Exp. No.: 01 Date: 30/01/2023	VOLTAGE REGULATOR POWER SUPPLY USING LM317

OBJECTIVES:

(i) To construct a voltage regulator using LC317

(ii) To construct and test the application circuit using the power supply.

AIM:

(i)To design and construct a voltage using LM317.

(ii)To construct the blinking LED using power supply.

APPARATUS REQUIRED:

S.No.	Components	Value/Range	Quantity	Specification/Remark
1	Resistors	220Ω	01	1W,5%tolerance
		1KΩ	02	1W,5%tolerance
		1mΩ	01	1/4 W,1% tolerance
2	Potentiometer	5KΩ	01	_
3	Capacitors	2200µF	01	Electrolytic
		0.01 µF	01	Electrolytic
		0.3 µF	01	Electrolytic
		10 µF	01	Electrolytic
		2.2 µF	01	Electrolytic
4	Transformer	2:1	01	230v
5	IC	LM317	01	-
6	Transistor	BC547, BC558	Each 01	-
6	Bread Board	-	01	-
7	LED	-	01	-
8	DIODE	1B4B42	04	-
9	AC Power	230V	-	-

THEORY:

LM317 is a very famous voltage regulator IC available in many different packages. Today we are going to discuss about LM317 pinout, equivalent, uses, features and other details about this IC. Previously we have discussed some fixed voltage regulator ICs like LM7805, LM7806, LM7809, LM7812 and LM7815. All these ICs are designed to provide a fixed output voltage. LM317 is a widely used adjustable positive voltage regulator IC available in many different packages. The main reason for its wide usage is because it contains entire adjustable regulated power supply circuitry in a single chip due to which one can easily make a low cost and reliable adjustable power supply of 1.5V to 3V DC output voltage with up to 1.5A output current capability and very low external components.

CIRCUIT DESCRIPTION:

VOLTAGE REGULATOR USING LM1084

The LM1084 can be configured in a variety of ways to produce a regulated output voltage. The output voltage is set by a resistive divider network connected between the output and the adjustable terminal. The resistors in the divider network determine the regulated output voltage, with the formula Vout = 1.25 x (1 + R2/R1).

It is important to choose the right components for the resistive divider network to ensure stability and to minimize the output voltage ripple. A ceramic capacitor of 10uF to 100uF should be connected between the input and ground, as close as possible to the regulator, to reduce output voltage ripple. Figure 1.1 has been explain the process of voltage regulator power supply.

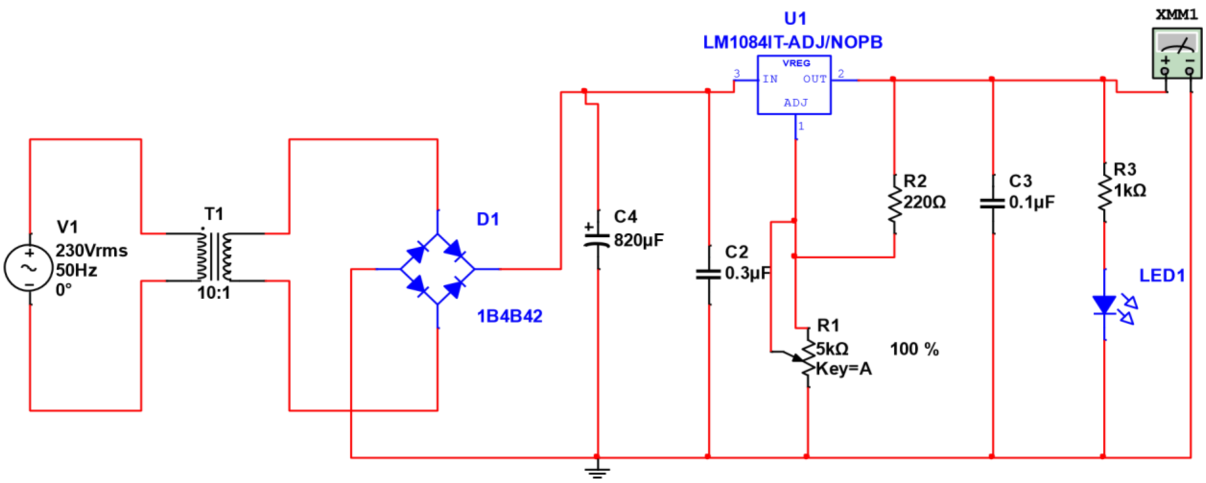

Fig.no:1.1 Circuit diagram of adjustable voltage regulator

PROCEDURE:

1.Ensure the power supply is OFF before giving the connections.

2.Check the component status and identify the terminals.

3.Connect the input voltage to the input terminal of the LM317.

4.Connect the output terminal of the LM317 to the load.

5.Connect the adjustable terminal of the LM317 to the resistive divider network, consisting of two resistors R1 and R2.

6.Connect a ceramic capacitor of 10uF to 100uF between the input and ground, as close as possible to the regulator.

BLINKING LED:

- Dancing LED circuit can be used for any visual sign indication in any highways or it can be used in advertisement hoarding also.
- LED blinking circuit can be used in signaling purpose (It can be used as signal for help, if you are in danger)
- LED blinking circuit can be used as flashing beacon.
- LED blinking circuit can be used as vehicle indicator when it is broke down in the middle of the road. It can be used in operation theaters or offices as an indication that you are engaged in work. The application of the voltage regulator power supply has shown in fig 1.2.

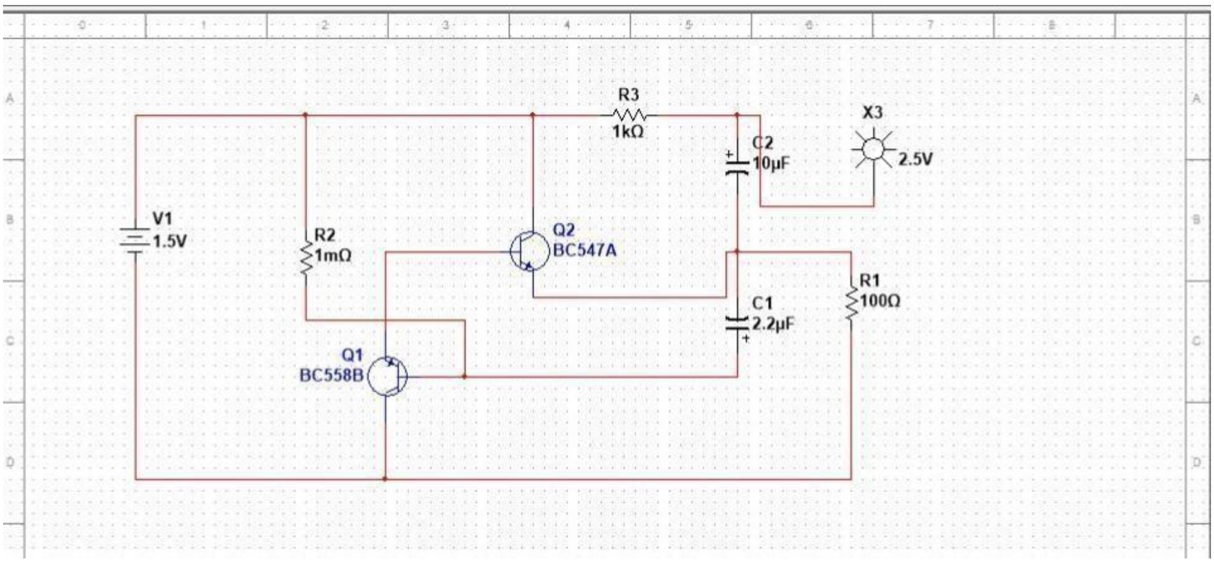

Fig.no:1.2 Blinking LED using power supply

TECHNICAL SPECIFICATION:

LM317:

The LM317 device is an adjustable three-terminal positive-voltage regulator capable of supplying more than 1.5 A over an output-voltage range of 1.25 V to 37 V. It requires only two external resistors to set the output voltage. The device features a typical line regulation of 0.01% and typical load regulation of 0.1%.

The LM317 is an adjustable 3−terminal positive voltage regulator capable of supplying in excess of 1.5 A over an output voltage range of 1.2 V to 37 V. This voltage regulator is exceptionally easy to use and requires only two external resistors to set the output voltage. Further, it employs internal current limiting, thermal shut down and safe area compensation, making it essentially blow−out proof. The LM317 serves a wide variety of applications including local, on card regulation. This device can also be used to make a programmable output regulator, or by connecting a fixed resistor between the adjustment and output, the LM317 can be used as a precision current regulator.

- Output Current in Excess of 1.5 A
- Output Adjustable between 1.2 V and 37 V
- Internal Thermal Overload Protection
- Internal Short Circuit Current Limiting Constant with Temperature
- Output Transistor Safe−Area Compensation
- Floating Operation for High Voltage Applications
- Eliminates Stocking many Fixed Voltages.

5K OHM POTENTIOMETER:

Type: Rotary Radio POT.

Available in different resistance values like 500Ω, 1K, 2K, 5K, 10K, 22K, 47K, 50K, 100K, 220K, 470K, 500K, 1 M.

Power Rating: 0.3W.

Maximum Input Voltage: 200Vdc.

Rotational Life: 2000K cycles.

STEP DOWN TRANSFORMER:

A step-down transformer is an electrical device that is used to convert a high voltage, low current AC power supply into a low voltage, high current AC power supply. It is called a "stepdown" transformer because it steps down the voltage from the input to the output.

The basic principle behind the operation of a step-down transformer is electromagnetic induction. A step-down transformer consists of two coils of wire wound around a common magnetic core. The coil that is connected to the AC power source is called the primary coil, while the coil that is connected to the load is called the secondary coil. When an AC voltage is applied to the primary coil, it creates an alternating magnetic field around the coil. This magnetic field induces a voltage in the secondary coil, which is proportional to the number of turns in the secondary coil.

