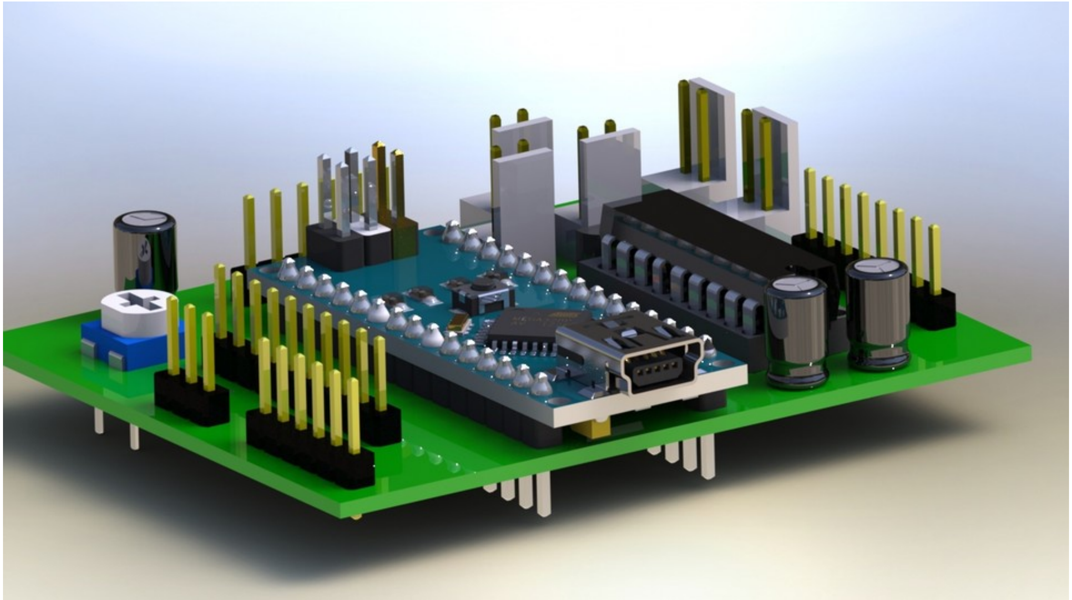
MakerNova

TEAM
PCB DESIGNING



MENTORS -

Aman Kapuriya
Aastha Basunde
Anushka Purohit
Atharav Karhad

TEAM MEMBERS -
Aashish Gupta
Vyom Vyas
Harsh Raj Singh
Honey Bohra
Chahat Bedi
Vaishnavi Tammana
Darshit Desai
Haritha Veeravalli
Krishna Meghwal
Meet Chudasama
Tej Hothi
Shivani Parihar
Nimesh Parmar
Shivam Randive
Shivam Singh
Krishna Ghetia

Acknowledgement

I’d like to acknowledge the invaluable guidance and support provided by our mentors throughout the process of designing the PCB for this project. Their expertise and insights were instrumental in shaping the project’s success.

Additionally, I am grateful to the Drishti Club for providing a platform that fostered collaborative learning and allowed us to delve into the world of PCB designing.

CONTENTS

ABSTRACT

This project focuses on the design and development of a Printed Circuit Board (PCB) tailored for microcontroller-based embedded systems. The objective is to create a versatile and efficient platform capable of accommodating a wide range of applications, while ensuring optimal performance, power efficiency, and ease of integration. This project contributes to the advancement of microcontroller-based PCB design methodologies, promoting efficient and reliable embedded systems.

1 INTRODUCTION

In today's fast-paced technological landscape, electronic devices have become increasingly complex and demanding high-speed data transmission.Printed Circuit Boards (PCBs) are at the core of these devices, acting as the central platform for interconnecting components and facilitating signal flow. This project aims to tackle the challenges associated with microcontroller-based PCB design by developing an optimized and adaptable platform that can serve as a foundation for various embedded system applications. The primary focus lies in achieving superior performance, ensuring seamless integration with diverse peripherals and sensors, and enhancing the design's ease of use and scalability.

A printed circuit board (PCB) is an electronic assembly that uses copper conductors to create electrical connections between components. Printed circuit boards provide mechanical support for electronic components so that a device can be mounted in an enclosure. A printed circuit board design must include a specific set of steps that aligns with the manufacturing process, integrated circuit packaging, and the structure of the bare circuit board.

1.1 SOFTWARE REQUIREMENTS

1.EAGLE SOFTWARE

Eagle is a widely-used electronic design automation (EDA) software for schematic capture and PCB layout, developed by Autodesk.

-Installation link for Eagle Software:-

https://eagle-updates.circuits.io/downloads/9_6_2/Autodesk_EAGLE_9.6.2_English_Win_64bit.exe

2.Arduino IDE

The Arduino IDE (Integrated Development Environment) app is a user-friendly software tool that simplifies the process of programming and developing projects for microcontrollers. It provides a convenient interface for writing, compiling, and uploading code to microcontroller boards, such as those based on the Arduino platform.

-Installation link for Arduino IDE:-
https://downloads.arduino.cc/arduino-1.8.19-windows.exe

1.2 MANUFACTURING

The printed circuit board (PCB) manufacturing process is complex and requires multiple steps to ensure performance. It involves designing layouts, preparing circuit images, etching copper traces, drilling holes, and applying solder mask and silkscreen layers. Different PCB structures may require 20 or more steps.

1.3 METHODS INVOLVED IN MANUFACTURING

1) Photographic Processing:- Photographic Processing is a traditional technique for creating PCB copper traces and patterns using light-sensitive resist, exposing designs to UV light, and undergoing chemical development, etching, and resist removal.
2) Milling machine:- Milling machine method removes copper from copper-clad substrate using CNC machines, resulting in precise traces and patterns for low volume prototype PCBs.
3) Silk screening Process:- Stencil application on PCB involves applying solder mask or silkscreen layer, allowing ink or mask material to penetrate openings, providing protection and labeling, enhancing soldering and component identification.

1.4 PARTS OF PCB

A Printed Circuit Board (PCB) consists of several essential parts that together form a functional electronic circuit. The main parts of a PCB are:
1) Substrate: The base material on which the PCB is constructed, often made of fiberglass-reinforced epoxy known as FR-4 or other specialized materials.
2) Copper Traces: Thin copper conductive paths that form the electrical connections between different components on the PCB.
3) Pads: Small metal areas on the PCB surface used to connect components through soldering or other mounting methods.
4) Through-Holes: Holes drilled through the PCB to allow for component leads to pass through and create electrical connections between different layers.
5) Components: Electronic devices such as resistors, capacitors, integrated circuits (ICs), transistors, and other active and passive components that perform specific functions on the PCB.

6) Solder Mask: A protective layer applied to cover the copper traces and prevent accidental short-circuits during soldering.
7) Silkscreen: A layer of ink used for labeling components, test points, and other information on the PCB, providing visual references for assembly and troubleshooting.
8) Vias: Plated holes that provide electrical connections between different layers of a multi-layer PCB.
9) Power and Ground Planes: Copper layers dedicated to provide a stable power supply and grounding to the components on the PCB.
10) Surface Finishes: Coatings applied to exposed copper traces to protect them from oxidation and ensure better solderability.
11) Solder Paste: A mixture of solder alloy and flux used to temporarily hold surface-mount components in place during assembly.
All these parts work together to form a complete and functional electronic circuit on the PCB, enabling the proper flow of electrical signals and power throughout the system.

