GEETHANJALI COLLEGE OF ENGINEERING AND TECHNOLOGY

Cheeryal (V), Keesara (M), R. R. District

ELECTRONIC CIRCUITS LAB

STUDENTS' MANUAL

…striving toward perfection

DEPARTMENT OF

ELECTRONICS AND COMMUNICATION ENGINEERING

Lab Incharge: **HOD-ECE:**

J. Mrudula **Dr.P.Srihari**

Associate Professor

GEETHANJALI COLLEGE OF ENGINEERING & TECHNOLOGY

Cheeryal (V), Keesara (M), R. R. District

…striving toward perfection

LABORATORY MANUAL

FOR

ELECTRONIC CIRCUITS LAB

Prepared by:
Ms. J. Mrudula
Assoc.Prof

Checked by:
Prof. K. Somasekhara Rao,
Dean of Academics, Dept., of ECE

Approved by:
Dr.P.Srihari,HOD
Dept., of ECE

GEETHANJALI COLLEGE OF ENGINEERING AND TECHNOLOGY
DEPARTMENT OF *Electronics and Communications Engineering*

(Name of the Subject / Lab Course) : Electronic circuits Lab

(JNTU CODE -) A40484 **Programme : *UG***

Branch: ECE	**Version No : *01***
Year: II	**Updated on : 02/12/2015**
Semester: II	**No. of pages :**

Classification status (Unrestricted / Restricted): Unrestricted

Distribution List :Department , Lab, Library, Lab incharge

Prepared by : 1) Name : J. Mrudula	**1) Name :**
2) Sign :	**2) Sign :**
3) Design : Associate professor	**3) Design :**
4) Date : 02/12/2015	**4) Date : 02/12/2015**

Verified by : 1) Name :Prof. K. Somasekhara Rao	*** For Q.C Only.**
2) Sign :	**1) Name :**
3) Design : Professor	**2) Sign :**
4) Date :	**3) Design :**
	4) Date :

Approved by : (HOD) 1) Name : Dr.P.Srihari

2) Sign :

3) Date :

ECE DEPARTMENT

Vision of the Department

To impart quality technical education in Electronics and Communication Engineering emphasizing analysis, design/synthesis and evaluation of hardware/embedded software using various Electronic Design Automation (EDA) tools with accent on creativity, innovation and research thereby producing competent engineers who can meet global challenges with societal commitment.

Mission of the Department

i. To impart quality education in fundamentals of basic sciences, mathematics, electronics and communication engineering through innovative teaching-learning processes.
ii. To facilitate Graduates define, design, and solve engineering problems in the field of Electronics and Communication Engineering using various Electronic Design Automation (EDA) tools.
iii. To encourage research culture among faculty and students thereby facilitating them to be creative and innovative through constant interaction with R & D organizations and Industry.
iv. To inculcate teamwork, imbibe leadership qualities, professional ethics and social responsibilities in students and faculty.

<u>Program Educational Objectives of B. Tech (ECE) Program:</u>

I. To prepare students with excellent comprehension of basic sciences, mathematics and engineering subjects facilitating them to gain employment or pursue postgraduate studies with an appreciation for lifelong learning.
II. To train students with problem solving capabilities such as analysis and design with adequate practical skills wherein they demonstrate creativity and innovation that would enable them to develop state of the art equipment and technologies of multidisciplinary nature for societal development.
III. To inculcate positive attitude, professional ethics, effective communication and interpersonal skills which would facilitate them to succeed in the chosen profession exhibiting creativity and innovation through research and development both as team member and as well as leader.

Program Outcomes of B.Tech ECE Program:

1. An ability to apply knowledge of Mathematics, Science, and Engineering to solve complex engineering problems of Electronics and Communication Engineering systems.
2. An ability to model, simulate and design Electronics and Communication Engineering systems, conduct experiments, as well as analyze and interpret data and prepare a report with conclusions.
3. An ability to design an Electronics and Communication Engineering system, component, or process to meet desired needs within the realistic constraints such as economic, environmental, social, political, ethical, health and safety, manufacturability and sustainability.
4. An ability to function on multidisciplinary teams involving interpersonal skills.
5. An ability to identify, formulate and solve engineering problems of multidisciplinary nature.
6. An understanding of professional and ethical responsibilities involved in the practice of Electronics and Communication Engineering profession.
7. An ability to communicate effectively with a range of audience on complex engineering problems of multidisciplinary nature both in oral and written form.
8. The broad education necessary to understand the impact of engineering solutions in a global, economic, environmental and societal context.
9. A recognition of the need for, and an ability to engage in life-long learning and acquire the capability for the same.
10. A knowledge of contemporary issues involved in the practice of Electronics and Communication Engineering profession
11. An ability to use the techniques, skills and modern engineering tools necessary for engineering practice.
12. An ability to use modern Electronic Design Automation (EDA) tools, software and electronic equipment to analyze, synthesize and evaluate Electronics and Communication Engineering systems for multidisciplinary tasks.
13. Apply engineering and project management principles to one's own work and also to manage projects of multidisciplinary nature.

SYLLABUS

JAWAHARLAL NEHRU TECHNOLOGICAL UNIVERSITY HYDERABAD

II Year B.Tech. ECE-II Sem

	L	T/P/D	C
	-	-/3/-	2

(A40484) ELECTRONIC CIRCUITS AND PULSE CIRCUITS LAB

List of Experiments (16 experiments to be done):

PART –I: ELCTRONIC CIRCUITS

Minimum eight experiments to be conducted:

I) Design and Simulation in Simulation Laboratory using a Simulation Software (Minimum 6 Experiments):

1. Common Emitter Amplifier
2. Common Source Amplifier
3. Two Stage RC Coupled Amplifier
4. Current shunt and Voltage Series Feedback Amplifier
5. Cascode Amplifier
6. Wien Bridge Oscillator using Transistors
7. RC Phase Shift Oscillator using Transistors
8. Class A Power Amplifier (Transformer less)
9. Class B Complementary Symmetry Amplifier
10. Common Base (BJT) / Common Gate (JFET) Amplifier.

II) Testing in the Hardware Laboratory (Minimum 2 Experiments)

1. Class A Power Amplifier (with transformer load)
2. Class C Power Amplifier
3. Single Tuned Voltage Amplifier
4. Hartley & Colpitt's Oscillators
5. Darlington Pair
6. MOS Common Source Amplifier

Equipment required for the Laboratory:

1. For software simulation of Electronic circuits
   - i) Computer Systems with latest specifications
   - ii) Connected in LAN (Optional)
   - iii) Operating system (Windows XP)
   - iv) Suitable Simulations software
2. For Hardware simulations of Electronic Circuits
   - i) Regulated Power Supply (0-30V)
   - ii) CRO's

iii) Functions Generators
iv) Multimeters
v) Components

3. Win XP/ Linux etc.

PART –II: PULSE CIRCUITS

Minimum eight experiments to be conducted:

1. Linear Wave Shaping
   a. RC Low Pass Circuit for different time constants
   b RC High Pass Circuit for different time constants
2. Non-linear wave shaping
   a. Transfer characteristics and response of Clippers:
      i) Positive and Negative Clippers
      ii) Clipping at two independent levels
   b The steady state output waveform of clampers for a square wave input
      i) Positive and Negative Clampers
      ii) Clamping at reference voltage
3. Comparison Operation of Comparators
4. Switching characteristics of a transistor
5. Design a Bistable Multivibrator and draw its waveforms
6. Design an Astable Multivibrator and draw its waveforms
7. Design a Monostable Multivibrator and draw its waveforms
8. Response of Schmitt Trigger circuit for loop gain less than and greater than one
9. UJT relaxation oscillator
10. The output- voltage waveform of Boot strap sweep circuit
11. The output- voltage waveform of Miller sweep circuit

Equipment required for Laboratories:

Regulated Power Supply	- 0 – 30 V
CRO	- 0 – 20 M Hz.
Function Generators	- 0 – 1 M Hz
Components	
Multi Meters	

INSTRUCTIONS

Instruction for students:-

1. Do not handle any equipment without reading the instructions /Instruction manuals.
2. Observe type of sockets of equipment power to avoid mechanical damage.
3. Do not insert connectors forcefully in the sockets.
4. Strictly observe the instructions given by the Teacher/ Lab Instructor.
5. After the experiment is over, the students must hand over the Bread board, Trainer kits, wires, CRO probes and other components to the lab assistant/teacher.
6. It is mandatory to come to lab in a formal dress (Shirts, Trousers, ID card, and Shoes for boys). Strictly no Jeans for both Girls and Boys.
7. It is mandatory to come with observation book and lab record in which previous experiment should be written in Record and the present lab's experiment in Observation book.
8. Observation book of the present lab experiment should be get corrected on the same day and Record should be corrected on the next scheduled lab session.
9. Mobile Phones should be Switched OFF in the lab session.
10. Students have to come to lab in-time. Late comers are not allowed to enter the lab.
11. Prepare for the viva questions. At the end of the experiment, the lab faculty will ask the viva questions and marks are allotted accordingly.
12. Bring all the required stationery like graph sheets, pencil & eraser, different color pens etc. for the lab class.
13. While shorting 2 or more wires for common connections like grounding, do not twist wires. Use shorting link on the bread board.

Instructions to Laboratory Teachers:-

1. Observation book and lab records submitted for the lab work are to be checked and signed before the next lab session.
2. Students should be instructed to switch ON the power supply after the connections are checked by the lab assistant / teacher.
3. The promptness of submission of records/ observation books should be strictly insisted by awarding the marks accordingly.
4. Ask viva questions at the end of the experiment.
5. Do not allow students who come late to the lab class.
6. Encourage the students to do the experiments innovatively.

ELECTRONIC CIRCUITS

INDEX.

ELECTRONIC CIRCUITS

Exp 1. TRANSISTOR CE AMPLIFIER

OBJECTIVE: 1. To Design Transistor CE amplifier and simulating in MULTISIM software and find its voltage gain.

2. To draw the frequency response curve of CE amplifier.

APPARATUS:

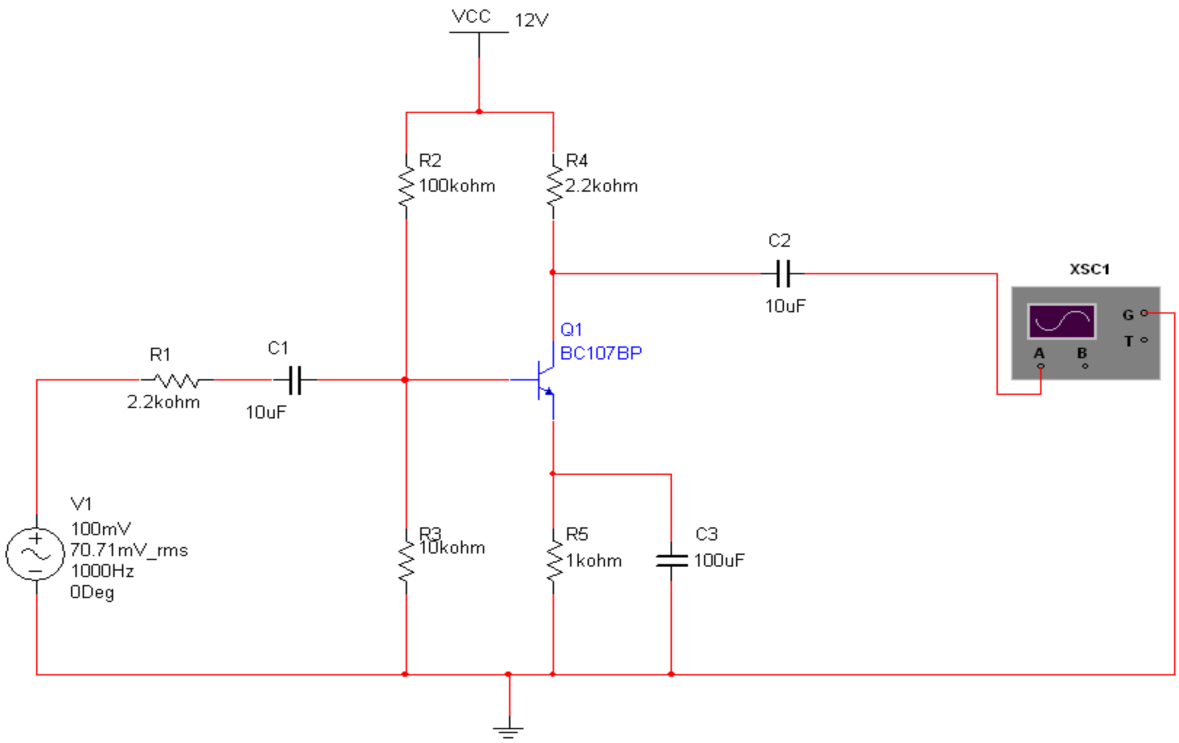
MULTISIM 2001 SOFTWARE

PC

THEORY:

The CE amplifier provides high gain and wide frequency response. The emitter lead is common to both input and output circuits and is grounded. The emitter-base circuit is forward biased. The collector current is controlled by the base current rather than emitter current. The input signal is applied to base terminal of the transistor and amplifier output is taken across collector terminal. A very small change in base current produces a much larger change in collector current. When +ve half-cycle is fed to the input circuit, it opposes the forward bias of the circuit which causes the collector current to decrease, it decreases the voltage more –ve. When input cycle varies through a -ve half-cycle, it increases the forward bias of the circuit, which causes the collector current to increases thus the output signal in common emitter amplifier is in out of phase with the input signal.

