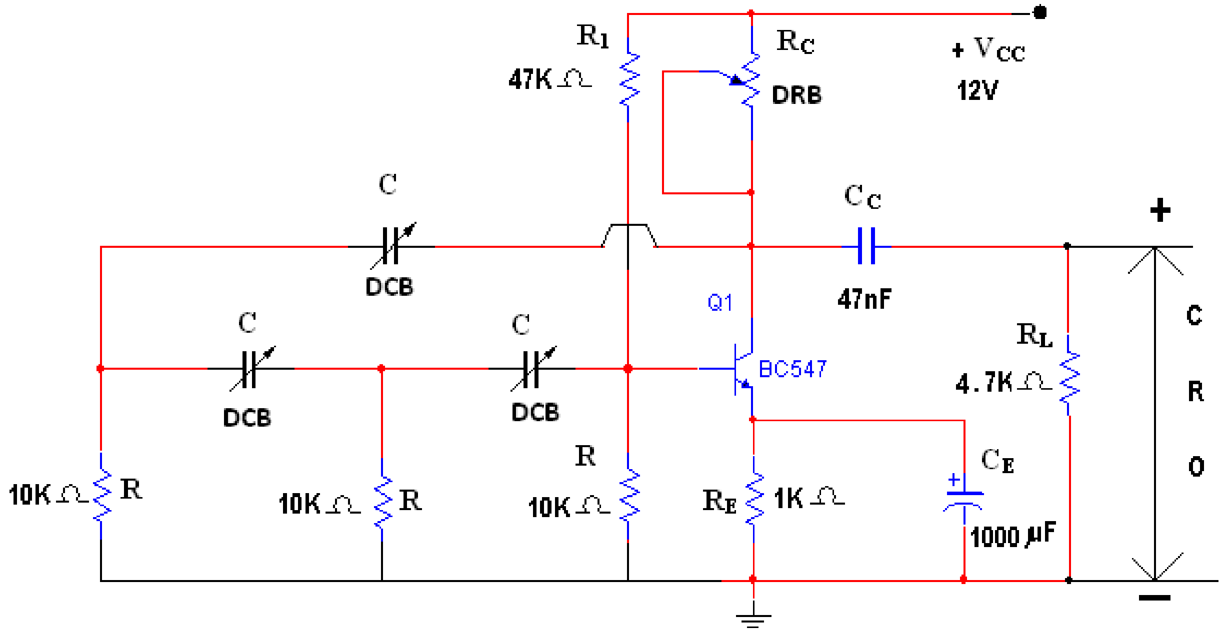


Figure: Circuit diagram of RC phase shift oscillator.

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.

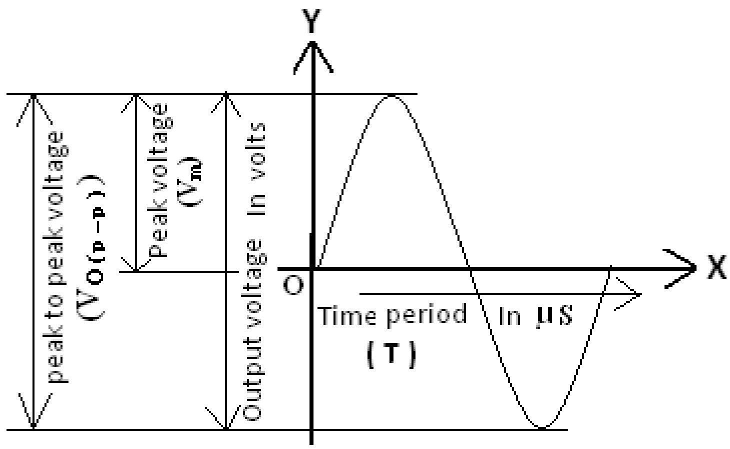
TABULAR FORM / CALCULATIONS :

Sl. No.	Rei stor (R) In KΩ	Capa-citor (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)	Pract-Ical 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 VOICE Questions :

Note : The questions which are avail in *Software section-A* under the same experiment are applicable here also.

Experiment No. : 10 **Date :**

Name of the Experiment : COLPITTS OSCILLATOR

(By using hardware)

AIM :

To draw the *sine wave form* and to calculate its frequency values of a given *Colpitts 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 No.
   1.5 KΩ --- 1 No.
   10 KΩ --- 1 No.
   47 KΩ -- 1 No.
2. Capacitors : 0.1μF -- 1 No.
   0.01μF -- 1 No.
3. Transistor : BC547 -- 1 No.

THEORY :

Note : Theory had given in *Software section-A* of this same experiment. That same theory can apply here also.