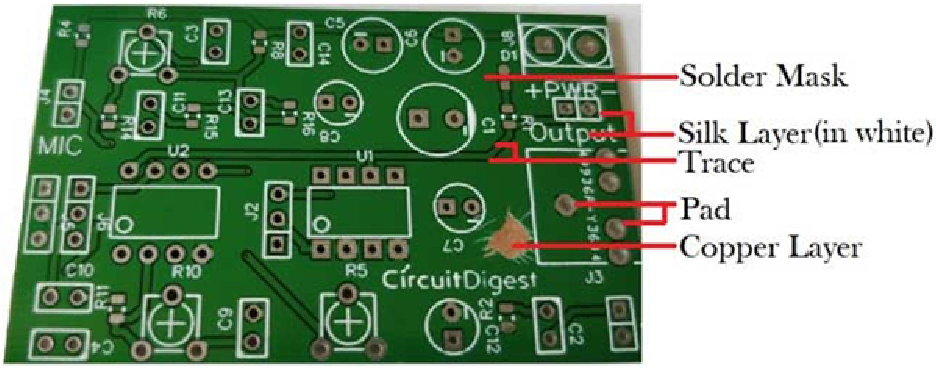


Figure 1.1. PARTS OF PCB

1.5 TYPES OF PCB

There are several types of PCBs, each designed to cater to specific applications and requirements. The main types of PCBs are:
1) Single-Sided PCB: Contains copper traces on one side of the substrate. Suitable for simple electronic circuits with few components.
2) Double-Sided PCB: Has copper traces on both sides of the substrate, connected through vias. Provides more flexibility for circuit design and accommodates a higher density of components.
3) Multilayer PCB: Consists of multiple layers of copper traces and substrates, connected by vias. Ideal for complex circuits with high component density and signal integrity requirements.

4) Rigid PCB: Made from inflexible materials like FR-4 and suitable for most electronic devices and applications.

5) Flexible PCB (FPC): UtilisesUtilizes flexible materials like polyimide, allowing the PCB to bend or twist. Ideal for applications with limited space or requiring flexibility.

6) Rigid-Flex PCB: Combines rigid and flexible PCB sections in a single board, offering versatility and improved reliability for complex designs.

1.6 ADVANTAGES OF PCB DESIGNING

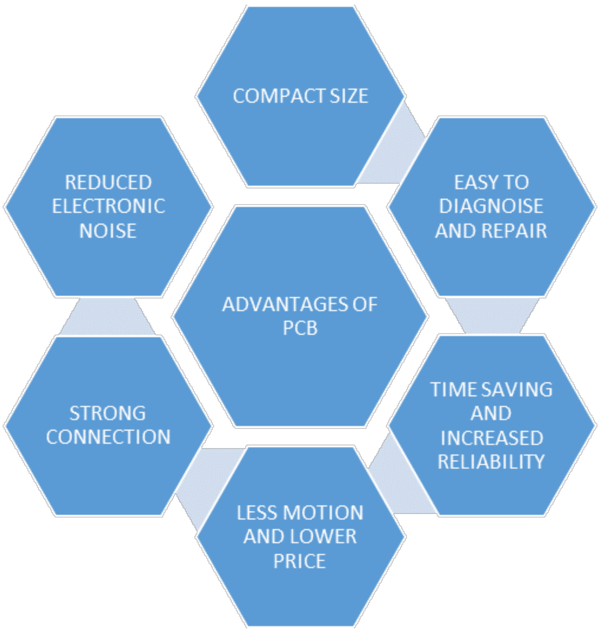


Figure 1.2. ADVANTAGES OF PCB DESIGNING

2 INTRODUCTION TO EAGLE

EAGLE is an EDA software for designing printed circuit boards (PCBs), assisting engineers and hobbyists in creating circuit schematics and layouts. It includes component placement, connection, and design rule checking, supports single and multi-layer designs, and offers customization options for components. Autodesk owns the software.

Figure 2.1. EAGLE SOFTWARE

2.1 LIBRARIES

The Libraries used in the project are “adafruit” and “sparkfun”. Adafruit and SparkFun offer EAGLE libraries with footprints and symbols for electronics components, simplifying PCB design, saving time, and ensuring accurate connections for makers and DIY enthusiasts.

2.2 SCHEMATIC

The schematic part of EAGLE is where you create and organize the visual representation of your electronic circuit design. It allows you to design and connect various components, such as resistors, capacitors, ICs (integrated circuits), connectors, and more, to form a coherent circuit diagram. Schematic parts contain features like Components Library, Symbol Creation, Wiring and Connections , Netlist Generation , Names and Labels, Net and Junctions.

Interface of schematic part of eagle :

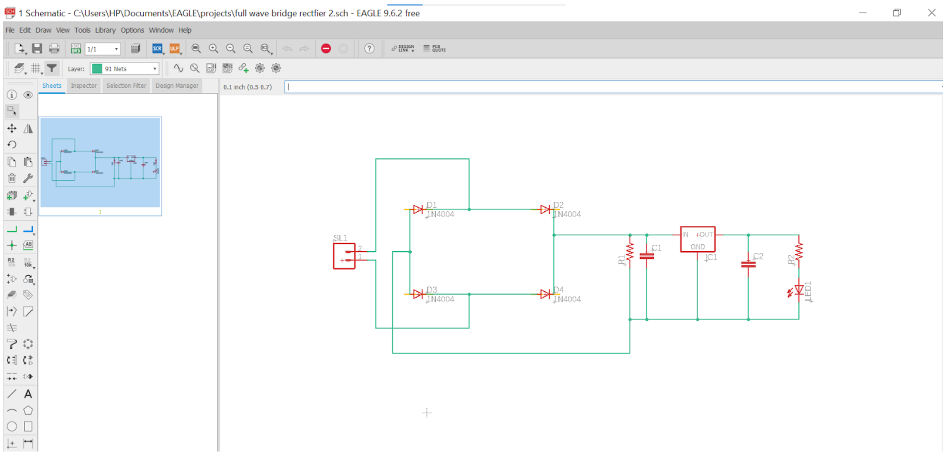


Figure 2.2. SCHEMATIC OF RECTIFIER CIRCUIT

2.3 LAYOUT

The layout page in Eagle is where you transform your schematic design into a physical representation on a printed circuit board (PCB). It's the stage where you arrange components, define copper traces, and optimize the layout for manufacturing and functionality. The layout process involves converting the logical connections from your schematic into physical connections on the PCB, ensuring proper spacing, alignment, and adherence to design rules.

Layout interface of eagle :

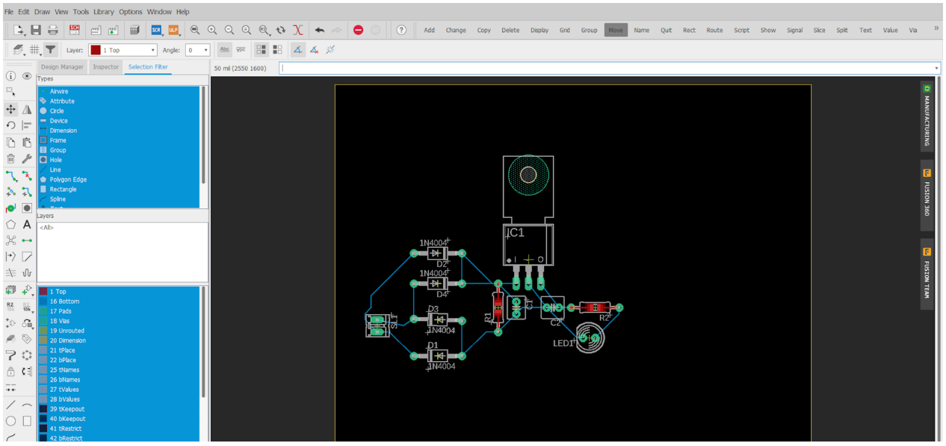


Figure 2.3. LAYOUT OF RECTIFIER CIRCUIT

2.3.1 ROUTING

Define copper traces to establish connections between components. You'll use routing tools to draw these traces while considering signal integrity, avoiding noise, and minimizing interference.