When an AC voltage is applied to the primary coil, it creates an alternating magnetic field around the coil. This magnetic field induces a voltage in the secondary coil, which is proportional to the number of turns in the secondary coil.

OUTPUT ANALYSIS:

Hardware output of power supply

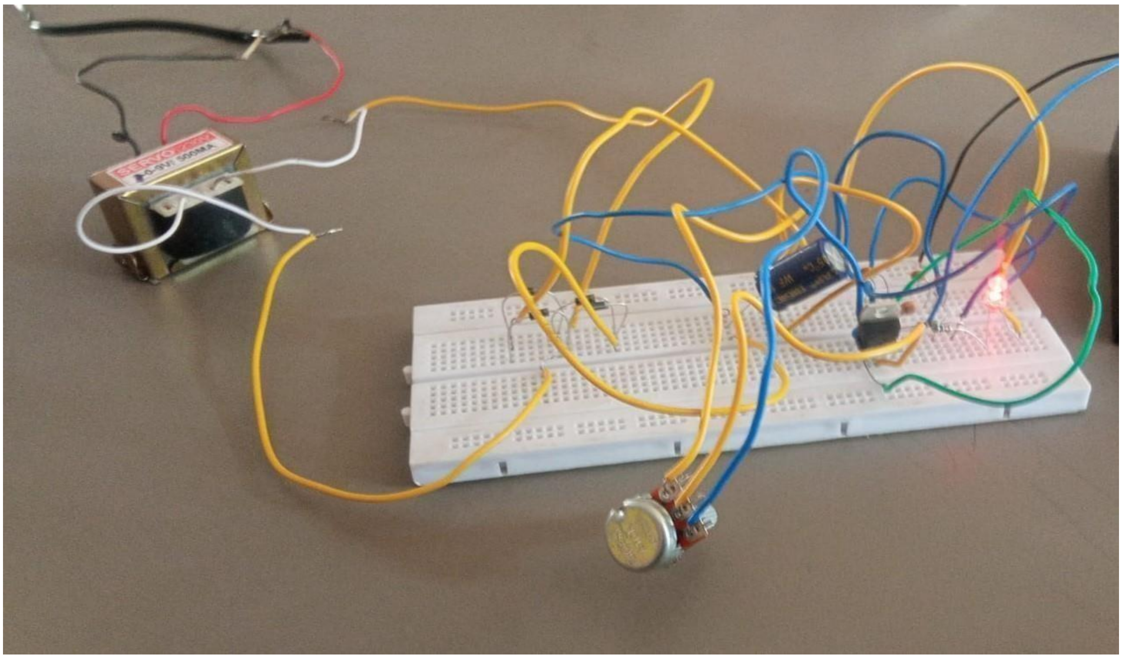

Figure:1.3 Output of LED glowing

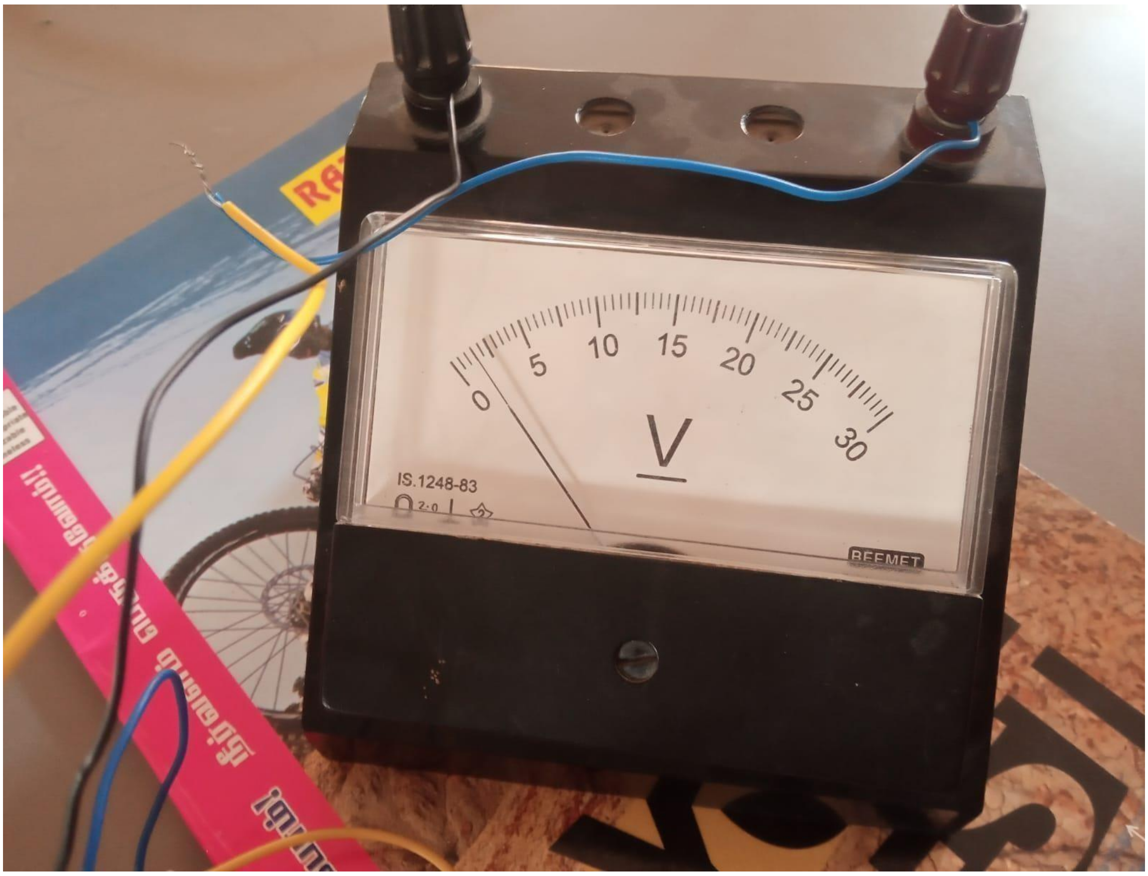

Figure:1.4 Output of voltage display

PRE LABQUESTIONS:

1.What are the components of power supply?

The components used in this power supply circuits are

1)Diodes,

2)Resistors,

3)Capacitors,

4) IC's

5) Transformers

2.What is IC317?

The LM317 is a precision voltage regulator that can provide both positive and negative output voltages.

3.What is a bridge rectifier?

A bridge rectifier is an electronic component that converts an alternating current (AC) into a direct current (DC). It is essentially a circuit made up of four diodes arranged in a specific configuration that allows current to flow in one direction while blocking it in the other.

POST LAB QUESTIONS

1.How does the transformer works?

Transformers are found everywhere Alternating Current (AC) electrical energy is used. A transformer is an electrical device that trades voltage for current in a circuit, while not affecting the total electrical power. This means it takes high-voltage electricity with a small current and changes it into low-voltage electricity with a large current, or vice versa. One thing to know about transformers is that they only work for Alternating Current (AC).

2.What is the difference between AC and DC current?

The primary difference between AC and DC current is in the direction that the current flows. In DC current, the flow of electric charge is always in one direction, from the positive (+) to the negative (-) terminal. In contrast, in AC current, the flow of electric charge periodically changes direction, oscillating between positive and negative.

3.List the practical applications and limitations of power supply applications:

The primary function of a power supply is to convert electric current from a source to the correct voltage, current, and frequency to power the load. As a result, power supplies are sometimes referred to as electric power converters.

LIMITATIONS:

These limitations to power supplies include size, high heat loss, and lower efficiency levels when compared to a switch-mode power supply. The problem with linear power supply units, when used in a high power application, is that it requires a large transformer and other large components to handle the power.

RESULT:

Thus the power supply of 1.5V has been designed and the output has been verified successfully. For 1.5V of blinking and flashing LED indicator circuit was also constructed and the output was verify.

MARK ALLOCATION:

S. No	Parameters	Mark Allotted	Mark Awarded
1	Circuit Design/ code developing and debugging / Trouble shooting	0-3	
2	Implementation and Demonstration	0-3	
3	Discussion	0-3	
4	Report writing & Presentation	0-3	
5	Contribution & Team Dynamics	0-3	
Total		15	

Signature of Lab In-charge

Exp. No: 02 Date:	LOW NOISE MICROPHONE PRE-AMPLIFIER WITH NE5534 IC

OBJECTIVES:

(i) To construct the low noise high performance amplifier circuit using NE5534 IC

(ii) To construct the headphone amplifier circuit using 3transistor.

AIM:

(i) To design a low noise high performance frequency amplifier circuit using NE5534 IC.

(ii) To design the headphone amplifier circuit using 3transistor.

APPARATUS REQUIRED:

S.No.	Components	Value/Range	Quantity	Specification/Remark
1	Resistors	10KΩ	03	0.5W,5%tolerance
		68KΩ	03	¼ W, 5%tolerance
		100kΩ	01	
		330KΩ	01	
		4.7KΩ,3.3KΩ,1.5KΩ ,2.2KΩ	Each 1	
2	IC	NE5534	01	-
3	Capacitors	10μF	02	Electrolytic
		1μF	02	Electrolytic
		100μF	02	Electrolytic
		470μF	01	Electrolytic
4	Electret mic	-	01	-
5	Audio jack	-	01	-
6	Transistor	BC237, BC239	01	-
7	Battery	9v-10v	01	-
8	Bread Board	-	01	-

THEORY:

In a communication system, the receiver section needs amplification for the weak signal which is received from the antenna. So this amplification can be accomplished through the main component called Low Noise Amplifier or LNA. The characteristics of this amplifier can be described by certain parameters like gain, noise figure, chip area, linearity, power consumption & bandwidth.