CIRCUIT DIAGRAM :

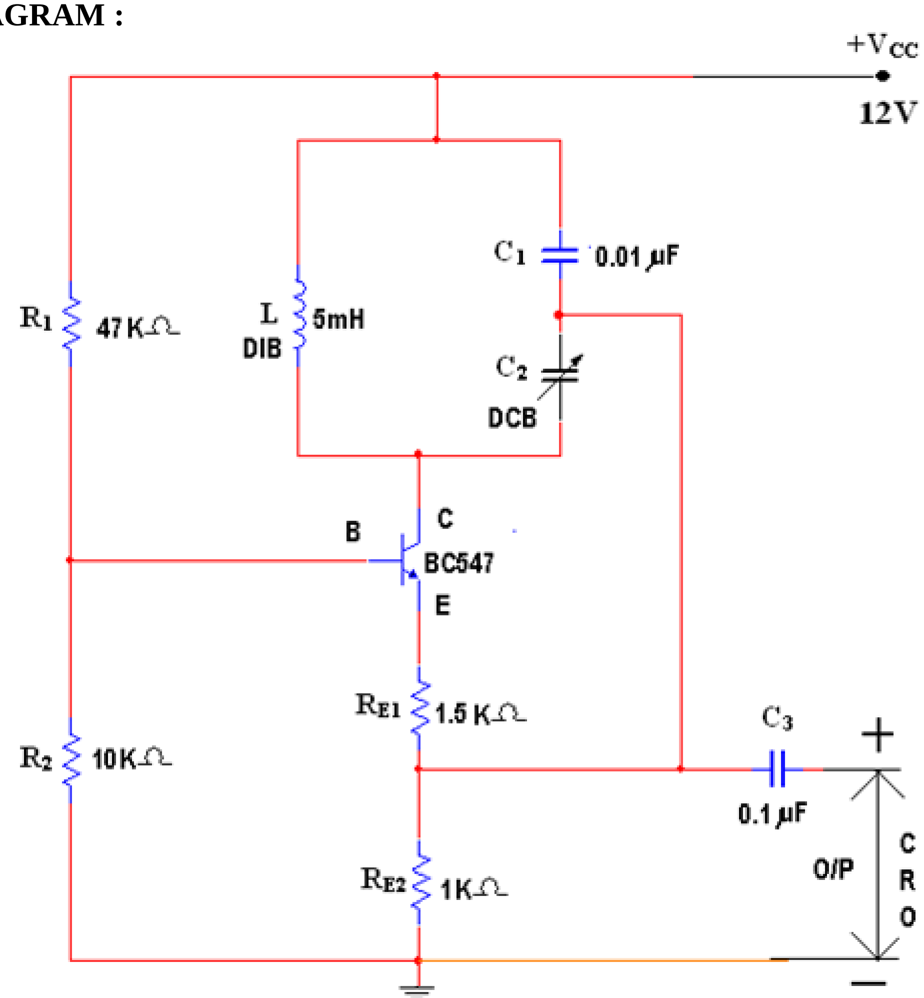


Figure: Circuit diagram of Colpitt's oscillator.

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.

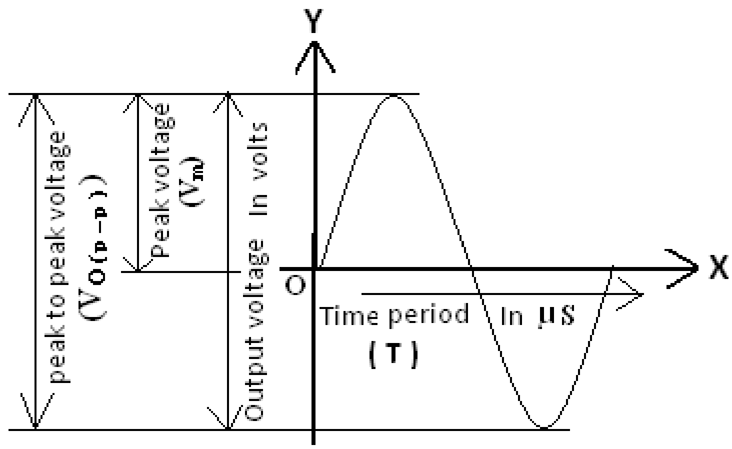
12. I compared that *theoretical frequency* value (F_O) and *practical frequency* values are approximately same.

TABULAR FORM / CALCULATIONS :

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) = $\frac{1}{2\pi\sqrt{L C_T}}$ In KHz	Pract-ical – 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 *Colpitts Oscillator*



RESULT :

I have drawn the output signal and calculated the frequency values of a given *Colpitts Oscillator.*

VIVA VOICE Questions :

Note : The questions which are avail in *Software section-A* under the same experiment are applicable here also.