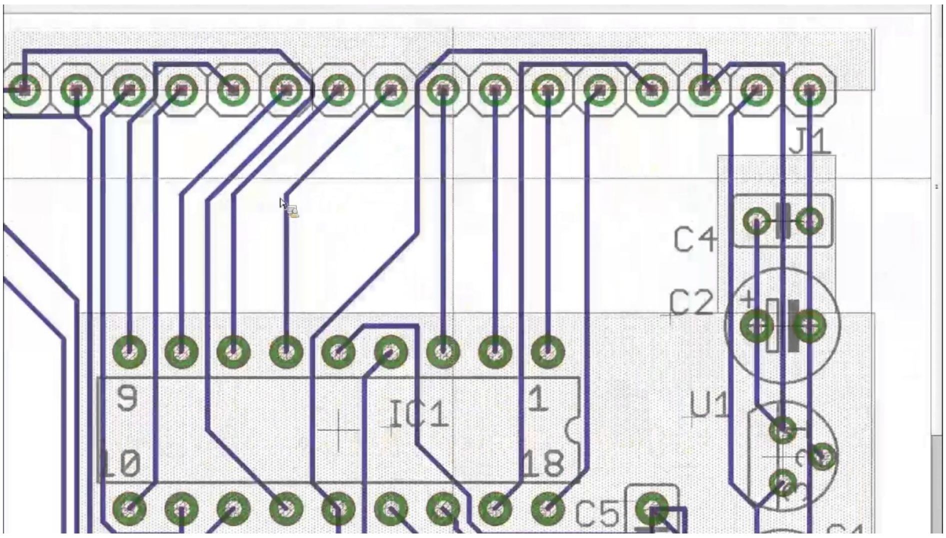


Figure 2.4. CONSIDERATION TAKEN WHILE ROUTING

2.3.2 SURFACE MOUNT COMPONENTS

SMCs are compact, light electronic parts soldered onto circuit boards using a specific heating process. They enable smaller devices and are assembled using machines. Proper handling, placement, and fixing are crucial for improving technology and creating small gadgets.

2.4 RECTIFIER CIRCUIT

2.4.1 INTRODUCTION

To get used to this eagle software we first implemented the circuit of Full Wave Rectifier. A full-wave rectifier is an electronic circuit that converts alternating current (AC) into direct current (DC). It rectifies both the positive and negative halves of the AC input waveform, resulting in a continuous unidirectional flow of current in the output.

The circuit diagram of this circuit is given below:-

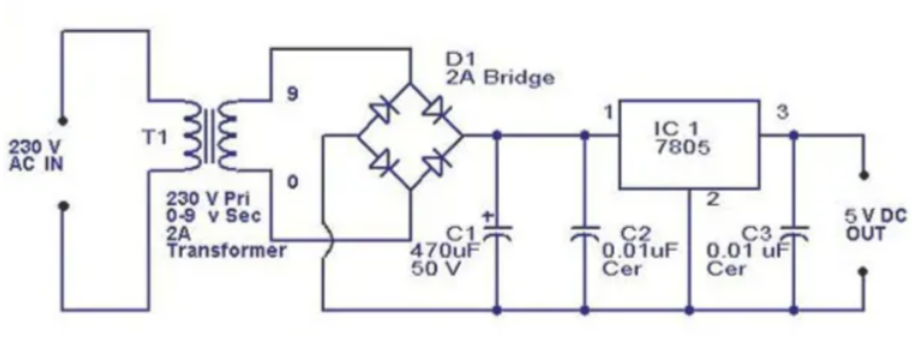


Figure 2.5. RECTIFIER CIRCUIT

2.4.2 SCHEMATIC

With the aid of Eagle software, we successfully created the schematic of the circuit mentioned above.

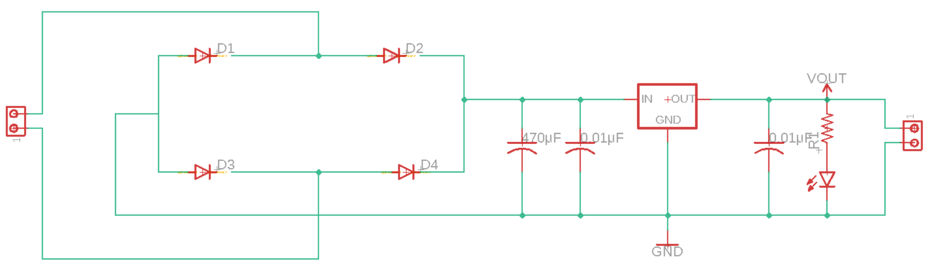


Figure 2.6. SCHEMATIC OF RECTIFIER CIRCUIT

2.4.3 LAYOUT

From the schematic provided, we proceed to implement the circuit layout, converting the abstract diagram into a physical representation of the components and their connections on the Printed Circuit Board (PCB).
Furthermore, we recognized the advantages of optimizing our layout to be as compact as possible, as it not only saves valuable space but also enhances the overall efficiency and functionality of the circuit design.

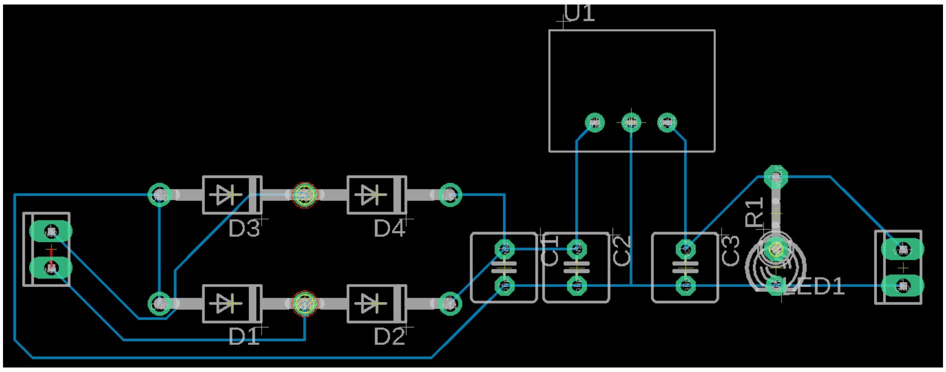


Figure 2.7. LAYOUT OF RECTIFIER CIRCUIT

3 MICROCONTROLLER

3.1 INTRODUCTION

A microcontroller is a small chip that integrates a processor, memory, and various input/output interfaces onto a single circuit. Microcontrollers are programmed to perform specific tasks . They are essential components in modern electronics, enabling efficient control and operation of devices and processes.

3.1.1 ATmega328P

We used the Atmel ATmega328P as our microcontroller in our PCB because it is a robust and reliable microcontroller that strikes a good balance between features, performance, ease of use, and cost. Here are some general features of this microcontrollers:-
-The ATmega328P has a total of 28 pins out of which 23 are programmable I/O lines.
-6 PWM channels
-Operating Voltage (V) is +2.7V to +5.5 V (+5.5V being absolute maximum)

3.1.2 PIN DESCRIPTION

The Atmega328P has three ports: Port B (PB0 to PB7), Port C (PC0 to PC5), and Port D (PD0 to PD7). Each port consists of eight pins and can be individually configured as input or output. Here's a detailed explanation of each port:
(1) Port B (PB0 to PB7):

- PB0 to PB5 (D 0 to D 5): General-purpose digital I/O pins. They can be configured as inputs or outputs. These pins can be used to connect various digital devices such as buttons, switches, LEDs, etc.
- PB6 (D 6) and PB7 (D 7): These pins have special functions:
  i)PB6 (D 6) is the I^2C serial clock pin (SCL). It is used for I^2C communication when the Atmega328P is acting as an I^2C master or slave.