CIRCUIT DIAGRAM:



PROCEDURE:

1. Start MULTISIM. A blank circuit window will appear on the screen along with a component tool bar.
2. Using component tool bar, place all the components on the circuit window and wire the circuit.
3. Connect the circuit as shown in circuit diagram
4. Apply the input of 50mV peak-to-peak and 1 KHz frequency sinusoidal, using AC Voltage Source
5. Measure the Output Voltage Vo (p-p) for different frequencies
6. Tabulate the readings in the tabular form.
7. The voltage gain can be calculated by using the expression

$$A_v = (V_0/V_i)$$

8. For plotting the frequency response the input voltage is kept constant at 50mV peak-to-peak and the frequency is varied from 10Hz to 1MHz using function generator
9. Note down the value of output voltage for each frequency.
10. All the readings are tabulated and voltage gain in dB is calculated by usingThe expression

`

$$A_v = 20 \log_{10} (V_0/V_i)$$

11. A graph is drawn by taking frequency on X-axis and gain in dB on Y-axis on a Semi-log graph.

The band width of the amplifier is calculated from the graph using the expression,

Bandwidth, $BW=f_2-f_1$

Where f_1 lower cut-off frequency of CE amplifier, and

Where f_2 upper cut-off frequency of CE amplifier

The Gain bandwidth product of the amplifier is calculated using the expression

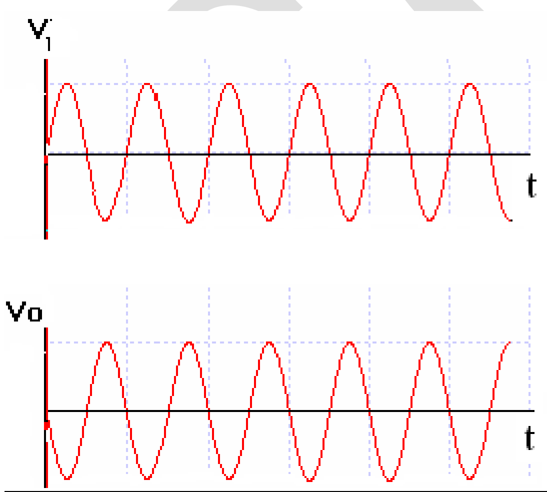
Gain Bandwidth product=3-dB midband gain X Bandwidth

OBSERVATIONS:

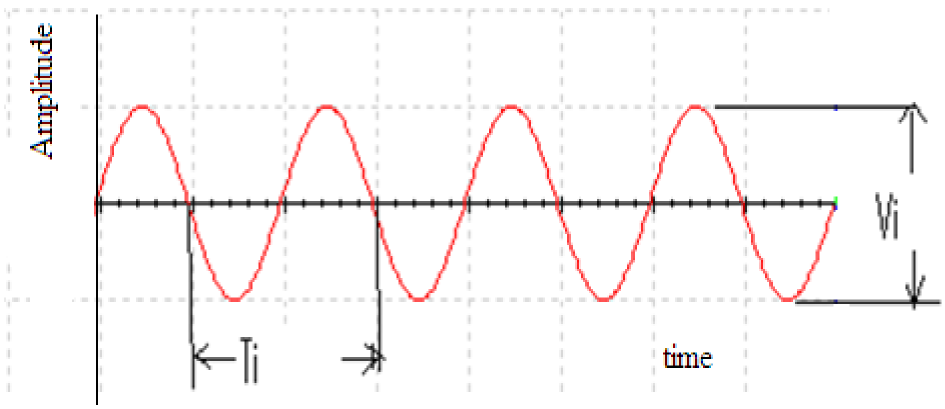
Input voltage V_{in} =50mV

FREQUENCY(Hz)	OUTPUT VOLTAGE (V_o)	Gain = V_o/V_{in}	GAIN IN dB $A_v=20\log_{10}(V_0/V_{in})$
10			
50			
100			
200			
400			
600			
800			
1K			
5K			
10K			
50K			
100K			
200K			
400K			
600K			
800K			
1M			

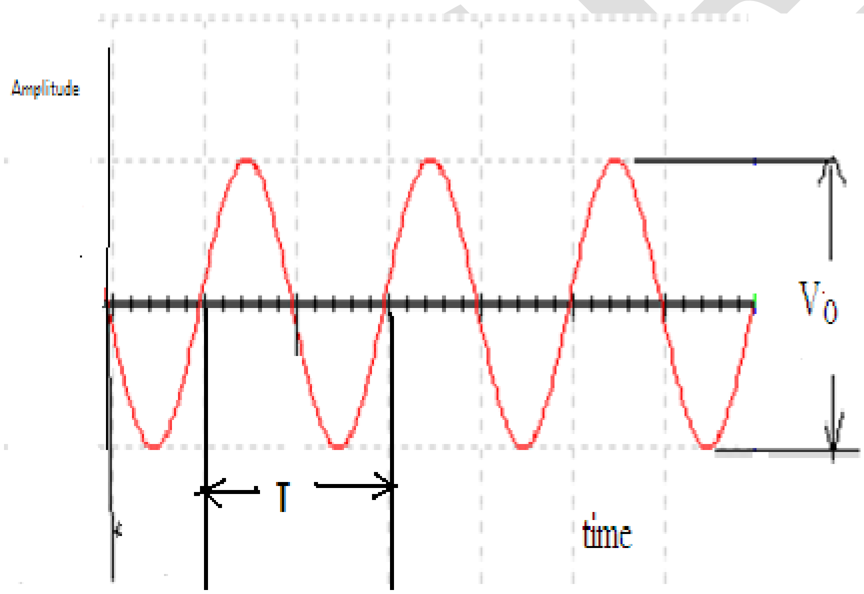
MODELWAVE FORMS:



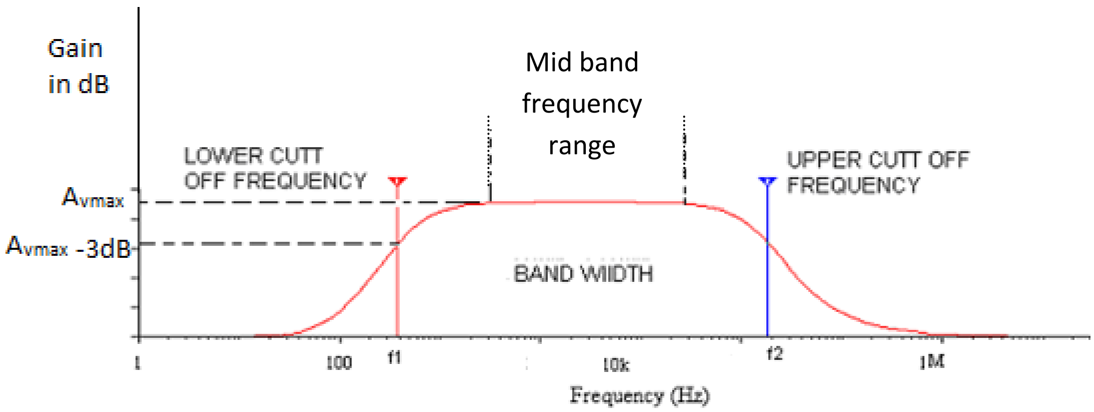
INPUT WAVE FORM:

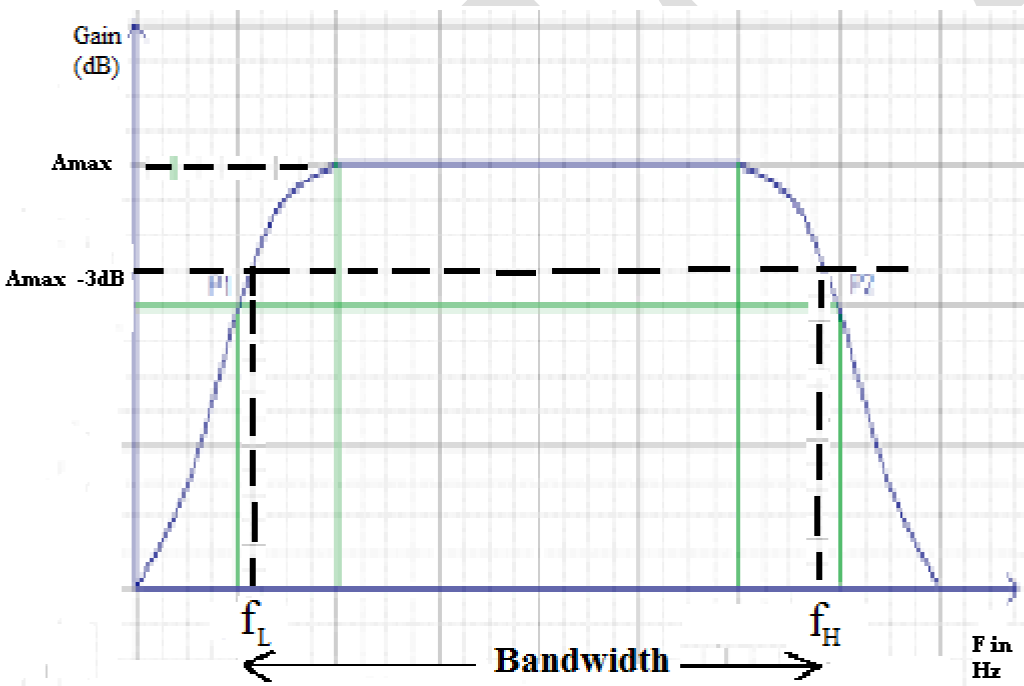


OUTPUT WAVE FORM



FREQUENCY RESPONSE:





RESULT: The voltage gain and frequency response of the CE amplifier are obtained. Also gain bandwidth product of the amplifier is calculated.

LEARNING OUTCOMES:

Students are able to

1. Design and analyze CE amplifier using MULTISIM software.
2. Calculate voltage gain and bandwidth of CE amplifier from the observations made.

VIVA QUESTIONS:

1. What is phase difference between input and output waveforms of CE amplifier?
2. What type of biasing is used in the above given circuit?
3. If the given transistor is replaced by a p-n-p, can we amplify the signals and explain?
4. What is effect of emitter-bypass capacitor on frequency response?
5. What is the effect of coupling capacitor?
6. What is the region of the transistor so that it can be operated as an amplifier?
7. How does transistor acts as an amplifier?
8. Draw the h-parameter model of CE amplifier?
9. What type of transistor configuration is used in intermediate stages of a multistage amplifier?
10. What is Early effect?