The block diagram of the low noise amplifier is shown below. The designing of a low noise amplifier can be done by using a common gate amplifier, active inductor & common drain stage. Generally, the common gate amplifier is mainly used at the input stage whereas the common drain amplifier is used at the output stage to provide the best input as well as output matching. The low noise amplifier is bound with particular characteristics like gain & noise figure but the selection of LNA mainly depends on some specific parameters like power supply, bandwidth, chip area & linearity.

CIRCUIT DESCRIPTION:

The main function of this low noise microphone pre-amplifier circuit is to amplify the weak audio signals. Once an audio source like the microphone has less sound range then this circuit is helpful in amplifying that low signal or weak signals & transmits that signal for further amplification. The above circuit can be designed with an IC instead of a transistor because this IC will provide a much better distortion filter. This low noise amplifier IC includes low noise, high unity-gain, low power consumption with short circuit protection, and low harmonic distortion.

The mic preamplifier specifications may change based on the mic used to capture the audio signals & the type of sound being recorded. The essential specifications to be considered within a pre-amplifier circuit are noise, distortion & gain. This circuit operates with a 9V to 12V battery. Here, Electret Mic generates audio signals that are transmitted to the input of IC. Here you can use any audio source instead of a microphone. This circuit’s overall voltage gain mainly depends on the resistors connected at the non-inverting terminal of the IC. The output signal which is generated from this IC can be transmitted through one more capacitor to filter out any noise. This circuit is very useful where low noise audio preamplifier is necessary like live music, cell phones, soundcard within laptops, and audio recording application. Figure 2.1 has been clearly explain the process of low noise pre amplifier circuit.

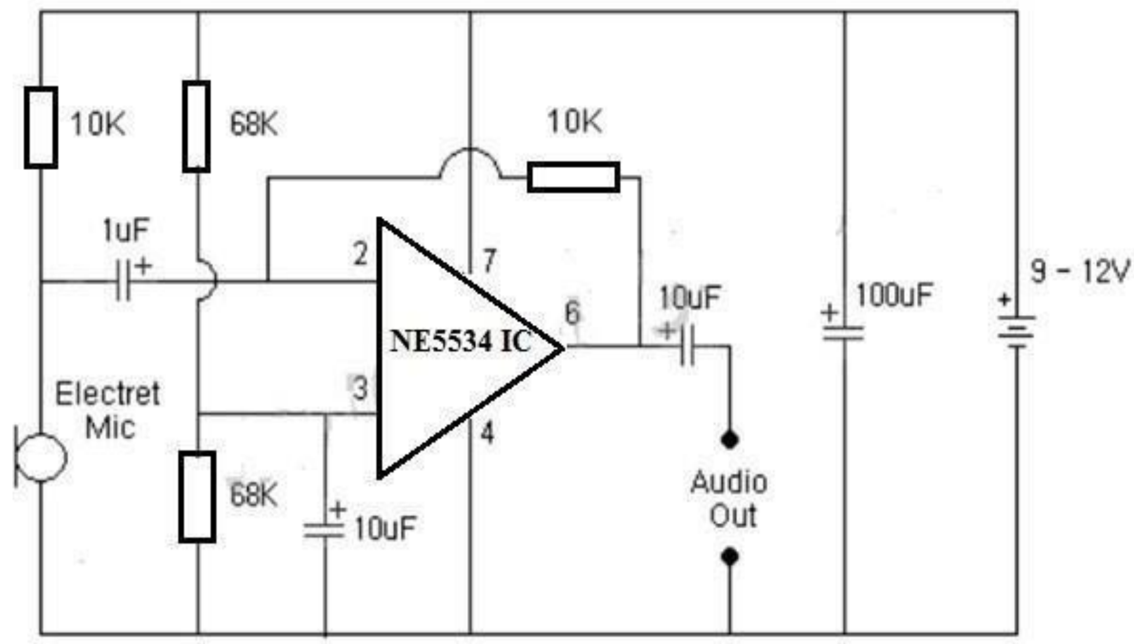

Fig.no:2.1 Circuit diagram low noise high performance amplifier

PROCEDURE:

1.Ensure the power supply is OFF before giving the connections.

2.Check the component status and identify the terminals.

3.Make the connection about the reference of circuit diagram.

4.Connect the input voltage to the input terminal of the NE5534.

5.Connect the output terminal of the NE5534 to the audio out.

6.Connect the of the NE5534 to the resistive divider network, consisting of two resistors R1 and R2.

HEADPHONE AMPLIFIER CIRCUIT USING 3 TRANSISTORS:

A headphone amplifier is a relatively low-power amplifier that boosts the low-voltage audio signal from a source device (be it a turntable, laptop, or smartphone) to a sufficient level, such that it converts (or transduces) into sound waves by the speakers inside your headphones.

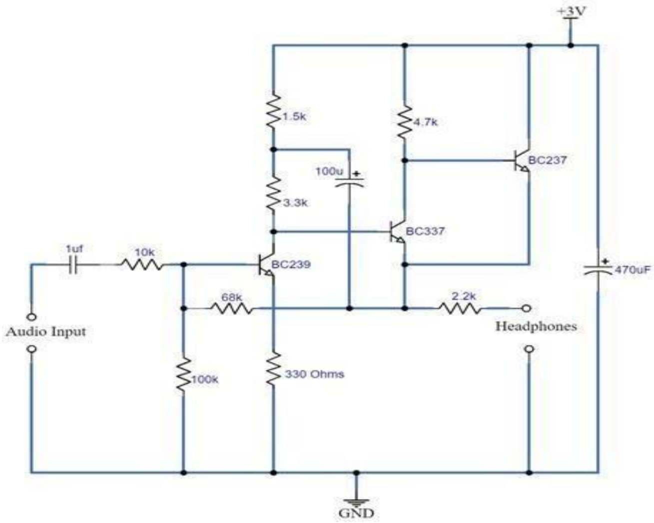

Figure:2.2 Headphone amplifier circuit diagram

It works like the amps which power full-sized speakers but also operate at a lower scale. In this project, we are going to build a simple headphone amplifier circuit using some transistors.

A Headphone amplifier circuit is integral when it comes to mass media production & recording. They are small, easy to use and design & provide low power consumption. The application of the low noise pre amplifier circuit has shown in fig 2.2

TECHNICAL SPECIFICATION:

IC NE5534:

The NE/SA/SE5534/5534A are single high-performance low noise operational amplifiers. Compared to other operational amplifiers, such as TL083, they show better noise performance, improved output drive capability, and considerably higher small-signal and power bandwidths. This makes the devices especially suitable for application in high quality and professional audio equipment, in instrumentation and control circuits and telephone channel amplifiers. The op amps are internally compensated for gain equal to, or higher than, three. The frequency response can be optimized with an external compensation capacitor for various applications (unity gain amplifier, capacitive load, slew rate, low overshoot, etc.).

FEATURES:

- Small-Signal Bandwidth: 10 MHz
- Output Drive Capability: 600, 10 VRMS at VS = 18 V
- Input Noise Voltage: 4 nV Hz
- DC Voltage Gain: 100000
- AC Voltage Gain: 6000 at 10 kHz
- Power Bandwidth: 200 kHz
- Slew Rate: 13 V/s
- Large Supply Voltage Range: 3.0 to 20 V

ELECTRET MICROPHONE:

The resistor R1 (10K) is used to limit the current flowing through the microphone (maximum should be 0.5mA) and the capacitor C1 (1uF) is used to filter the DC noise that might be coupled along with the analog electrical signals (output). Also, note that the capacitor is also polarity sensitive and the positive should be connected to microphone Output pin. This capacitor is rated for 20Hz to 16,000Hz so any sound waves in that range will be picked up the microphone.

FEATURES:

- Operating Voltage: 2V to 10V
- Current consumption: 0.5mA (max)
- Recommended operating voltage: 2V
- Operating Frequency: 20Hz to 16,000Hz
- Impedance: <2.2kΩ

OUTPUT ANALYSIS:

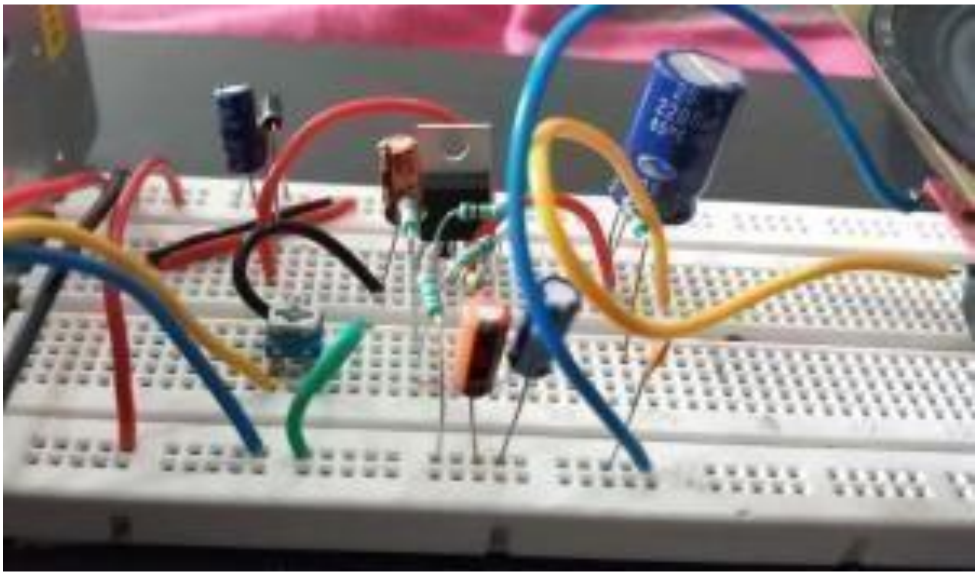

Figure:2.3 Output for low noise microphone preamplifier circuit

PRELABQUESTIONS:

1. How to design a Low noise amplifier?

A low noise amplifier can be designed using either a negative feedback topology or one without any feedback. The former approach is more commonly used because it can provide better performance when compared to an equivalent amplifier without feedback.