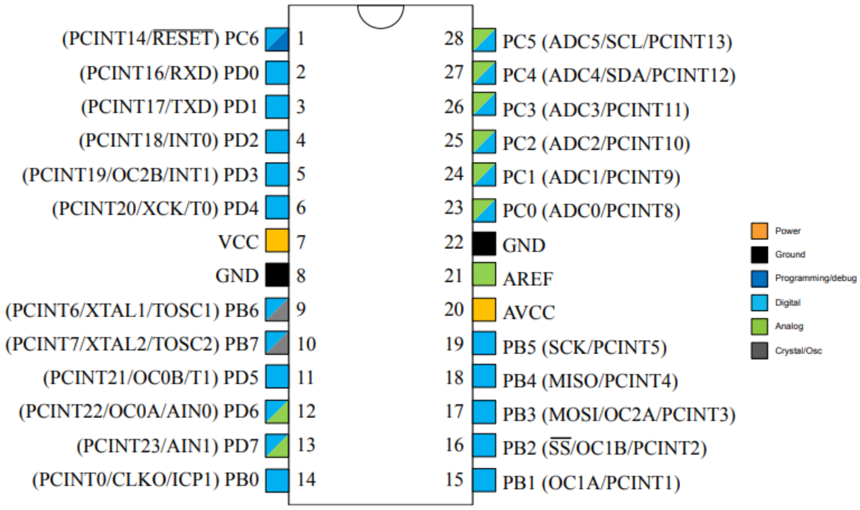


Figure 3.1. PIN DESCRIPTION OF ATMEGA328P

ii)PB7 (D 7) is the I^{2}C serial data pin (SDA). It is used for I^{2}C communication when the Atmega328P is acting as an I^{2}C master or slave.

(2) Port C (PC0 to PC5):

- PC0 to PC5 (A 0 to A 5): These pins are analog input channels used for the built-in Analog-to-Digital Converter (ADC). They can also be used as digital I/O pins if not required for analog input.

(3) Port D (PD0 to PD7):

- PD0 to PD7 (D8 to D15): General-purpose digital I/O pins. They can be configured as inputs or outputs and used for various digital tasks.

- PD0 (D8) and PD1 (D9): These pins have special functions:
  i)PD0 (D8) is the RXD (Receive Data) pin for serial communication. It is used for UART communication (asynchronous serial communication) to receive data from other devices or microcontrollers.
  ii)PD1 (D9) is the TXD (Transmit Data) pin for serial communication. It is used for UART communication to transmit data to other devices or microcontrollers.

- PD2 (D 10): This pin is used as an external interrupt pin (INT0). It can generate interrupts based on external events, allowing the microcontroller to respond to specific signals quickly.

- PD3 (D 11) to PD7 (D 15): These pins are general-purpose digital I/O pins, similar to the other PD pins.

3.1.3 CRYSTAL OSCILLATORS

The Atmega328P microcontroller, like many other microcontrollers, requires an accurate and stable clock source to operate correctly.

The Atmega328P offers several clock source options, including an internal RC (Resistor-Capacitor) oscillator and an external crystal oscillator. The External crystal oscillator is preferred in many applications for the following reasons:

- Accuracy and Stability
- Consistent Timing
- Higher Clock Frequencies
- Reduced Power Consumption
- Easy to Interface
- Compatibility with Peripheral Modules

In conclusion, Some applications, such as real-time systems, communication protocols (e.g., UART, I2C), and precise measurements, require consistent and predictable timing. The crystal oscillator ensures that the microcontroller operates at a precise frequency, meeting the timing requirements of and providing a greater clock frequency than the internal oscillator.

Figure 3.2. CRYSTAL OSCILLATOR

3.2 ATMEGA 328P BASED DEVELOPMENT BOARD CIRCUIT

3.2.1 INTRODUCTION

An ATmega328-based development board serves as a versatile and user-friendly platform for creating and experimenting with electronic projects. At the heart of this board lies the ATmega328 microcontroller, a powerful and popular component known for its capabilities in processing data, controlling peripherals, and interfacing with the world around it.
This development board provides an accessible environment for both beginners and experienced enthusiasts to explore the world of embedded systems and microcontroller programming. With various input and output pins, built-in components like LEDs and buttons, and the ability to connect external sensors and modules, it enables a wide range of applications and experiments.

Sr. no	Components	Value	Qty
1	IC	ATmega328p	1
2	USB TO TTL Module	-	1
3	Crystal Oscillator	16MHz	1
4	Capacitor	100nF,22pf/0.1uf	2
5	Resistor	10k,220 Ω	1
6	Pin Header	-	-
7	Breadboard	-	1

Figure 3.3. LIST OF COMPONENTS

3.2.2 CIRCUIT DIAGRAM

Below we have the schematic for this configuration. As you can see we need a supply of 5 volts. This supply has to be very well regulated with no voltage spikes. For that add an extra 10uF capacitor between 5V and GND. Also, the reset pin is negatively enabled. So, in order to have it disabled, we need to apply 5V to it. For that, a 10k ohms resistor is placed between RESET and Vcc.

Also, the ATMega328 usually works at 16MHz. For that, between pins 9 and 10 we place a 16MHz crystal. But this crystal, in order to oscillate, needs two capacitors of exactly 22pF/0.1uF connected to GND. In the figure above, you have all the pins of the chip.