<u>**Exp.2. COMMON SOURCE FET AMPLIFIER**</u>

OBJECTIVE: 1. To obtain the frequency response of the Common Source FET Amplifier

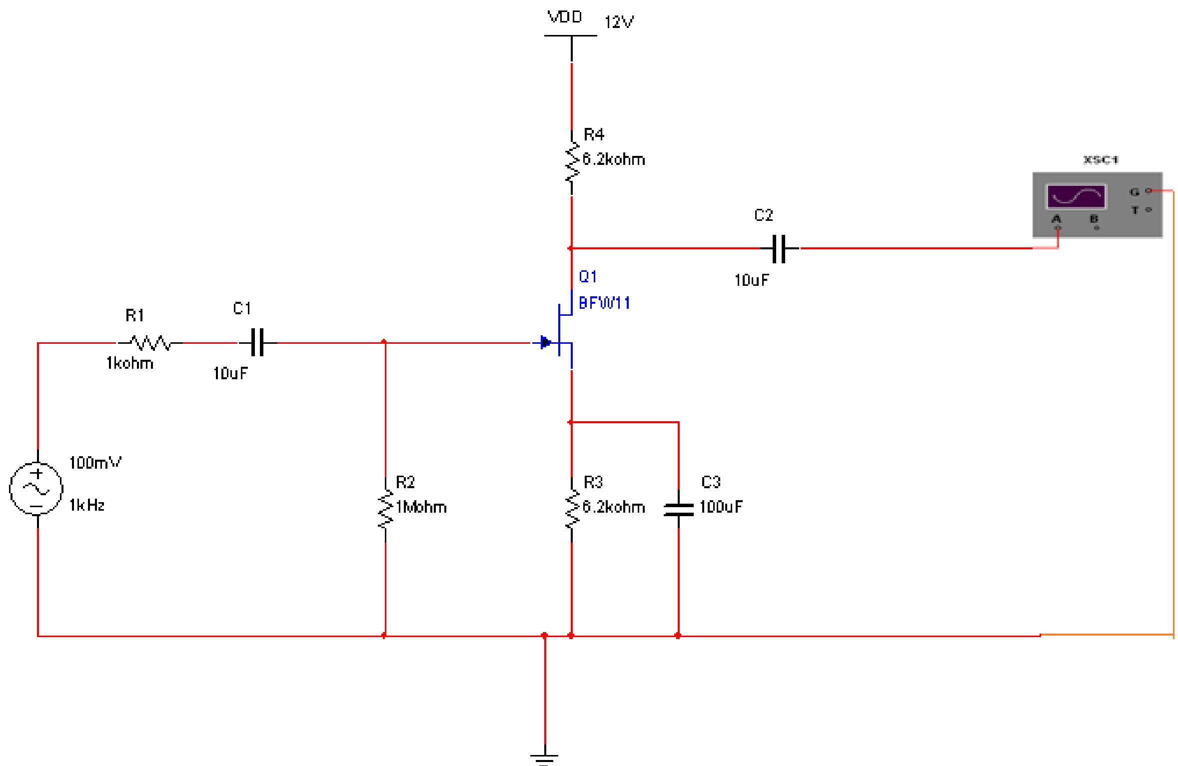
2. To find the Bandwidth.

APPRATUS:

MULTISIM 2001 SOFTWARE

PC

CIRCUIT DIAGRAM:



THEORY:

A field-effect transistor (FET) is a type of transistor commonly used for weak-signal amplification (for example, for amplifying wireless (signals)). The device can amplify analog or digital signals. It can also switch DC or function as an oscillator. In the FET, current flows along a semiconductor path called the *channel*. At one end of the channel, there is an electrode called the *source*. At the other end of the channel, there is an electrode called the *drain*. The physical diameter of the channel is fixed, but its effective electrical diameter can be varied by the application of a voltage to a control electrode called the *gate*. Field-effect transistors exist in two major classifications. These are known as the *junction FET (JFET) and the metal-oxide-*

semiconductor FET (MOSFET). The junction FET has a channel consisting of N-type semiconductor (N-channel) or P-type semiconductor (P-channel) material; the gate is made of the opposite semiconductor type. In P-type material, electric charges are carried mainly in the form of electron deficiencies called *holes*. In N-type material, the charge carriers are primarily electrons. In a JFET, the junction is the boundary between the channel and the gate. Normally, this P-N junction is reverse-biased (a DC voltage is applied to it) so that no current flows between the channel and the gate. However, under some conditions there is a small current through the junction during part of the input signal cycle. The FET has some advantages and some disadvantages relative to the bipolar transistor. Field-effect transistors are preferred for weak-signal work, for example in wireless, communications and broadcast receivers. They are also preferred in circuits and systems requiring high impedance. The FET is not, in general, used for high-power amplification, such as is required in large wireless communications and broadcast transmitters.

Field-effect transistors are fabricated onto silicon integrated circuit (IC) chips. A single IC can contain many thousands of FETs, along with other components such as resistors, capacitors, and diodes.

PROCEDURE:

1. Start MULTISIM. A blank circuit window will appear on the screen along with a component tool bar.
2. Using component tool bar place all the components on the circuit window and wire the circuit.
3. A sinusoidal signal of 1 KHz frequency and 200mV peak-to-peak is applied at the input of amplifier.
4. Output is taken at drain and gain is calculated by using the expression,

   $$A_v=V_0/V_i$$

5. Voltage gain in dB is calculated by using the expression,

   $$A_v=20\log 10(V_0/V_i)$$

6. Repeat the above steps for different frequencies.
7. For plotting the frequency response the input voltage is kept Constant at 50mV peak-to-peak and the frequency is varied from 10Hz to 1MHz using AC Voltage Source.
8. The Bandwidth of the amplifier is calculated from the graph using the
9. Expression,

   $$\text{Bandwidth } BW=f_2-f_1$$

   Where f_1 is lower 3 dB frequency

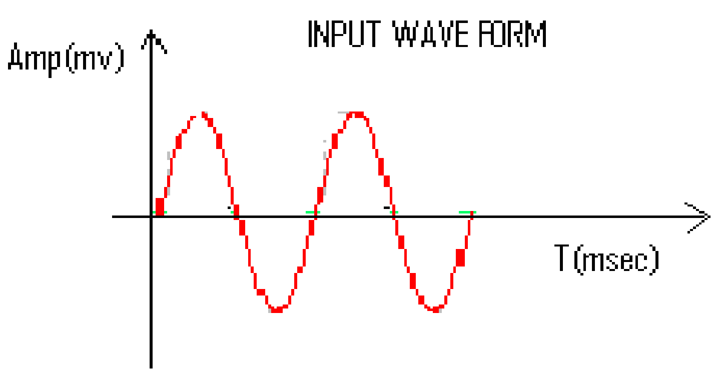
   f_2 is upper 3 dB frequency

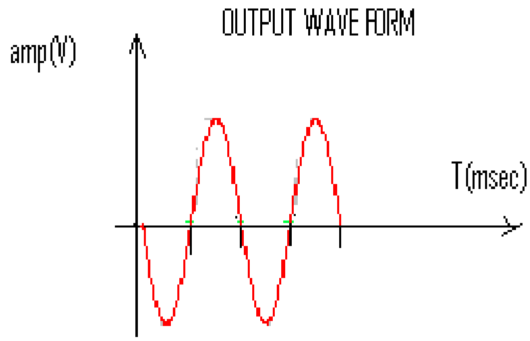
OBSERVATIONS:

Input Voltage given V_{in}:50mV

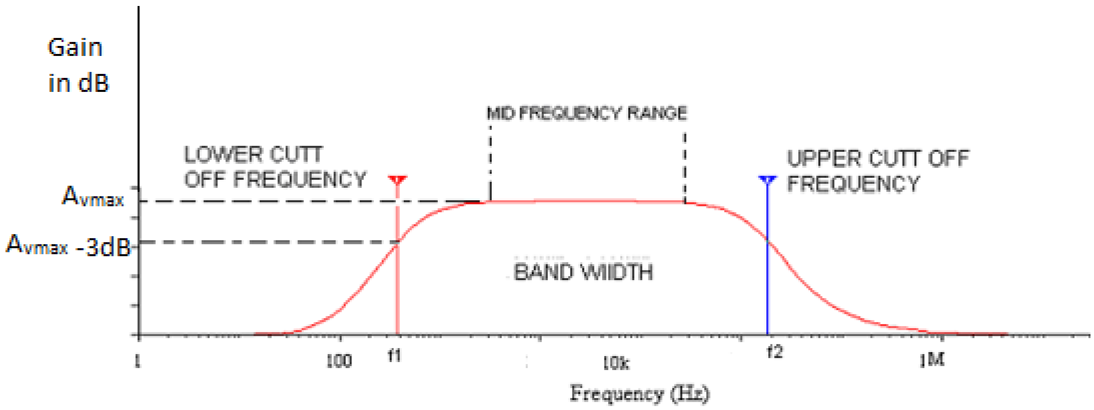
Frequency(Hz)	Output Voltage (V_o)	Gain= V_o/V_{in}	Av in dB 20 log (V_o/V_{in})
10			
50			
100			
200			
400			
600			
800			
1K			
5K			
10K			
50K			
100K			
200K			
400K			
600K			
800K			
1M			

MODEL GRAPH:





FREQUENCY RESPONSE:



PRECAUTIONS

1. All the connections are to be connected properly.
2. Transistor terminals must be identified properly

RESULT: The frequency response of the common source FET Amplifier and Bandwidth is obtained.

LEARNING OUTCOMES:

Students are able to

1. Design and analyze CE amplifier using MULTISIM software.

2. Calculate voltage gain and bandwidth of CE amplifier from the observations made.

VIVA QUESTIONS

1. What are the differences between FET and BJT?
2. Is FET an unipolar or bipolar device ?
3. Draw the symbol of FET?
4. What are the applications of FET?
5. Is FET a voltage controlled or current controlled device?
6. Draw the equivalent circuit of common source FET amplifier?
7. What is the voltage gain of the FET amplifier?
8. What is the input impedance of FET amplifier?
9. What is the output impedance of FET amplifier?
10. What are the FET parameters?

Exp.3. TWO STAGE RC COUPLED AMPLIFIER

OBJECTIVES: 1.To obtain the frequency response of the Two Stage RC Coupled Amplifier

2. To find the Voltage Gain and Bandwidth.

APPARATUS:

MULTISIM 2001 SOFTWARE

PC

THEORY:

This is most popular type of coupling as it provides excellent audio fidelity. A coupling capacitor is used to connect output of first stage to input of second stage. Resistances R_5, R2, R_3 form biasing and stabilization network for Q_1 and R_7, R_8 and R_9 for Q_2. Emitter bypass capacitor offers low reactance paths to signal. coupling Capacitor transmits AC signal, blocks DC. Cascaded stages amplify the signal, the overall gain is increased. Thus coupling is done for more gain and overall gain of two stages equals to A=A1*A2

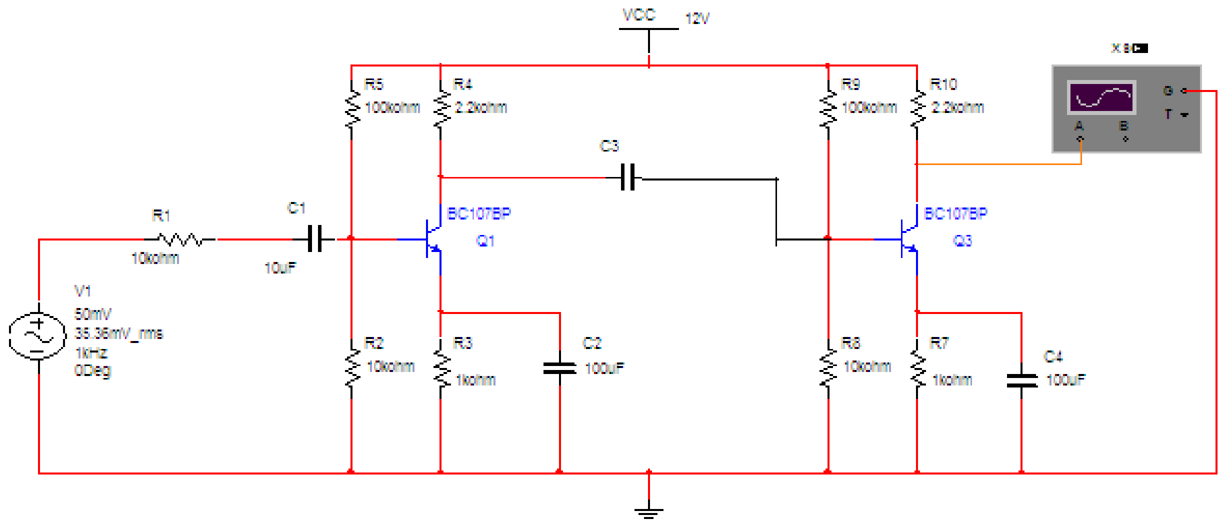
A_1=voltage gain of first stage=V_{01}/V_i

A_2=voltage gain of second stage= V_{02}/V_{01}

$$A= V_{02}/V_i$$

When ac signal is applied to the base of the transistor, its amplified output appears across the collector resistor Rc. It is given to the second stage for further amplification and signal appears with more strength. Frequency response curve is obtained by plotting a graph between frequency and gain in dB .The gain is constant in mid frequency range and gain decreases on both sides of the mid frequency range. The gain decreases in the low frequency range due to coupling capacitor C_c and at high frequencies due to junction capacitance C_{be}.

CIRCUIT DIAGRAM:



PROCEDURE:

1. Start MULTISIM. A blank circuit window will appear on the screen along with a component tool bar.
2. Using component tool bar, place all the components on the circuit window and wire the circuit.
3. Apply input 50mVp-p by using AC Voltage Source to the circuit.
4. Observe the output waveform on CRO.
5. Measure the voltage at
    a. Output of first stage
    b. Output of second stage.
6. From the readings calculate voltage gain of first stage, second stage and overall gain of two stages. Disconnect second stage and then measure output voltage of first stage calculate voltage gain.
7. Compare it with voltage gain obtained when second stage was connected.
8. Note down various values of output voltage for different frequencies from 10Hz to 1MHz from the function generator.
9. A graph is plotted between frequency and voltage gain.