2. What is IC NE 5534?

The NE5534, NE5534A, SA5534, and SA5534A devices are high-performance operational amplifiers combining excellent dc and ac characteristics. Some of the features include very low noise, high output-drive capability, high unity-gain and maximum-output-swing bandwidths, low distortion, and high slew rate.

3. What is use of electret microphone?

An electret microphone is a type of electrostatic capacitor-based microphone, which eliminates the need for a polarizing power supply by using a permanently charged material. Another term for an electret microphone is a 'permanently polarized condenser' or 'permanently biased condenser'. However, electret microphones do feature an internal preamp, which does need power.

POST LAB QUESTIONS:

1. What is the purpose of a low noise amplifier?

- To increase the signal-to-noise ratio
- To increase the gain of an amplifier
- To remove unwanted noise from an electromagnetic wave • To attenuate the input of an amplifier.

2. What is the difference between noise and interference?

Noise is unwanted signal that can interfere with the operation of a circuit. It is usually caused by thermal agitation, voltage fluctuations, or other electrical effects. Interference is an unwanted signal that interferes with the operation of another circuit.

3. What is the Advantage and disadvantage of low noise amplifier?

Advantage:

- These are designed to reduce that extra noise.
- These electronic amplifiers are helpful in amplifying very low-strength signals.
- This amplifier is used once the SNR or signal-to-noise ratio is high & needs to be decreased by almost 50% while power is enhanced.

Disadvantage:

- Low noise amplifiers are expensive. • These are very sensitive to flicker noise & DC offsets.

RESULT:

Thus the low noise high performance frequency amplifier circuit using NE5534 IC has been verified successfully.

MARK ALLOCATION:

S. No	Parameters	Mark Allotted	Mark Awarded
1	Circuit Design/ code developing and debugging / Trouble shooting	0-3	
2	Implementation and Demonstration	0-3	
3	Discussion	0-3	
4	Report writing & Presentation	0-3	
5	Contribution & Team Dynamics	0-3	
Total		15	

Signature of Lab In-charge

Exp. No.: 03 Date:	DIGITAL DELAY TIMER (LS7212) IN VERILOG HDL

OBJECTIVES:

(i) To construct the Verilog code using HDL

(ii) To design a Verilog code for delay timer using HDL.

AIM:

(i) To design and construct the Verilog code for delay timer.

SOFTWARE REQUIRED:

Xilinx ISE

THEORY:

CODE DESCRIPTION:

The digital delay timer being implemented is CMOS IC LS7212 which is to generate programmable delays. The specification of the delay timer can be easily found here. Basically, the delay timer has 4 operating modes: one-shot (OS), Delayed Operate (DO), Delayed Release (DR), Dual Delay (DD). Those four modes will be selected by inputs mode_a and mode_b. The wb [7:0] input is to program the delays according to given equations in the specification of the delay timer.

FUCNTION TABLE:

Table:1 Function table

mode_a	mode_b	Mode
0	0	One-shot
0	1	Delayed operate
1	0	Delayed release
1	1	Dual delay

BLOCK DIAGRAM:

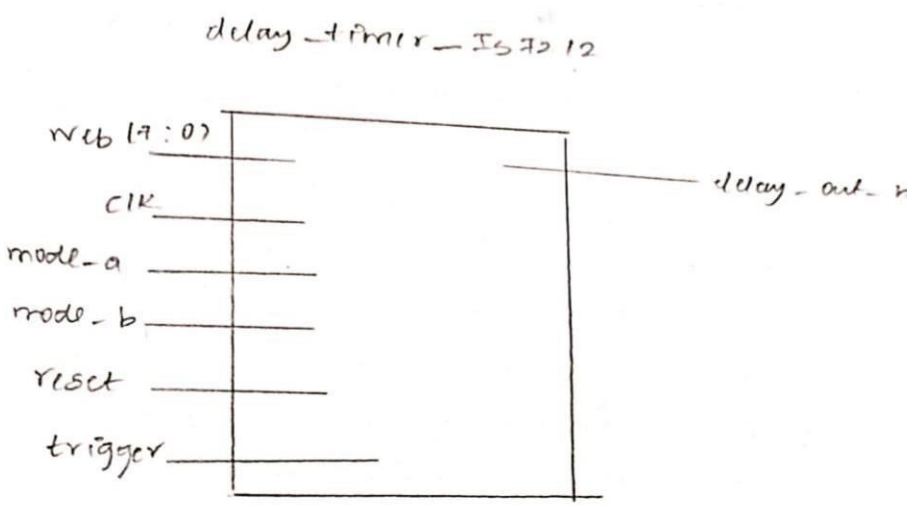

Figure:3.1 Block diagram of LS7212

VERILOG CODE FOR DELAY TIMER:

```
module delay_timer_ls7212
 (
   input [7:0] wb,
input clk,        input reset,
input trigger,        input
mode_a, mode_b,
output reg delay_out_n
 );
 reg[7:0] PULSE_WIDTH ;
reg [7:0] DELAY;
 reg [7:0] TIMER=0;
 reg trigger_sync_1=0,trigger_sync_2=0;
wire trigger_rising,trigger_falling;    reg
timer_start=0,out_low=0;    wire
timer_clear2,timer_clear3,timer_clear;    reg
[1:0] mode;
```

```
 reg reset_timer1=0,reset_timer2=0,reset_timer=0;
wire      reset_timer3,reset_det;                        reg
reset_det1=0,reset_det2=0;
always @(posedge clk)    begin
          trigger_sync_1 <= trigger;
trigger_sync_2 <= trigger_sync_1;
reset_timer1 <= reset_timer;
reset_timer2 <= reset_timer1;
reset_det1 <= reset;
   reset_det2 <= reset_det1;
 end
assign trigger_rising = trigger_sync_1 & (~trigger_sync_2);
assign trigger_falling = trigger_sync_2 & (~trigger_sync_1);
assign reset_timer3 = reset_timer1 & (~reset_timer2);
assign reset_det = reset_det2 & (~reset_det1);
always @(trigger_rising,trigger_falling,mode_a,mode_b,wb)    begin
     if(trigger_falling == 1 || trigger_rising == 1) begin
          PULSE_WIDTH = wb;
DELAY = (2*wb + 1)/2;              mode
= {mode_a,mode_b};
     end
end
always
@(mode,reset,trigger_falling,trigger_rising,TIMER,reset,trigger,PULSE_WIDTH,DELAY,re
set_det)    begin         case(mode)
              2'b00: // One-Shot Mode
begin
                              if(reset) begin
out_low <= 0;
timer_start <= 0;
reset_timer <= 1;                              end
```

```
                    else if(trigger_rising==1) begin
out_low <= 1;                                timer_start
<= 1;                                 reset_timer <= 1;
end
                    else if(TIMER>=PULSE_WIDTH) begin
                       out_low   <=   0;
timer_start               <=              0;
reset_timer <= 1;                    end
end
                  begin
                    if(reset) begin
out_low <= 0;
timer_start <= 0;
reset_timer <= 1;
      end
else if(reset_det==1 && trigger==1) begin
 timer_start<=1;
reset_timer <= 0;
        end
  else if(trigger_rising==1) begin
timer_start <= 1;
reset_timer <= 0;                          end
 else    if(trigger_falling==1    ||    trigger    ==    0)    begin
out_low <= 0;
    reset_timer <= 1;
timer_start <= 0;                      end
   else if(TIMER >= DELAY) begin
out_low <= 1;
   timer_start <= 0;
    reset_timer <= 1;                        end
```

```
            reset_timer <= 0;
end
                        begin
if(reset) begin
out_low <= 0;
timer_start <= 0;
reset_timer <= 1;
end
 else if(trigger_rising==1 || trigger == 1) begin
out_low <= 1;                               end
  else if(trigger_falling==1 ) begin
timer_start <= 1;
reset_timer <= 0;
                 end
else if(TIMER>=DELAY) begin
out_low <= 0;
        timer_start <= 0;
      reset_timer <= 1;                               end
end
begin                                if(reset)
begin                                     out_low
<= 0;
timer_start <= 0;
reset_timer <= 1;
end
else if(reset_det==1 && trigger==1) begin
timer_start <= 1;
reset_timer <= 0;
 end
```

```
else    if(trigger_falling==1    ||    trigger_rising==1    )    begin
timer_start <= 1;
  reset_timer <= 0;
  end
 else
 if(TIMER>=DELAY)
 begin
     out_low          <=          trigger;
timer_start <= 0;                    reset_timer
<= 1;                    end
    end
endcase    end
 always @(posedge clk or posedge timer_clear)
begin    if(timer_clear)       TIMER <= 0;    else
if(timer_start)
    TIMER <= TIMER + 1;
end
 assign timer_clear = reset_timer3 | trigger_rising == 1 | timer_clear3 ;
assign timer_clear2 = (trigger_rising == 1)|(trigger_falling == 1);    assign
timer_clear3 = timer_clear2 & (mode == 2'b11);
if(out_low    ==    1)
delay_out_n    <=    0;
else
delay_out_n <= 1;    end
endmodule
```

TIME BENCH:

```
module tb_ls7212;
reg    [7:0]    wb;
reg clk;
```

```
reg reset; reg
trigger; reg
mode_a; reg
mode_b;
delay_out_n;
delay_timer_ls7212 uut (
        .wb(wb),
        .clk(clk),
        .reset(reset),
        .trigger(trigger),
        .mode_a(mode_a),
        .mode_b(mode_b),
        .delay_out_n(delay_out_n)
    ); initial begin
wb = 10;
mode_a = 0;
mode_b = 0;
reset = 0;
trigger = 0;
#500; trigger =
1; #15000;
trigger = 0;
#15000; trigger = 1;
        #2000 ;
trigger = 0;
#2000;
trigger = 1;
#2000;
trigger = 0;
#20000;
```

```
trigger = 1;
#30000;
trigger = 0;
#2000;
trigger = 1;
#2000;
trigger = 0;
#4000;
trigger = 1;
#10000;           reset
= 1;            #10000;
reset = 0;
        end     initial begin
clk = 0;     forever #500
clk = ~clk;        end
 endmodule
```

TECHNICAL SPECIFICATION:

In this Verilog code, we have defined a module called Digital Delay Timer. It takes several inputs: clk(clock signal), reset (reset signal), enable (enable signal), and delay value (the desired delay value in number of clock cycles). It provides one output, timer done, which indicates when the delay has completed.