PART – C

Beyond the Syllabus

(Using Simulation software & Hardware)

Experiment No. : 11 **Date :**

Name of the Experiment : CE - CB CASCODE AMPLIFIER
(Beyond the Syllabus-Using Software & Hardware)

AIM :

1). To obtain the frequency response of *CE – CB cascade amplifier* using Software and Hardware
2). To calculate the band width of this amplifier.

APPARATUS :

Software :

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

Hardware :

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 :

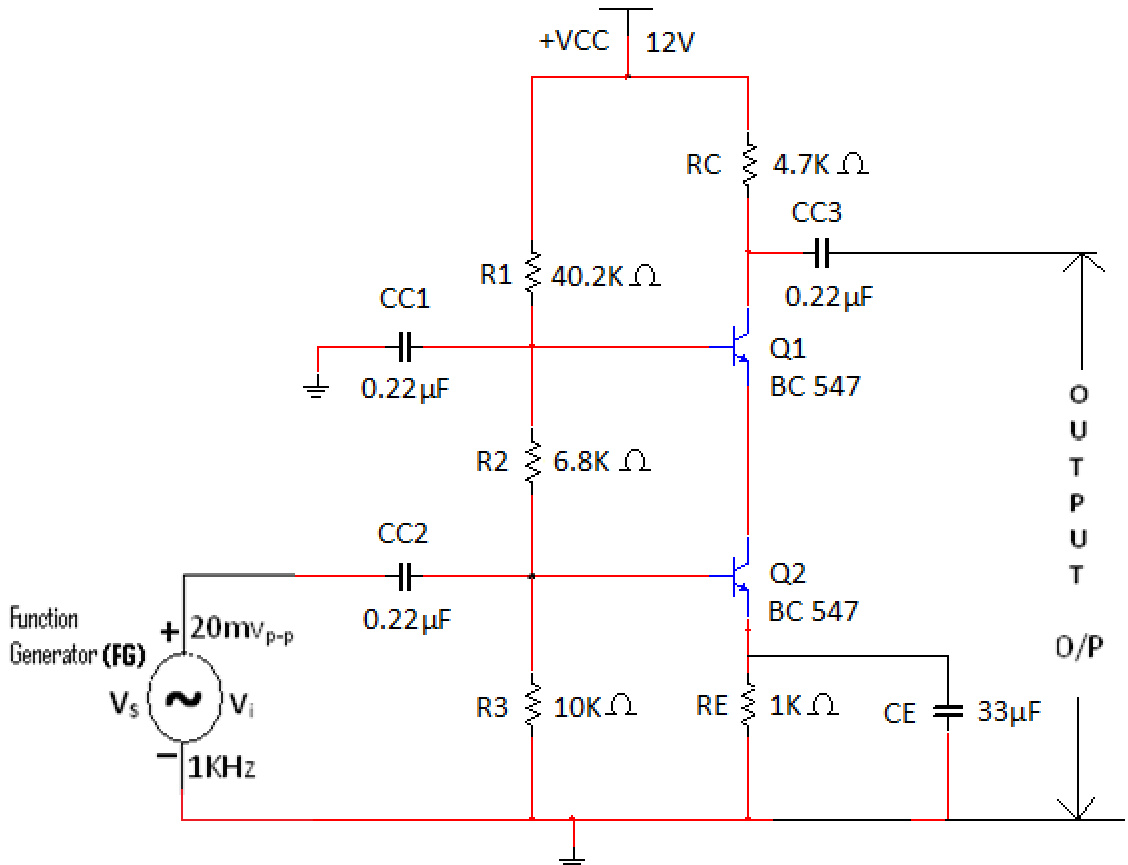
1). Transistor BC 547--1 No.

2) Carbon fixed resistors	a).	47KΩ, ½W	-------- 1 No.
	b).	40.2KΩ	-------- 1 No.
	c).	10KΩ , ½W	-------- 1 No.
	d).	6.8 KΩ , ½W	-------- 1 No.
	e).	4.7 KΩ , ½W	-------- 1 No.
	f).	1 KΩ , ½W	-------- 1 No.
3). Capacitors	g).	0.22μF	-------- 3 No.
	h).	33μF	-------- 1 No.

THEORY :

Note : Theory had given in *Software section-A* of this same experiment. That same theory can apply here also.