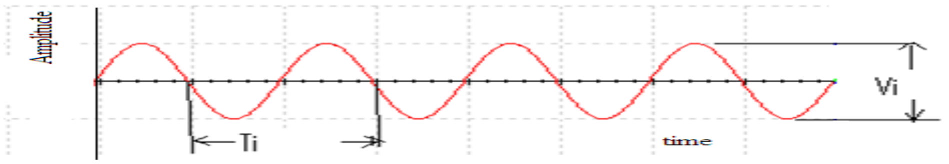
OBSERVATIONS: -

Vin = 50mVp-p

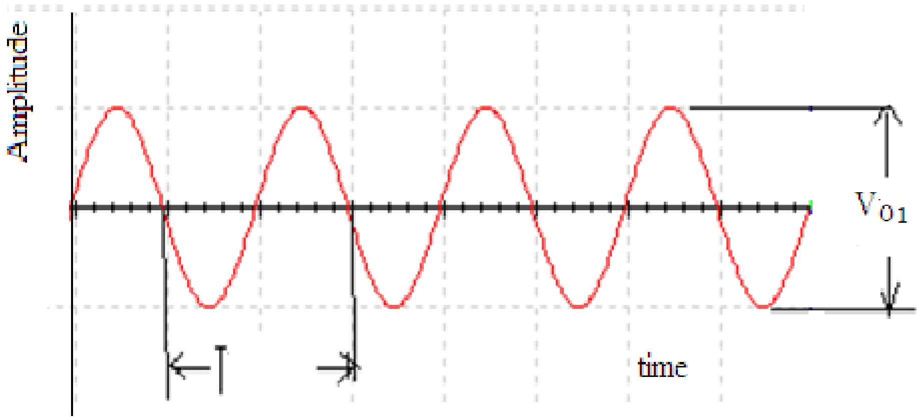
FREQUENCY(HZ)	O/P VOLTAGE (V_o)	Gain = V_o/V_{in}	VOLTAGE GAIN in dB (20 $\log_{10} V_o/V_{in}$)
10			
50			
100			
200			
400			
600			
800			
1K			
5K			
10K			
50K			
100K			
200K			
400K			
600K			
800K			
1M			

MODELGRAPHS:-

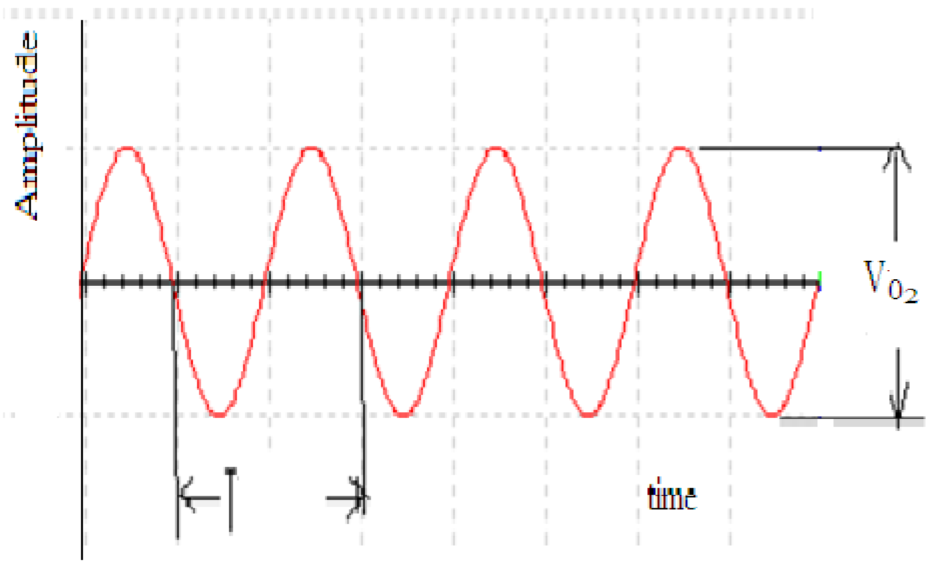
INPUT WAVE FORM:



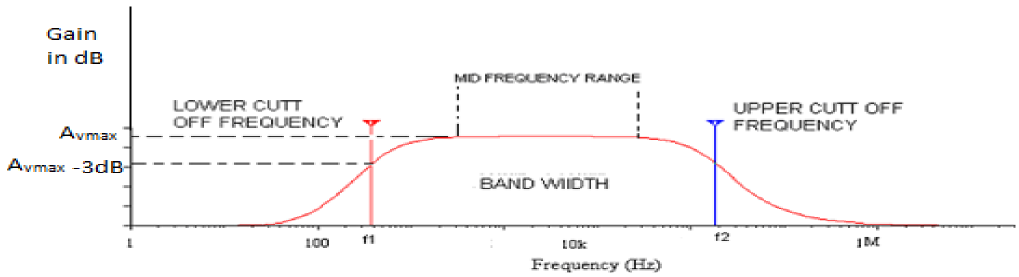
FIRST STAGE OUTPUT:



SECOND STAGE OUTPUT:



FREQUENCY RESPONSE:



PRECAUTIONS:

1) All connections are to be connected properly.
2) Transistor terminals must be identified properly.

RESULT: Thus voltage gain and bandwidth of two stage RC coupled amplifier is calculated and its frequency response is observed.

LEARNING OUTCOMES:

Students are able to

Design and analyze Two stage RC-Coupled amplifier.

VIVA QUESTIONS:

1. What is the necessity of cascading?
2. What is 3dB bandwidth?
3. Why RC coupling is preferred in audio range?
4. Which type of coupling is preferred and why?
5. Explain various types of Capacitors?
6. What is loading effect?
7. Why it is known as RC coupling?
8. What is the purpose of emitter bypass capacitor?
9. Which type of biasing is used in RC coupled amplifier?
10. What is the phase difference between the input and the output of the individual stages and for overall amplifier.
11. What is Cascode amplifier?

Exp.4 RC PHASE SHIFT OSCILLATOR

OBJECTIVE:

To calculate the frequency of oscillations of RC phase shift oscillator & to compare theoretical and practical value.

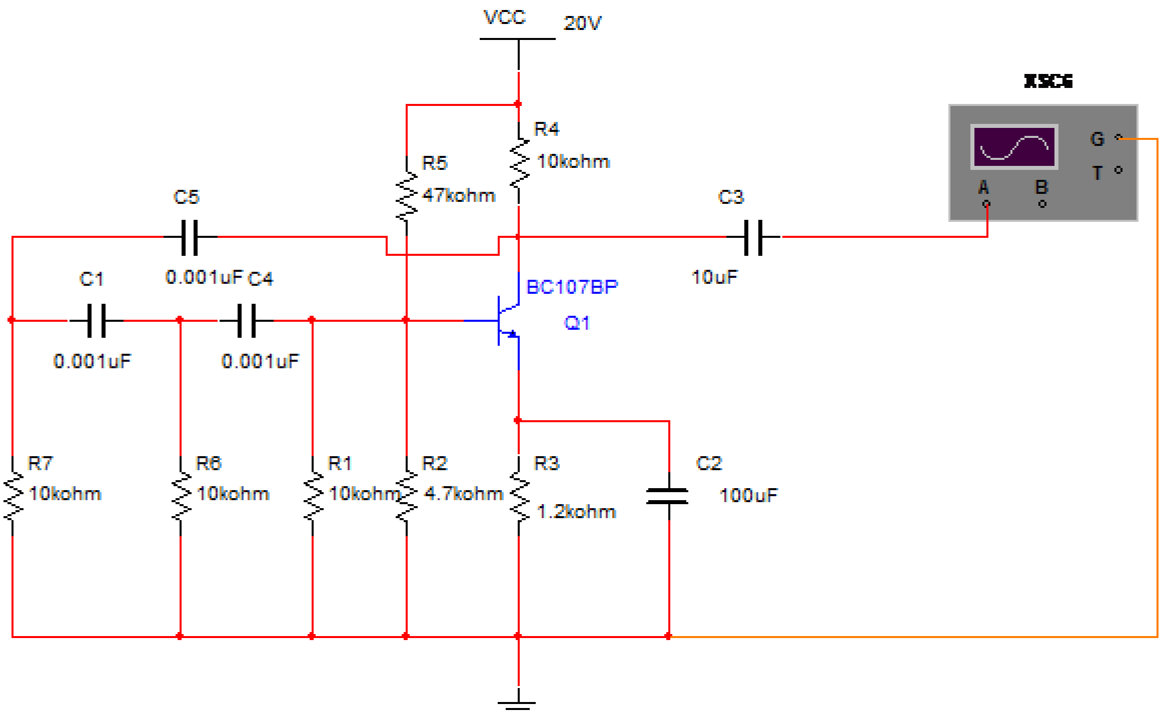
Objective:

APPARATUS:

MULTISIM 2001 SOFTWARE

PC

CIRCUIT DIAGRAM:



THEORY:

RC-Phase shift Oscillator has a CE amplifier followed by three sections of RC phase shift feedback Networks. The output of the last stage is returned to the input of the amplifier. The values of R and C are chosen such that the phase shift of each RC section is 60º.Thus The RC ladder network produces a total phase shift of 180º between its input and output voltage for the given frequencies. Since CE Amplifier produces 180 º phase shift, the total phase shift from the base of the transistor around the circuit and back to the base will be exactly 360º or 0º. This satisfies the Barkhausen condition for sustaining oscillations.Total loop gain of this circuit is greater than or equal to 1, this condition used to generate the sinusoidal oscillations.

The frequency of oscillations of RC-Phase Shift Oscillator is,

$$f = 1/2\pi RC\sqrt{6}$$

PROCEDURE:

1. Start MULTISIM. A blank circuit window will appear on the screen along with a component tool bar.
2. Using component tool bar place all the components on the circuit window and wire the circuit.
3. Observe the output signal and note down the output amplitude and time period (T_d).
4. Calculate the frequency of oscillations theoretically and verify it practically ($f=1/T_d$).
5. Repeat the above procedure for different values of R & C.

THEORITICAL CALCULATIONS:

$R = 10K\Omega$, $C = 0.001\ \mu f$

$f = 1/2\pi RC\sqrt{(6+4k)}$ where $k = R_c/R$

PRACTICAL CALCULATIONS:

T_d = time period of oscillations

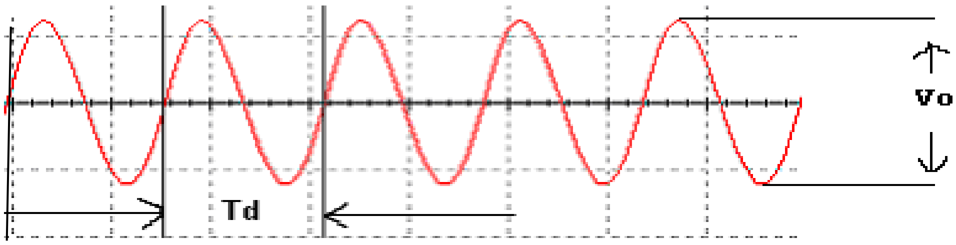
$f = 1/T_d$

Tabular column:

Resistance(Ω)	**Theoretical frequency (KHz)**	**Practical frequency (KHz)**
10k **1k** **15k** **22k**		

MODEL WAVE FORMS:

OUT PUT WAVE FORM :



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

LEARNING OUTCOMES:

Students are able to
Design and analyze RC-phase shift oscillator.

VIVA QUESTIONS:

1. What are the conditions for generating oscillations?
2. Give the formula for frequency of oscillations for RC phase shift oscillator?
3. What is the total phase shift produced by the RC ladder network?
4. Whether the RC phase shift oscillator is connected with positive feedback or negative feedback?
5. What are the different types of oscillators?
6. What is the gain of RC phase shift oscillator?
7. What is the difference between damped oscillations and undamped oscillations?
8. What are the applications of RC phase shift oscillators?
9. How many resistors and capacitors are used in RC phase shift network and why?
10. How the Barkhausen criterion is satisfied in RC phase shift oscillator?
11. What are the drawbacks of RC phase shift oscillator?