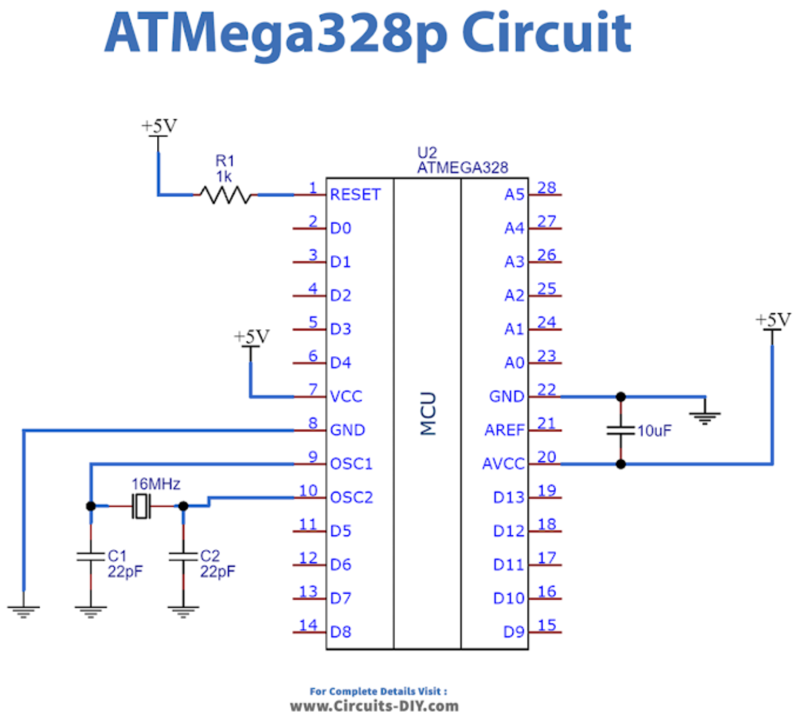


Figure 3.4. ATMEGA CIRCUIT DIAGRAM

3.2.3 SCHEMATIC DIAGRAM

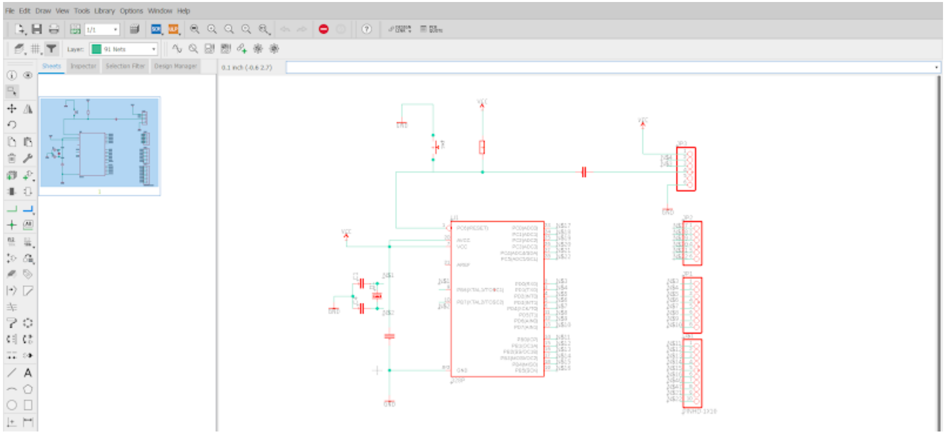


Figure 3.5. SCHEMATIC OF ATMEGA328P CIRCUIT

3.2.4 USB TO TTL ADAPTER

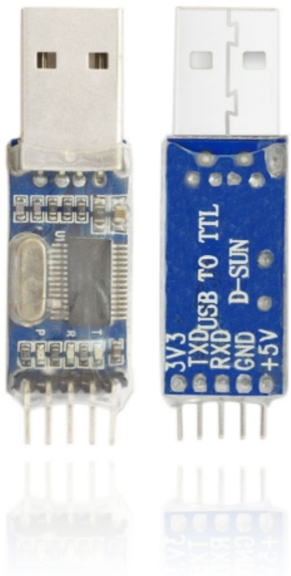


Figure 3.6. USB TO TTL MODULE

Use of USB to TTL adapter to upload code :
Uploading code to an ATmega328P microcontroller using a USB to TTL adapter (also known as a USB to serial adapter) involves a few steps. This method is commonly used for programming Arduino boards, which are built around the ATmega328P.

USB TO TTL ADAPTER TO UPLOAD CODE

Materials Needed:

- ATmega328P microcontroller (or Arduino board based on ATmega328P)
- USB to TTL (Serial) adapter (such as FTDI or CH340G)
- Breadboard and jumper wires (if not using an Arduino board)
- Arduino IDE installed on your computer Connections:

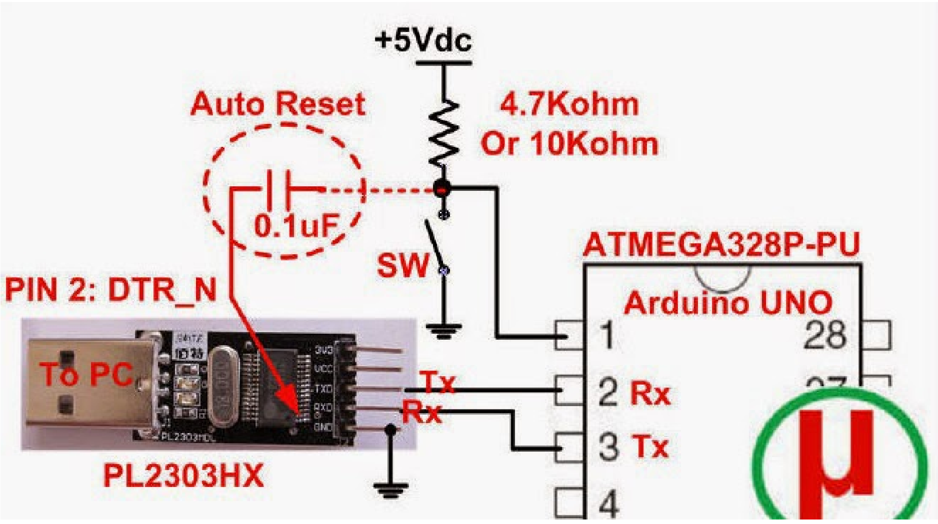


Figure 3.7. CONNECTIONS DIAGRAM OF USB TO TTL ADAPTER WITH ATMEGA328P

3.2.5 USBASP

USBasp is a USB in-circuit programmer for Atmel AVR controllers. It simply consists of an ATMega88 or an ATMega8 and a couple of passive components. The programmer uses a firmware-only USB driver, no special USB controller is needed.

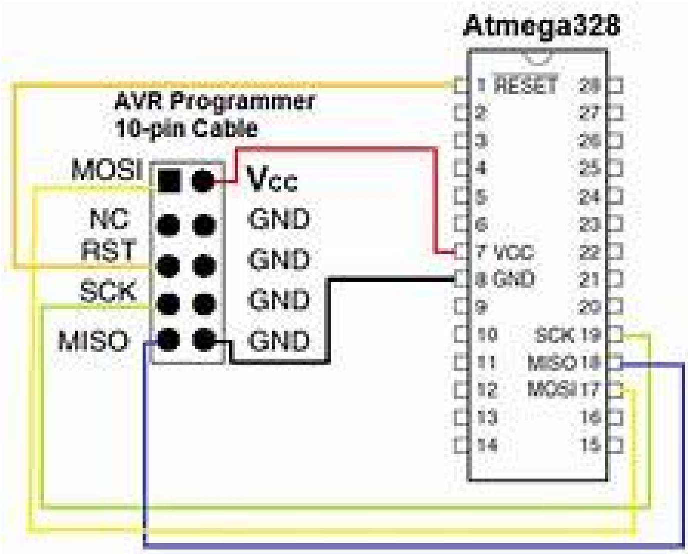


Figure 3.8. CONNECTIONS DIAGRAM OF USBasp WITH ATMEGA328P

4 DEVELOPMENT BOARD CIRCUIT

4.1 SCHEMATIC

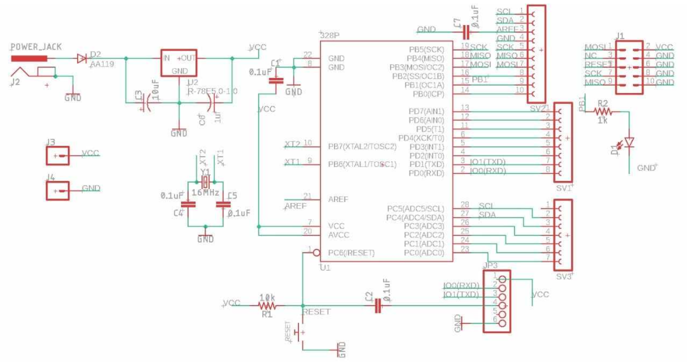


Figure 4.1. SCHEMATIC OF DEVELOPMENT BOARD CIRCUIT

4.2 ROUTING

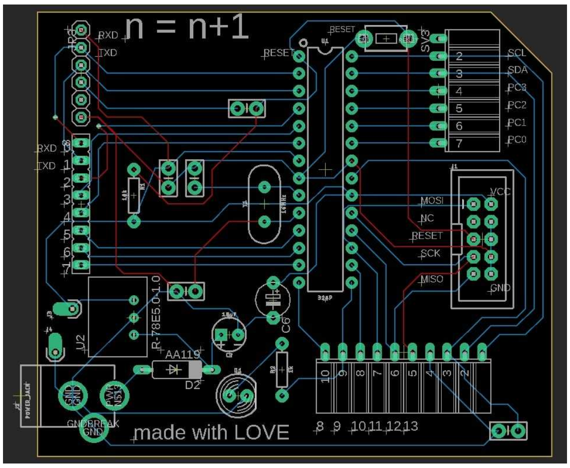


Figure 4.2. ROUTING OF DEVELOPMENT BOARD CIRCUIT

4.3 CLEARANCES

There were some clearance parameters that we had to check before finalising our PCB circuit .
1) Distance check :-
Distance check is checked in the communication pins of the Atmega328p And the headers . It is checked between MISO-MOSI,SCL-SDA,TXD-RXD the oscillator pins . The minimum allowed difference of the distance of two pins is 3 to 4 mm. 7We can check the distance of the route from one pin to another by giving the command “run length-freq-ri” in the eagle software.
2) Clearance check :- It is the minimum distance check between the two routes of the PCB. The minimum distance can be checked by the formula-spacing = 0.6 + (0.005 X Vpk), where Vpk is the peak voltage is the circuit which is 5.5v. Thus from this formula we found out that the minimum distance is 0.625mm. And when we convert it to mils(1/1000inch) we get 24.6 mils.