CIRCUIT DIAGRAM – SOFTWARE & HARDWARE :



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. Set the input signal as *sine wave form which is having the* value 20mV$_{P-P}$ as constant in the functiongenerator.
4. Initially set the input signal frequency value is 1KHz in the function generator.
5. To simulate the circuit clicked on *run option* through *execute button* in *tool bar*.
6. We have seen the ***sine wave*** on the ***CRO*** screen as o/p signal.
7. Calculated the *peak to peak voltage* ($V_{O(p-p)}$) and noted down in the tabular form Against the column of1KHz.
8. Stopped the simulation by clicked on *run option* through *execute button* in the *tool bar*.
9. Repeat the same procedure from points 7 to 9 for the corresponding frequency values by setting in thefunction generator for the following steps, 20Hz, 100Hz, 200Hz, 1KHz, 200KHz, 400KHz,600KHz, 920KHz,1MHz, 100MHz, 500MHz. in the function generator.
10. Observed the graph for *frequency* Vs *amplitude* through the *AC Analysis*.
11. Finally shut down the system safely.
12. We have observed that, the graph which is drawn by manually is same to the graph which is obtained fromthe *AC Analysis*.
13. 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,

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

14). 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.

PROCEDURE – HARDWARE :

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 thewrong 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, 1KHz, 200KHz, 400KHz, 600KHz, 920KHz,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.								
	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 = $20log_{10}(A_V)$**		**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	1 KHz.								
5	200KHz.								
6	400KHz.								
7	600KHz.								

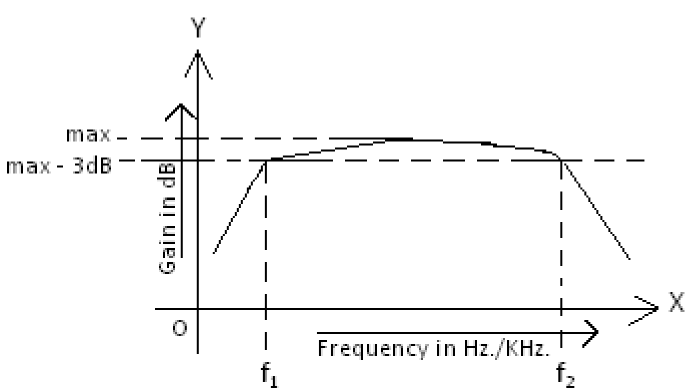
-------- Table form will continue in next page --------

-----Table form Continued -----

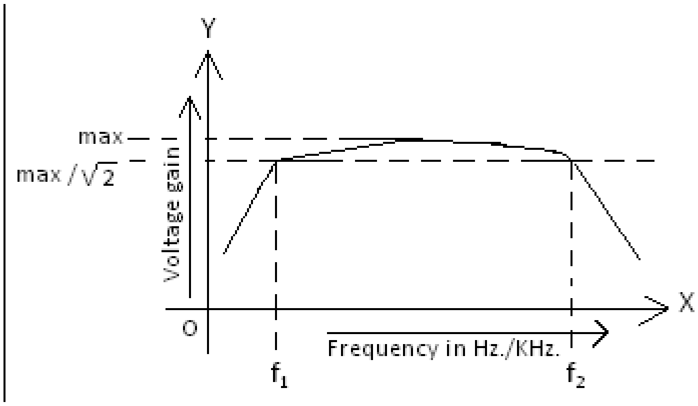
	Input Voltage (V_i) = 20 mV_{P-P} (0.02V) is constant for all readings.								
	For Software :					For Hardware :			
Sl. No.	**Frequ-ency In Hz/KHz.**	**Output Voltage (V_O) In mVolts.**	**Voltage gain $A_V = V_o/V_i$**	**Gain in dB = $20\log_{10}(A_V)$**		**Frequ-ency In Hz/KHz.**	**Output Voltage (V_O)In mVolts.**	**Voltage gain $A_V = V_o/V_i$**	**Gain in dB = $20\log_{10}(A_V)$**
8	920KHz.								
9	1 MHz.								
10	100 MHz					-------	-------	-------	-------
11	500MHz.					-------	-------	-------	-------

EXPECTED GRAPHS – SOFTWARE & HARDWARE :

A). Frequency response curve For frequency verses gain in dB.



B). Frequency response curve For frequency verses voltage gain.



PARAMETERS – SOFTWARE & HARDWARE :

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 –SOFTWARE & HARDWARE :

We have obtained the frequency response curves of *CE-CB cascade Amplifie* for frequency verses gain in dB & frequency verses voltage gain and calculated the band width of both of them.The band width values are given below,

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 VOICE Questions :

Note : The questions which are avail in *Software section-A* under the same experiment are applicable here also.

Experiment No. : 12 **Date :**

Name of the Experiment : ASTABLE MULTIVIBRATOR

(Beyond the Syllabus-Using Software & Hardware)

AIM :

To conduct and verify the Astable multi vibrator and to draw the waveforms using software and hardware

APPARATUS :

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

COMPONENTS :

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

THEORY :

Note : Theory had given in *Software section-A* of this same experiment. That same theory can apply here also.

