Electronic Circuits and Pulse Circuits

LAB MANUAL

Academic Year	**:**	**2017 - 2018**
Course Code	**:**	**AEC102**
Regulations	**:**	**IARE - R16**
Class	**:**	**IV Semester**
Branch	**:**	**ECE**

Prepared By

Mr. K Ravi, Assistant Professor, ECE
Mr. N Nagaraju, Assistant Professor, ECE
Mr. C Srihari, Assistant Professor, ECE
Ms. N Anusha, Assistant Professor, ECE
Ms. P Saritha, Associate Professor, ECE
Mr. B Naresh, Associate Professor, ECE

Department of Electronics & Communication Engineering

INSTITUTE OF AERONAUTICAL ENGINEERING
(AUTONOMUS)
Dundigal, Hyderabad – 500 043

INSTITUTE OF AERONAUTICAL ENGINEERING

(Autonomous)

Dundigal, Hyderabad - 500 043

Electronic Ciruits & Pulse Circuits Lab

WORK BOOK

Name of the Student			
Roll No.			
Branch			
Class		**Section**	

INSTITUTE OF AERONAUTICAL ENGINEERING

(Autonomous)

Dundigal, Hyderabad - 500 043

Vision

To bring forth professionally competent and socially sensitive engineers, capable of working across cultures meeting the global standards ethically.

Mission

To provide students with an extensive and exceptional education that prepares them to excel in their profession, guided by dynamic intellectual community and be able to face the technically complex world with creative leadership qualities.

Further, be instrumental in emanating new knowledge through innovative research that emboldens entrepreneurship and economic development for the benefit of wide spread community.

Quality Policy

Our policy is to nurture and build diligent and dedicated community of engineers providing a professional and unprejudiced environment, thus justifying the purpose of teaching and satisfying the stake holders.

A team of well qualified and experienced professionals ensure quality education with its practical application in all areas of the Institute.

Philosophy

The essence of learning lies in pursuing the truth that liberates one from the darkness of ignorance and Institute of Aeronautical Engineering firmly believes that education is for liberation.

Contained therein is the notion that engineering education includes all fields of science that plays a pivotal role in the development of world-wide community contributing to the progress of civilization. This institute, adhering to the above understanding, is committed to the development of science and technology in congruence with the natural environs. It lays great emphasis on intensive research and education that blends professional skills and high moral standards with a sense of individuality and humanity. We thus promote ties with local communities and encourage transnational interactions in order to be socially accountable. This accelerates the process of transfiguring the students into complete human beings making the learning process relevant to life, instilling in them a sense of courtesy and responsibility.

INSTITUTE OF AERONAUTICAL ENGINEERING

(Autonomous)

Dundigal, Hyderabad - 500 043

Certificate

This is to Certify that it is a bonafied record of Practical work done by Sri/Kum.________________________ bearing the Roll No. _______________________ of _____________ Class ____ Branch in the ___________________________laboratory during the Academic year __________under our supervision.

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

External Examiner **Internal Examiner**

COURSE OVERVIEW

This laboratory course builds on the lecture course "Electronic circuit analysis" and “pulse and digital circuits” which is mandatory for all students of electronics and communication engineering. The course aims at practical experience with the characteristics and theoretical principles of linear and non linear devices and pulse circuits.

OBJECTIVE

1. To understand different amplifier circuits.
2. To understand different oscillating circuits.
3. To indentify the linear and non linear wave shaping.
4. To observe the applications of diodes like clippers and clampers..
5. To analyze the switching characteristics of transistor. .
6. To design and illustrate the characteristics of multivibrators.

COURSE OUT COMES

1. **Analyze** and **Design** various amplifiers
2. **Analyze** and **Design** various oscillators.
3. **Analyze** the RC circuit characteristics.
4. **Analyze** the diode and transistor applications.
5. **Create** the different oscillations and timing circuits using multivibrators.
6. **Identify** the applications of UJT.

Electronics & Communication Engineering

INSTRUCTIONS TO THE STUDENTS

1. Students are required to attend all labs.
2. Students should work individually in the hardware and software laboratories.
3. Students have to bring the lab manual cum observation book, record etc along with them whenever they come for lab work.
4. Should take only the lab manual, calculator (if needed) and a pen or pencil to the work area.
5. Should learn the prelab questions. Read through the lab experiment to familiarize themselves with the components and assembly sequence.
6. Should utilize 3 hour's time properly to perform the experiment and to record the readings. Do the calculations, draw the graphs and take signature from the instructor.
7. If the experiment is not completed in the stipulated time, the pending work has to be carried out in the leisure hours or extended hours.
8. Should submit the completed record book according to the deadlines set up by the instructor.
9. For practical subjects there shall be a continuous evaluation during the semester for 30 sessional marks and 70 end examination marks.
10. Out of 30 internal marks, 20 marks shall be awarded for day-to-day work and 10 marks to be awarded by conducting an internal laboratory test.

ELECTRONICS & COMMUNICATION ENGINEERING

Program Outcomes	
PO1	**Engineering knowledge**: Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.
PO2	**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.
PO3	**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.
PO4	**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.
PO5	**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.
PO6	**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.
PO7	**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.
PO8	**Ethics:** Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.
PO9	**Individual and Team Work**: Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.
PO10	**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.
PO11	**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 and in multidisciplinary environments.
PO12	**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.
Program Specific Outcomes	
PSO1	**Professional Skills:** An ability to understand the basic concepts in Electronics & Communication Engineering and to apply them to various areas, like Electronics, Communications, Signal processing, VLSI, Embedded systems etc., in the design and implementation of complex systems.
PSO2	**Problem-solving skills:** An ability to solve complex Electronics and communication Engineering problems, using latest hardware and software tools, along with analytical skills to arrive cost effective and appropriate solutions.
PSO3	**Successful career and Entrepreneurship:** An understanding of social-awareness & environmental-wisdom along with ethical responsibility to have a successful career and to sustain passion and zeal for real-world applications using optimal resources as an Entrepreneur.

ELECTRONIC CIRCUITS AND PULSE CIRCUITS LAB SYLLABUS

Recommended Systems/Software Requirements:

Intel based desktop PC with minimum of 166 MHZ or faster processor with at least 64 MB RAM and 100MB free disk space. Multisim software, Electronic components, Analog Discovery Kits,Digilint Software.

S. No.	List of Experiments	Page No.
	ELECTRONIC CIRCUITS LAB	
1	Common Emitter and Common Base amplifier	
2	Two Stage RC Coupled Amplifier	
3	Single Tuned Voltage Amplifier	
4	Current shunt and voltage series Feedback Amplifier	
5	RC Phase Shift Oscillator	
6	Hartley and Colpitts oscillator	
7	Class A power amplifier (transformer less) and Class B power amplifier	
	PULSE CIRCUITS LAB	
1	RC low pass and high pass circuit for different time constants.	
2	Transfer characteristics and response of Clippers. The steady state output waveform of clampers for a square wave input.	
3	Transistor as a switch.	
4	Design a Astable Multivibrator and draw its waveforms.	
5	Schmitt trigger.	
6	UJT relaxation oscillator.	
7	Boot strap sweep circuit	
8	Comparator	

ATTAINMENT OF PROGRAM OUTCOMES & PROGRAM SPECIFIC OUTCOMES

Exp. No.	Experiment	Program Outcomes Attained	Program Specific Outcomes Attained
1	Common Emitter and Common Base amplifier	PO1, PO2, PO11	PSO1, PSO2
2	Two Stage RC Coupled Amplifier	PO1, PO2, PO11	PSO1, PSO2
3	Single Tuned Voltage Amplifier	PO1, PO2, PO11	PSO1, PSO2
4	Current shunt and voltage series Feedback Amplifier	PO1, PO2, PO11	PSO1, PSO2
5	RC Phase Shift Oscillator	PO1, PO2, PO11	PSO1, PSO2
6	Hartley and Colpitts oscillator	PO1, PO2, PO11	PSO1, PSO2
7	Class A power amplifier (transformer less) and Class B power amplifier	PO1, PO2, PO11	PSO1, PSO2
8	RC low pass and high pass circuit for different time constants.	PO1, PO2, PO11	PSO1, PSO2
9	Transfer characteristics and response of Clippers. The steady state output waveform of clampers for a square wave input.	PO1, PO2, PO11	PSO1, PSO2
10	Transistor as a switch.	PO1, PO2,PO5, PO12	PSO1, PSO2
11	Design a Astable Multivibrator and draw its waveforms.	PO1, PO2, PO11	PSO1, PSO2
12	Response of Schmitt Trigger circuit for loop gain less than and greater than one	PO1, PO2, PO11	PSO1, PSO2
13	UJT relaxation oscillator.	PO1, PO2, PO11	PSO1, PSO2
14	Boot strap sweep circuit	PO1, PO2, PO11	PSO1, PSO2
15	Comparator	PO1, PO2, PO11	PSO1, PSO2

EC LAB

EXPERIMENT NO: 1

CE AND CB AMPLIFIER

1.1 AIM:

To plot the frequency response of CE amplifier and calculate gain bandwidth.

1.2 SOFTWARE REQUIRED:

MultiSim Analog Devices Edition 13.0

1.3 COMPONENTS & EQUIPMENTS REQUIRED:

S.No	Apparatus	Range/ Rating	Quantity (in No.s)
1	CE Amplifier trainer Board with DC power supply DC power supply NPN transistor Carbon film resistor (e)Carbon film resistor (f) Capacitor.	 12V 5V BC 107 100KΩ, 1/2W 2.2KΩ, 1/2W 0.1μF	 1 1 1 1 1 2
2	Cathode Ray Oscilloscope.	(0-20)MHz	1
3	Function Generator.	0.1 Hz-10 MHz	1
4	BNC Connector		2
5	Connecting Wires	5A	5

1.4 THEORY:

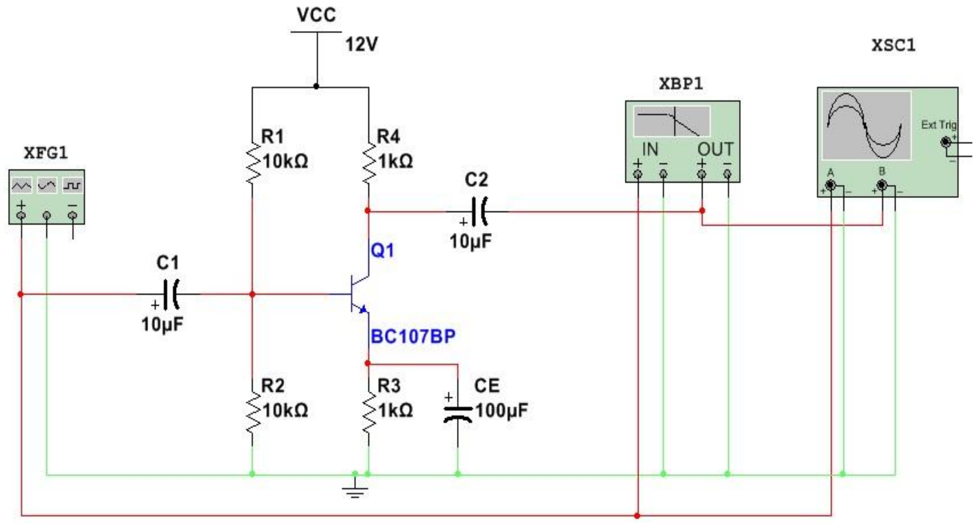
The CE amplifier provides high gain & wide frequency response. The emitter lead is common to both input and output circuits and is grounded. The emitter base is forward biased. The collector current is controlled by the base current rather than emitter current. The input signal is applied to base terminal 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. Frequency response of an amplifier is defined as the variation of gain with respective frequency. The gain of the amplifier increases as the frequency increases from zero till it becomes maximum at lower cut-off frequency and remains constant till higher cut-off frequency and then it falls again as the frequency increases.

At low frequencies the reactance of coupling capacitor CC is quite high and hence very small part of signal will pass through from one stage to the next stage.

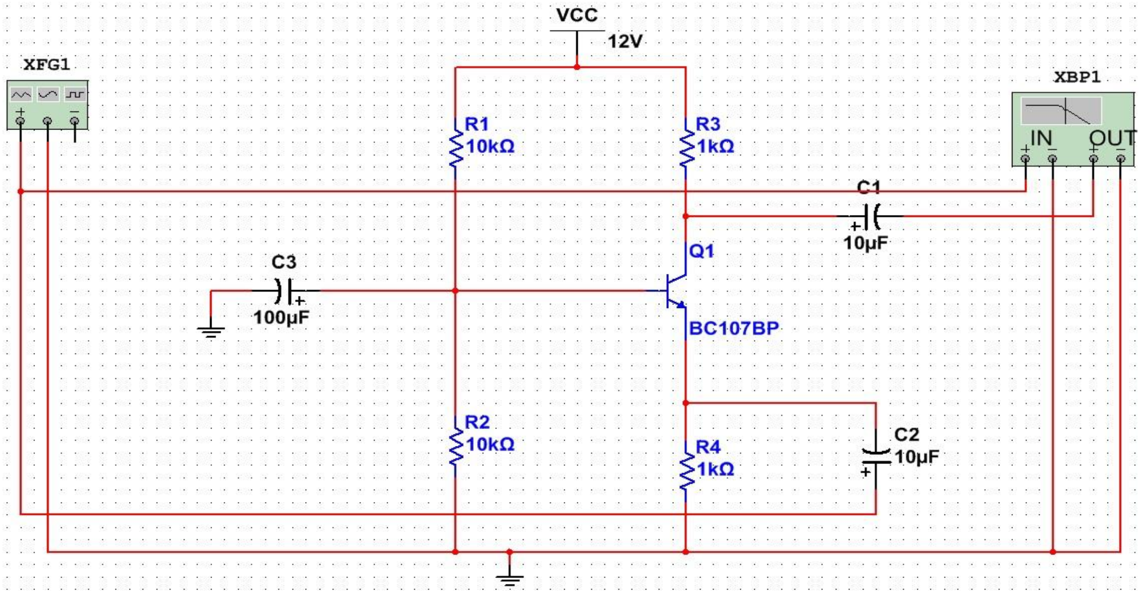
At high frequencies the reactance of inter electrode capacitance is very small and behaves as a short circuit. This increases the loading effect on next stage and service to reduce the voltage gain due to these reasons the voltage gain drops at high frequencies.

At mid frequencies the effect of coupling capacitors is negligible and acts like short circuit, where as inter electrode capacitors acts like open circuit. So, the circuit becomes resistive at mid frequencies and the voltage gain remains constant during this range.

1.4 CIRCUIT DIAGRAM:

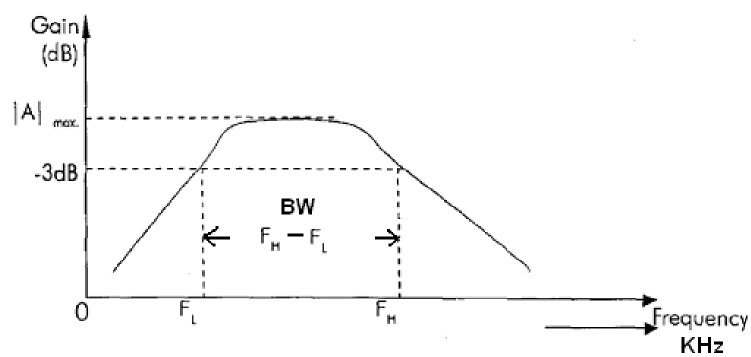


CE AMPLIFIER



CB AMPLIFIER

1.5 EXPECTED GRAPH:



1.6 PROCEDURE:

1. Connect the circuit diagram as shown in figure.
2. Adjust input signal amplitude in the function generator and observe an amplified voltage at the output without distortion.
3. By keeping input signal voltages at 50mV, vary the input signal frequency from 0 to 1MHz in steps as shown in tabular column and note the corresponding output voltages.
4. Save the circuit and simulate.
5. Calculate the maximum gain and bandwidth using bode plotter. Compare the values with the practical circuit values.