Exp.5 .WEIN BRIDGE OSCILLATOR

OBJECTIVE::

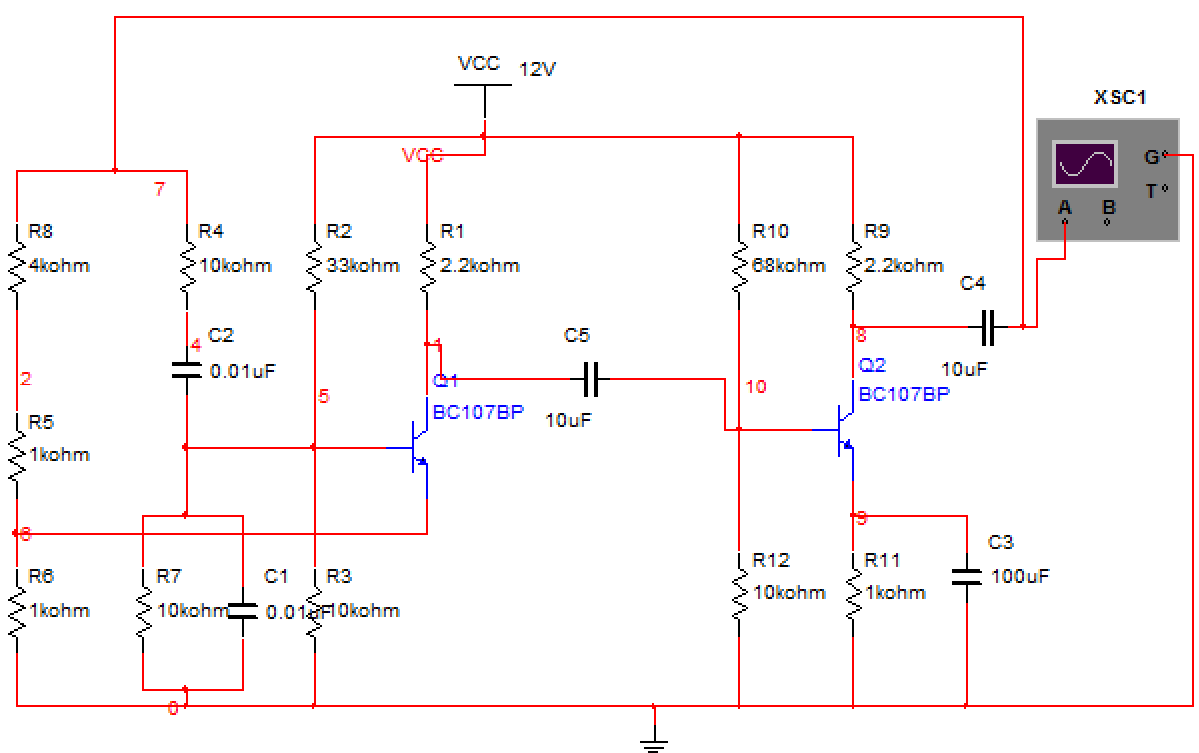
To study and calculate frequency of oscillations of Wein Bridge Oscillator and compare it with theoretical value.

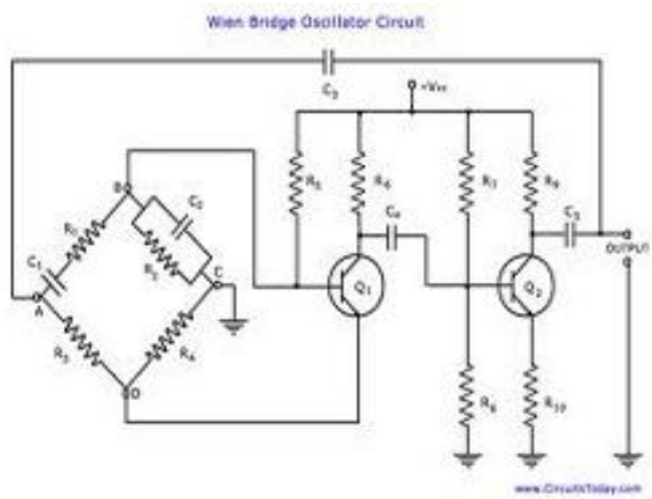
APPARATUS:

MULTISIM 2007 SOFTWARE

PC

CIRCUITDIAGRAM:





Circuit highlighting the Bridge formation.

THEORY:

The Wein Bridge oscillator is a standard circuit for generating low frequencies in the range of 10 Hz to 1MHz.The method used for getting +ve feedback in Wein Bridge oscillator is to use two stages of an RC-coupled amplifier. Since one stage of the RC-coupled amplifier introduces a phase shift of 180 deg, two stages will introduces a phase shift of 360 deg. At the frequency of oscillations *f* the +ve feedback network shown in fig makes the input & output in the phase. The formula for frequency of oscillations is given as

$$f = 1/2\pi\sqrt{R_1C_1R_2C_2}$$

If $R_1C_1 = R_2.C_2 = RC$, then $f = 1/2\pi RC$

In the above circuit, $f=1/2\pi\sqrt{R_4C_2R_7C_1}$

PROCEDURE:

1. Start MULTISIM. A blank circuit window will appear on the screen along with a component tool bar.
2. Using component tool bar place all the components on the circuit window and wire the circuit.
3. Feed the output of the oscillator to a C.R.O by making adjustments in the Potentiometer connected in the +ve feedback loop, try to obtain a stable sine wave.
4. Measure the time period of the waveform obtained on CRO. & calculate the Frequency of oscillations.
5. Repeat the procedure for different values of capacitances.

OBSERVATION:

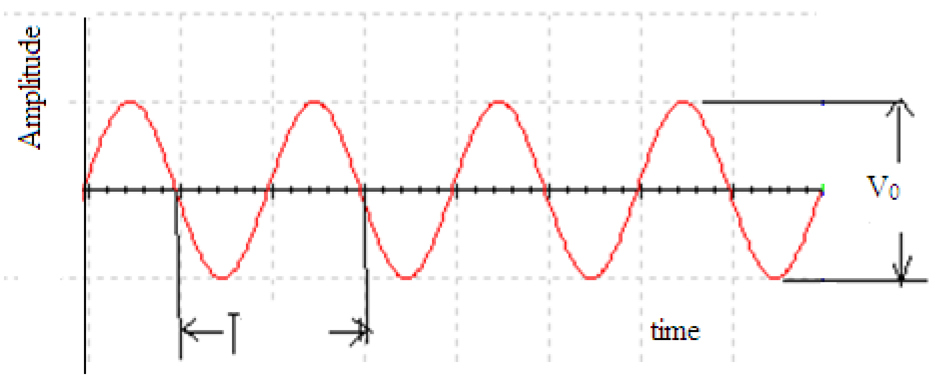
Given R=10kΩ, C=0.01µF

$\mathbf{f_T} = 1/ 2\pi RC$

$\mathbf{f_P} = \frac{1}{T} =$

Amplitude, V_0 =

MODEL WAVE FORMS:



RESULT:

The frequency of oscillations of the Wein Bridge oscillator is calculated and is verified.

LEARNING OUTCOMES:

Students are able to
Design and analyze Wein Bridge oscillator.

VIVA QUESTIONS:

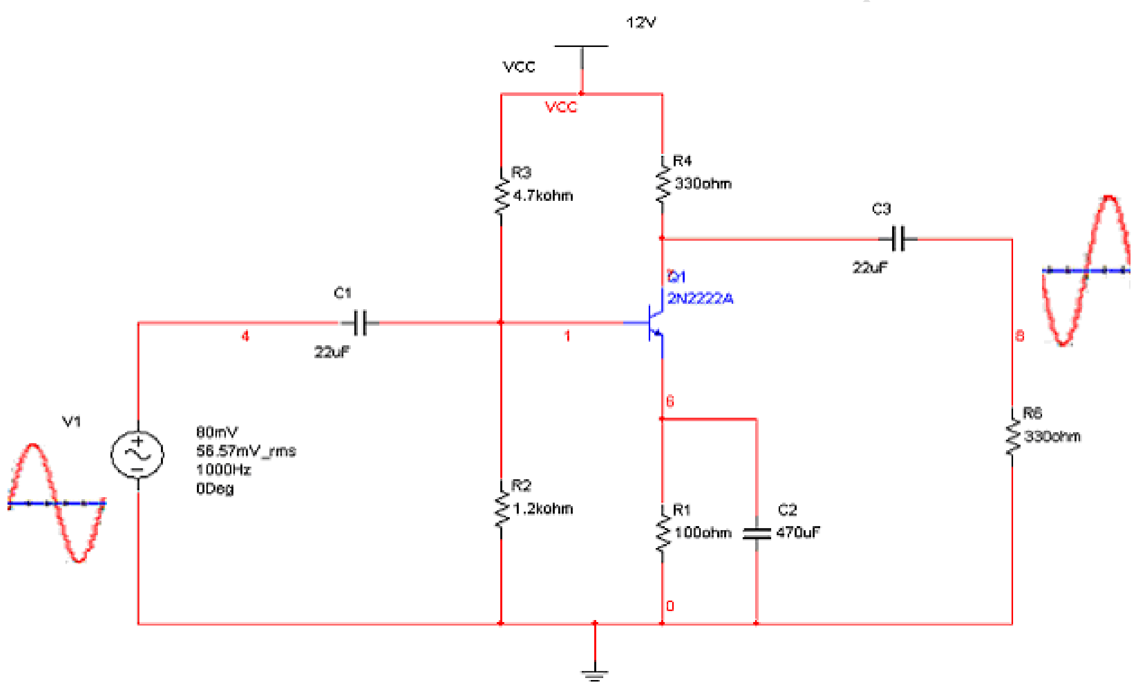
1. Give the formula for frequency of oscillations in Wein Bridge Oscillator circuit?
2. What is the condition for Wien Bridge oscillator to generate oscillations?
3. What is the total phase shift provided by the Wein Bridge oscillator?
4. What is the function of lead-lag network in Wein Bridge oscillator?
5. Which type of feedback is used in Wein Bridge oscillator
6. What is the gain of Wein Bridge oscillator?
7. What are the applications of Wein Bridge oscillator
8. What is the condition for generating oscillations?
9. What is the difference between damped oscillations undamped oscillations?
10. Wein Bridge oscillator is either LC or RC oscillator.
11. What are the drawbacks in using Wein Bridge Oscillators?

EXP.6 . CLASS A POWER AMPLIFIER

OBJECTIVE : To determine efficiency of class A power amplifier.

APPARATUS REQUIRED: MULTISIM 2007

CIRCUIT DIAGRAM:



THEORY:

The function of power amplifier is to raise the power level of input signal. Class A power amplifier is one in which the output current flows during the entire cycle of input signal. Thus the operating point is selected in such a way that the transistor operates only over the linear region of its load line. So this amplifier can amplify input signal of small amplitude. As the transistor operates over the linear portion of load line the output wave form is exactly similar to the input wave form. Hence this amplifier is used where freedom from distortion is the prime aim.

PROCEDURE:

1. Select different components and place them in the grid.
2. Apply the input ac signal voltage of 160mV (p-p) and simulate the circuit.