CIRCUIT DIAGRAMS – SOFTWARE & HARDWARE:

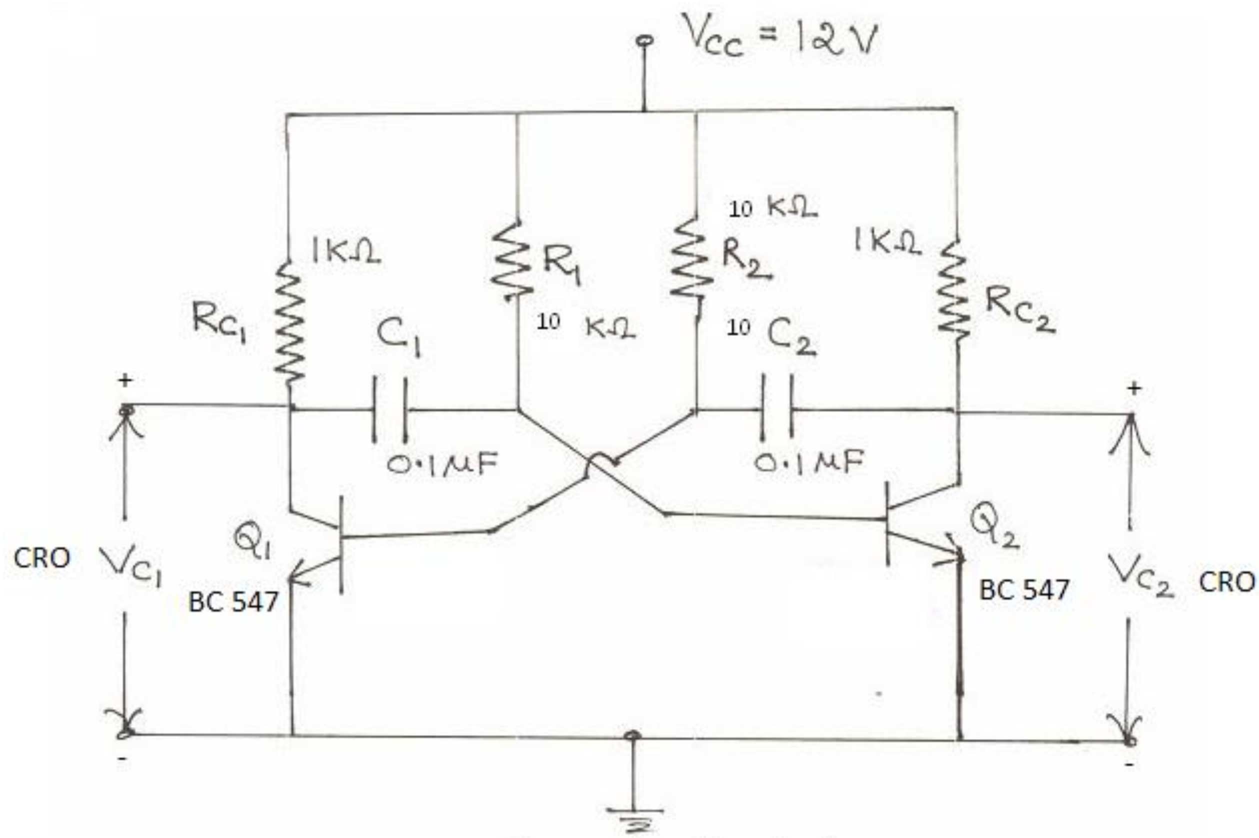


Fig : Circuit diagram of Astable Multivibrator

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 = 7.24KΩ (Practically choose 10KΩ) i.e., R1 and R2 resistors = 10KΩ

Let $I_{Cmax} = 10mA$

$R_C = \frac{V_{cc} - V_{cesat}}{I_{cmax}} = \frac{12 - 0.3}{0.01}$ = 1.17KΩ (Practically choose 1KΩ) i.e., Rc1 and Rc2 resistors = 1KΩ

Theoretical calculations :

F = 1/ T = (1/1.38RC)

R = 10KΩ C = $0.1\mu F$

PROCEDURE - SOFTWARE :

1. I 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 picked up.
3. Connected the CRO across VC1 and VC2 .
6. To simulate the circuit clicked on *run option* through *execute button* in *tool bar*.
7. I have observed the wave forms as shown under the heading of Expected graphs .
8. Observed the Base Voltage and Collector Voltages of Q1 & Q2 on CRO in DC mode and measured the frequency (f = 1/T).
9. Traced the waveforms at collector and base as each transistor with the help of dual trace CRO and plot the waveforms.
10. Verified the practical output frequency with theoretical values f = 1/T, where T = 1.38RC
11. Shut down the system safely.
12. Plotted the graphs for VB1 & VC1 and VB2 and VC2 by taking the Time period on X-axis and Voltage on Y-axis for all graphs as per shown in the Expected graphs heading.
13. Noted the practical Time period T values at VB1 & VC1 and VB2 and VC2 and noted down in the corresponding columns of the Tabular form.
14. Calculated the practical frequency values by using formula 1/T and noted down in the corresponding columns of the Tabular form.
15. I Compared the Theoretical and practical values are approximately same.