The module includes two always blocks. The first one is responsible for incrementing the count register when the timer is enabled and running. The second always block controls the state of the timer running register based on the enable signal and the count reaching the desired delay value.

OUTPUT ANALYSIS:

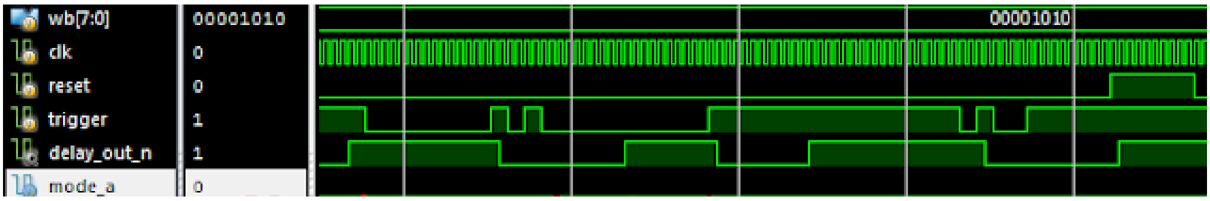

Figure: 3.2 Output of digital delay timer

PRE LABQUESTIONS:

1.What is Verilog HDL?

The Verilog Hardware Description Language (Verilog HDL) is a language that describes the behaviour of electronic circuits, most commonly digital circuits.

2.What is main use age of Verilog HDL software?

Verilog, standardized as IEEE 1364, is a hardware description language (HDL) used to model electronic systems. It is most commonly used in the design and verification of digital circuits at the register-transfer level of abstraction. Verilog HDL for designing hardware and for creating test entities to verify the behaviour of a piece of hardware. Verilog HDL is used as an entry format by a variety of EDA tools, including synthesis tools.

POST LAB QUESTIONS:

1.What is use of delay timer in Verilog HDL?

The digital delay timer being implemented is CMOS IC LS7212 which is to generate programmable delays. The specification of the delay timer can be easily found here. Basically, the delay timer has 4 operating modes: one-shot (OS), Delayed Operate (DO), Delayed Release (DR), Dual Delay (DD).

2.Advantage and disadvantage of Verilog HDL?

Advantages

- Verilog HDL allows different levels of abstraction to be mixed in the same model.
- Thus, a designer can define a hardware model in terms of switches, gates, RTL, or behavioral code.
- A designer needs to learn only one language for stimulus and hierarchical design. of choice for designers.

Disadvantages

- The control logic is designed with traditional techniques.
- The intent of design gets lost in complexity and details.
- The schematics are accompanied with documentation.
- Portability is an issue.
- Simulation environment for schematic capture design is not same.

RESULT:

Thus the digital delay timer (LS7212) in Verilog HDL has been stimulated and the output has been verified successfully.

MARK ALLOCATION:

S. No	Parameters	Mark Allotted	Mark Awarded
1	Circuit Design/ code developing and debugging / Trouble shooting	0-3	
2	Implementation and Demonstration	0-3	
3	Discussion	0-3	
4	Report writing & Presentation	0-3	
5	Contribution & Team Dynamics	0-3	
Total		15	

Signature of Lab In-charge

Exp. No.: 04 Date:03/04/23	**PROTOTYPE DESIGN FOR LOW NOISE PREAMPLIFIER CIRCUIT**

OBJECTIVE:

To construct a prototype design for a low noise pre amplifier circuit.

AIM:

To design a low noise high performance frequency amplifier circuit using NE5534 IC.

SOFTWARE REQUIRED:

ALTIUM DESIGNER 2022.

THEORY:

In a communication system, the receiver section needs amplification for the weak signal which is received from the antenna. So this amplification can be accomplished through the main component called Low Noise Amplifier or LNA. The characteristics of this amplifier can be described by certain parameters like gain, noise figure, chip area, linearity, power consumption & bandwidth.

The block diagram of the low noise amplifier is shown below. The designing of a low noise amplifier can be done by using a common gate amplifier, active inductor & common drain stage. Generally, the common gate amplifier is mainly used at the input stage whereas the common drain amplifier is used at the output stage to provide the best input as well as output matching. The low noise amplifier is bound with particular characteristics like gain & noise figure but the selection of LNA mainly depends on some specific parameters like power supply, bandwidth, chip area & linearity.

PCB DESIGN:

PCB (Printed Circuit Board) design is the Signal Integrity Theory. Signal integrity refers to the quality of electrical signals as they travel through a PCB. It focuses on minimizing noise, interference, and distortion to ensure that signals remain accurate and reliable. According to the

Signal Integrity Theory, there are several factors that need to be considered during PCB design to maintain signal integrity:

Trace Routing: Proper routing of traces is crucial to minimize signal degradation. High-speed signals should be routed as short and direct as possible to reduce signal reflections and crosstalk. Differential pairs should be routed in close proximity and with equal length to maintain signal balance.

Impedance Matching: Impedance matching ensures that the impedance of the transmission line matches the impedance of the source and load components. This helps to minimize signal reflections, which can cause signal distortion and loss.

Grounding and Power Distribution: Proper grounding and power distribution are essential for maintaining a stable reference voltage and reducing noise. Ground planes should be used to provide a low impedance return path for signals and to minimize ground loops. Decoupling capacitors should be placed near high-speed components to provide clean power and suppress voltage fluctuations.

EMI/EMC Considerations: Electromagnetic interference (EMI) and electromagnetic compatibility (EMC) issues can arise due to high-speed signals. Careful PCB layout and component placement can help reduce EMI/EMC problems. Techniques like shielding, signal isolation, and controlled impedance routing can be employed to minimize electromagnetic emissions and susceptibility.

Thermal Considerations: High-power components can generate heat, which needs to be managed to prevent thermal issues. Proper placement of heat sinks, thermal vias, and consideration of the PCB's thermal conductivity are important to dissipate heat effectively and prevent component failure.

Design for Manufacturability: PCB designs should be manufactural within the constraints of the chosen fabrication and assembly processes. DFM considerations involve optimizing component placement, trace widths, and clearances to ensure ease of manufacturing, inspection, and testing. These are some of the key aspects of the Signal Integrity Theory in PCB design. By understanding and implementing these principles, designers can improve signal quality, minimize errors, and ensure reliable operation of electronic devices.

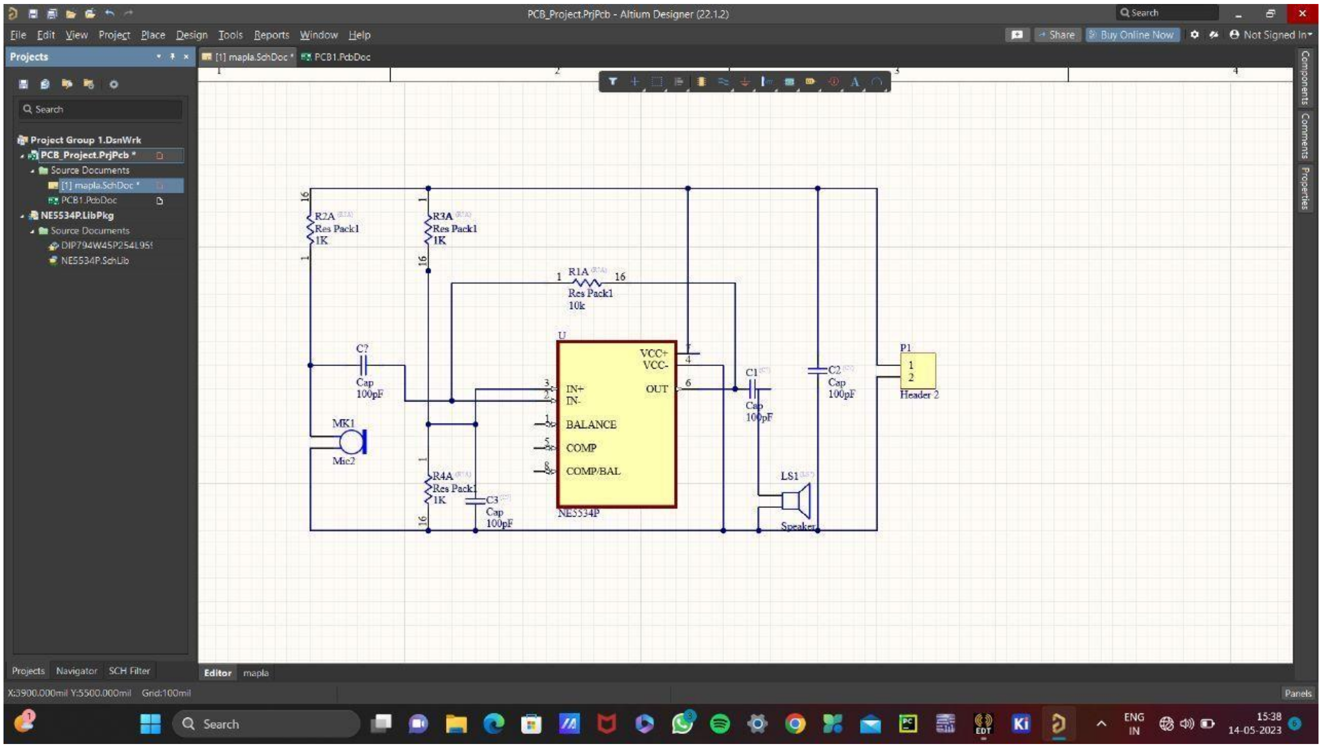

Figure 4.1: Schematic diagram of Low noise pre amplifier circuit.