4.4 TEXT COMMAND IN EAGLE SOFTWARE

After giving the text command we can add a text on the PCB circuit so that we can print the numbers text that we want to in the silkscreen on the PCB.

4.5 GERBER FILE

After checking and correcting all the clearances. We have to create a gerber file from the board file (.brd->.grb). It is created by selecting the CAM processor in the eagle software. The after selecting process the gerber file is created. We can view the gerber files by an online viewer(PCB way).
Here are some pictures of the gerber files viewed from online viewer:-

4.6 COPPER TOP

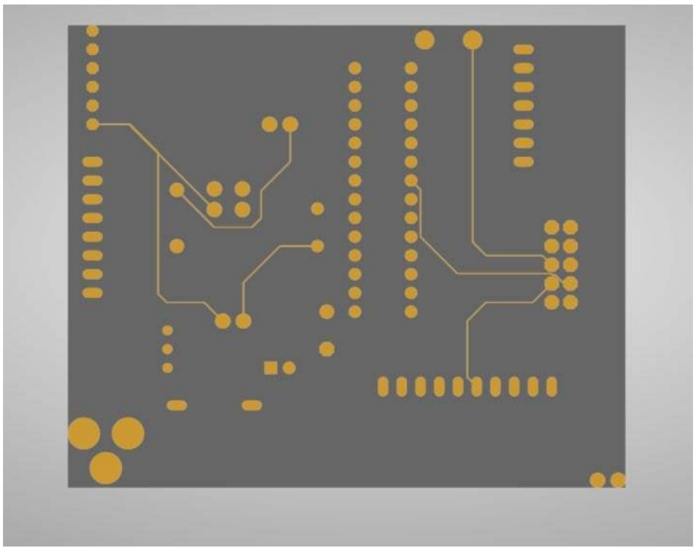


Figure 4.3. COPPER TOP OF DEVELOPMENT BOARD CIRCUIT

4.7 COPPER BOTTOM

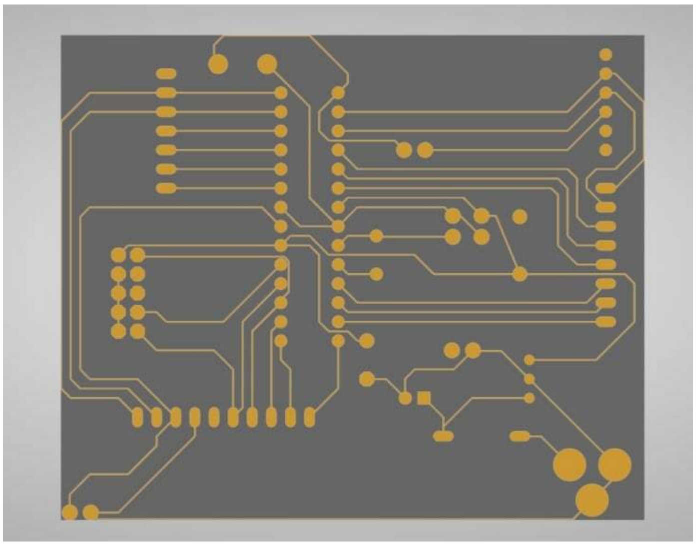


Figure 4.4. COPPER BOTTOM OF DEVELOPMENT BOARD CIRCUIT

4.8 SILKSCREEN TOP

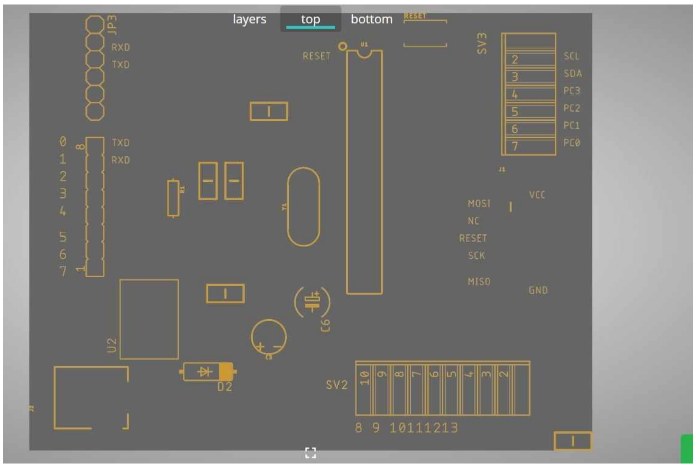


Figure 4.5. SILKSCREEN TOP OF DEVELOPMENT BOARD CIRCUIT

4.9 SILKSCREEN BOTTOM

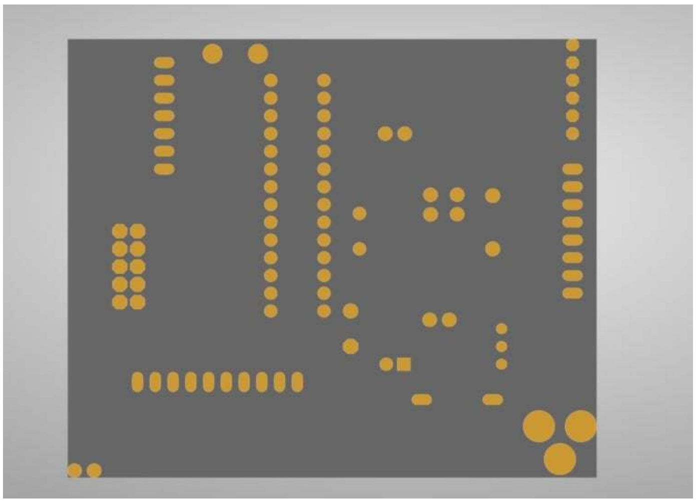


Figure 4.6. SILKSCREEN BOTTOM OF DEVELOPMENT BOARD CIRCUIT

5 AUDIO AMPLIFIER

5.1 INTRODUCTION

Amplifier is the generic term used to describe a circuit which produces and increased version of its input signal.