PROCEDURE - HARDWARE :

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
5. Switched off the RPS and CRO.
6. Plotted the graphs for VB1 & VC1 and VB2 and VC2 by taking the Time period on X-axis and Voltage on Y-axis for all graphs as per shown in the Expected graphs heading.
7. Noted the practical Time period T values at VB1 & VC1 and VB2 and VC2 and noted down in the corresponding columns of the Tabular form.
8. Calculated the practical frequency values by using formula 1/T and noted down in the corresponding columns of the Tabular form.
9. I Compared the Theoretical and practical values are approximately same.

TABULAR FORMS – SOFTWARE & HARDWARE :

	Software					Hardware			
	At VB1	At VC1	At VB2	At VC2		At VB1	At VC1	At VB2	At VC2
Theoretical Time period (T)									
Theoretical Frequency (f) = 1/T									
Practical Time period (T)									
Practical Frequency (f) = 1/T									

EXPECTED WAVEFORM - SOFTWARE & HARDWARE :

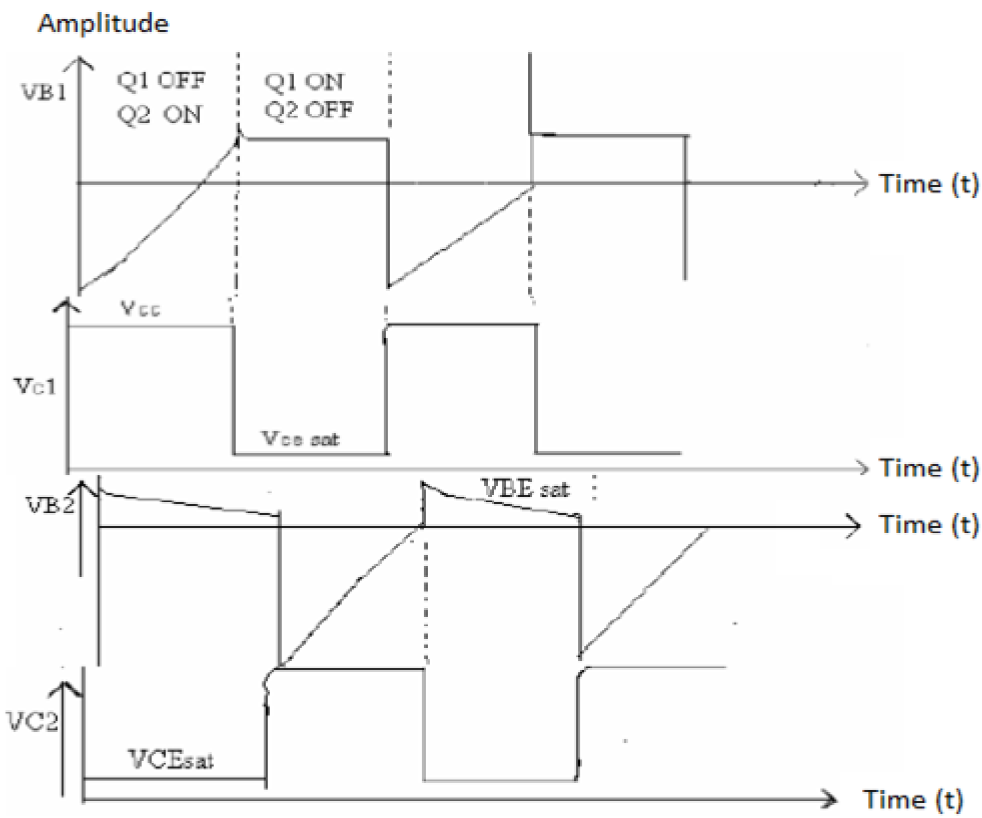


Fig : Wafeforms of Astable Multivibrator

RESULT : I have conducted and verified the Astable Multivibrator.

VIVA VOICE Questions :

Note : The questions which are avail in *Software section-A* under the same experiment are applicable here also.