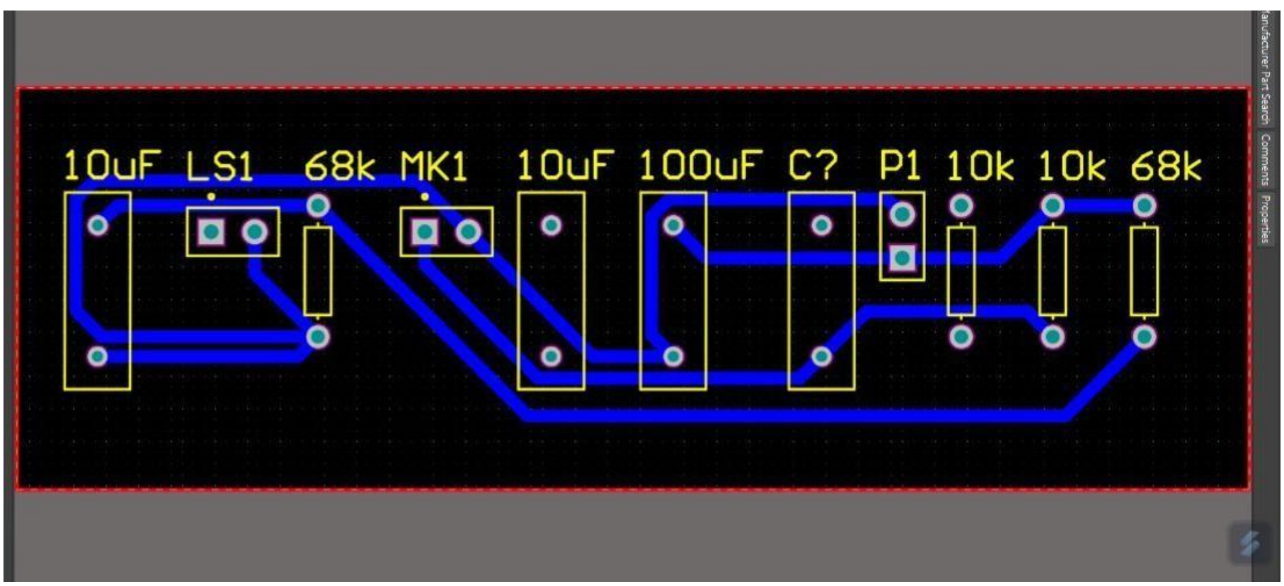

Figure 4.2: PCB layout diagram of Low noise pre amplifier circuit.

PROCEDURE:

The procedure for PCB design for a bipolar voltage regulator involves several steps. Here's a general outline of the process:

1. Schematic Design: Start by creating a schematic diagram of the bipolar voltage regulator circuit. This schematic will serve as a blueprint for the PCB layout. Include all the necessary components such as transistors, resistors, capacitors, and voltage references.
2. Component Selection: Choose appropriate components for your voltage regulator circuit based on your requirements, such as input/output voltage levels, current ratings, and power dissipation. Consider factors like package size and availability for ease of PCB layout.
3. PCB Footprint Creation: Create or select the appropriate PCB footprints for each component in your schematic. The footprint defines the physical dimensions and pin arrangement of the component on the PCB.
4. PCB Layout: Begin the PCB layout process by placing the components on the PCB. Consider factors like component orientation, signal flow, and thermal considerations. Group related components together to minimize trace lengths and optimize signal integrity.
5. Power and Ground Planes: Designate power and ground planes on the PCB. Power planes provide a low impedance path for power distribution, while ground planes serve as a stable reference for signals and minimize noise. Properly connect power and ground planes to the regulator's input and output pins.
6. Trace Routing: Route the traces on the PCB to connect the components according to the schematic and layout requirements. Pay attention to trace widths, spacing, and impedance matching for high-frequency signals. Minimize trace lengths and avoid crossing sensitive analog signals with noisy digital signals.
7. Thermal Considerations: Ensure that components generating significant heat, such as power transistors, have sufficient thermal relief and are adequately connected to heat sinks or thermal vias to dissipate heat effectively

8. Design Rule Check (DRC): Perform a design rule check to ensure that your PCB layout adheres to manufacturing and assembly constraints. Check for clearance violations, minimum trace widths, and other design rules specified by the manufacturer or your PCB design software.
9. Gerber File Generation: Once the layout is complete and error-free, generate Gerber files. These files contain the necessary information for the PCB manufacturer to fabricate your PCB.
10. PCB Prototyping and Testing: Send your Gerber files to a PCB manufacturer for fabrication. Once you receive the fabricated PCB, assemble the components and perform thorough testing to verify the functionality and performance of the bipolar voltage regulator circuit.
11. Remember, this is a general procedure, and the specific details may vary depending on the complexity of your voltage regulator circuit and the tools and software you use for PCB design. It's important to consult the datasheets of your components and follow best practices for PCB layout to ensure a successful design.

PRELAB QUESTIONS:

1. How to design a Low noise amplifier?

A low noise amplifier can be designed using either a negative feedback topology or one without any feedback. The former approach is more commonly used because it can provide better performance when compared to an equivalent amplifier without feedback.

2. What is PCB design?

PCB design is the process of creating a layout for a printed circuit board that connects and supports electronic components, facilitating their proper functioning in an electronic device.

3. What is IC NE 5534?

The NE5534, NE5534A, SA5534, and SA5534A devices are high-performance operational amplifiers combining excellent dc and ac characteristics. Some of the features include very low noise, high output-drive capability, high unity-gain and maximum-output-swing bandwidths, low distortion, and high slew rate.

4.What is Gerber diagram?

A Gerber file is a standard file format used in the PCB manufacturing industry to describe the artwork of each layer of a printed circuit board. It contains the information necessary to produce the copper traces, pads, vias, and other features on the PCB during the manufacturing process. The Gerber file acts as a blueprint for the PCB manufacturer, guiding them on how to fabricate and assemble the board accurately.

POSTLAB QUESTIONS:

1. What is the purpose of a low noise amplifier?

- To increase the signal-to-noise ratio
- To increase the gain of an amplifier
- To remove unwanted noise from an electromagnetic wave
- To attenuate the input of an amplifier.

2.What is the difference between noise and interference?

Noise is unwanted signal that can interfere with the operation of a circuit. It is usually caused by thermal agitation, voltage fluctuations, or other electrical effects. Interference is an unwanted signal that interferes with the operation of another circuit.

3. What is the advantages and disadvantages of PCB design?

ADVANTAGES:

- Compact and Space-Efficient.
- Reliable and Consistent Connections.
- Ease of Manufacturing and Assembly.
- Signal Integrity and Noise Reduction.
- Scalability and Upgradability.

DISADVANTAGES:

- Complexity.
- Cost.
- Time-consuming.

- Manufacturing Limitations.

RESULT:

Thus I have completed a prototype design for low noise preamplifier circuit, verified and tested its output successfully.

MARK ALLOCATION:

S.No	**Parameters**	**Marks allotted**	**Marks awarded**
1	Circuit design / Code developing and debugging / Trouble shooting	**0-3**	
2	Implementation / Demonstration	**0-3**	
3	Discussion	**0-3**	
4	Report writing and presentation	**0-3**	
5	Contribution and Team dynamics	**0-3**	
Total		**0-15**	

Signature of Lab In-charge

Exp.No.:05 Date:24/04/23	**CONEWAVEGUIDE-TM02 MODE**

OBJECTIVE:

To construct a rectangular wave guide using TM_{02} mode.

AIM:

To design a rectangular wave guide using TM_{02} mode.

SOFTWARE REQUIRED:

ANSYSVERSION2022

THEORY:

An electromagnetic wave is a type of wave that consists of oscillating electric and magnetic fields that travel through space at the speed of light. These waves are generated by the movement of charged particles and can travel through a vacuum or through various materials, such as air or water. Electromagnetic waves have a wide range of frequencies and wavelengths, which determines their properties and uses. Some common examples of electromagnetic waves include radio waves, microwaves, infrared radiation, visible light, ultraviolet radiation, X-rays, and gamma rays.

TM_{02}MODE:

TM02 mode is a type of electromagnetic wave propagation in a cylindrical waveguide or resonant cavity. When a magnetic field is applied, it can affect the propagation characteristics of the TM01 mode. In the presence of a magnetic field, the TM01 mode becomes elliptically polarized, meaning that the electric and magnetic fields are no longer perpendicular to each other. The orientation of the ellipse changes as the wave travels along the waveguide or cavity. The strength of the magnetic field also affects the propagation characteristics of the TM02 mode. As the magnetic field strength increases, the frequency of the mode shifts towards higher values, a phenomenon known as the "magnetic field dispersion. In addition, the magnetic field can also affect the loss of the TM02 mode, with higher magnetic fields generally leading to lower losses. Overall, the magnetic field can have a significant impact on the behaviour of the TM02 mode in a waveguide or resonant cavity, making it an important factor to consider in many applications.