1.7 PRECAUTIONS:

1. Check whether the connections are made properly or not.

1.8 OBSERVATIONS:

Input voltage: $V_i = 50mV$

Frequency (in Hz)	Gain (in dB) = $20 \log_{10} V_O / V_i$
20	
600	
1K	
2K	
4K	
8K	
10K	
20K	
30K	
40K	
50K	
60K	
80K	
100K	
250K	
500K	
750K	
1000K	

1.9 CALCULATIONS

1.10 PRE LAB QUESTIONS

1. What are the advantages and disadvantages of single-stage amplifiers?
2. Why gain falls at HF and LF?
3. Why the gain remains constant at MF?

1.11 POST LAB QUESTIONS

1. Explain the function of emitter bypass capacitor, Ce?
2. How the band width will effect as more number of stages are cascaded?
3. Define frequency response?
4 What is the phase difference between input and output waveforms of a CE amplifier?
5 What is early effect?

1.12 RESULT:

Frequency response of CE amplifier is plotted.

Gain, A_V = ________dB.

Bandwidth= f_H - f_L =________Hz.

EXPERIMENT NO- 2

TWO STAGE RC COUPLED AMPLIFIER

2.1 AIM:

1. To plot the frequency response of a RC coupled amplifier with a pair of shunted emitter capacitors of 10µF and 100µF.
2. To calculate gain.
3. To calculate bandwidth.

2.2 SOFTWARE REQUIRED:

MultiSim Analog Devices Edition 13.0

COMPONENTS & EQUIPMENT REQUIRED:

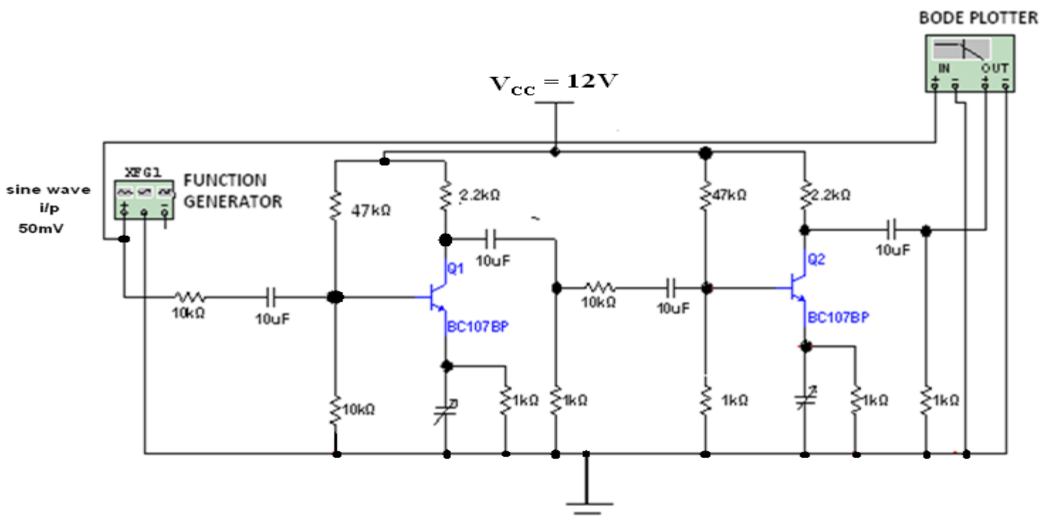
S.No	Device	Range/ Rating	Quantity (in No.s)
1	Trainer Board containing a) DC Supply voltage. b) NPN Transistor. c) Resistors. d) Capacitors.	12 V BC 107 47 KΩ 2.2 KΩ 1 KΩ 10 KΩ 100µF 10µF.	 1 2 2 2 5 2 6
2	Bode Plotter		1
3	Function Generator.	0.1 Hz-10 MHz	1

2.3 THEORY:

As the gain provided by a single stage amplifier is usually not sufficient to drive the load, so to achieve extra gain multi-stage amplifier are used. In multi-stage amplifiers output of one-stage is coupled to the input of the next stage. The coupling of one stage to another is done with the help of some coupling devices. If it is coupled by RC then the amplifier is called RC-coupled amplifier.

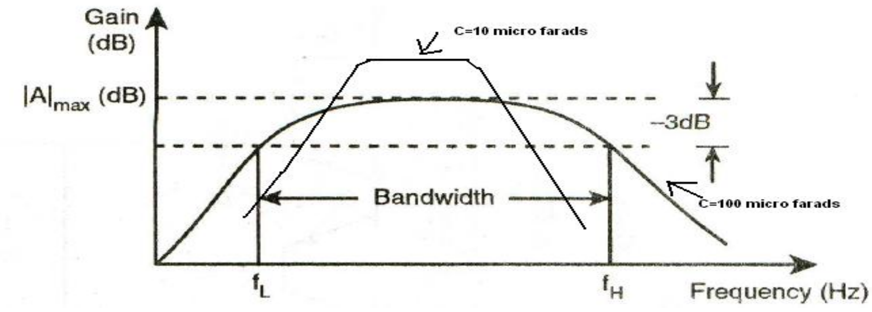
Frequency response of an amplifier is defined as the variation of gain with respective frequency. The gain of the amplifier increases as the frequency increases from zero till it becomes maximum at lower cut-off frequency and remains constant till higher cut-off frequency and then it falls again as the frequency increases. At low frequencies the reactance of coupling capacitor CC is quite high and hence very small part of signal will pass through from one stage to the next stage. At high frequencies the reactance of inter electrode capacitance is very small and behaves as a short circuit. This increases the loading effect on next stage and service to reduce the voltage gain due to these reasons the voltage gain drops at high frequencies. At mid frequencies the effect of coupling capacitors is negligible and acts like short circuit, where as inter electrode capacitors acts like open circuit. So, the circuit becomes resistive at mid frequencies and the voltage gain remains constant during this range.

2.4 CIRCUIT DIAGRAM



TWO STAGE RC COUPLED AMPLIFIER

2.5 EXPECTED GRAPH:



2.6 PROCEDURE:

1. Connect the circuit as shown in figure for 10 μF.
2. Adjust input signal amplitude in the function generator and observe an amplified voltage at the output without distortion.
3. By keeping input signal voltage, say at 50 mV, vary the input signal frequency from 0-1 MHz as shown in tabular column and note the corresponding output voltage.
4. Save the circuit and simulate.
5. Calculate the maximum gain and bandwidth using Bode plotter. Compare the values with the practical circuit values.
6. Repeat the same procedure for C=100μF.

2.7 PRECAUTIONS:

Check whether the connections are made properly or not.

2.8 TABULAR FORM:

V_{in} = 50 mV

C=10μF			C=100μF	
S.No	Frequency (in Hz)	Gain(dB) 20 log(V_o/ V_i)	Frequency (in Hz)	Gain(dB) 20 log(V_o/ V_i)
1	100			
2	200			
3	400			
4	800			

5	1K			
6	2K			
7	4K			
8	8K			
9	10K			
10	20K			
11	40K			
12	80K			
13	100K			
14	200K			
15	300K			
16	500K			
17	700K			
18	900K			
19	1M			

2.9 CALCULATIONS

2.10 PRELAB QUESTIONS

1. What is the need for Cascading?
2. What are the types of Coupling Schemes for Cascading?

2.11 POSTLAB QUESTIONS

1. What are the advantages of RC coupling
2. What is the effect of bypass Capacitor on frequency response
3. What is the effect of Coupling Capacitors

2.12 RESULT:

Hence, the frequency Response of RC coupled (2 stage) amplifier for 10µF and 100 µF is plotted.

1. For C=10 µF,

Gain= Bandwidth $=f_H - f_L =$

2. For C=100µF

Gain= Bandwidth $=f_H - f_L =$

EXPERIMENT NO-3
SINGLE TUNED VOLTAGE AMPLIFIER

3.1 AIM:

1. To study & plot the frequency response of a Single Tuned voltage amplifier.
2. To find the resonant frequency.
3. To calculate gain and bandwidth.

3.2 SOFTWARE REQUIRED:

MultiSim Analog Devices Edition 13.0

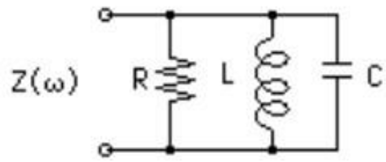
COMPONENTS & EQUIPMENT REQUIRED:

S.No	Apparatus	Range/ Rating	Quantity (in No.s)
1.	Trainer Board containing		
	a) DC Supply voltage.	12 V	1
	b) NPN Transistor.	BC 107	1
	c) Resistors.	47 KΩ	1
		150Ω	1
		1 KΩ	1
		10 KΩ	2
	d) Capacitor.	10uF	2
		22 uF.	1
		0.022 uF.	1
		0.033□F.	1
	e) Inductor.	1mH	1
2.	Cathode Ray Oscilloscope.	(0-20)MHz	1
3.	Function Generator.	0.1 Hz-10MHz	1
4.	BNC Connector		2
5.	Connecting Wires	5A	5

3.3 THEORY:

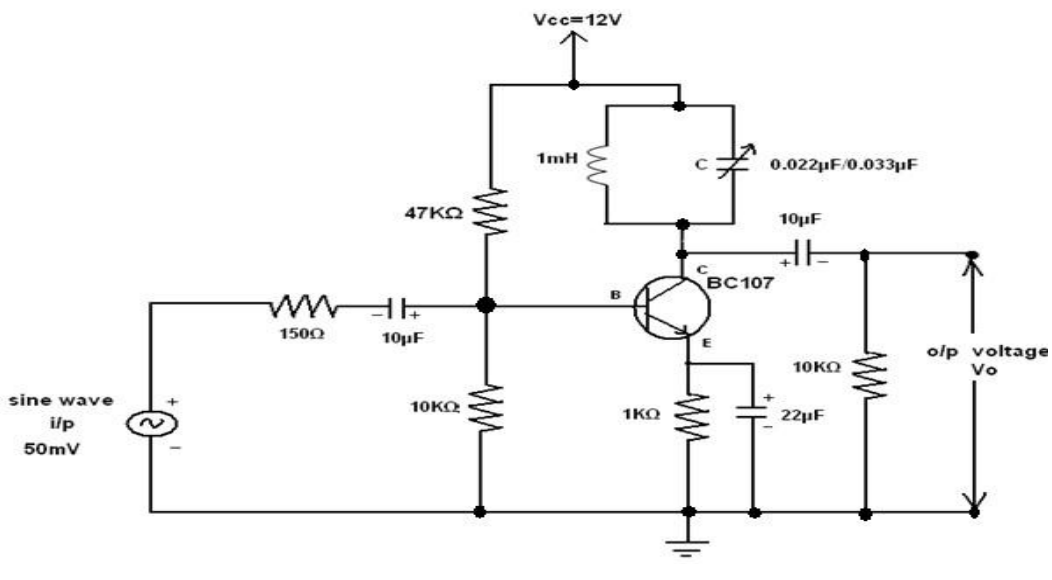
Tuned amplifiers are amplifiers involving a resonant circuit, and are intended for selective amplification within a narrow band of frequencies. Radio and TV amplifiers employ tuned amplifiers to select one broadcast channel from among the many concurrently induced in an antenna or transmitted through a cable. Selected aspects of tuned amplifiers are reviewed in this note. Parallel Resonant Circuit

An idealized parallel resonant circuit, i.e. one described by idealized circuit elements, is drawn below.

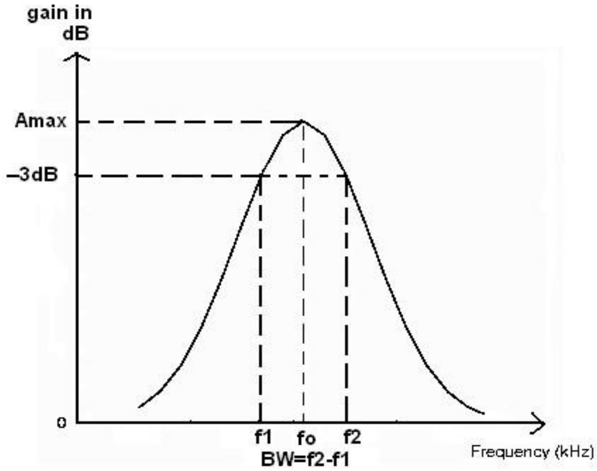


input impedance of this configuration, shown below the circuit diagram, is readily obtained. A modest algebraic restatement in convenient form also is shown. The significance of the definitions of the 'quality factor' Q and the resonant frequency ω_o will become clear from the discussion. The influence of the Q parameter on the tuned-circuit impedance for several values of Q is plotted below for a normalized response.

3.4 CIRCUIT DIAGRAM:



3.5 EXPECTED WAVEFORM:



3.6 TABULAR COLUMN :

C=0.022µF Vin = 50 mV					C== 0.033µF Vin = 50 mV			
S.No	Frequency (in Hz)	V_o (V)	Gain A = V_o/ V_i	Gain(dB) 20 log(V_o/ V_i)	Frequency (in Hz)	V_o (V)	Gain A = V_o/ V_i	Gain(dB) 20 log(V_o/ V_i)
1	100							
2	200							
3	400							
4	800							
5	1K							
6	2K							
7	4K							

8	8K							
9	10K							
10	20K							
11	40K							
12	80K							
13	100K							
14	200K							

3.7 PROCEDURE:

1. Connect the circuit as shown in figure.
2. Connect the 0.022μF capacitor
3. Adjust input signal amplitude in the function generator and observe an amplified voltage at the output without distortion.
4. By keeping input signal voltage, say at 50 mV, vary the input signal frequency from 0-100KHz as shown in tabular column and note the corresponding output voltage.
5. Repeat the same procedure for 0.033μF capacitor.
6. Plot the graph: gain (Vs) frequency.
7. Calculate the f_1 and f_2 and bandwidth.
8. Compare the resonant frequency with theoretical value in both the cases.

3.8 PRECAUTIONS: -

1. No loose contacts at the junctions.
2. Check the connections before giving the power supply
3. Observations should be taken carefully.

3.9 PRE LAB QUESTIONS:

1. What is the purpose of tuned amplifier?
2. What is Quality factor?
3. Why should we prefer parallel resonant circuit in tuned amplifier.
4. What type of tuning we need to increase gain and bandwidth.?
5. What are the limitations of single tuned amplifier?
6. What is meant by Stagger tuning?
7. What is the conduction angle of an tuned amplifier if it is operated in class B mode?

3.10 PRE LAB QUESTIONS:

1. What are the applications of tuned amplifier
2. What are the different types of tuned circuits ?
3. State relation between resonant frequency and bandwidth of a Tuned amplifier.
4. Differentiate between Narrow band and Wideband tuned amplifiers ?
5. Calculate bandwidth of a Tuned amplifier whose resonant frequency is 15KHz and Q-factor is 100.
6. Specify the applications of Tuned amplifiers.

3.11 RESULT:

Frequency response of RF Tuned voltage amplifier is plotted.

For 0.022µF, gain = ________dB Bandwidth= _________

For 0.033µF, gain = ________dB Bandwidth= _________

EXPERIMENT NO-4

CURRENT SHUNT AND VOLTAGE SERIES FEEDBACK AMPLIFIER

4.1 AIM:

To study and plot the frequency response of a current shunt and voltage series feedback amplifier.