3. Observe the output wave form on CRO and measure the output voltage V_0.
4. Now connect the ammeter at collector terminal of transistor.
5. Disconnect the ac signal from input and measure the collector current Ic in ammeter.
6. calculate the efficiency by using practical calculations compare it with theoretically calculated efficiency

OBSERVATION :

THEORITICAL CALCULATIONS :

$$I_{CQ} = \frac{V_{CC}/R_L}{2}$$

$$I_{CQ} = \frac{I_C}{2}$$

$$P_{in}(dc) = \frac{V_{CC} * V_{CC}}{2R_L} = \frac{V_{CC}^2}{2R_L}$$

$$Po(a.c) = \frac{(V_{max} - V_{min}) * (I_{max} - I_{min})}{8}$$

$$(I_{max} - I_{min}) = \frac{V_{CC}}{R_L}$$

$$(V_{max} - V_{min}) = V_{CC}$$

Po(a.c) = $\frac{V_{CC}}{8R_L}$ = $\frac{{V_{cc}}^2}{8R_L}$

% of efficiency = $\frac{P_o(ac)}{P_{in}(dc)}$*100= $\frac{{V_{CC}}^2/8R_L}{{V_{CC}}^2/2R_L}$ * 100=25%

PRACTICAL CALCULATIONS :

I_C =

Pin(d.c) = V_{CC}*I_{CQ} =

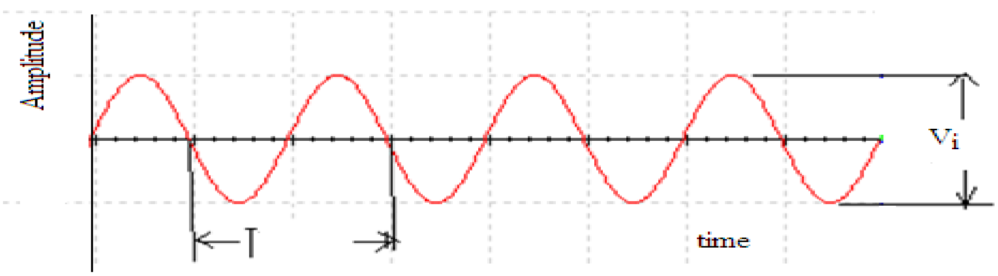
Po(a.c) = $\frac{{V_0}^2}{8R_L}$ =

% of efficiency = $\frac{Po(ac)}{P_{in(dc)}}$ *100 =

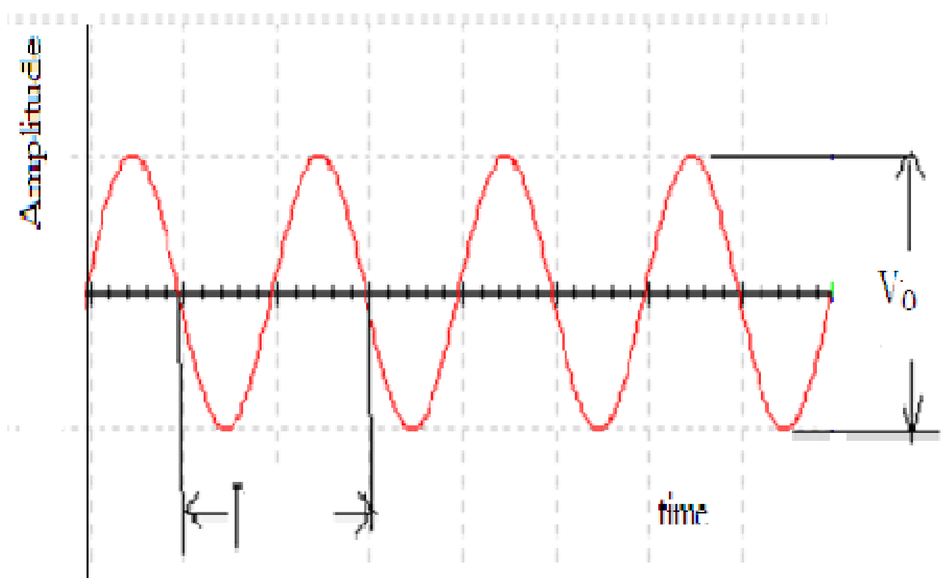
OBSERVATIONS:

	OUTPUT VOLTAGE (V_0)	Gain = Vo/Vin	GAIN IN dB A_v=20 $\log_{10}$ (V_0/V_i)

WAVEFORMS:



OUTPUT WAVEFORMS:



RESULT: The efficiency of class A Power amplifier is verified.

LEARNING OUTCOMES:

Students are able to
Design and analyze class A power amplifier.

VIVA QUESTIONS:

1. Define class A power amplifier?
2. Give the reason why class A power amplifier is called as directly coupled power amplifier?
3. What is the efficiency of class A power amplifier?
4. In a power transistor, when the maximum power dissipation takes place?
5. List out the different types of distortions?
6. Define Harmonic distortion?
7. What are Class B, Class C and Class AB amplifiers and which type is used for what application?

HARDWARE EXPERIMENTS

Exp 1. SINGLE TUNED VOLTAGE AMPLIFIER

OBJECTIVE: To plot the frequency response of a single tuned voltage amplifier.

APPARATUS :

Transistor BC107 - 1 No.

Resistors 33kΩ -1No.

56kΩ -1No.

100kΩ -1No.

560Ω -1No.

Capacitors 0.1µf -1No.

1µf 2 Nos.

Inductor 100 mH -1No.

CRO(Dual Channel) (0-20 MHz)

Function generator (1Hz to 1 MHz)

Regulated power supply (0-30V)

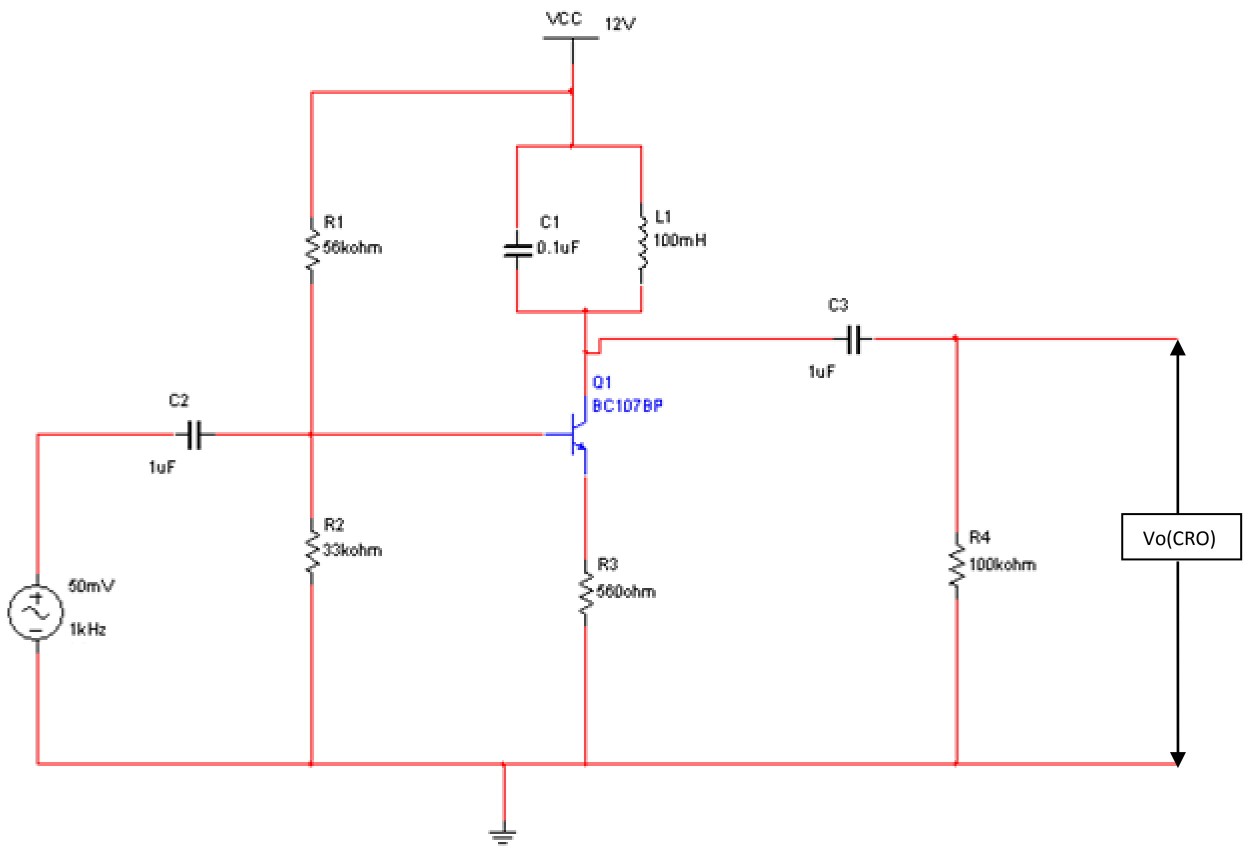
Breadboard

Connecting wires.

THEORY:

The signal to be amplified and it is applied between the terminals base and emitter. The tank circuit (i.e L & R) is located at collector terminal and it may be varied in such a way that the resonant frequency becomes equal to the frequency of the input signal. At resonant the tuned circuit offers high impedance and thus given input signal is amplified and thus appears with large value across it and other frequencies will be rejected. The response of the tuned amplifiers falls sharply below and above the resonant frequency. So the tuned circuit selects the desired frequency and rejects all other frequencies.

CIRCUIT DIAGRAM:



PROCEDURE:

1. Connections are to be made as per the circuit diagram on bread board properly.

2. Set input signal v_i=50mVusing the Signal Generator.

3. Keep the input voltage constant and vary the frequency from 10 Hz to 100kHz in steps as shown in the tabular column and note the output voltage.

4. Plot the graph of voltage gain in dB V_S frequency.

5. Calculate the bandwidth from the graph.

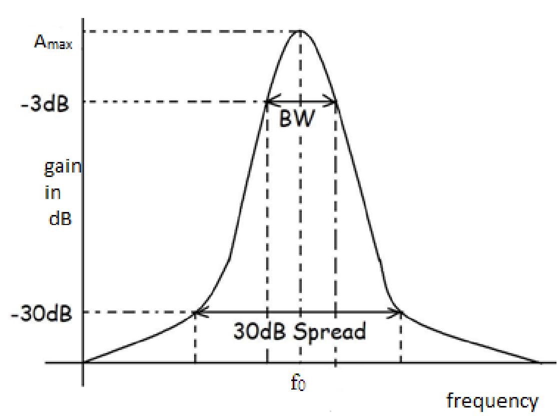
OBSERVATIONS:

Tabular form:

frequency	Gain=(V_o/V_i)	Gain in dB= 20log(gain)
10Hz		
100Hz		
500Hz		
1kHz		
1.2khz		
1.22kHz		
1.24kHz		
1.26kHz		
1.28kHz		
1.30kHz		
1.32kHz		
1.34kHz		
1.36kHz		
1.38kHz		
1.40kHz		
1.42kHz		
1.44kHz		
1.46kHz		
1.5kHz		
2kHz		
10kHz		
100kHz		

(the values for the above designed circuit will be obtained from 1.2 or 1.30KHz to 1.40KHz . before or after this values the output will not be seen .)

Frequency response:



THEORITICAL CALCULATIONS:

$L=100mH$, $C=0.1\mu F$

$f_o=1/2\pi\sqrt{(LC)}$

RESULT:

Thus the frequency response of single tuned voltage amplifier has been studied and its bandwidth has been calculated.

LEARNING OUTCOMES:

Students are able to
Design and analyze single tuned amplifier.

VIVA QUESTIONS:

1) What do you mean by tuned amplifiers?
2) What are advantages and disadvantages of tuned amplifiers?
3) What are the applications of tuned amplifiers?
4) Define resonant frequency?
5) Define Quality factor?

Exp.2 a). HARTLEY OSCILLATOR

OBJECTIVES: To study and calculate frequency of oscillations of Hartley oscillator. Compare the frequency of oscillations, theoretically and practically.

APPARATUS:

Transistor BC 107 -1 No.

Capacitors 0.1μF, 10 μF -1 No. each

0.22 μF -2 Nos

Resistors

47Kohm, 1Kohm and 10Kohm -1 No.

Decade inductance box (DIB) -1 No.

Decade resistance box (DRB) -1 No.

Inductors : 5mH – 2 Nos.

CRO(Dual Channel) (0-20 MHz)

Function generator (1Hz to 1 MHz)

Regulated power supply (0-30V)

Bread board

Connecting wires

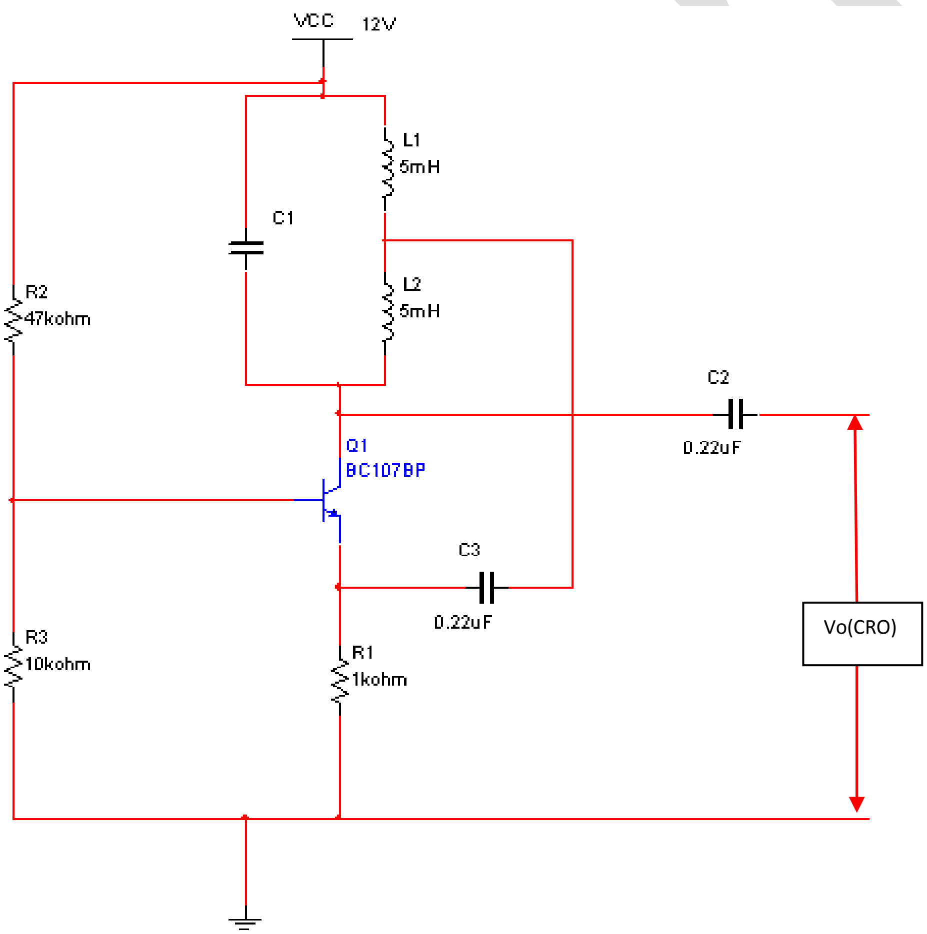
THEORY:

Hartley oscillator is very popular and is commonly used as a local oscillator in radio receivers. It has two main advantages viz… Adaptability to wide range of frequencies and easy to tune. The tank circuit is made up of L1, L2, and C1. The coil L1 is inductively coupled to coil L2, the combination of two functions acts as auto transformer. The resistances R2 and R3 provide the necessary biasing. The capacitance C2 blocks the d.c component. The frequency of oscillations is determined by the values of L1, L2 and C1 and is given by,

$$F=1/(2\pi(C_1(\sqrt{L_1}+L_2)))$$

The energy supplied to the tank circuit is of correct phase. The auto transformer (L1 and L2 forms as an auto transformer) provides 180˚ out of phase. Also another 180˚ is produced by the transistor. In this way, energy feedback to the tank circuit is in phase with the generated oscillations

CIRCUIT DIAGRAM:



PROCEDURE:

1. Connections are to be made as per the circuit diagram properly (C1 value to be selected as 0.1µF and10µF as required).

2. Connect CRO at output terminals and observe the wave form.

3. Calculate practically the frequency of oscillations by using the expression.

f=1/T, Where T= Time period of the waveform

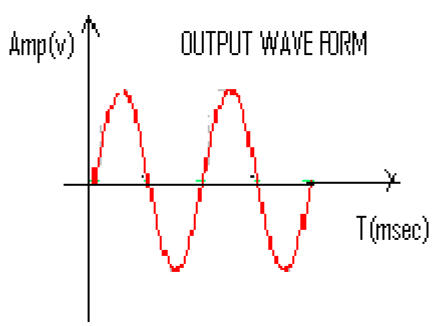
4. Repeat the above steps 2, 3 for different values of L1 and note down practical values of oscillations of Hartley oscillator.

5. Compare the values of frequency of oscillations both theoretically and Practically.

OBSERVATIONS:

CAPACITANCE(µF)	**Theoretical frequency (KHz)**	**Practical frequency (KHz)**

MODEL GRAPH:



PRECAUTIONS:

1. All the connections are to be connected properly.
2. Transistor terminals must be identified properly.

3. Reading should be taken without any parallax error.

RESULT: Frequency of oscillations is calculated and compared with theoretical values.

LEARNING OUTCOMES:

Students are able to
Design and analyze Hartley oscillator

Viva questions:

1. What are the applications of LC oscillator?
2. What type of feedback is used in oscillators?
3. What is the loop gain of an oscillator?
4. What is the difference between amplifier and oscillator?
5. What is the condition for oscillations?

Exp. 2 b).COLPITT'S OSCILLATOR

OBJECTIVE: To study and calculate frequency of oscillations of Colpitt's oscillator.

APPARATUS:

Transistor BC 107 -1 No.
Capacitors 0.1μF - 1 No.
0.01μF - 2 Nos

Inductor 5mH - 1 No.

Resistors 47KOhm, 1kΩ, 10kΩ, 1.5K -1 No.

Decade inductance box

CRO(Dual Channel) (0-20 MHz)

Function generator (1Hz to 1 MHz)

Regulated power supply (0-30V)

Connecting Wires

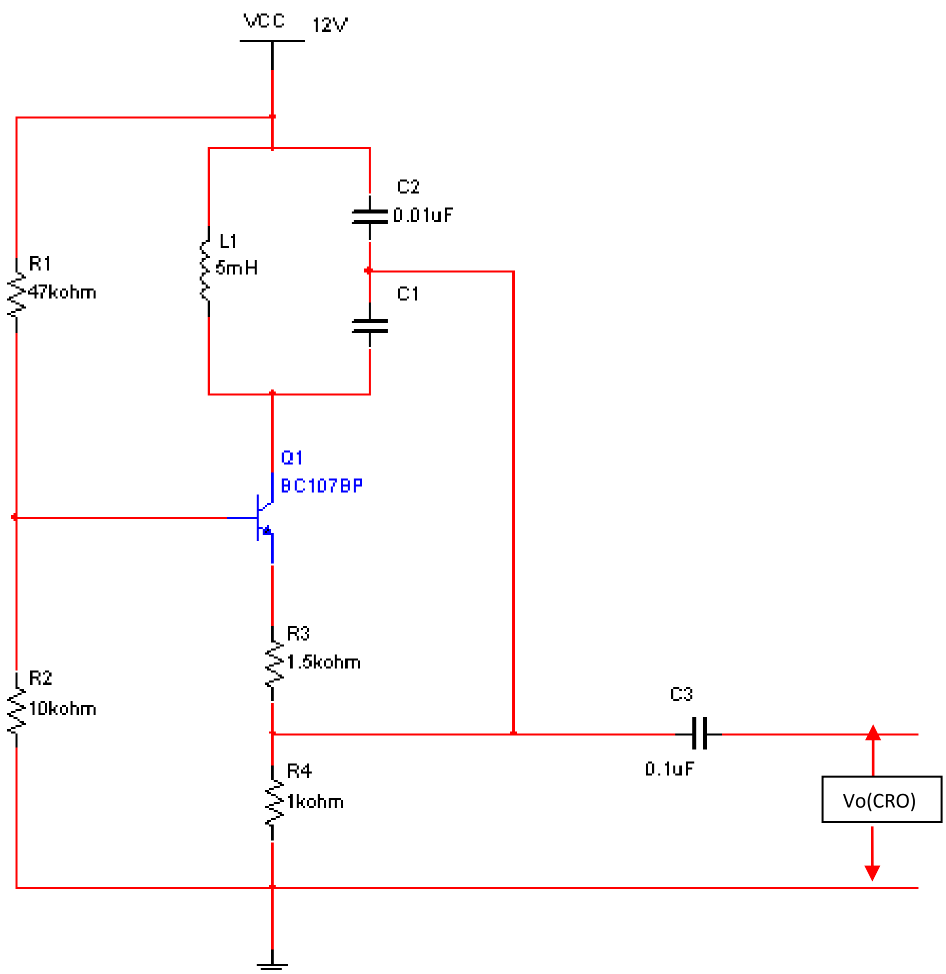
THEORY:

The tank circuit is made up of L_1,C_1 and C_2 .The resistance R_2 and R_3 provides the necessary biasing. The capacitance C_3 blocks the D.C component. The frequency of oscillations is determined by the values of L_1,C_4 and C_5, and is given by

$$f = 1 / (2\pi (C_T L1)^{1/2}) \text{ Where } C_T = C_1 C_2 / (C_1 + C_2)$$

The energy supplied to the tank circuit is of in phase. The tank circuit provides 180^0 out of phase. Also the transistor provides another 180^0. In this way, energy feedback to the tank circuit is in phase with the generated oscillations.

CIRCUITDIAGRAM:



PROCEDURE:

1. Connections are to made as per circuit diagram.
2. Connect CRO output terminals and observe the waveform.
3. Calculate practically the frequency of oscillations by using the expression

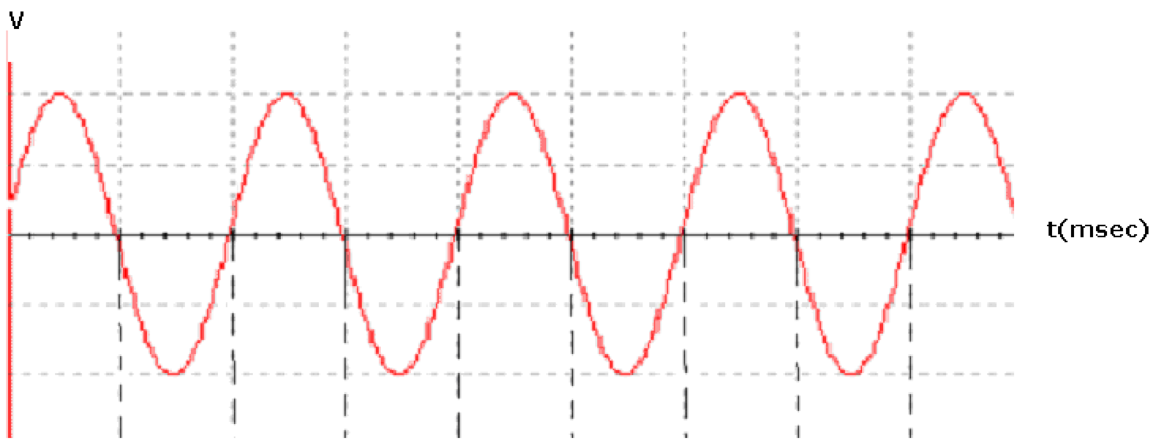
   f = 1 / T (T= Time period of the waveform)

4. Repeat the above steps 2,3 for different values of L, and note down the practical values of oscillations of the Collpitt's oscillator.
5. Compare the values of oscillations both theoretically and practically.

OBSERVATIONS:

Inductance (mH)	Theoretical Frequency (Hz)	Practical Frequency (Hz)

MODELWAVEFORM:



PRECAUTIONS:

1. The connections are to be connected properly.
2. Transistor terminals should be identified properly.
3. Readings should be taken without parallax error.

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

LEARNING OUTCOMES:

Students are able to
Design and analyze Colpitts oscillator.

VIVA QUESTIONS:

1. What are the applications of LC oscillator?
2. What type of feedback is used in oscillators?
3. What is the loop gain of an oscillator?
4. What is the difference between amplifier and oscillator?
5. What is the condition for oscillations?

ADDITIONAL EXPERIMENT

MOS AMPLIFIER

OBJECTIVE: To study the frequency response of MOS amplifier.

Apparatus: Transistor TSC7000K-1 No.

Capacitors 10µF - 2 Nos
100µF - 1 No
Resistors 100kΩ, -2 Nos
47kΩ, 220kΩ, 10kΩ -1 No.
100Ω -2 Nos

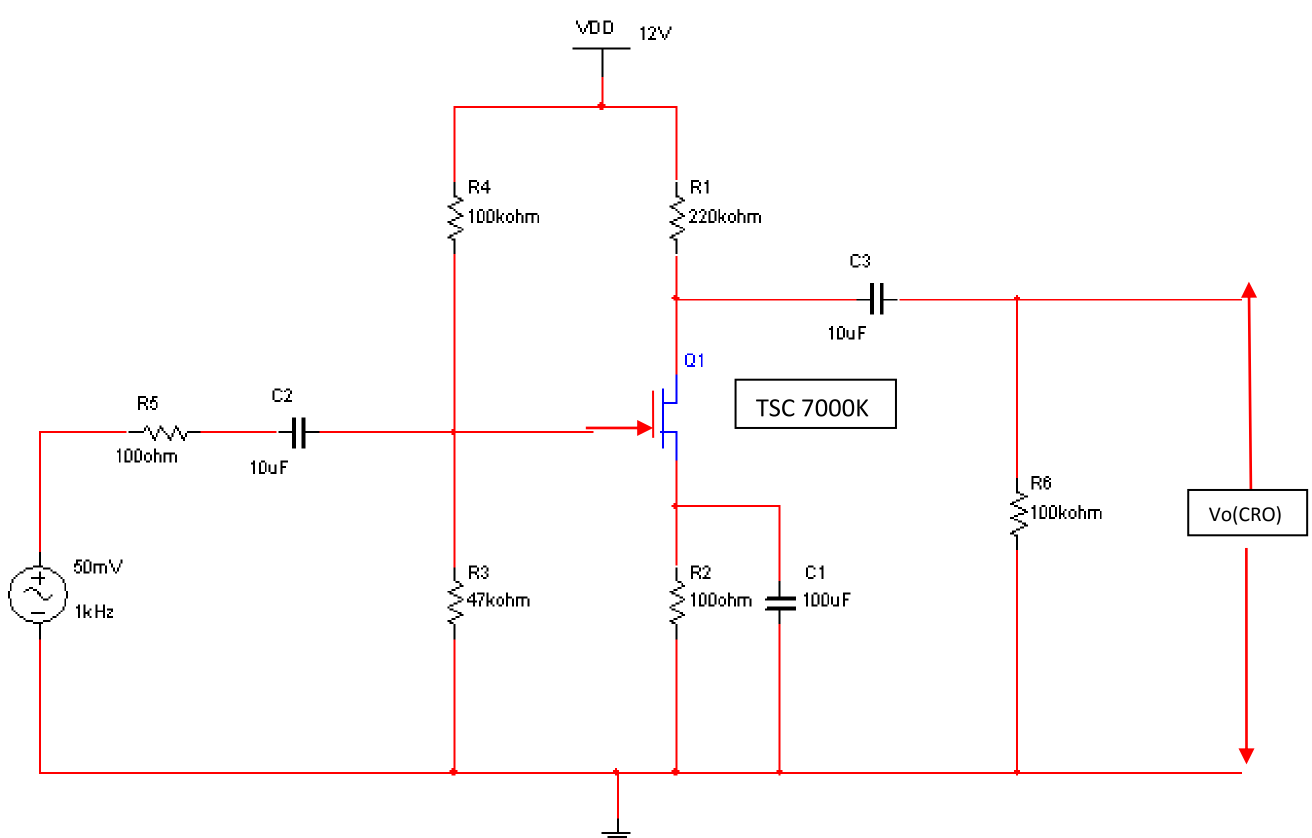
CRO(Dual Channel) (0-20 MHz)