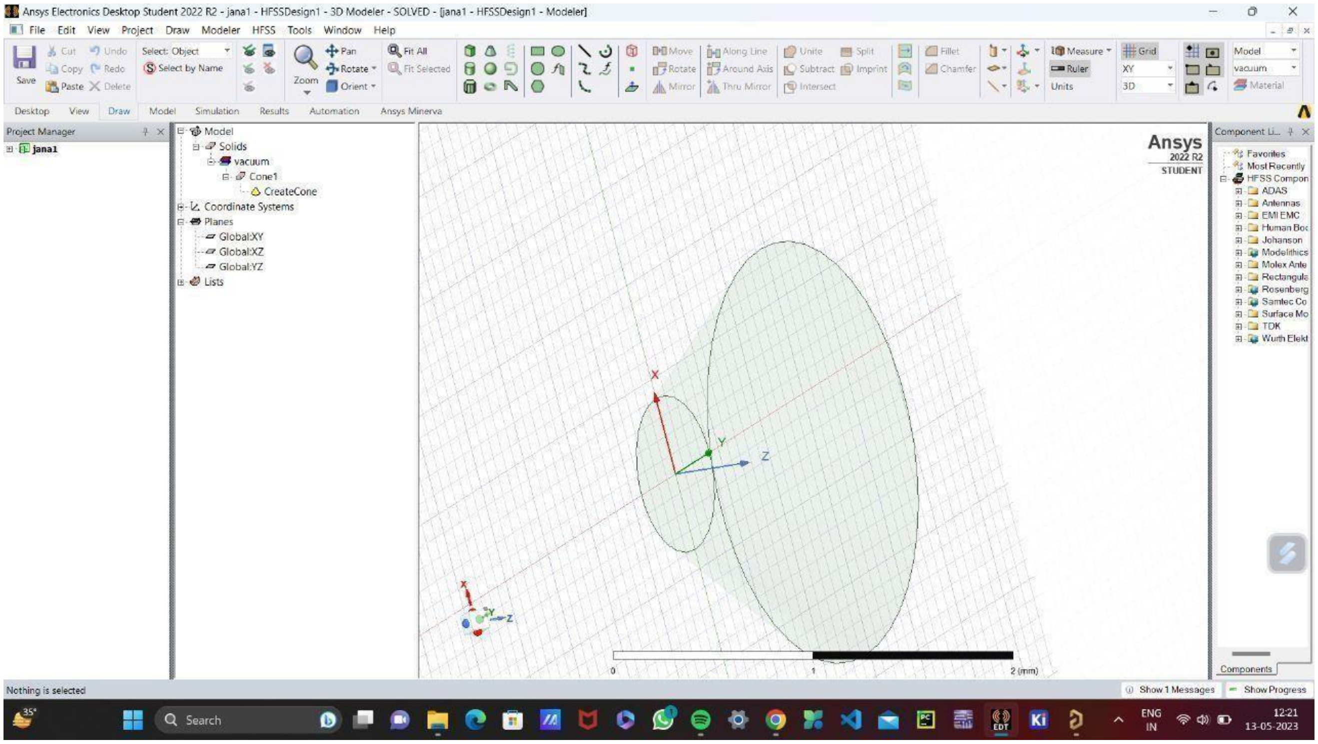

Figure: 5.1 Cone waveguide

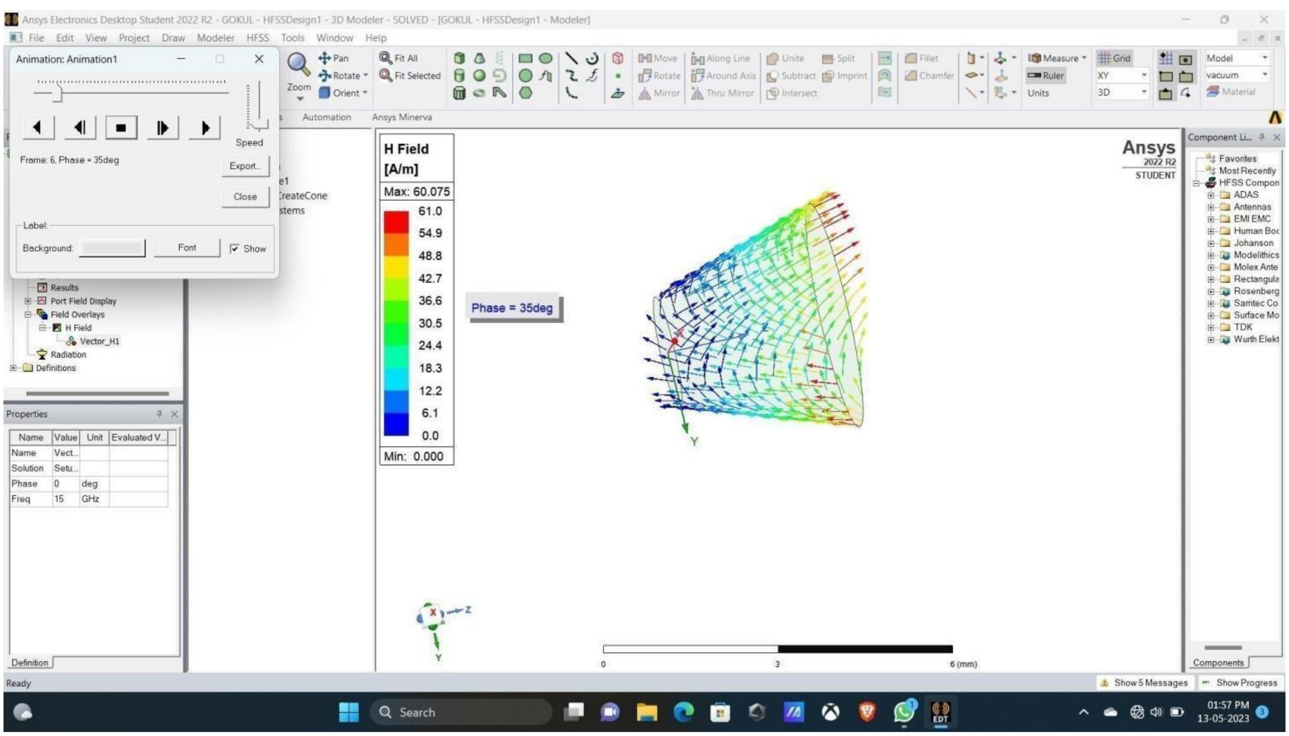

Figure: 5.2 Cone waveguide TM02 mode (35degree)

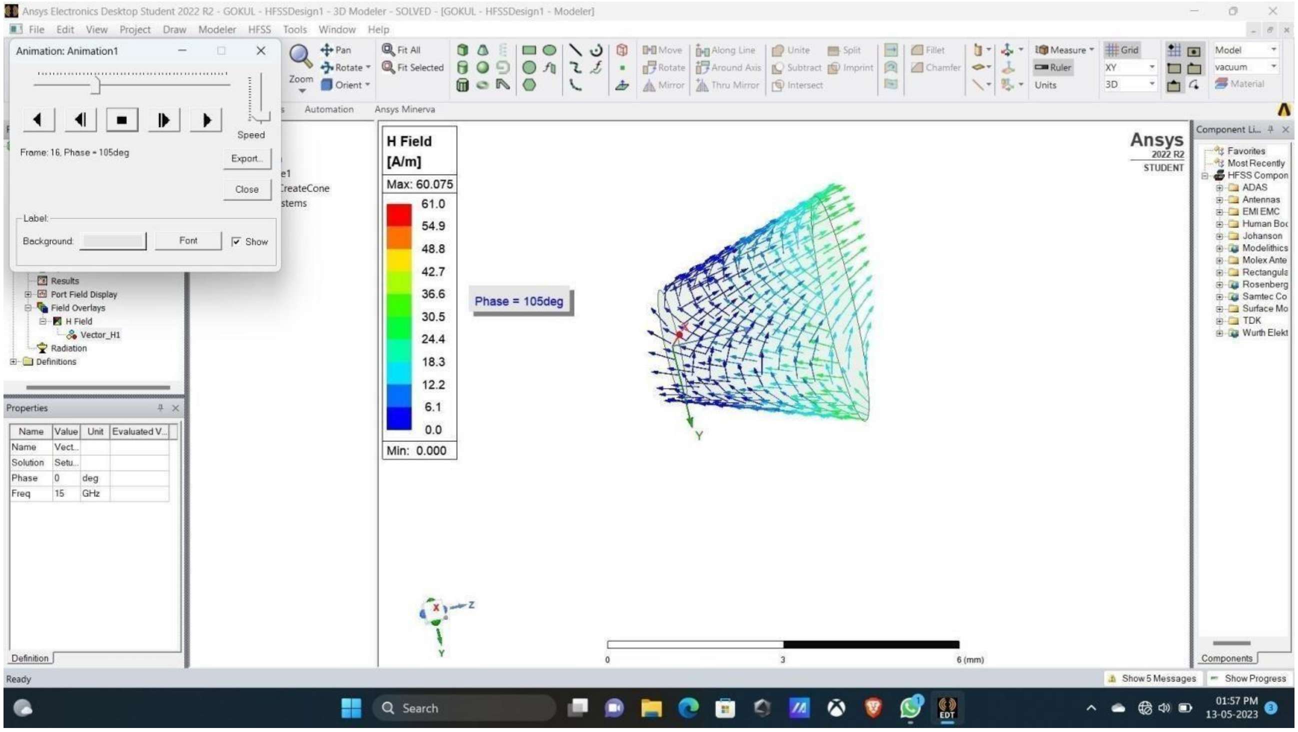

Figure: 5.3 Cone waveguide TM02 mode (105degree)