4.2 SOFTWARE REQUIRED:

MultiSim Analog Devices Edition 13.0

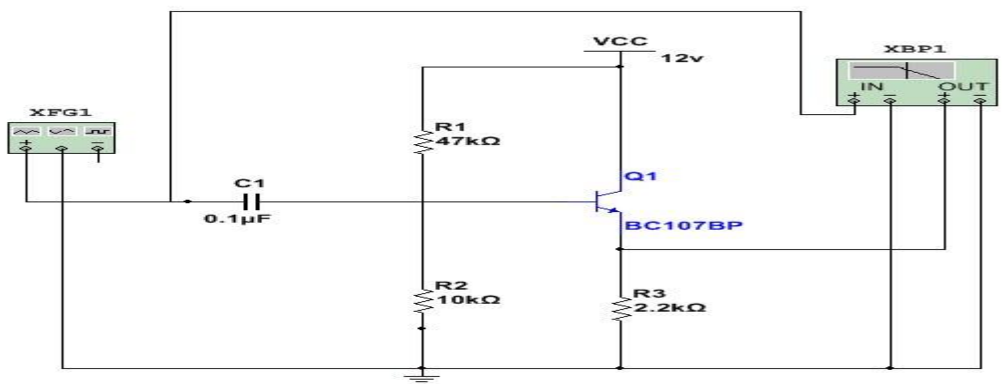
4.3.COMPONENTS & EQUIPMENT REQUIRED:

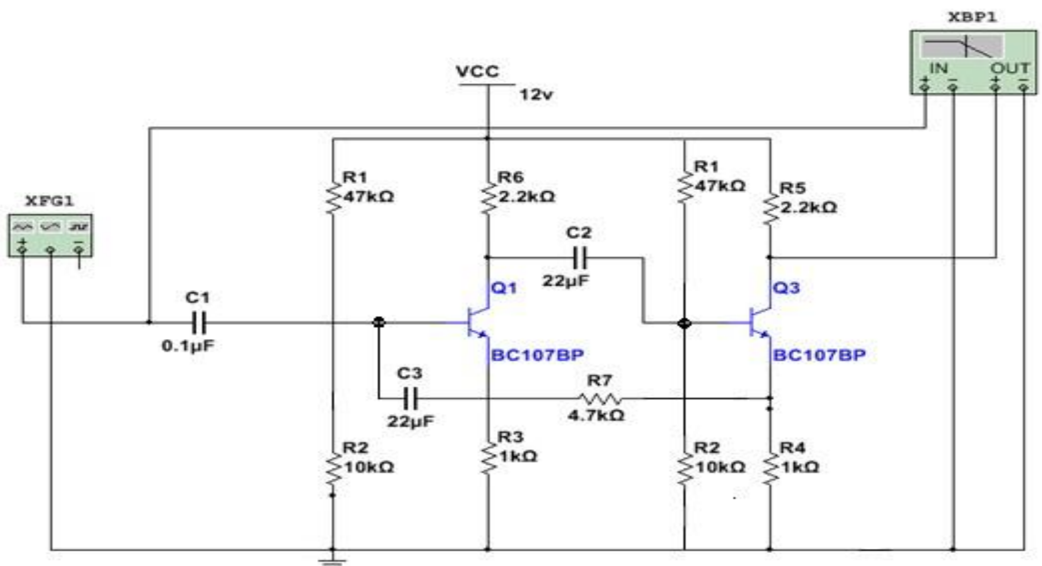
S.No	Apparatus	Range/ Rating	Quantity (in No.s)
1.	a) DC Supply voltage. b) NPN Transistor. c) Resistors. d) Capacitor.	12 V BC 107 47kΩ 2.2KΩ 10kΩ 1k 0.1μF. 22μF.	1 2 2 2 1 2 1 3
3.	Bode plotter		1
4.	Function Generator.	0.1 Hz-10 MHz	1

4.4 THEORY:

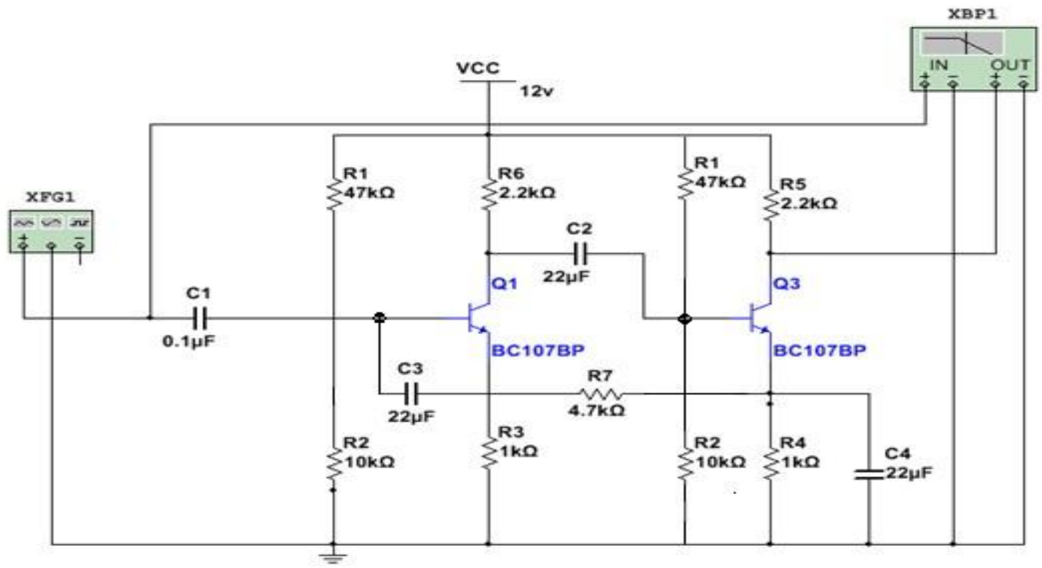
Feedback plays a very important role in electronic circuits and the basic parameters, such as input impedance, output impedance, current and voltage gain and bandwidth, may be altered considerably by the use of feedback for a given amplifier. A portion of the output signal is taken from the output of the amplifier and is combined with the normal input signal and thereby the feedback is accomplished. There are two types of feedback. They are i) Positive feedback and ii) Negative feedback. Negative feedback helps to increase the bandwidth, decrease gain, distortion, and noise, modify input and output resistances as desired. A current shunt feedback amplifier circuit is illustrated in the figure. It is called a series-derived, shunt-fed feedback. The shunt connection at the input reduces the input resistance and the series connection at the output increases the output resistance. This is a true current amplifier.

4.5 CIRCUIT DIAGRAM:



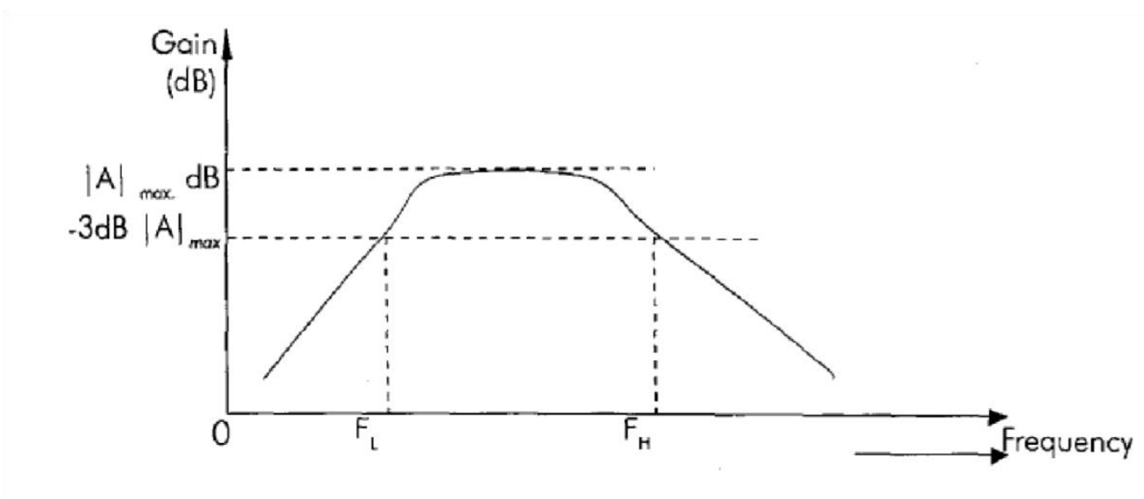


Current shunt (with out capacitor)



Current shunt (with capacitor)

4.6 EXPECTED GRAPH:



4.7 TABULAR FORM:

Input voltage = 50mv

Voltage series feedback			Current shunt (without capacitor)		Current shunt(with capacitor)	
Frequency (Hz)	Out put	gain	Output	gain	Output	Gain
20						
40						
60						
100						
200						
400						
600						
800						
1k						
2k						
5k						
8k						
10k						
20k						
40k						
60k						
100k						
400k						
600k						
800k						
1M						

4.8 PROCEDURE:

1. Connect the circuit as shown in figure
2. Adjust input signal amplitude in the function generator and observe an amplified voltage at the output without distortion.
3. By keeping input signal voltage, say at 50 mV, vary the input signal frequency from 0-1 MHz as shown in tabular column and note the corresponding output voltage.
4. Save the circuit and simulate.
5. For current shunt feedback amplifier with shunt capacitor (with and without capacitor) voltage series feedback amplifier (with and without feedback resistance). Repeat the above procedure.
6. Calculate the maximum gain and bandwidth using Bode plotter. Compare the values with the practical circuit values

4.9 PRECAUTIONS:

1. No loose contacts at the junctions.
2. Check the connections before giving the power supply
3. Observations should be taken carefully.

4.10 RESULT:

Frequency responses for voltage series (with and without feedback amplifier),

Frequency responses current shunt (with and without capacitor are plotted)

EXPERIMENT NO-5

RC PHASE SHIFT OSCILLATOR

5.1 AIM:

To find practical frequency of RC phase shift oscillator and to compare it with theoretical frequency for R=10KΩ and C = 0.01µF, 0.0022µF & 0.0033µF respectively

5.2 SOFTWARE REQUIRED:

MultiSim Analog Devices Edition 13.0

COMPONENTS AND EQUIPMENTS REQUIRED:

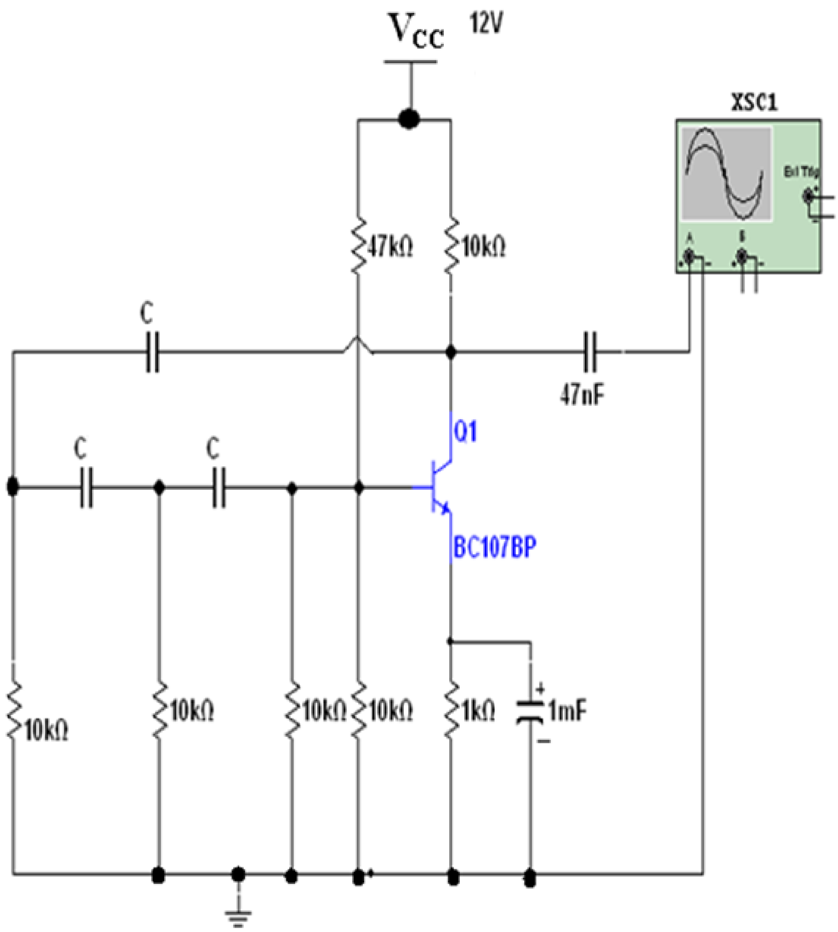
S.No	Device	Range/ Rating	Quantity (in No.s)
1	RC phase shift oscillator trainer board containing a) DC supply voltage b) Capacitor c) Resistor d) NPN Transistor	 12V----------- 1000µF------- 0.047µF------ 0.01µF-------- 0.0022µF------ 0.0033µF---- 1KΩ----------- 10KΩ--------- 47KΩ---------- 100KΩ--------- BC 107--------	 1 1 1 3 3 3 1 4 1 1 1
2	CRO		1

5.3 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 return to the input of the amplifier.the values of R and C are chosen such that the phase shift of each RC section is 600.thus,the RC ladder network produces a total phase shift of 1800 between its input and output voltage for the given frequencies since CE amplifier produces 1800 phase shift the total phase shift from the base of the transistor around the circuit and back to the transistor will be exactly 3600 or 00.The frequency of oscillation is given by

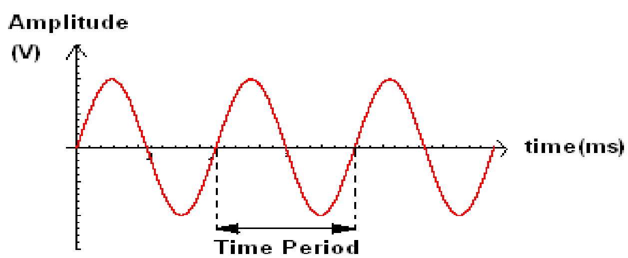
F = 1/2ΠRC√6

5.4 CIRCUIT DIAGRAM:



RC PHASE SHIFT OSCILLATOR

5.5 EXPECTED WAVEFORM:



5.6 PROCEDURE:

1. Connect the circuit as shown in figure.
2. Connect the 0.0022 μF capacitors in the circuit and observe the waveform.
3. Save the circuit and simulate.
4. Calculate the time period and frequency of the resultant wave form. Compare the values with the practical circuit values
5. Repeat the same procedure for C=0.033 μF and 0.01μF and calculate the frequency and tabulate as shown.
6. Find theoretical frequency from the formula $f = 1/2\Pi RC\sqrt{6}$ and compare theoretical and practical frequencies.

5.7 PRECAUTIONS:

1. No loose contacts at the junctions.
2. Check the connections before giving the power supply
3. Observations should be taken carefully.

5.8 OBSERVATIONS:

S.No	C (μF)	R (Ω)	Theoretical Frequency (KHz)	Practical Frequency (KHz)	V_o (p-p) (Volts)
1	0.0022	10K			
2	0.0033	10K			
3	0.01	10K			

5.9 CALCULATIONS

5.10 PRE LAB QUESTIONS

1. What is the frequency of RC phase shift oscillator?
2. What is a phase shift oscillator?
3. Why RC oscillators cannot generate high frequency oscillations?
4. What are the applications of RC phase shift oscillators?

5.11 POST LAB QUESTIONS

1. What phase shift does RC phase shift oscillator produce?
2. Why we need a phase shift between input and output signal?
3. How is phase angle determined in RC phase shift oscillator?
4. How can we get a maximum phase angle of 90 degrees in RC phase shift oscillator?