5.2 CIRCUIT DIAGRAM

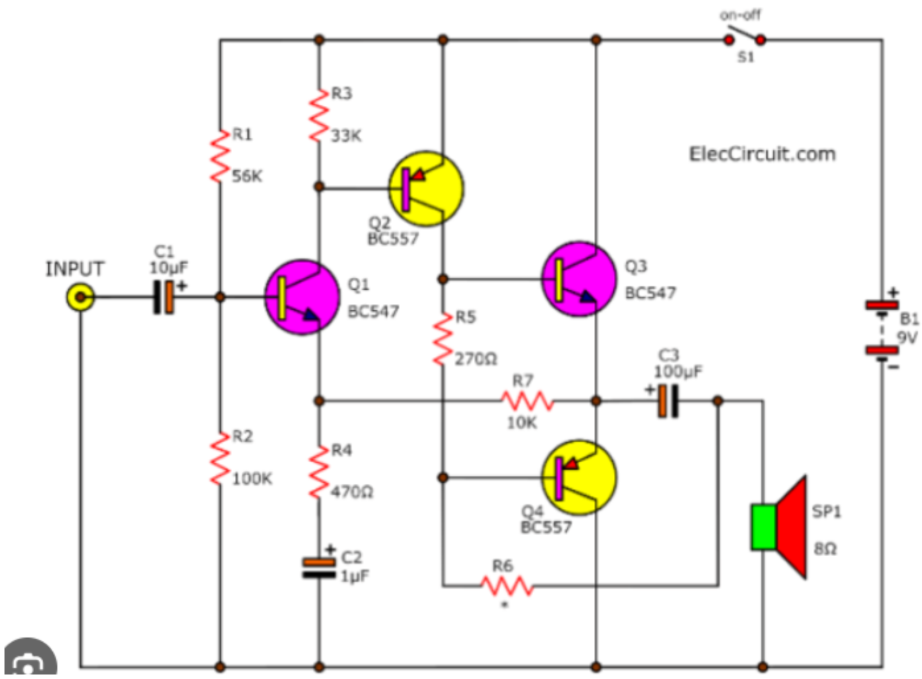


Figure 5.1. CIRCUIT DIAGRAM OF AUDIO AMPLIFIER

5.3 COMPONENTS REQUIRED

i) Q1,Q3: BC547 or equivalent, 45V 0.1A, NPN Transistor
Q2,Q4: BC557 or equivalent, 45V 0.1A, PNP Transistor
ii) 0.25W Resistors tolerance: 5
iii) R1: 56K
iv) R2: 100K
v) R3: 33K
vi) R4: 470 ohms
vi) R5: 270 ohms
vii) R6: 1.5K
vii) R7: 10K Electrolytic Capacitors
ix) C1: 10µF 25V
x) C2: 1µF 25V
xi) C3: 100µF 25V
xii) B1: 9-volt batteries Or 9V power supply circuit
xiii) SP1:8 ohms 0.25” Speaker

5.4 WORKING

• Both transistors Q3 and Q4 are arranged as a complementary pair operating in push-pull. Each output transistor deals with one half of the audio cycle, one being cut-off when the other conduct.
• Next, the Q1 and Q2 transistors run as a pre-amp to increase the incoming voltage to drive the output pair.
• Then, at Q1 the bias of the whole circuit commences with the voltage divider made up of the 56K and 100K resistors.
• This provides the base with a bias voltage of 5.5V. The emitter voltage of 0.6V less than this, and will be 4.9V.
• And next, The Q2-transistor is biased so that it provides a voltage across the 270 ohms load resistor, which will give the output transistors. There is a voltage differential of 0.6V between their base and emitter leads.
• This needs to reduce cross-over distortion which occurs whenever two transistors are connected in push-pull.
• The 100uF electrolytic protect DC from appearing on the speaker, the require the speaker to oscillate around this new position.

5.5 SCHEMATIC

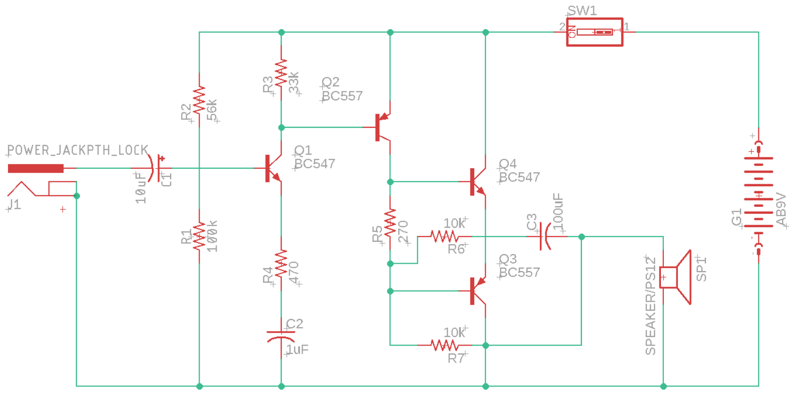


Figure 5.2. SCHEMATIC OF AUDIO AMPLIFIER

5.6 LAYOUT

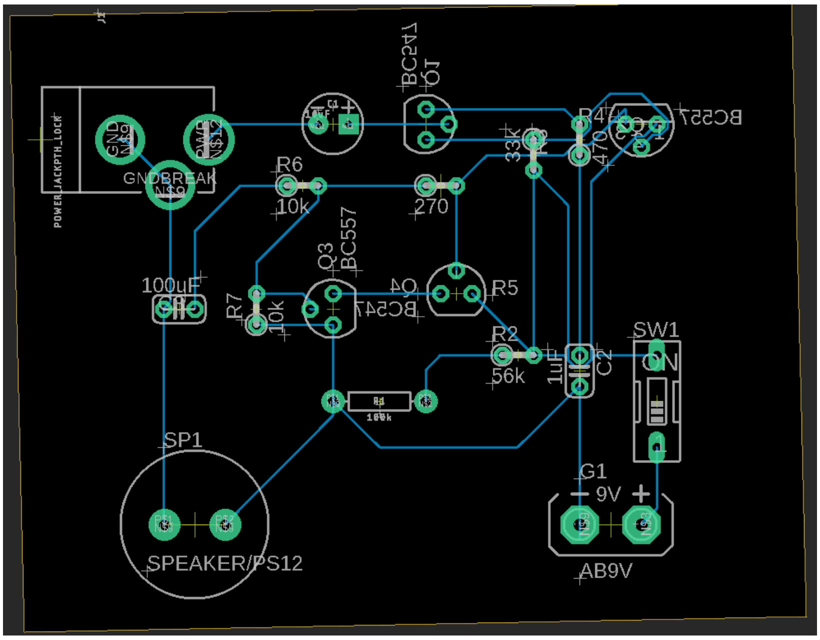


Figure 5.3. LAYOUT OF AUDIO AMPLIFIER

6 LED BLINKING

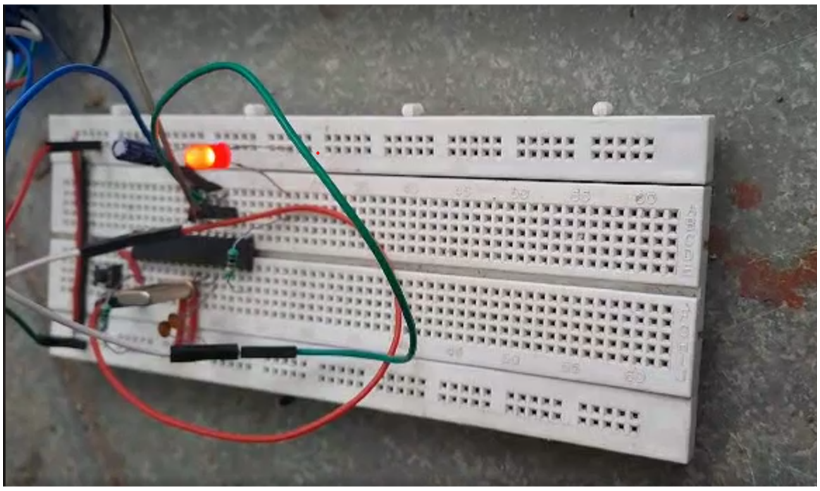


Figure 6.1. BLINKING OF LED

7 CLAP SWITCH USING 555 TIMER AND BC-547 TRANSISTORS

7.1 REQUIRED COMPONENTS

i) 1k, 4.7k, 47k, 330 and 470 ohms resistors
ii) 10µF and 2 100nF capacitors
iii) Electric condenser Mic
iv) Two BC547 transistors
v) LED
vi) 555 Timer
vii) 9V battery

7.2 WORKING PRINCIPLE

This circuit is made of Sound activated sensor, which senses the sound of Clap as input and processes it to the circuit to give the Output. When sound is given as the input to the Electric Condenser Mic, it is changed into the Electrical Energy as the LED turns on. LED turns ON, as we give sound input and it turns OFF automatically after a few seconds. Turn-On LED timer can be changed by varying the value of 100mF capacitor as it is connected with 555 timer whose main purpose is to generate the pulse.
This circuit is mainly based on transistors because the negative terminal of Mic is directly connected with the transistor. In this circuit, we haven't used any Electronic Switch to turn on/off the circuit, so when you are connecting the battery with the circuit, it means your circuit is now turned ON and it will take the inputs in the form of Sound Energy.
As we give the sound input to the circuit, it amplifies the sound signals and proceeds them to the 555 timer which generates the pulse to the LED, making it turn ON.The negative side of the Condenser mic is connected with the amplifier or the circuit will heat-up and may not

work with different models of transistors etc.