PART – D

Data sheets, Rules & Syllabus Copy

A. DATA SHEETS

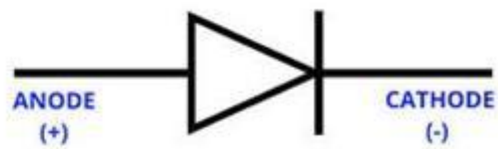
PN JUNCTION DIODE :

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	800	800	1000	V
RMS Reverse Voltage	$V_{R(RMS)}$	35	70	140	280	420	580	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

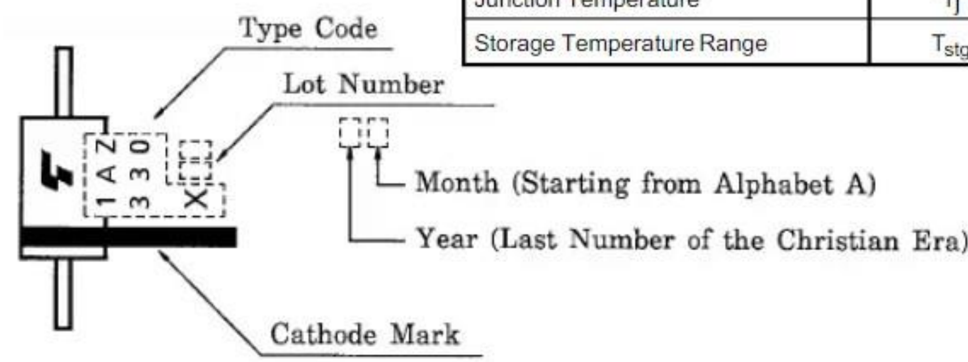
CONSTANT VOLTAGE REGULATION
TRANSIENT SUPPRESSORS

- 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

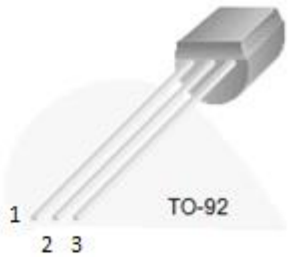
BIPOLAR JUNCTION TRANSISTORS (BJT) :

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 kΛ		2.0	10.0	dB
		BC549 / BC550			1.2	4.0	
		BC549	V_{CE} = 5 V, I_C = 200 μA, R_G = 2 kΛ, f = 30 to 15000 MHz		1.4	4.0	
		BC550			1.4	3.0	

h_{FE} Classification

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

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)

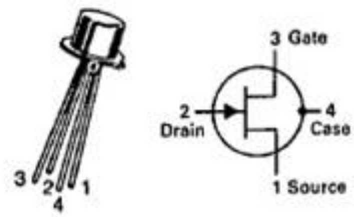
SYMBOL	TEST CONDITIONS	2N2646 MIN	2N2646 MAX	2N2647 MIN	2N2647 MAX	UNITS
η	V_{B2B1}=10V	0.56	0.75	0.68	0.82	
R_{BB}	V_{B2B1}=3.0V	4.7	9.1	4.7	9.1	kΩ
I_{EB2O}	V_{B2E}=30V	-	12	-	0.2	µA
I_V	V_{B2B1}=20V, R_{B2}=100Ω	4.0	-	8.0	18	mA
I_P	V_{B2B1}=25V	-	5.0	-	2.0	µA
V_{OB1}	V_1=20V	3.0	-	6.0	-	V

FIELD EFFECT TRANSISTOR (FET) :

MOTOROLA SC {XSTRS/R F}

BFW10
BFW11

CASE 20-03, STYLE 1
TO-72 (TO-206A)



JFET
VHF/UHF AMPLIFIER

N-CHANNEL — DEPLETION

MAXIMUM RATINGS

Rating	Symbol	Value	Unit
Drain-Source Voltage	V_{DS}	30	Vdc
Drain-Gate Voltage	V_{DG}	30	Vdc
Reverse Gate-Source Voltage	V_{GSR}	−30	Vdc
Forward Gate Current	I_{GF}	10	mAdc
Total Device Dissipation @ T_A = 25°C Derate above 25°C	P_D	300 1.71	mW mW/°C
Operating and Storage Junction Temperature Range	T_J, T_{stg}	−65 to +150	°C