5.12 RESULT:

1. For C = 0.0022μF & R=10KΩ

   Theoretical frequency= Practical frequency=

2. For C = 0.0033μF & R=10KΩ

   Theoretical frequency= Practical frequency=

3. For C = 0.01μF & R=10KΩ

   Theoretical frequency= Practical frequency=

EXPERIMENT NO-6

(A) HARTLEY OSCILLATOR

6A.1 AIM:

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

6A.2 SOFTWARE REQUIRED:

MultiSim Analog Devices Edition 13.0

COMPONENTS AND EQUIPMENTS REQUIRED:

S.No	Device	Range/ Rating	Quantity (in No.s)
1	Hartley Oscillator trainer board containing a) DC supply voltage b) Inductors c) Capacitor d) Resistor e) NPN Transistor	 12V 5mH 0.22uF 0.01uF 0.033uF 0.047uF 1K 10K 47K BC 107	 1 2 2 1 1 1 1 1 1 1
2	Cathode Ray Oscilloscope	(0-20) MHz	1
3.	BNC Connector		1
4	Connecting wires	5A	4

6A.3 THEORY:

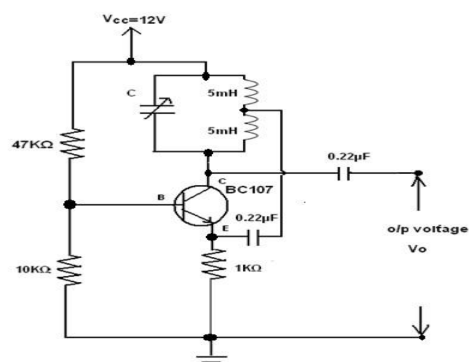
The **Hartley oscillator** is an electronic oscillatorcircuitin which the oscillation frequency is determined by a tuned circuitconsisting of capacitorsand inductors, that is, an LC oscillator. The circuit was invented in 1915 by American engineer Ralph Hartley. The distinguishing feature of the Hartley oscillator is that the tuned circuit consists of a single capacitor in parallel with two inductors in series (or a single tapped inductor), and the feedbacksignal needed for oscillation is taken from the center connection of the two inductors. The frequency of oscillation is approximately the resonant frequencyof the tank circuit. If the capacitance of the tank capacitor is *C* and the total inductanceof the tapped coil is *L* then $f = \frac{1}{2\pi\sqrt{LC}}$

If two *uncoupled* coils of inductance L_1 and L_2 are used then $L = L_1 + L_2$

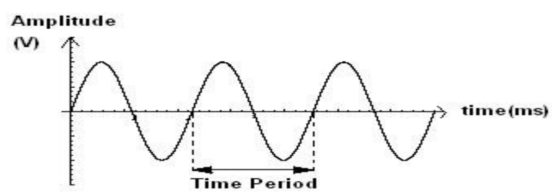
However if the two coils are magnetically coupled the total inductance will be greater because of mutual inductance*k*. $L = L_1 + L_2 + k\sqrt{L_1 L_2}$

6A.4 CIRCUIT DIAGRAM:

HARTLEY OSCILLATOR



6A.5 EXPECTED WAVEFORM:



6A.6 TABULATIONS:

S.No	L_T(mH)	C (uF)	Theoretical frequency (KHz)	Practical waveform time period (Sec)	Practical frequency (KHz)	Vo (V) (ptp)
1	10	0.01				
2	10	0.033				
3	10	0.047				

6A.7 PROCEDURE:

1. Connect the circuit as shown in figure.
2. Connect 0.01uF capacitor in the circuit and observe the waveform.
3. Note the time period of the waveform and calculate the frequency: f = 1/T .
4. Now connect the capacitance to 0.033 uF and 0.047uF and calculate the frequency and tabulate the readings as shown.
5. Find the theoretical frequency from the formula

$$f = \frac{1}{2\pi\sqrt{LC}}$$

Where $L_T = L_1 + L_2$ = 5 mH + 5mH = 10 mH and compare theoretical and practical values.

6A.8 PRECAUTIONS:

1. No loose contacts at the junctions.
2. Check the connections before giving the power supply
3. Observations should be taken carefully.

6A.9 PRE LAB QUESTIONS:

1. What are the types of sinusoidal oscillator? Mention the different types of sinusoidal oscillator?
2. What is Barkhausan criterion?
3. Name two high frequency Oscillators.
4. What are the essential parts of an Oscillator?

6A.10 POST LAB QUESTIONS:

1. How many inductors and capacitors are used in Hartley Oscillator?
2. How the oscillations are produced in Hartley oscillator?

6A.11 RESULT:

For C = 0.01uF, & L_T = 10 mH;

Theoretical frequency = Practical frequency =

For C = 0.033uF, & L_T = 10 mH;

Theoretical frequency = Practical frequency =

For C = 0.047uF, & L_{Ts} = 10 mH;

Theoretical frequency = Practical frequency =

(B) COLPITTS OSCILLATOR

6B.1 AIM:

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

6B.2 SOFTWARE REQUIRED:

MultiSim Analog Devices Edition 13.0

COMPONENTS & EQIUPMENT REQUIRED: -

S.No	Device	Range/ Rating	Quantity (in No.s)
1	Colpitts Oscillator trainer board containing		
	a) DC supply voltage	12V	1
	b) Inductors	5mH	1
	c) Capacitor	0.01uF	1
		0.1uF	1
		100 uF	1
		0.001u	1
		0.0022u	1
		0.0033 u	1
	d) Resistor	1K	1
		1.5K	1
		10K	1
		47K	1
	e) NPN Transistor	BC 107	1
2	Cathode Ray Oscilloscope	(0-20) MHz	1
3.	BNC Connector		1
4	Connecting wires	5A	4

6B.3 THEORY:

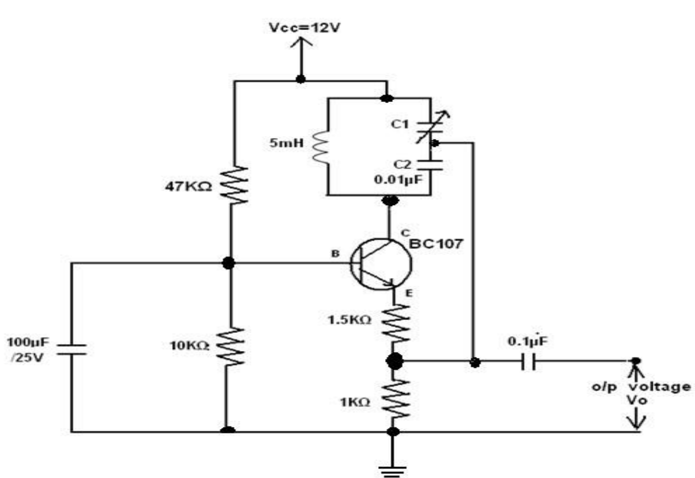
A **Colpitts oscillator**, invented in 1918 by American engineer Edwin H. Colpitts,is one of a number of designs for LC oscillators,electronic oscillatorsthat use a combination of inductors(L) and capacitors(C) to produce an oscillation at a certain frequency. The distinguishing feature of the Colpitts oscillator is that the feedbackfor the active device is taken from a voltage dividermade of two capacitors in series across the inductor.The frequency of oscillation is approximately the resonant frequency of the LC circuit, which is the series combination of the two capacitors in parallel with the inductor

$$f_0 = \frac{1}{2\pi\sqrt{L\left(\frac{C_1C_2}{C_1+C_2}\right)}}$$

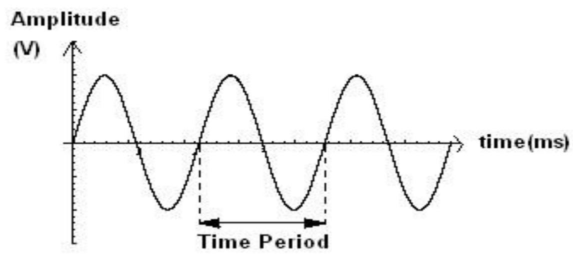
The actual frequency of oscillation will be slightly lower due to junction capacitances and resistive loading of the transistor.As with any oscillator, the amplification of the active component should be marginally larger than the attenuation of the capacitive voltage divider, to obtain stable operation. Thus, a Colpitts oscillator used as a variable frequency oscillator(VFO) performs best when a variable inductance is used for tuning, as opposed to tuning one of the two capacitors. If tuning by variable capacitor is needed, it should be done via a third capacitor connected in parallel to the inductor (or in series as in the Clapp oscillator).

6B.4 CIRCUIT DIAGRAM:

COLPITTS OSCILLATOR



6B.5 EXPECTED WAVEFORM:



6B.6 TABULAR COLUMN:

S.NO	L(mH)	C_1 (uF)	C_2 (uF)	C_T (uF)	Theoretical Frequency (KHz)	Practical Frequency (KHz)	Vo(V) Peak to peak
1	5	0.01	0.001				
2	5	0.01	0.0022				
3	5	0.01	0.0033				

6B.7 PROCEDURE:

1. Connect the circuit as shown in the figure
2. Connect C_2= 0.001uF in the circuit and observe the waveform.
3. Calculate the time period and frequency of the waveform (f=1/T)
4. Now, fix the capacitance to 0.002 uF and then to 0.003 uF and calculate the frequency and tabulate the reading as shown.
5. Find theoretical frequency from the formula

$$f_0 = \frac{1}{2\pi\sqrt{L\left(\frac{C_1 C_2}{C_1+C_2}\right)}}$$

6. Compare theoretical and practical values.
7. Plot the graph o/p voltage vs time period and practical frequency

6B.8 PRECAUTIONS:

1. No loose contacts at the junctions.
2. Check the connections before giving the power supply
3. Observations should be taken carefully.

6B.9 PRE LAB QUESTIONS:

1. What are the applications of LC oscillations?
2. What type of feedback is used in oscillators?
3. Whether an oscillator is dc to ac converter. Explain?
4. What is the loop gain of an oscillator?
5. What is the difference between amplifier and oscillator?
6. What is the condition for sustained oscillations?

6B.10 POST LAB QUESTIONS:

1. What is the difference between damped oscillations undamped oscillations?
2. How does Colpitt's differ from Hartley?
3. What is the expression for the frequency of oscillations of Colpitt's and Hartley oscillator?

6B.11 RESULT:

Hence, the frequency of oscillations of Colpitts oscillator is measured practically and compared with theoretical values .

1. For C=0.0022uF & L= 5mH

Theoretical frequency = Practical frequency =

2. For C=0.0033uF & L= 5mH

Theoretical frequency = Practical frequency =

3. For C=0.001uF & L= 5mH

Theoretical frequency = Practical frequency =

EXPERIMENT NO-7(A)

CLASS A POWER AMPLIFIER

7A.1 AIM:

To study and plot the frequency response of a Class A Power Amplifier.

To calculate efficiency of Class A Power Amplifier.

7A.2 COMPONENTS & EQUIPMENT REQUIRED:

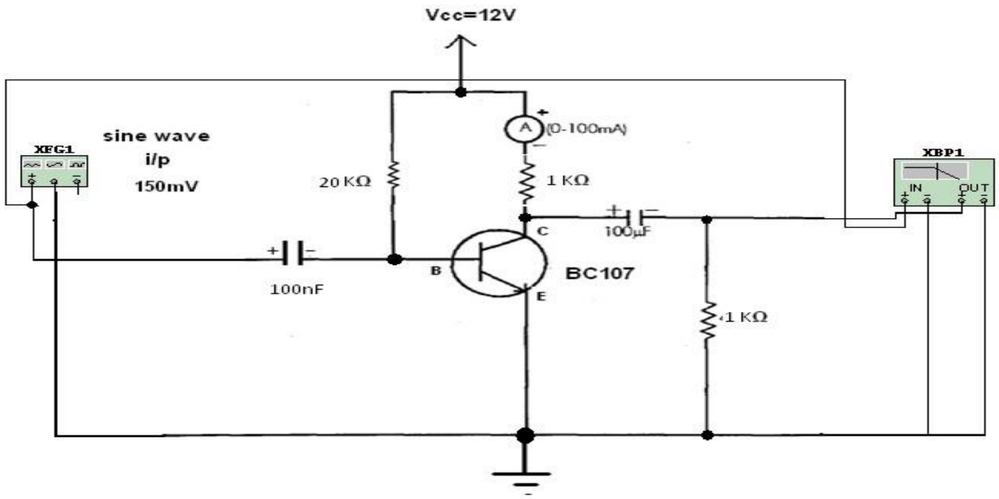
MultiSim Analog Devices Edition 13.0

S.No	Apparatus	Range/ Rating	Quantity (in No.s)
1.	Trainer Board containing a) DC Supply voltage. b) NPN Transistor. c) Resistors. d) Capacitor. e) Inductor.	 12 V BC 107 560Ω 100KΩ 470Ω 22 □F. 50mH	 1 1 1 1 1 1 1
2.	D.C milli ammeter	0-100mA	1
3.	Cathode Ray Oscilloscope.	(0-20)MHz	1
4.	Function Generator.	0.1 Hz-10 MHz	1
5.	BNC Connector		2
6.	Connecting Wires	5A	5

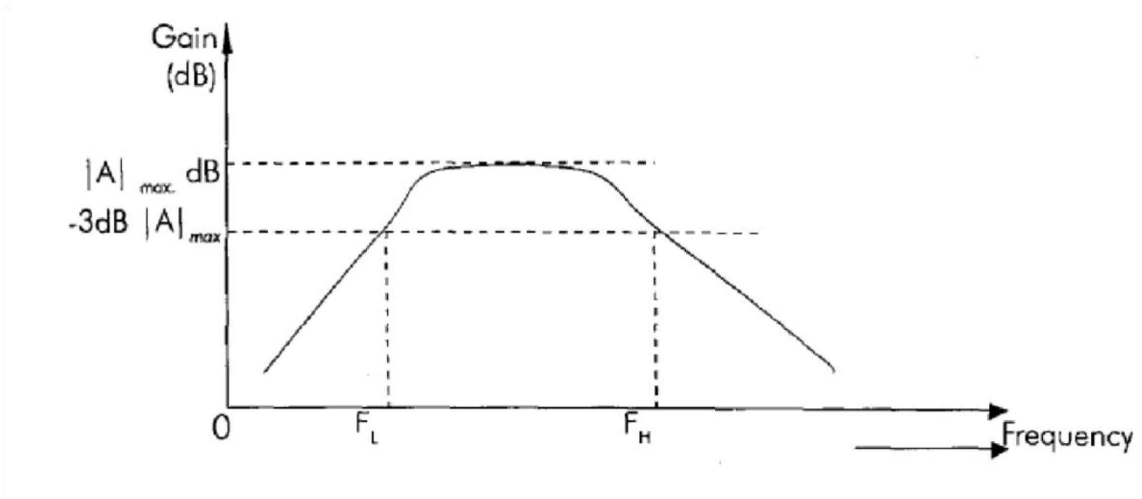
7A.3 THEORY:

Power amplifiers are mainly used to deliver more power to the load. To deliver more power it requires large input signals, so generally power amplifiers are preceded by a series of voltage amplifiers. In class-A power amplifiers, Q-point is located in the middle of DC-load line. So output current flows for complete cycle of input signal. Under zero signal condition, maximum power dissipation occurs across the transistor. As the input signal amplitude increases power dissipation reduces. The maximum theoretical efficiency is 50%.

7A.4 CIRCUIT DIAGRAM:



7A.5 EXPECTED GRAPH:



Bandwidth=$f_H - f_L$

7A.6 TABULAR FORM:

V_{in} = 50 mV

S.No	Frequency (in Hz)	Gain(dB) Av = 20 log(V_o/ V_i)
1	100	
2	200	
3	400	
4	800	
5	1K	
6	2K	
7	4K	
8	8K	
9	10K	
10	20K	
11	40K	
12	80K	
13	100K	
14	200K	

7A.7 CALCULATIONS:

Efficiency is defined as the ratio of AC output power to DC input power

DC input power = Vcc x I_{CQ}

AC output power = V_{P-P}^2 / $8R_L$

Under zero signal condition:

Vcc = IBRB + VBE

IBQ =(Vcc - VBE) / RB

ICQ = β x IBQ

VCE = Vcc - ICRC

7A.8 PROCEDURE:

1. Connect the circuit as shown in figure.
2. Adjust input signal amplitude in the function generator and observe an amplified voltage at the output without distortion.
3. By keeping input signal voltage, say at 150 mV, vary the input signal frequency from 0-1 MHz as shown in tabular column and note the corresponding output voltage.
4. Measure and note down the zero signal dc current by disconnecting the function generator from the circuit.
5. Calculate the efficiency according to the expressions given.
6. Plot the graph between the o/p gain and frequency and calculate the bandwidth.

7A.9 PRECAUTIONS:

1. No loose contacts at the junctions.
2. Check the connections before giving the power supply
3. Observations should be taken carefully.

7A.10 RESULT:

1. Frequency Response of CLASS-A Power amplifier is plotted.
2. Efficiency of CLASS A Power amplifier is found to be___________
3. Bandwidth $f_H - f_L$ = ____________

7A.11 VIVA QUESTIONS:

1. Differentiate between voltage amplifier and power amplifier
2. Why power amplifiers are considered as large signal amplifier?
3. When does maximum power dissipation happen in this circuit ?.
4. What is the maximum theoretical efficiency?
5. Sketch wave form of output current with respective input signal.
6. What are the different types of class-A power amplifiers available?
7. What is the theoretical efficiency of the transformer coupled class-A power amplifier?
8. What is difference in AC, DC load line?.
9. How do you locate the Q-point ?
10. What are the applications of class-A power amplifier?
11. What is the expression for the input and output power in class A power amplifier?