7.3 CIRCUIT DIAGRAM

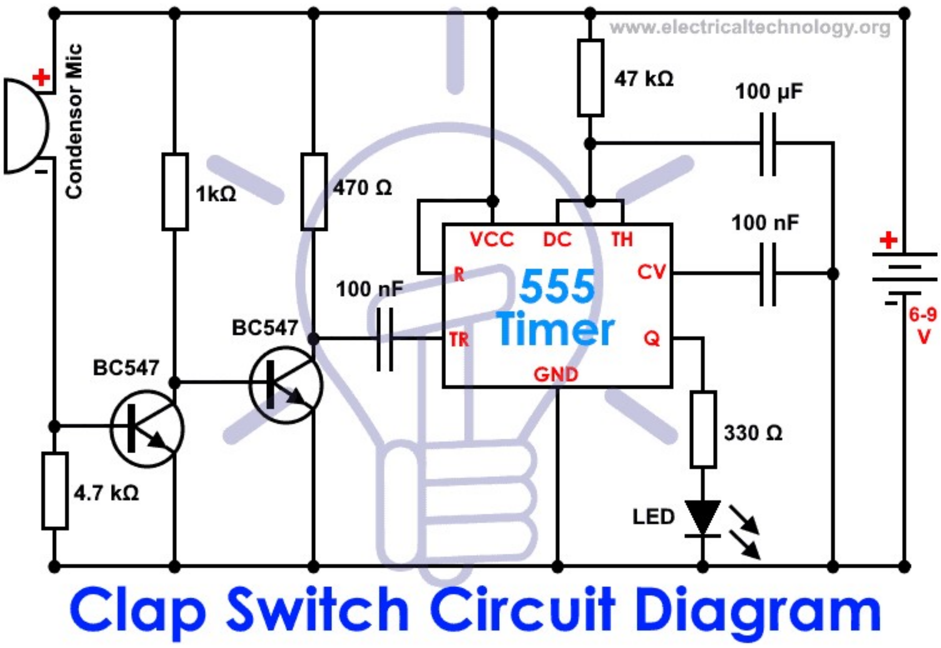


Figure 7.1

8 ULTRASONIC SENSOR

8.1 COMPONENTS REQUIRED

i) ATmega328P microcontroller (or Arduino board with ATmega328P)
ii) Ultrasonic sensor module (e.g., HC-SR04)
iii) Breadboard and jumper wires
iv) Power source (e.g., USB cable or battery)

8.2 WORKING PRINCIPLE

The sensor's receiver detects the reflected waves and measures the time it takes for them to return. Since the speed of sound is relatively constant in a given medium (such as air), the sensor can calculate the distance to the object by using the formula:
Distance = (Speed of Sound × Time) / 2
The division by 2 is necessary because the sound waves travel to the object and then back to the sensor. By knowing the speed of sound and the time it took for the waves to return, the sensor can accurately determine the distance between itself and the object

9 OUR ERRORS AND HOW DID WE OVERCOME?

9.1 FAILED TO UPLOAD CODE FROM ARDUINO USING USBasp

Before we can program our code into the ATmega328 we need to do some additional setup in the Arduino IDE to get the Arduino Uno to function as a programmer.
• Download the file from the given link and extact the file `breadboard-1-6-x`
• We even need to download drivers from Zadic

9.2 FAILED TO IMPLEMENT CLAP SWITCH CIRCUIT ON BREADBOARD

Reasons:
In an ideal scenario, conductors and connecting wires are devoid of internal resistance. However, in practical situations, it is a reality that every connecting wire possesses its own internal resistance. The current within the circuit of the device, measuring in the range of pico amperes to femto amperes, encounters impedance due to the internal resistance inherent in the conductors. Due to this reason our circuit was not implemented on breadboard.

9.3 PROBLEMS FACED DURING ROUTING

1) As micro-controller circuit is complicated we were unable to do single sided PCB. We used top and bottom layers during routing.
2)We even need to maintain certain gap between the two wires. This was done by going into

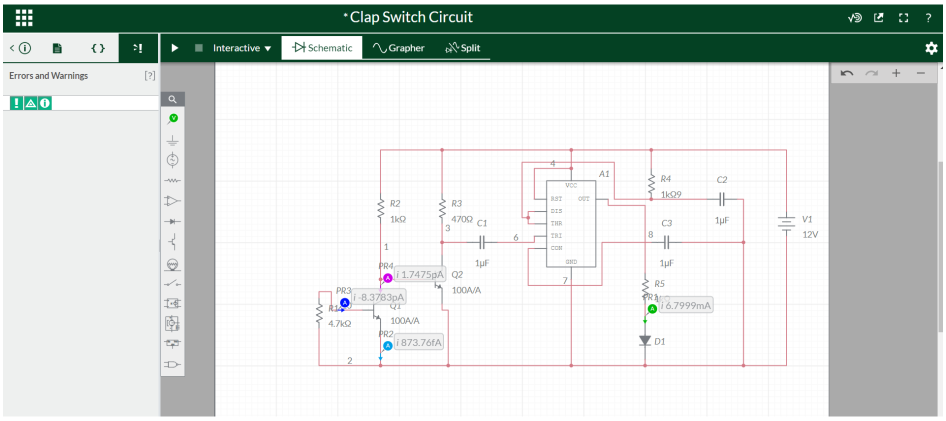


Figure 9.1

clearance. It will be showing our errors if any.
3)The length of the trace between the two pins of the oscillator connecting to atmega328p shoud be same.Same goes with (MISO, MOSI) ; (Rx,Tx) ; (SCL,SGA).

9.4 FAILED TO SIMULATE ULTRASONIC SENSOR ON PROTEUS

Failed to simulate ultrasonic on bread board so decided to stimulate on the proteus But were not able to upload code. As came upon the solution to complile the code in the aurdiuno and with.hex file uploaded the code on the atmega328p but the simulation didn't workout and on process.

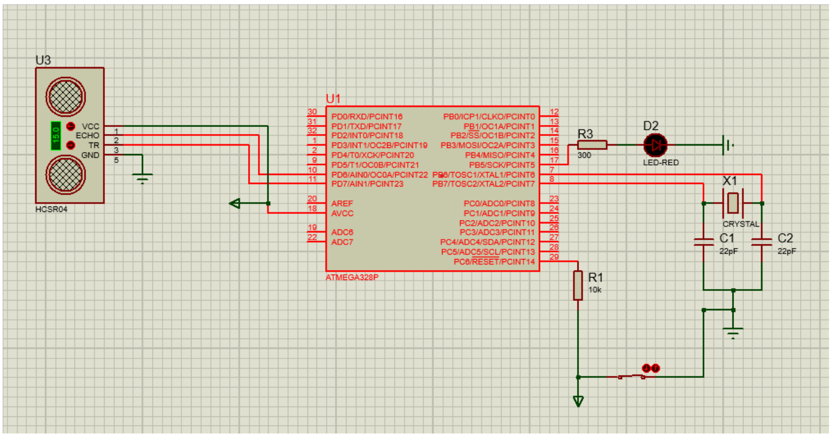


Figure 9.2