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

Characteristic		Symbol	Min	Typ	Max	Unit
OFF CHARACTERISTICS						
Gate-Source Breakdown Voltage (I_G = 10 μAdc, V_{DS} = 0)		$V_{(BR)GSS}$	30	—	—	Vdc
Gate-Source Cutoff Voltage (V_{DS} = 15 Vdc, I_D = 0.5 nAdc)	BFW10 BFW11	$V_{GS(off)}$	—	—	8 6	Vdc
Gate Reverse Current (V_{GS} = 20 Vdc, V_{DS} = 0)		I_{GSS}	—	—	0.1	nAdc
Gate-Source Voltage (V_{DS} = 15 Vdc, I_D = 400 μAdc)	BFW10	V_{GS}	2	—	7.5	Vdc
Gate-Source Voltage (V_{DS} = 15 Vdc, I_D = 50 μAdc)	BFW11	V_{GS}	1.25	—	4	Vdc
ON CHARACTERISTICS						
Zero-Gate Voltage Drain Current (V_{DS} = 15 Vdc, V_{GS} = 0)	BFW10 BFW11	I_{DSS}	8 4	— —	20 10	mAdc
SMALL-SIGNAL CHARACTERISTICS						
Forward Transadmittance (V_{DS} = 15 Vdc, V_{GS} = 0, f = 1 kHz)	BFW10 BFW11	Y_{fs}	3.5 3.0	— —	6.5 6.5	mmhos
Output Admittance (V_{DS} = 15 Vdc, V_{GS} = 0, f = 1.0 kHz)	BFW10 BFW11	Y_{os}	— —	— —	85 50	μmhos
Input Capacitance (V_{DS} = 15 Vdc, V_{GS} = 0 Vdc, f = 1.0 MHz)		C_{iss}	—	—	5.0	pF
Reverse Transfer Capacitance (V_{DS} = 15 Vdc, V_{GS} = 0 Vdc, f = 1.0 MHz)		C_{rss}	—	—	0.8	pF
Forward Transadmittance (V_{DS} = 15 Vdc, V_{GS} = 0, f = 200 MHz)		Y_{fs}	3.2	—	—	mmhos
Equivalent Noise Voltage (V_{DS} = 15 Vdc, V_{GS} = 0, f = 25 Hz)		e_n	—	—	75	nV/$\sqrt{Hz}$
Noise Figure (V_{DS} = 15 Vdc, V_{GS} = 0 V, see Figures 1, 2, 3)		NF	—	—	2.5	dB

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

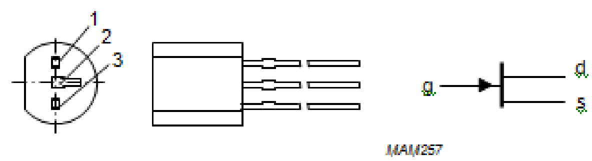


Fig.1 Simplified outline (TO-92 variant) and symbol.

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
$R_{th\ j\text{-}a}$	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

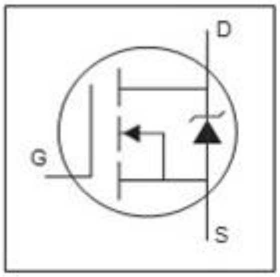
Common source; T_{amb} = 25 °C; unless otherwise specified.

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

MOSFET IRFZ 44N :

IRFZ44NPbF

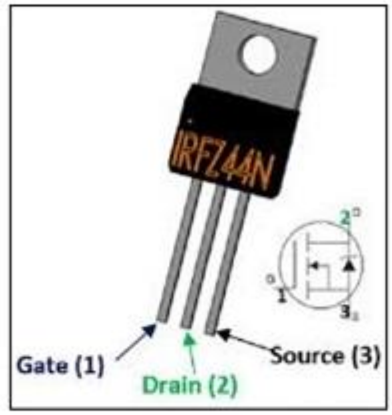
HEXFET® Power MOSFET



$V_{DSS} = 55V$

$R_{DS(on)} = 17.5m\Omega$

$I_D = 49A$



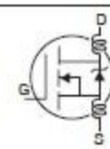
Absolute Maximum Ratings

	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)	

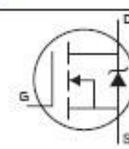
Thermal Resistance

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

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 $T_J = 175°C$.

RULES TO BE FOLLOWED WHILE OPERATING THE REGULATED POWERSUPPLY(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*.

JAWAHARLAL NEHRU TECHNOLOGICAL UNIVERSITY ANANTAPUR

B.Tech –II-II Sem 15A04404, ANALOG CIRCUIT ANALYSIS LABORATORY

LIST OF EXPERIMENTS Branch : For ECE only R15

Syllabus – List of experiments :
(Minimum 10 experiments to be done using simulation software and hardware)

1. Determination of f_T of a given transistor.
2. Voltage-Series Feedback Amplifier.
3. Current-Shunt Feedback Amplifier.
4. RC Phase Shift/Wein Bridge Oscillator.
5. Hartley/Colpitt's Oscillator.
6. Two Stage RC Coupled Amplifier.
7. Darlington Pair Amplifier.
8. Bootstrapped Emitter follower.
9. Class A Series-fed Power Amplifier.
10. Transformer-coupled Class A Power Amplifier.
11. Class B Push-Pull Power Amplifier.
12. Complementary Symmetry Class B Push-Pull Power Amplifier.
13. Single Tuned Voltage Amplifier.
14. Double Tuned Voltage Amplifier.