EXPERIMENT NO-7(B)

CLASS B POWER AMPLIFIER

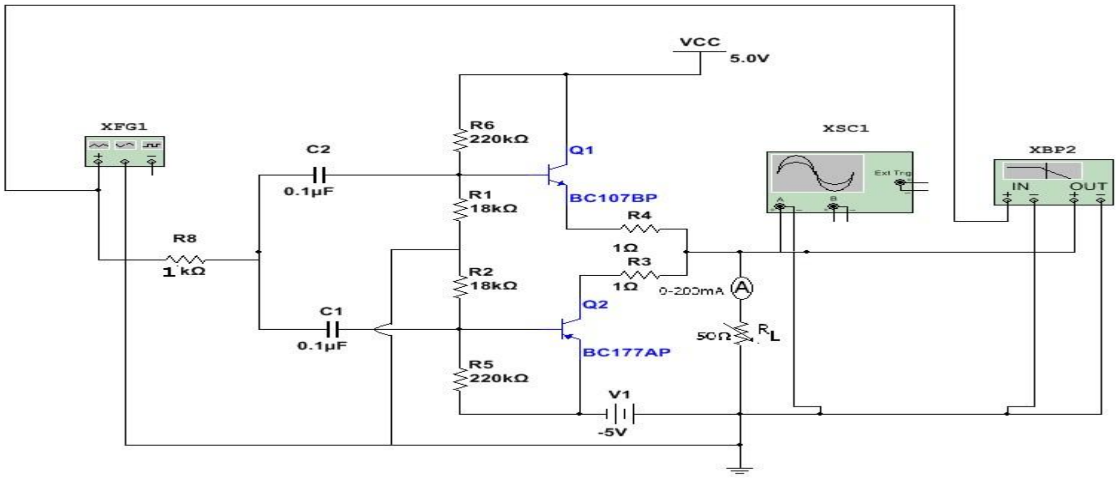
7B.1 AIM: To study the CLASS B Complementary Symmetry amplifier and to calculate its efficiency.

7B.2 SOFTWARE REQUIRED: MultiSim Analog Devices Edition 13.0

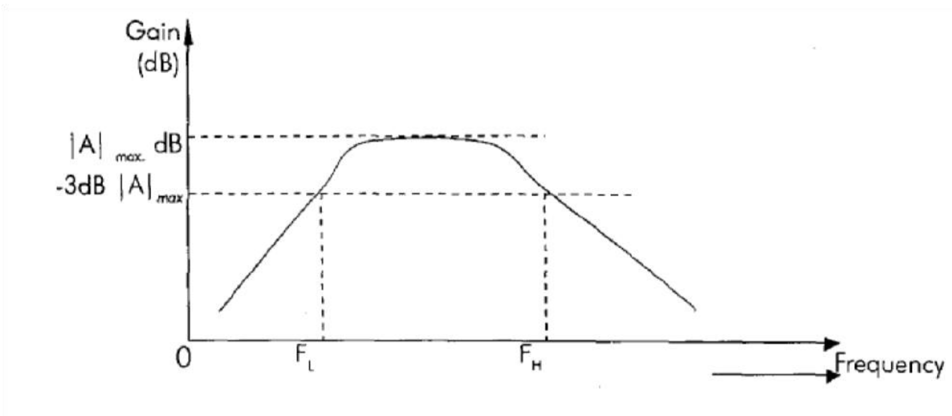
7B.3 COMPONENTS & EQUIPMENT REQUIRED:

S.No	Apparatus	Range/ Rating	Quantity (in No.s)
1.	a) DC Supply voltage. b) NPN Transistor. c) Resistors. d) Capacitor.	12 V BC 107 220KΩ 1KΩ 1Ω 18KΩ 0.1 □F.	1 2 1 1 1 2 2
2.	D.C Milliammeter	0-100mA	1
3.	Bode plotter		1
4.	Function Generator.	0.1 Hz-10 MHz	1

7B.4 CIRCUIT DIAGRAM:



7B.5 EXPECTED GRAPH:



7B.6 THEORY:

Power amplifiers are designed using different circuit configuration with the sole purpose of delivering maximum undistorted output power to load. Push-pull amplifiers operating either in class-B are classAB are used in high power audio system with high efficiency. In complementary-symmetry class-B power amplifier two types of transistors, NPN and PNP are used. These transistors acts as emitter follower with both emitters connected together.

In class-B power amplifier Q-point is located either in cut-off region or in saturation region. So, that only 180o of the input signal is flowing in the output. In complementary-symmetry power amplifier, during the positive half cycle of input signal NPN transistor conducts and during the negative half cycle PNP transistor conducts. Since, the two transistors are complement of each other and they are connected symmetrically so, the name complementary symmetry has come

Theoretically efficiency of complementary symmetry power amplifier is 78.5%.

7B.7 PROCEDURE:

1. Switch ON the CLASS B amplifier trainer.
2. Connect Milliammeter to (A) terminals and DRB to the R_L terminals and fix R_L=50Ω.
3. Apply the input voltage from the signal generator to the V_s terminals.
4. Connect channel 1 of CRO to the V_s terminals and channel 2 across the load.
5. By varying the input voltage, observe the maximum distortion less output waveform and note down the voltage reading.
6. Calculate the efficiency.

7B.8 OBSERVATIONS:

V_s=2v

FREQUENCY	V_o (volts)	I_{dc} (mA)	Efficiency
10 KHz			

7B.9 CALCULATIONS:

Pin=Vcc x Idc

I_{dc}= V_0/R_L

$P_{out} = V_0^2 / 8R_L$

Efficiency= P_o/P_i x100

7B.10 RESULT: Thus efficiency of CLASS B amplifier calculated.

7B.11 VIVA

1.Classfide large signal amplifier based of operating point.

2.state the advantages of push pull class b power amplifier over class b power amplifier .

3. what is harmonic distortion how even harmonic is eliminated using push pull

4. list advantages of complementary symmetry configuration over push pull amplifier.

5. What is covertion efficiency of class B power amplifier.

PC LAB

EXPERIMENT NO.1

A) LINEAR WAVE SHAPING

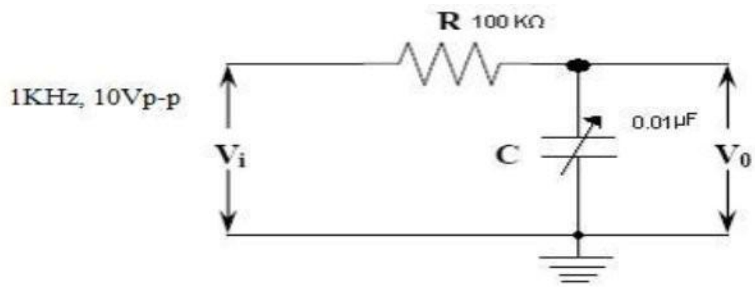
1A.1 AIM

To design low pass RC circuits for different time constants and verify their responses for a square wave input of given frequency.

1A.2 APPARTUS REQUIRED

S.NO	COMPONENT	VALUE	QUANTITY
1	Resistor	100 kΩ	1
2	Capacitor	0.1 µF, 0.01 µF, 0.001 µF	1
3	Digilent analog kit with PC		1
4	Bread Board		1
5	Connecting wires	-	Required

1A.3 CIRCUIT DIAGRAM



RC Low pass circuit

1A.4 THEORY

LowPass RC circuit : The reactance of the capacitor depends upon the frequency of operation. At very high frequencies, the reactance of the capacitor is zero. Hence the capacitor in fig.1.2 acts as short circuit. As a result, the output will fall to zero. At low frequencies, the reactance of the capacitor is infinite. So the capacitor acts as open circuit. As a result the entire input appears at the output. Since the circuit allows only low frequencies, therefore it is called as low pass RC circuit.

1A.5 DESIGN

RC low pass circuit: (Design procedure for RC low pass circuit)

Choose input time period is 1 msec.

i) Long time constant: RC >> T ; Where RC is time constant and T is time period of input signal.

Let RC = 10 T, Choose R = 100kΩ, f = 1kHz.

$C = 10 / 10^3 X\ 100X10^3 = 0.1\mu f$

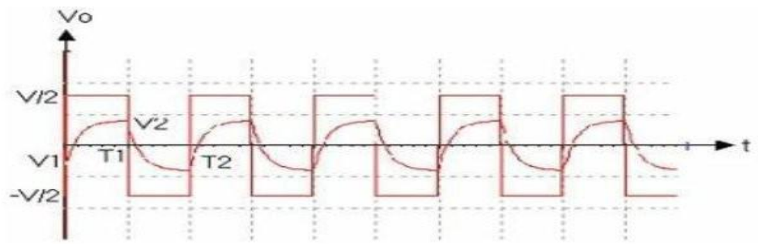
ii) Medium time constant: RC = T

$C = T/R = 1/\ 10^3 X100X10^3 = 0.01\mu f$

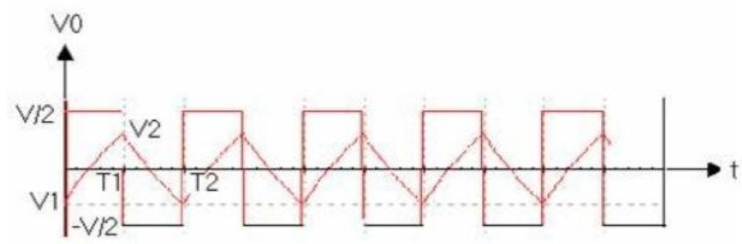
iii) Short time constant: RC << T

RC = T/10; C = T/10R = 1/ 10X103X100X103 = 0.001 μf.

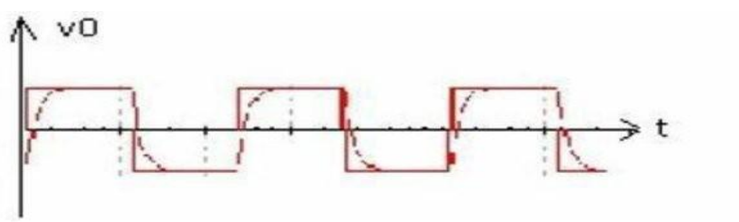
a) RC=T



b) RC >>T



c) RC<< T



1A.6 PROCEDURE

1. Connect the circuit, as shown in figure.
2. Apply the Square wave input to the circuit ($V_i = 10\ V_{P-P}$, f = 1KHz)
3. Calculate the time constant of the circuit by connecting one of the Capacitor provided.
4. Observe the output wave forms for different input frequencies (RC<<T,RC=T,RC> T) as shown in the tabular column for different time constants.
5. Plot the graphs for different input and output waveforms.

1A.7 PRECAUTIONS

1. Avoid loose and wrong connections.
2. Avoid eye contact errors while taking the observations in CRO.

1A.8 OBSERVATIONS

Low pass RC circuit

R	C	τ=RC	Practical time period	Condition
100 KΩ	0.01μF			
100 KΩ	0.01μF			
100 KΩ	0.01μF			

1A.9 Calculations

1A.10 PRE LAB QUESTIONS

1. Name the signals which are commonly used in pulse circuits and define any two of them?
2. Define linear wave shaping?
3. Define attenuator and types of attenuator?
4. Distinguish between the linear and non-linear wave shaping circuits.
5. Define Percentage Tilt and Rise time?

1A.11 LAB ASSIGNMENT

Design low pass filter with a cut-off frequency of 2KHz.

1A.12 POST LAB QUESTIONS

1. Explain the fractional tilt of a high pass RC circuit. Write the Expression.
2. State the lower 3-db frequency of high-pass circuit.
3. Prove that for any periodic input wave form the average level of the steady state output signal from an RC high pass circuit is always zero.
4. Show that a low pass circuit with a time constant acts as Integrator.
5. Name a wave shaping circuit which produces a Ramp wave as an output by taking
6. a step signal as input and draw its output for a sinusoidal wave?

1A.13 RESULT

EXPERIMENT NO.1

B) RC high pass circuit

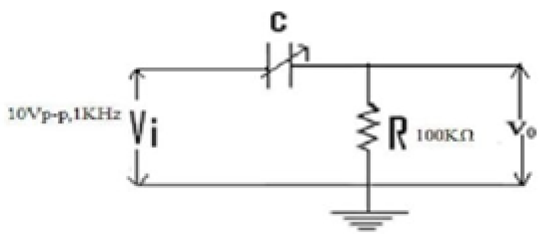
1B.1 AIM

To design high pass RC circuits for different time constants and verify their responses for a square wave input of given frequency.

1B.2 APPARTUS REQUIRED

S.NO	COMPONENT	VALUE	QUANTITY
1	Resistor	100 kΩ	1
2	Capacitor	0.1 µF, 0.01 µF, 0.001 µF	1
3	Digilent analog kit with PC		1
4	Bread Board		1
5	Connecting wires	-	Required

1B.3 CIRCUIT DIAGRAM



RC HIGHPASS FILTER

1B.4 Theory

High Pass RC circuit: The reactance of the capacitor depends upon the frequency of operation. At very high frequencies, the reactance of the capacitor is zero. Hence the capacitor in fig.1.1 acts as short circuit. As a result the entire input appears at the output. At low frequencies, the reactance of the capacitor is infinite. So the capacitor acts as open circuit. Hence no input reaches the output. Since the circuit allows only high frequencies, therefore it is called as high pass RC circuit.

1B.5 DESIGN

RC high pass circuit

Long time constant: RC >> T. Where RC is time constant and T is time period of input signal.

i) Let RC = 10 T, Choose R = 100kΩ, f = 1kHz.

$C = 10 / 10^3 X\ 100X10^3 = 0.1\mu f$

ii) Medium time constant: RC = T

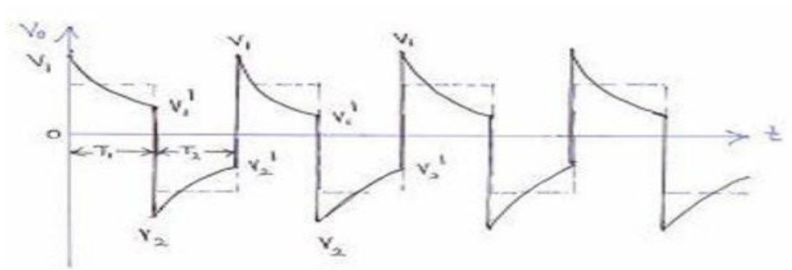
$C = T/R = 1/\ 10^3 X100X10^3 = 0.01\mu f$

iii) Short time constant: RC << T

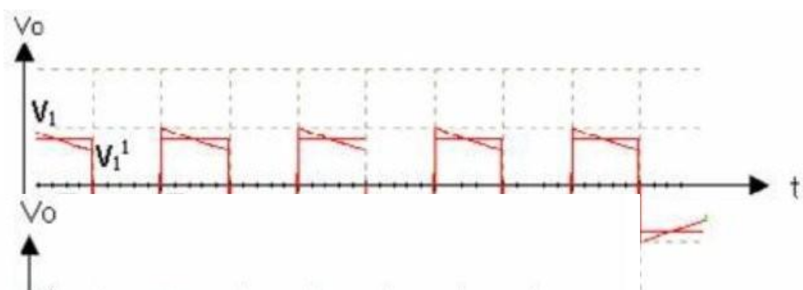
RC = T/10; C = T/10R = 1/ 10X103X100X103 = 0.001 µf.

1B.6 EXPECTED WAVEFORMS

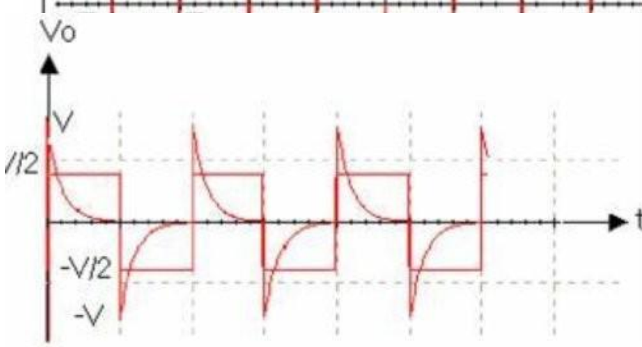
a) RC=T



b) RC >>T



c) RC<<T



1B.7 PROCEDURE

1. Connect the circuit, as shown in figure.
2. Apply the Square wave input to the circuit (Vi = 10 V_{P-P}, f = 1KHz)
3. Calculate the time constant of the circuit by connecting one of the Capacitor provided.
4. Observe the output wave forms for different input frequencies (RC<<T,RC=T,RC>T) as shown in the tabular column for different time constants.
5. Plot the graphs for different input and output waveforms.