Function generator (1Hz to 1 MHz)

Regulated power supply (0-30V)

Connecting Wires

CIRCUIT DIAGRAM:



THEORY:

MOSFET belongs to the family of FET. Its abbreviation is Metal Oxide Semiconductor Field Effect Transistor. The control terminal is called the gate. Remember that the base terminal of a bipolar transistor passes a small amount of current. The gate on the FET passes virtually no current when driven with D.C. When driving the gate with high frequency pulsed D.C. or A.C. there may be a small amount of current flow. The transistor's "turn on" (threshold) voltage varies from one FET to another but is approximately 3.3 volts with respect to the source. When FETs are used in the audio output section of an amplifier, the V_{GS} (voltage from gate to source) is rarely higher than 3.5 volts. When FETs are used in switching power supplies, the V_{GS} is usually much higher (10 to 15 volts). When the gate voltage is above approximately 5 volts, it becomes more efficient (which means less voltage drop across the FET and therefore less power dissipation).

MOSFETs are commonly used in electronics circuitry because they are easier to drive in high current applications (such as the switching power supplies found in car audio amplifiers). If a bipolar transistor is used, a fraction of the collector/emitter current must flow through the base junction. In high current situations where there is significant collector/emitter current, the base current may be significant. FETs can be driven by very little current (compared to the bipolar transistors). The only current that flows from the drive circuit is the current that flows due to the capacitance. As you already know, when DC is applied to a capacitor, there is an initial surge then the current flow stops. When the gate of an FET is driven with a high frequency signal, the drive circuit essentially sees only a small value capacitor. For low to intermediate frequencies, the drive circuit has to deliver little current. At very high frequencies or when many FETs are being driven, the drive circuit must be able to deliver more current.

Note:

The gate of a MOSFET has some capacitance which means that it will hold a charge (retain voltage). If the gate voltage is not discharged, the FET will continue to conduct current. This doesn't mean you can charge it and expect the FET to continue to conduct indefinitely but it will continue to conduct until the voltage on the gate is below the threshold voltage. You can make sure it turns off if you connect a pull down resistor between the gate and source.

High Current Terminals:

The 'controlled' terminals are called the source and the drain. These are the terminals responsible for conducting the current through the transistor.

PROCEDURE:

1. Connections are to be made as per the circuit diagram on the breadboard.
2. A signal of 1 KHz frequency and 50mV peak-to-peak is applied at the input of amplifier.
3. Output is taken at drain and gain is calculated by using the expression,

$$A_v = V_0/V_i$$

4. Voltage gain in dB is calculated by using the expression,

$$A_v = 20\log 10(V_0/V_i)$$

5. Repeat the above steps for various input frequencies from 10 Hz to 1MHz in semilog scale steps.
6. Plot A_v vs. Frequency
7. The Bandwidth of the amplifier is calculated from the graph using the expression,

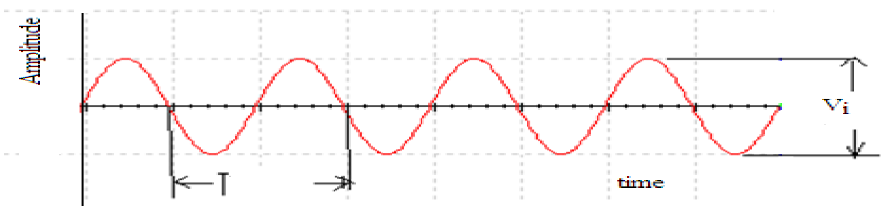
Bandwidth $BW = f_2 - f_1$

Where f_1 is lower 3 dB frequency

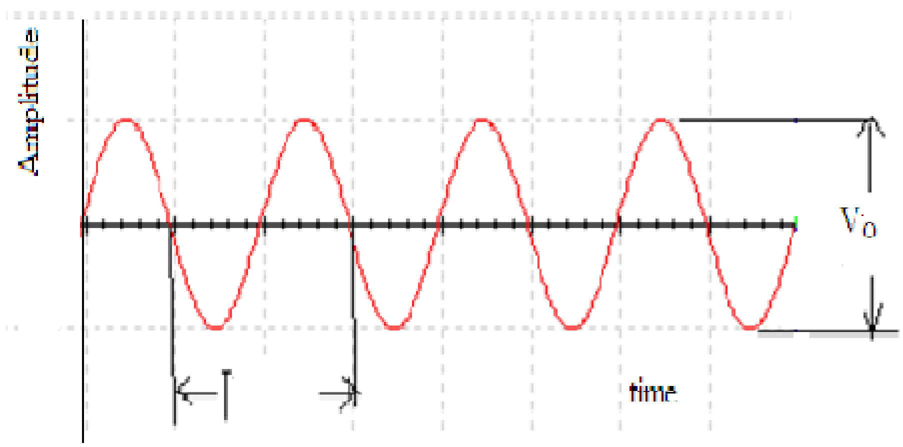
f_2 is upper 3 dB frequency

WAVEFORMS:

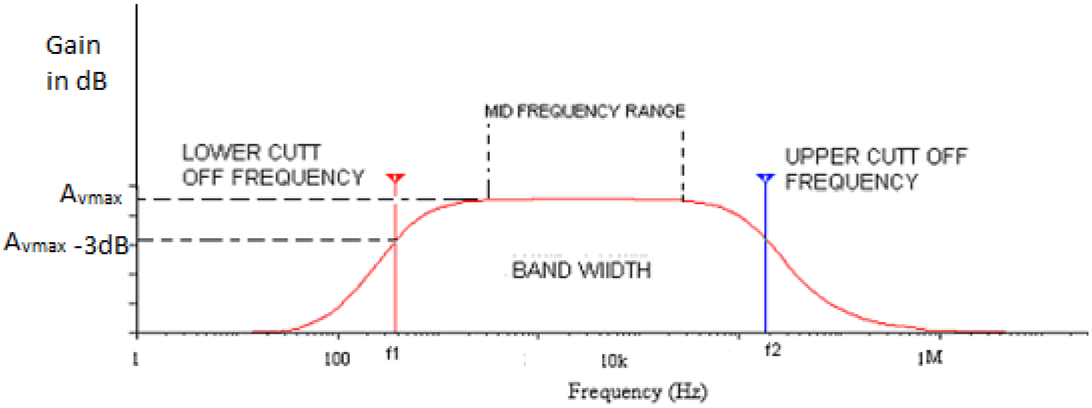
INPUT WAVEFORMS:



OUTPUTWAVE FORM:



FREQUENCY RESPONSE:



OBSERVATIONS:
Input voltage: 50 mV

S.NO	Frequency(Hz)	OUTPUT VOLTAGE(V_0)	VOLTAGE GAIN $A_v= (V_0/V_i)$	VOLTAGE GAIN in dB ($20 \log_{10} V_o/V_{in}$)
	10			
	50			
	100			
	200			
	400			
	600			
	800			
	1K			
	5K			
	10K			
	50K			
	100K			
	200K			
	400K			
	600K			
	800K			
	1M			

RESULT: The voltage gain and frequency response of the MOS amplifier are obtained. Also gain bandwidth product of the amplifier is calculated.

LEARNING OUTCOMES:

Students are able to
Design and analyze MOS amplifiers.

VIVA QUESTIONS:

1) What are the different types of MOSFETS?
2) How is MOSFET different from JFET?
3) What are the advantages of MOSFET?
4) What are the difference between depletion mode MOSFET and enhancement mode MOSFET ?

DESIGN EXPERIMENT

EXP .NO. 1 SHUNT VOLTAGE REGULATOR

OBJECTIVES :

To design a transistorized shunt voltage regulator and observing the regulation action for

i. Different values of input voltages
ii .Different values of load resistors and also to find percentage regulation.

EQUIPMENTS AND COMPONENTS:

APPARATUS REQUIRED: MULTISIM 2007 SOFTWARE

PC

THEORY:

A voltage regulator is a device or a combination of devices, designed to maintain the output voltage of a power supply as nearly constant as possible even if there are changes in the load or in input voltage. In shunt voltage regulator, transistor Q1 acts as control element, which is in shunt with load voltage.

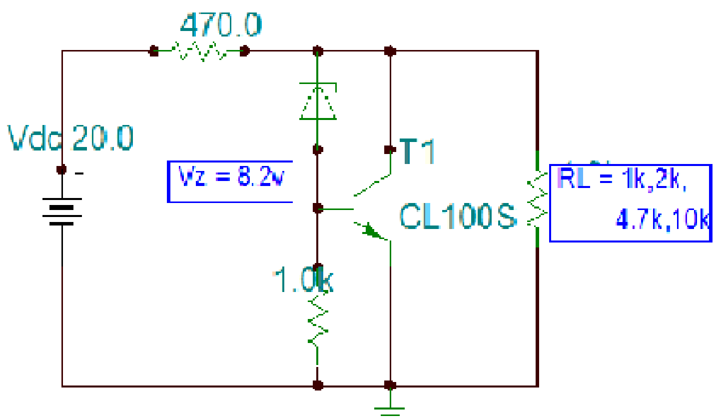
The output voltage is given as

$$Vo = Vz + V_{R1} = Vz$$

The regulation action of the circuit is explained below.

Since V_z is constant, any changes in output voltage reflects a propositional change in R1. If the output voltage decreases, voltage across R1 decreases which in turn decreases the base voltage of . As a result the base current of Q1 decreases which allows the load voltage to rise and makes it constant the same regulation action follows even if the output voltage increases.

CIRCUIT DIAGRAM:



PROCEDURE:

i. Connect the circuit as shown in the circuit diagram on bread board..
i. Apply the input voltage from power supply.
ii. For a specific value of load resistor, vary the input voltage from zero to a maximum of 20 Volts and note the values of output voltage.
iv. Change the load resistor and repeat steps 2 and 3.
v. Remove the load resistor and note down the voltage at no load.
vi. Find percentage regulation.

$$\text{Percentage load regulation} = \frac{V_{NL} - V_{FL}}{V_{FL}} x100$$

vii. Plot the graph for load regulation and line regulation.

OBSERVATIONS:

VOLTAGE AT NO-LOAD =

S.NO	V_{IN}	OUTPUT VOLTAGE		
		R_L=	R_L=	R_L=

CALCULATIONS:

$$\text{Percentage regulation} = \frac{V_{NL} - V_{FL}}{V_{FL}} x100$$

RESULT: Thus the shunt voltage regulator is designed.

What about line regulation??

LEARNING OUTCOMES:

Students are able to

design and analyze shunt voltage regulator .

PRECAUTIONS:

i. Proceed on the experiment only after obtaining expected DC voltages. Do not apply more than 20 V without connecting load on the output as this would result in maximum current in shunt transistors.

ii. Shorting the output will result in overheating series resistors which may burn at high voltage.

iii. Reversing the Zener may not damage the circuit but result in output voltage to drop 2 V or less.

APPLICATIONS:

1. Low current applications.
2. Fixed voltage applications

VIVA QUESTIONS:

1. Define the line regulation?
2. Define load regulation?
3. Mention the applications of shunt voltage regulator?
4. Define a voltage regulator?