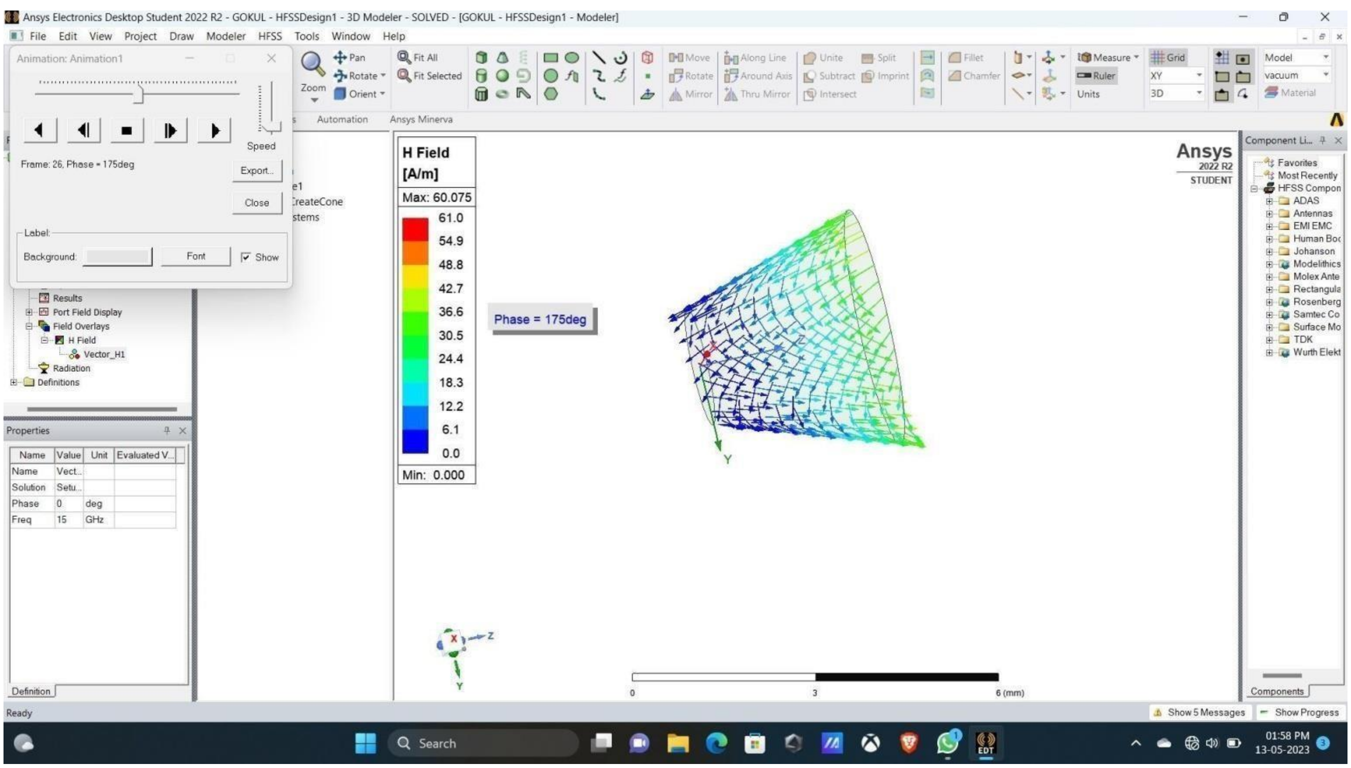

Figure: 5.4 Cone waveguide TM02(175degree)

PROCEDURE:

1. Create a new project: Open ANSYS software and create a new project. Save it with an appropriate name and location
2. Set the working environment: Select the working environment as "HFSS" (High Frequency Structural Simulator) to work with electromagnetic simulations.
3. Create a new design: Click on "Insert" and select "Design" from the dropdown menu. Then, select "Rectangular Waveguide" and enter the dimensions of the waveguide in the respective fields.
4. Create boundaries: To simulate the electromagnetic behaviour of a waveguide, you need to define the boundaries of the simulation. Click on "Boundaries" and create perfect electric conductor (PEC) boundary conditions on all sides of the waveguide.
5. Define materials: Define the material properties of the waveguide by clicking on "Material Properties". Select "Add Material" and choose the appropriate material for the waveguide.
6. Create a waveguide port: Click on "Excitations" and select "Waveguide Port". Select the mode of operation for the port and specify the dimensions of the port.
7. Mesh the design: Click on "Mesh" and select "Mesh Operations". Choose the appropriate meshing technique for the design and set the mesh parameters.
8. Solve the simulation: Click on "Solve" and select "Solve Setup". Choose the appropriate solver settings and start the simulation.
9. Analyze the results: Once the simulation is complete, click on "Results" to view the simulation results. You can analyze the S-parameters, radiation patterns, and other electromagnetic properties of the waveguide.
10. Optimize the design: You can optimize the design by tweaking the dimensions of the waveguide and port until you achieve the desired performance.

FIELD EQUATION:

- For TE mn mode, the field equations for a rectangular waveguide are:[2]
- E_{x} = (jco*mu)/(h ^ 2) * ((n*pi)/b) * cos((m*pi*x)/a) * sin((n*pi*y)/b)
- E_{y} = (- j * omega*mu)/(h ^ 2) * ((m*pi)/a) * sin((m*pi*x)/a) * cos((n*pi*y)/b)
- E_{sigma} = 0
- H_{x} = (j*beta)/(h ^ 2) * ((m*pi)/a) * sin((m*pi*x)/a) * cos((n*pi*y)/b)
- H_{J} = (j*beta)/(h ^ 2) * ((n*pi)/b) * cos((m*pi*x)/a) * sin((n*pi*y)/b)
- H_{z} = cos((m*pi*x)/a) * sin((n*pi*y)/b)
- For TM ma mode, the field equations for a rectangular waveguide are:
- E_{x} = (- j * beta)/(h ^ 2) * ((m*pi)/a) * cos((m*pi*x)/a) * sin((m*pi*y)/b)
- E_{y} = (- j * beta)/(h ^ 2) * ((n*pi)/b) * sin((m*pi*x)/a) * cos((n*pi*y)/b)
- E_{s} = sin((m*pi*x)/a) * sin((n*pi*y)/b)
- H pi = j omega e hbar^ 2 ((n*pi)/b) * sm((m*pi*x)/a) * cos((n*pi*y)/b)
- H_{x} = (- j * sis)/(h ^ 2) * ((pi*pi*x)/a) * cos((pi*a*pi*x)/a) * sin((pi*pi*y)/b)
- H_{z} = 0

PRELABQUESTIONS:

1. **Define dominant mode.**

The dominant mode is the mode of propagation that has the lowest cut off frequency and the highest characteristic impedance. It is the mode that can propagate through the waveguide with the least attenuation and distortion, and it is the mode that carries the majority of the power in the waveguide.

2. **What are dominant mode and degenerate mode in cone waveguide?**

The dominant mode in a waveguide is the primary mode of propagation that has the lowest cut off frequency and the highest power transmission capability. It is the mode that can propagate with the least amount of energy loss and distortion. In the case of a CONE waveguide, the dominant mode is typically the TE11 mode, which refers to the transverse electric mode with one variation in the electric field across the waveguide cross-section. This mode is commonly used in practical applications due to its desirable propagation characteristics.

3. **Define wave impedance.**

Wave impedance is defined as the ratio of the magnitude of the electric field to the magnitude of the magnetic field of an electromagnetic wave propagating through a medium.

4. **What is dominant mode of TE and TM waves in circular wave guide.**

For the TE modes, the dominant mode is the TE11 mode, which is the lowest-order mode that has no azimuthal variation in the electric field. The TE11 mode has a single radial electric field component and no magnetic field component along the direction of propagation. It is the mode that carries the majority of the power in the waveguide and is the most commonly used mode in practical applications.

POSTLABQUESTIONS:

1. **TEM waves is not possible through hollow rectangular wave guides. justify.**

In hollow waveguides (single conductor), TEM waves are not possible, since Maxwell's Equations will give that the electric field must then have zero divergence and zero curl and be equal to zero at boundaries, resulting in a zero field with boundary conditions guaranteeing only the trivial solution).

2. **If the dimensions of the wave guide increases what happens on cut off frequency?**

The cut off frequency will decrease. This is because as the waveguide dimensions increase, the wavelength of the propagating electromagnetic wave increases. At a certain point, the wavelength of the wave becomes too large to fit within the waveguide dimensions, and the wave cannot propagate in the waveguide without significant attenuation.

3. **What is the difference between TE wave and TM wave?**

In a TE wave, the electric field is transverse to the direction of propagation and the magnetic field is parallel to the direction of propagation. The magnetic field does not need to be zero inside the conductor for the wave to exist. Same is true for TM wave. So, both modes exist in this case.

RESULT:

Designing a cone waveguide TM02 mode using Ansys software involves setting up a 3D model of the waveguide, applying the appropriate boundary conditions for the TM02 mode. Ansys software provides advanced simulation capabilities to analyze the waveguide's performance, including the propagation characteristics, attenuation, and mode conversion. The results of the simulation can provide valuable insights into the waveguide's behaviour, including the field distributions, the transmission coefficient, and the reflection coefficient. These results can help optimize the design of the waveguide for specific application sand fields.

MARK ALLOCATION:

S. No	Parameters	Mark Allotted	Mark Awarded
1	Circuit Design/ code developing and debugging / Trouble shooting	0-3	
2	Implementation and Demonstration	0-3	
3	Discussion	0-3	
4	Report writing & Presentation	0-3	
5	Contribution & Team Dynamics	0-3	
Total		15	

Signature of Lab In-charge