1B.8 OBSERVATIONS

R	C	τ=RC	Practical time period	Condition
100 KΩ	0.01μF			
100 KΩ	0.01μF			
100 KΩ	0.01μF			

1B.9 CALCULATIONS

1B.10 PRE LAB QUESTIONS

1. When HP-RC circuit is used as Differentiator?
2. Draw the responses of HPF to step, pulse, ramp inputs?
3. Why noise immunity is more in integrator than differentiator?
4. Why HPF blocks the DC signal?
5. Define time constant?

1B.11 LAB ASSIGNMENT

Design HPF with a cut off frequency 100HZ.

1B.12 POST LAB QUESTIONS

1. Draw the responses of HPF to step, pulse, ramp inputs?
2. Define % tilt and rise time?
3. What is the working principle of high pass and low pass RC circuits for non

sinusoidal signal inputs.

1B.13 RESULT

EXPERIMENT NO: 2

NON LINEAR WAVE SHAPING

A) CLIPPERS

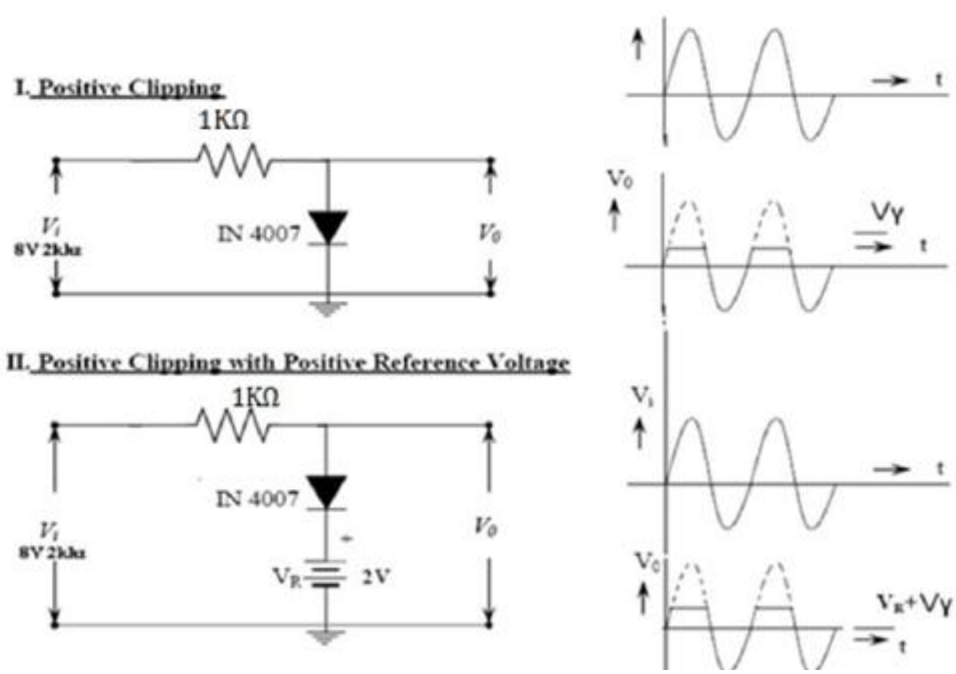
2A.1 AIM

To study the various clipper circuits and to plot the output waveforms for a sinusoidal input signal.

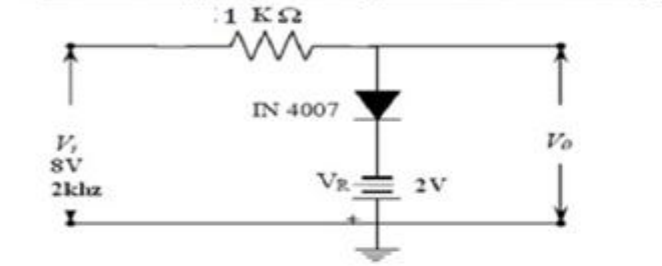
2A.2 APPARATUS REQUIRD

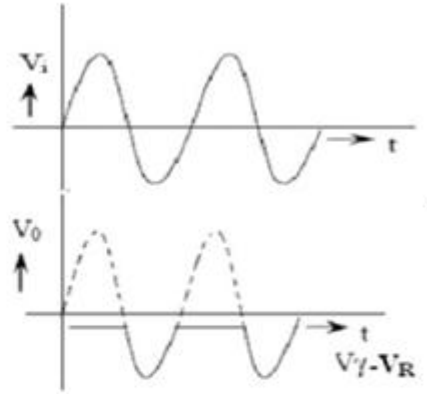
S.NO	COMPONENT	VALUE	QUANTITY
1	Resistor	1 kΩ	1
2	DIODE	IN4007	1
3	Digilent analog discovery kit with PC		1
4	Dual DC Power Supply	0 – 20 V	1
5	Bread Board		1
6	Connecting wires	-	Required

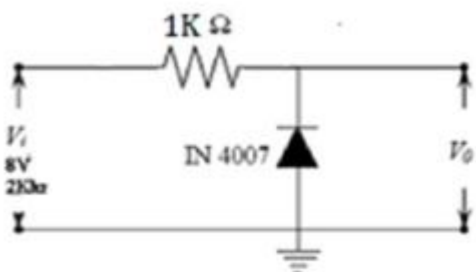
2A.3 CIRCUIT DIAGRAMS& EXPECTED WAVEFORMS

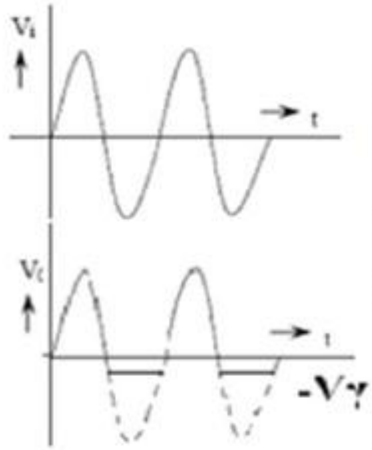


III. Positive Clipping with Negative Reference Voltage

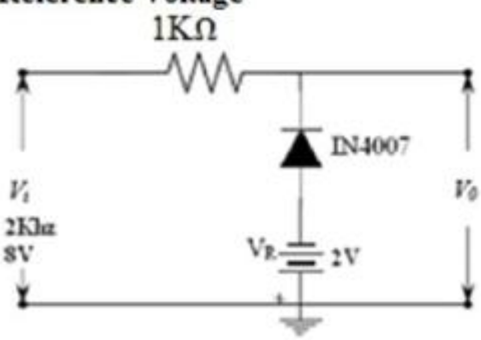


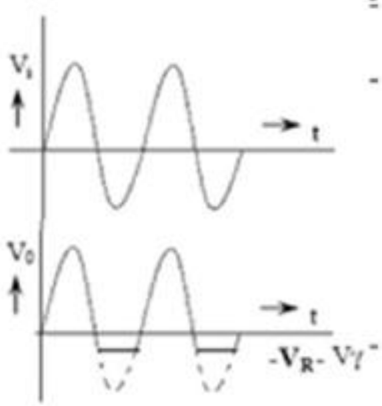


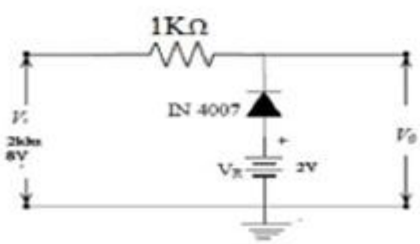


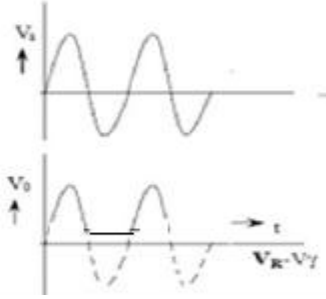


Negative Clipper wth Negative Reference Voltage









2A.4 PROCEDURE

1. Connect the circuit as shown in figure
2. Apply the input Sine wave to the circuit. (8Vp-p, 2 KHz)
3. Switch on the power supply and adjust the output of AF generator to 8V (peak to peak)
4. Observe the input and output waveforms on CRO and note down the readings.
5. Plot the graphs of input Vs output waveforms for different clipping circuits.

2A.5 OBSERVATIONS

S. No.	Type of Clipper	Reference Voltage	Practical Clipping Voltage levels
1	Positive Clipper	0V	
		2V	
		-2V	
2	Negative Clipper	0V	
		2V	
		-2V	

2A.6 PRECAUTIONS

1. Avoid loose and wrong connections.
2. Avoid parallax errors while taking the readings using CRO.

2A.7 RESULT

EXPERIMENT NO: 2

NON LINEAR WAVE SHAPING

B) CLAMPERS

2B.1 AIM

To study the various clamping circuits and to plot the output waveforms for a sinusoidal input of given peak amplitude. (Choose f=1 kHz, V_{p-p} =8V)

2B.2 APPARATUS REQUIRED

S.NO	COMPONENT	VALUE	QUANTITY
1	Resistor	100 kΩ	1
2	Capacitor	0.1 uF	
3	DIODE	IN4007	1
4	Digilent analog discovery kit with PC		1
5	Dual DC Power Supply	0 – 20 V	1
6	Connecting wires	-	Required
7	Bread Board		

2B.3 THEORY

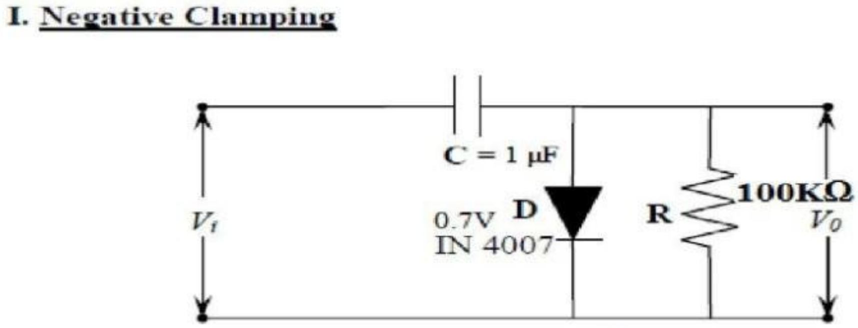
The process whereby the form of sinusoidal signals is going to be altered by transmitting through a non-linear network is called non-linear wave shaping. Non- linear elements in combination with resistors and capacitors can function as clamping circuit. A Clamping circuit is one that takes an input waveform and provides an output i.e a faithful replica of its shape, but has one edge clamped to the voltage reference point. The clamping circuit introduces the d.c component at the output side, for this reason the clamping circuits are referred to as d.c restorer or d.c reinserted.

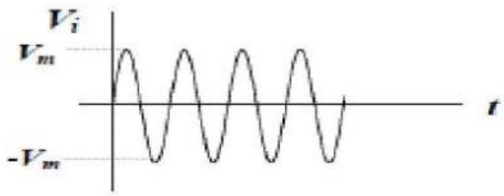
Clamping circuits are classified as two types.

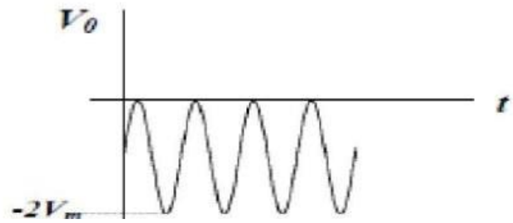
i) Negative Clampers ii) Positive Clampers

2B.4 CIRCUIT DIAGRAM

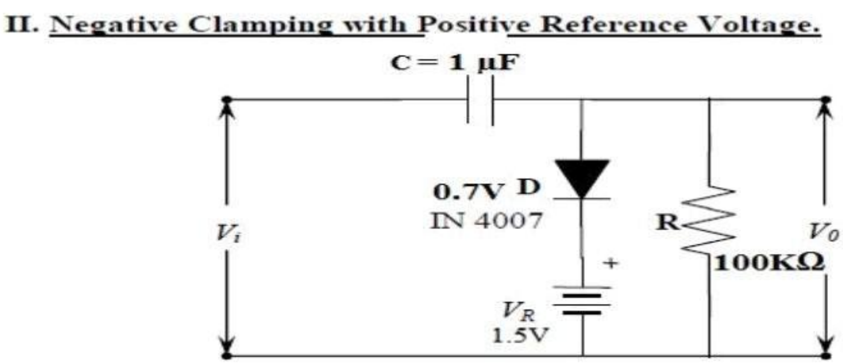
Negative clampers:

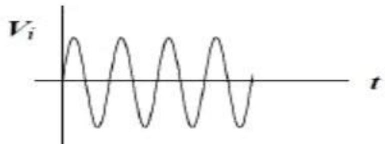


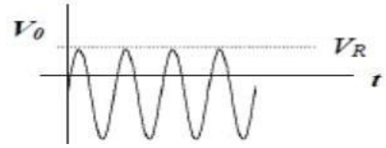




I/P Waveform O/P Waveform

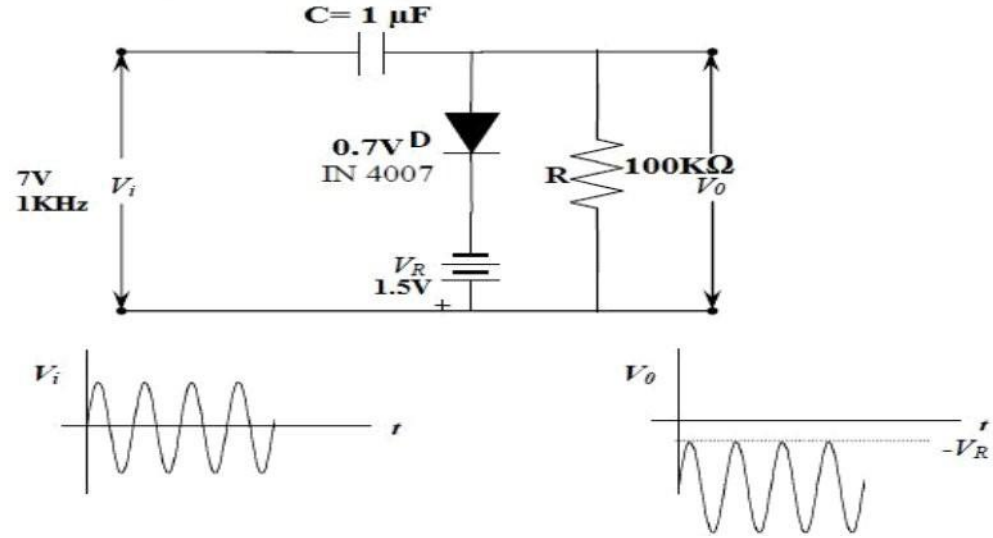




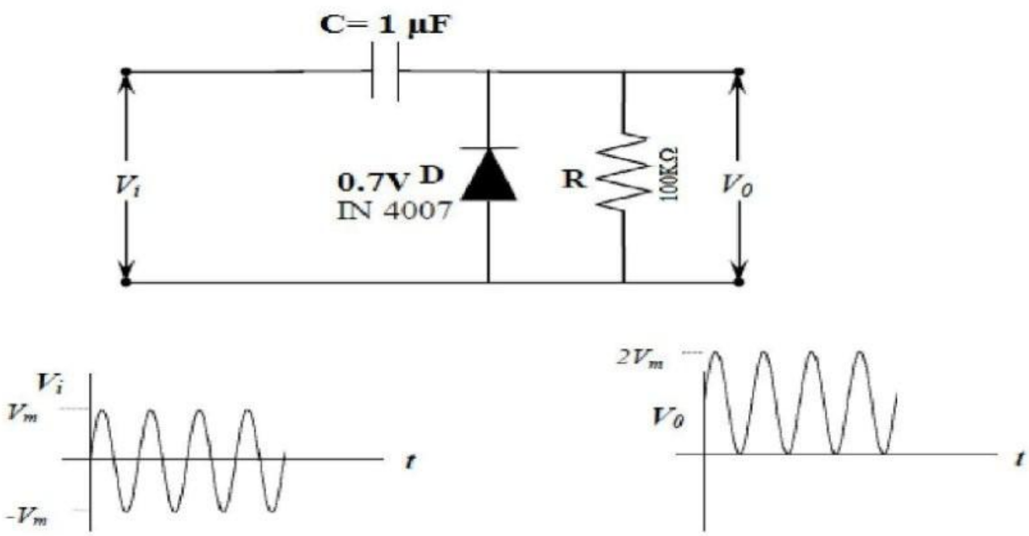


Input waveform output waveform

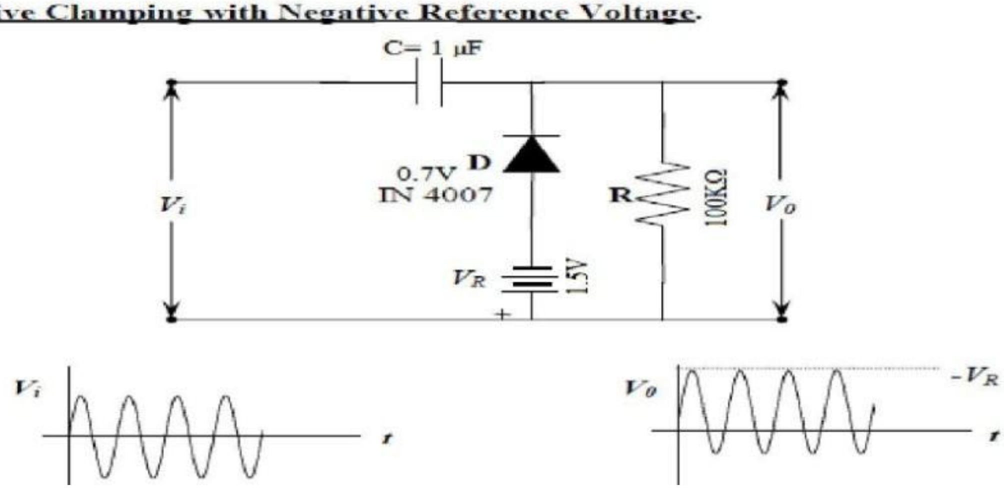
III. <u>Negative Clamping with Negative Reference Voltage.</u>



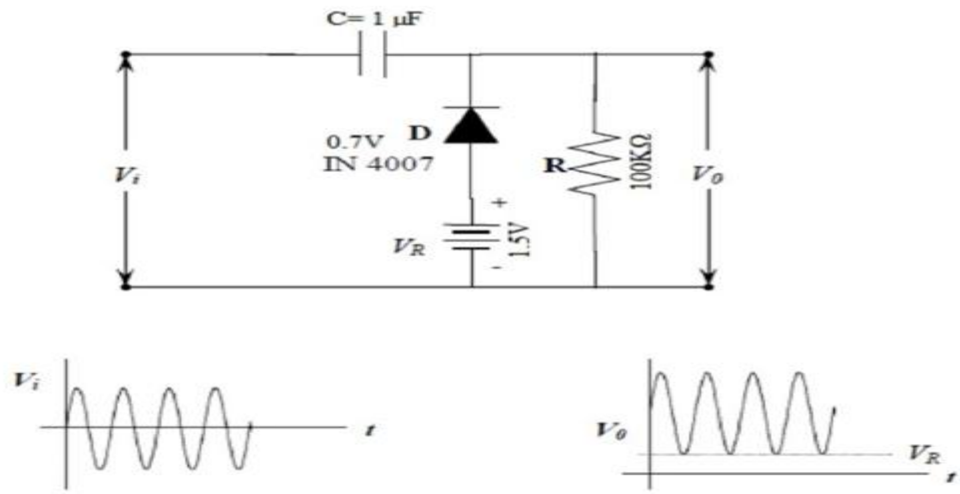
IV. <u>Positive Clamping.</u>



V. <u>Positive Clamping with Negative Reference Voltage.</u>



VI. Positive Clamping with Positive reference Voltage.



2B.5 THEORY

The process whereby the form of sinusoidal signals is going to be altered by transmitting through a non-linear network is called non-linear wave shaping. Non- linear elements in combination with resistors and capacitors can function as clamping circuit. A Clamping circuit is one that takes an input waveform and provides an output i.e a faithful replica of its shape, but has one edge clamped to the voltage reference point. The clamping circuit introduces the d.c component at the output side, for this reason the clamping circuits are referred to as d.c restorer or d.c reinserted.

Clamping circuits are classified as two types.

i) Negative Clampers   ii) Positive Clampers

2B.6 PROCEDURE

1. Connect the circuit as shown in figures
2. Switch on the power supply and adjust the output of AF generator to 8V (peak to peak)
3. Square wave input and observe the output waveforms on CRO and note down the readings.
4. Plot the graphs of input Vs output waveforms for different clamping circuits.

2B.7 OBSERVATIONS

S.No.	Type of Clamper	Reference Voltage	Practical Clamping reference Voltage level
1	Positive Clamper	0V	
		2V	
		-2V	
2	Negative Clamper	0V	
		2V	
		-2V	

2B.8 PRECAUTIONS

1. Avoid loose and wrong connections.

2. Avoid parallax errors while taking the readings using CRO.

2B.9 PRE LAB QUESTIONS

1. What are the applications of clamping circuits?
2. What is the synchronized clamping?
3. Why clamper is called as a dc inserter?
4. What is the function of capacitor?

2B.10 LAB ASSIGNMENT

Design a slicer circuit.

2B.11 POST LAB QUESTIONS

1. What is clamping circuit theorem. How the modified clamping circuit theorem does differs from this?
2. Differentiate –ve clamping circuit from +ve clamping circuits in the above circuits?
3. Describe the charging and discharging of a capacitor in each circuit?
4. What are the effects of diode characteristics on the o/p of the clamper?
5. Which kind of clipper is called a Slicer?

2B.12 RESULT

EXPERIMENT NO: 3

TRANSISTOR AS A SWITCH

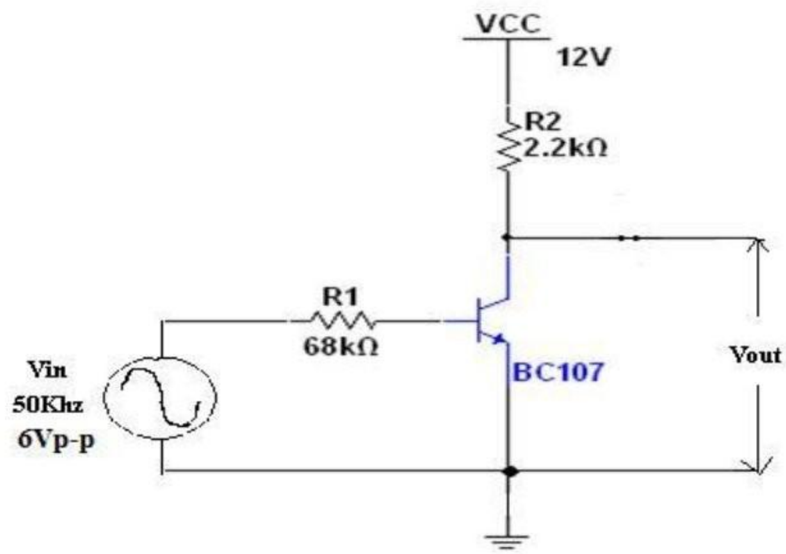
3.1 AIM

To study and observe the switching characteristics of a transistor.

3.2 APPARATUS REQUIRED

S.NO	COMPONENT	VALUE	QUANTITY
1	Resistor	2.2 KΩ, 68 KΩ	1
2	Transistor	BC 107	1
3	Bread Board		1
4	Digilent analog discovery kit with PC		1
5	Connecting wires	-	Required

3.3 CIRCUIT DIAGRAM



3.4 THEORY

The transistor Q can be used as a switch to connect and disconnect the load R_L from the source V_{CC}. When a transistor is saturated, it is like a closed switch from the collector to the emitter. When a transistor is cut-off, it is like an open switch

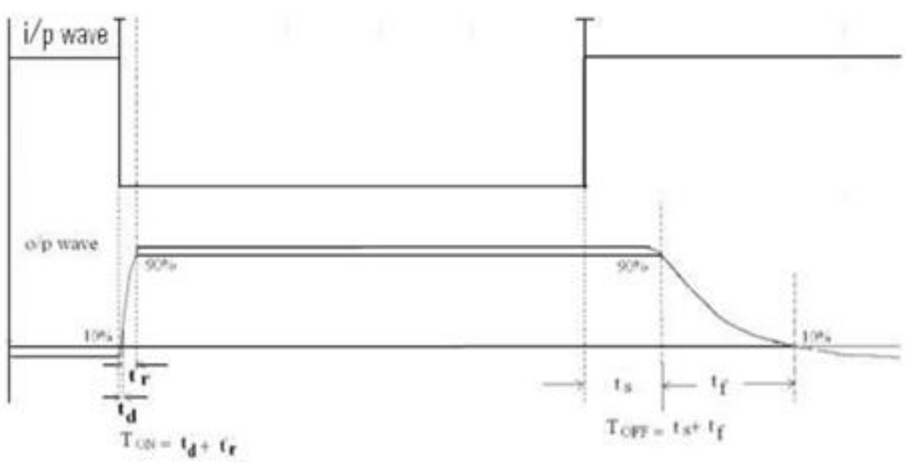
$$V_{CE} = V_{CC}$$

Saturation: The point at which the load line intersects the $I_B = 0$ curve is known as cut-off. At this point, base current is zero and collector current is negligible small i.e., only leakage current I_{CEO} exists. At cut-off, the emitter diode comes out of forward bias and normal transistor action is lost. The transistor appears like a closed switch.*VCE(sat)* ˜ *VCC*. The intersection of the load line and the $I_B = I_{B(sat)}$ is called saturation. At this point base current is $I_{B(sat)}$and the collector current is maximum. At saturation, the collector diode comes out of reverse bias, and normal transistor action is again lost.

3.5 PROCEDURE

1. Connect the circuit as shown in figure.
2. Switch on the power supply and observe the output of the function generator on CRO.
3. Adjust input signal amplitude such that output signal peak-to peak value is less than the Saturation level.
4. Observe output waveforms on CRO and note down the readings.
5. Plot the graphs between input and output waveforms at a giveninput frequency.

3.6 EXPECTED WAVEFORM



3.7 PRECAUTIONS

1. Avoid loose and wrong connections.
2. Aviod parallax error while taking the readings using CRO.

3.8 CALCULATIONS

3.9 PRE LAB QUESTIONS

1. Name the devices that can be used as switches?
2. Draw the Practical and piece-wise linear diode V-I characteristics?
3. Describe the two regions of a diode?
4. Define Forward recovery time and reverse recovery time?

3.10 LAB ASSIGNMENT

Design CE amplifier using C-B bias.

3.11 POST LAB QUESTIONS

1. Explain how a transistor can be used as a switch?
2. Write short notes on Transistor switching times?
3. Define ON time & OFF time of the transistor?

3.12 RESULT

EXPERIMENT NO: 4

ASTABLE MULTIVIBRATOR

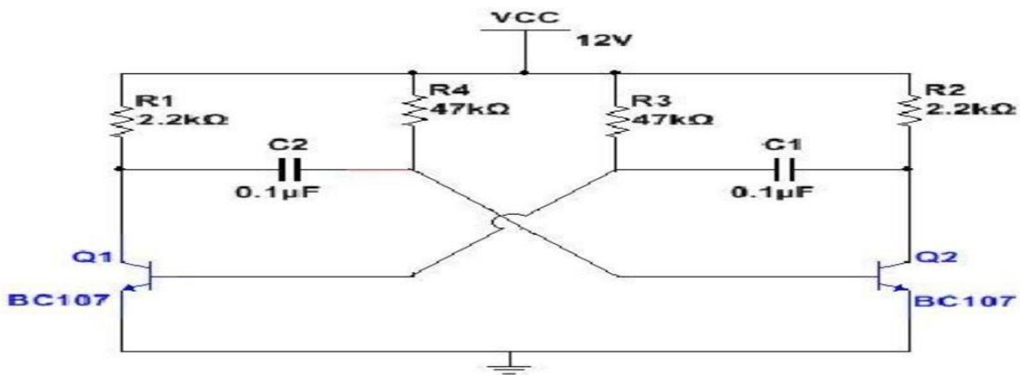
4.1 AIM:

To Study the Characteristics Of Astable Multivibrator Using Transistors.

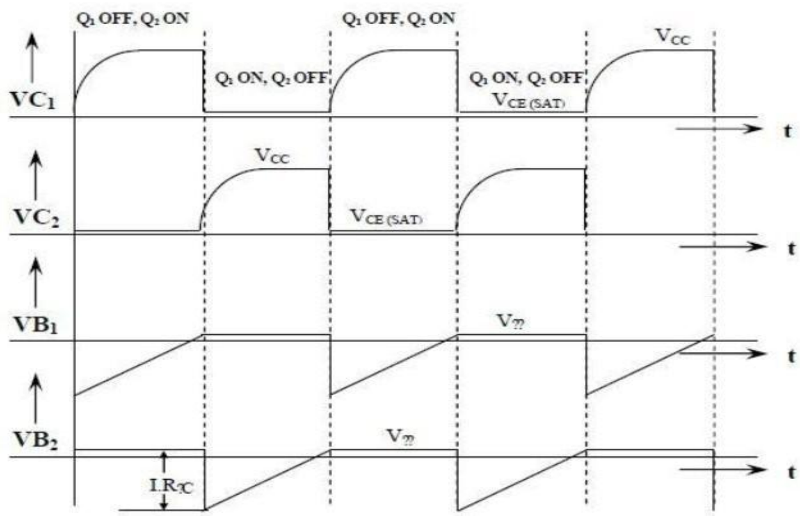
4.2 APPARATUS REQUIRED

S.NO	COMPONENT	VALUE	QUANTITY
1	Resistor	2.2 KΩ, 47 KΩ	1
2	Capacitor	0.1µF	1
2	Transistor	BC 107	2
3	Bread Board		1
4	Digilent analog discovery kit with PC		1
5	Connecting wires	-	Required

4.3 CIRCUIT DIAGRAM



4.4EXPECTEDGRAPH



4.5 THEORY

The Astable circuit has two quasi-stable states. Without external triggering signal the astable configuration will make successive transitions from one quasi-stable state to the other. The astable circuit is an oscillator. It is also called as free running multivibrator and is used to generate “Square Wave”. Since it does not require triggering signal, fast switching is possible.

Operation: When the power is applied, due to some importance in the circuit, the transistor Q_2 conducts more than Q_1 i.e. current flowing through transistor Q_2 is more than the current flowing in transistor Q_1. The voltage V_{C2} drops. This drop is coupled by the capacitor C_1 to the base by Q_1 there by reducing its forward base-emitter voltage and causing Q_1 to conduct less. As the current through Q_1 decreases, V_{C1} rises. This rise is coupled by the capacitor C_2 to the base of Q_2. There by increasing its base- emitter forward bias. This Q_2 conducts more and more and Q_1 conducts less and less, each action reinforcing the other. Ultimately Q_2 gets saturated and becomes fully ON and Q_1 becomes OFF. During this time C_1 has been charging towards V_{CC} exponentially with a time constant $T_1 = R_1C_1$. The polarity of C_1 should be such that it should supply voltage to the base of Q_1. When C_1 gains sufficient voltage, it drives Q_1 ON. Then V_{C1} decreases and makes Q_2 OFF. V_{C2} increases and makes Q_1 fully saturated. During this time C_2 has been charging through V_{CC}, R_2, C_2 and Q_1 with a time constant $T_2 = R_2C_2$. The polarity of C_2 should be such that it should supply voltage to the base of Q_2. When C_2 gains sufficient voltage, it drives Q_2 On, and the process repeats.

4.6 OBSERVATION TABLE

S.NO	OUTPUT VOLTAGES	TRANSISTOR IN ON	TRANSISTOR IS OFF
	VC1		
	VC2		
	VB1		
	VB2		

S.NO	Gate Width (Theoritical)	Gate Width (Practical)

4.7 PROCEDURE

1. Connect the circuit as shown in figure.
2. Observe the output of the circuit using oscilloscope and measure the time period of the signal and compare it with theoretical value by varying dc source v (5v to 10v) in steps (take minimum two readings).
3. Plot the output waveforms on the graph paper for one set of values.
4. Repeat the steps from 1 to 3 with timing capacitor 0.01μF.
5. Connect the circuit as shown in figure 2.
6. Repeat the steps from 1 to 4.

4.8 PRECAUTIONS

1. Avoid loose and wrong connections.
2. Aviod parallax errors while taking the readings using CRO.

4.9 CALCULATIONS

4.10 PRE LAB QUESTIONS

1. What are the other names of Astable multivibrator?
2. Define quasi stable state?
3. Is it possible to change time period of the waveform with out changing R&C?
4. Explain charging and discharging of capacitors in an Astable Multivibrator?
5. How can an Astable multivibrator be used as VCO?

4.11 LAB ASSIGNMENT

Design a astable multivibrator with a gate width of 6.4msec.

4.12 POST LAB QUESTIONS

1. Why do you get overshoots in the Base waveforms?
2. What are the applications of Astable Multivibrator?
3. How can Astable multivibrator be used as a voltage to frequency converter?
4. What is the formula for frequency of oscillations?

4.13 RESULT

EXPERIMENT NO: 5
SCHMITT TRIGGER

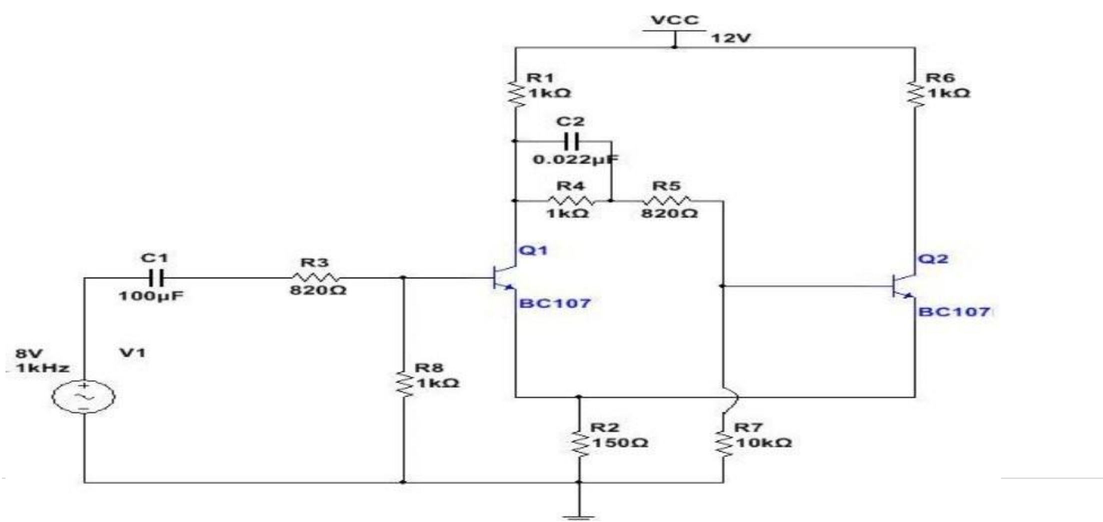
5.1 AIM

To observe and note down the output waveforms of Schmitt trigger using transistors.

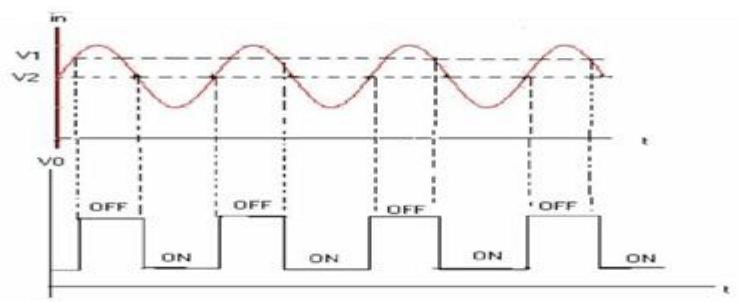
5.2 APPARATUS REQUIRED

S.NO	COMPONENT	VALUE	QUANTITY
1	Resistor	820Ω, 10KΩ 150 Ω, 1 KΩ	1
2	Capacitor	0.022 µF,100 µF	1
3	Transistor	BC 107	2
4	Bread Board		1
5	Digilent analog discovery kit with PC		1
6	Connecting wires	-	Required

5.3 CIRCUIT DIAGRAM



5.4 EXPECTED WAVEFORMS



5.5 THEORY

In digital circuit

s fast waveforms are required i.e, the circuit remain in the active region for a very short time (of the order of nano seconds) to eliminate the effects of noise or undesired parasitic oscillations causing malfunctions of the circuit. Also if the rise time of the input waveform is long, it requires a large coupling capacitor. Therefore circuits which can convert a slow changing waveform(long rise time) in to a fast changing waveform (small rise time) are required. The circuit which performs this operation is known as "Schmitt Trigger". In a Schmitt trigger circuit the output is in one of the two levels namely low or high. When the output voltage is raising the levels of the output changes. When the output passes through a specified voltage V_1 known as Upper trigger level, similarly when a falling output voltage passes through a voltage V_2 known as lower triggering level. The level of the output changes V_1 is always greater than V_2.The differences of these two voltages are known as "Hysteresis".

5.6 OBSERVATIONS

S.NO	OUTPUT VOLTAGES	TRANSISTOR IN ON	TRANSISTOR IS OFF
	VC1		
	VC2		

S.NO	LTP	UTP	V_H
	VC1		
	VC2		

5.7 PROCEDURE

Observation of UTP and LTP

1. Connect the circuit as per the circuit diagram.
2. Apply the square wave input of 1 KHz to the circuit.
3. Switch on the power supply and note down the amplitude and time period for the input square wave.
4. Observe the output waveform and note down the amplitude and time period.
5. Keep R_{e1} and R_{e2} in minimum condition (extremely in anticlockwise direction)
6. Initially keep DC source voltage at zero and observe the output of the Schmitt trigger (it will be in low state i.e. around 6V).

7. Vary the DC source output (i.e input voltage of the Schmitt trigger) slowly from zero.
8. Note down the input voltage value at which the output of the Schmitt trigger goes to high (UTP). Still increase (upto 10V) the input voltage and observe that the output is constant.
9. Now slowly decrease the input voltage and note down the value at which the output of the Schmitt trigger comes back to the original state (LTP).
10. Compare the values LTP and UTP with theoretical values.

5.8 PRECAUTIONS

1. Avoid loose and wrong connections.
2. Avoid parallax errors while taking the readings using CRO.

5.9 CALCULATIONS

5.10 LAB ASSIGNMENT

Design a Schmitt trigger with LTP is 2V and UTP is 4V.

5.11 RESULT

EXPERIMENT NO: 6

UJT RELAXATION OSCILLATOR

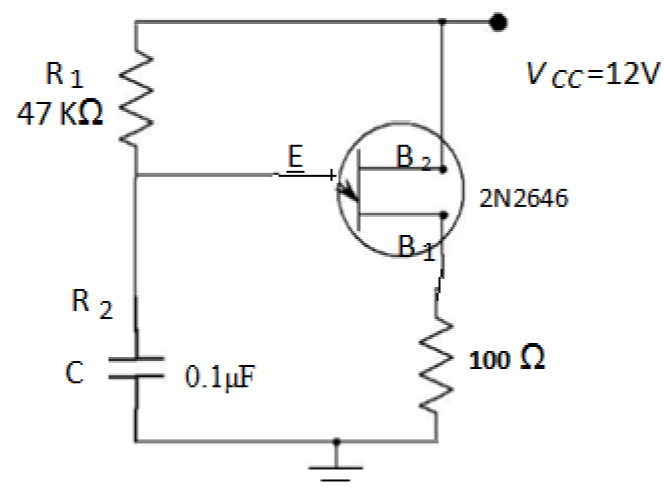
6.1 AIM

To obtain a saw tooth waveform using UJT and test its performance as an oscillator.

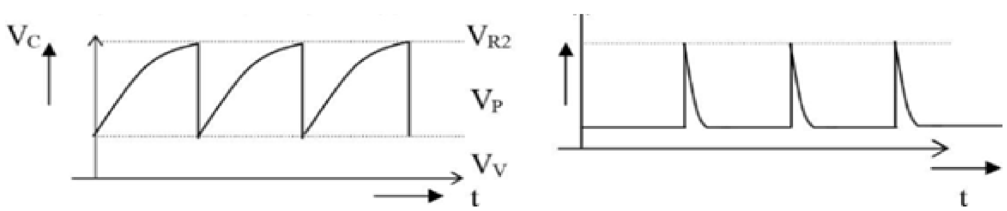
6.2 APPARATUS REQUIRED

S.NO	COMPONENT	VALUE	QUANTITY
1	Resistor	47 KΩ, 100 Ω	1
2	Capacitor	0.1µF	1
3	Transistor	2N2646	1
4	Bread Board		1
5	Digilent analog discovery kit with PC		1
6	Connecting wires	-	Required

6.3 Circuit Diagram



6.4 WAVEFORM



6.5 PROCEDURE

1. Connections are made as per the circuit diagram.
2. The Output Vo is noted, time period is also noted.
3. The theoretical time period should be calculated.
4. T=RTCT ln(1/1-n)
5. The Output at base 1 and base 2 should note.
6. Graph should be plotted and waveforms are drawn for V0, VB1,VB2.

6.6 THEORITICAL CALCULATIONS

$T = R_T C_T \ln(1/(1-n))$ $n = (V_P - V_D)/V_{BB}$

Let η=0.56 , R_T=24.7Kohm ,C_T=0.1microfarad Then T=

6.7 LAB ASSIGNMENT

If a 100nF capacitor is used to generate the timing pulses, calculate the timing resistor required to produce an oscillation frequency of 100Hz. (η = 0.65).

6.8 POST LAB QUESTIONS

1. Define peak voltage?
2. Define valley voltage?
3. Mention the names for negative resistances devices?

6.9 RESULT

EXPERIMENT No-7

Boot –strap sweep Circuit

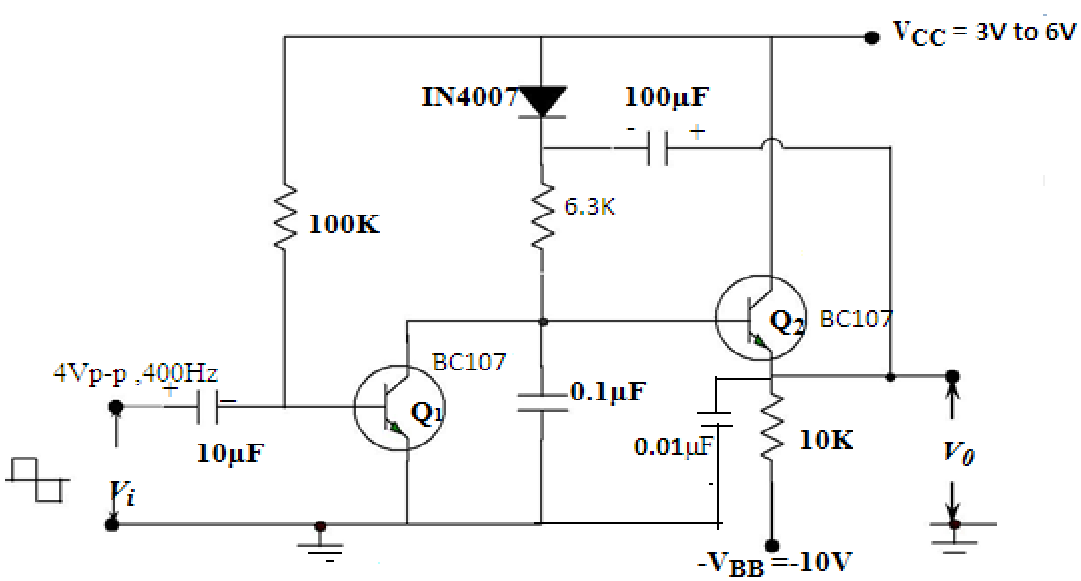
7.1 AIM

To design a Boot-strap Sweep and obtain a sweep wave form.

7.2 APPARATUS REQUIRED

S.NO	COMPONENT	VALUE	QUANTITY
1	Resistor	100 KΩ, 6.3KΩ, 10 KΩ	1
2	Capacitor	0.1µF, 0.01µF, 100 µF, 10µF	1
3	Transistor	BC107	2
4	Diode	IN4007	1
5	Bread Board		1
6	Digilent analog discovery kit with PC		1
7	Connecting wires	-	Required

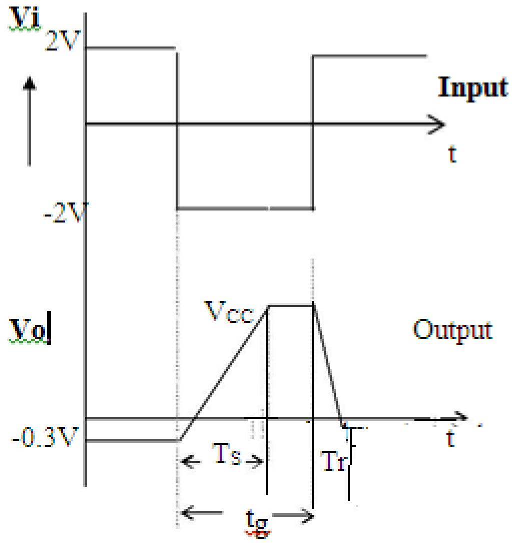
7.3 CIRCUIT DIAGRAM



7.4 PROCEDURE

i. Connect the circuit as show in figure.
ii. Apply the square wave or rectangular wave from at the input terminals.
iii. Connect the CRO at output terminals now plug the power card into line Switch on and observe the power indication.
iv. As motioned in circuit practical calculation .observe and record the Output. waveforms from CRO and compare with theoretical values.

7.5 EXPECTED WAVEFORMS



7.6 LAB ASSIGNMENT

Design Boot-strap Sweep Circuit with sweep amplitude of 8V, with sweep interval of 1ms neglect flyback time and $e_s = 0.25$.

7.7 POST LAB QUESTIONS

1. What are the various methods of generating time base wave-form?
2. Which amplifier is used in Boot-strap time base generator?
3. Which type of sweep does a bootstrap time-base generator produce?

7.8 RESULT

EXPERIMENT NO: 8

COMPARATOR

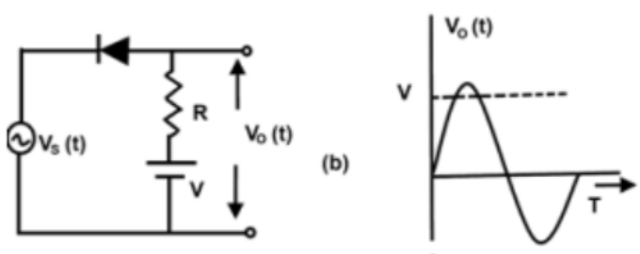
8.1 AIM

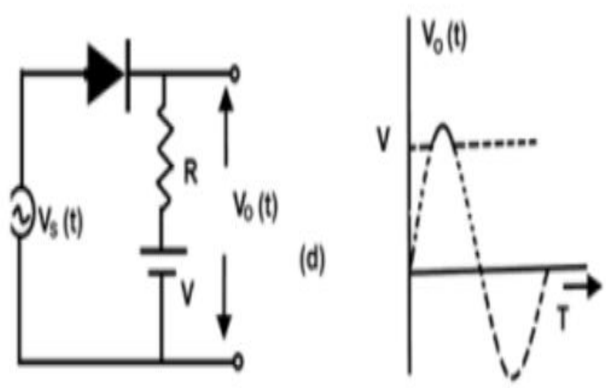
To Design a comparator circuit and plot the response with sinusoidal waveform with $8V_{P-P}$ and 2KHz

8.2 APPARATUS REQUIRED

S.NO	COMPONENT	VALUE	QUANTITY
1	Resistor	1KΩ	1
2	Diode	IN4007	1
3	Bread Board		1
4	Digilent analog discovery kit with PC		1
5	Connecting wires	-	Required

8.3 CIRCUIT DIAGRAM & WAVEFORMS:





8.4 PROCEDURE

1. Connect the circuit as shown in figure
2. Apply the input Sine wave to the circuit. (8Vp-p, 2 KHz).
3. Switch on the power supply and adjust the output of AF generatorto 8V.
4. Observe the input and output waveforms on CRO and note down the readings.
5. Plot the graphs of input Vs output waveforms for different clipping circuits.

8.5 OBSERVATIONS

Sl No.	Type of Clipper	Reference Voltage	Practical Clipping Voltage levels
1	Positive Clipper	0V	
		2V	
		-2V	
2	Negative Clipper	0V	
		2V	
		-2V	

8.6 PRECAUTIONS

1.Avoid loose and wrong connections.
2.Avoid parallax errors while taking the readings using CRO.

8.7 RESULT