Unbiased Electronic Dice with LEDs

A report submitted for implemention of Problem Based Learning (PBL) in the course Electronics Workshop for

Second Year Bachelor Of Technology

AY-2020-21, Sem-IV

By

Sakshi R. Nimje
(GR. No.: 21910310) (Roll No.: 212035)
(Group No.: B2)

Under the guidance of
Prof. Ashwini Navghane

Department of Electronics and Telecommunication Engineering[0.25in]
VISHWAKARMA INSTITUTE OF INFORMATION TECHNOLOGY, PUNE
(An Autonomous Institute Affiliated to Savitribai Phule Pune University)

APRIL 2021

Acknowledgement

Primarily I express my deep sense of gratitude to my advisor Prof. Ashwini Navghane to encourage and guide me to the highest peak to prepare the project.

I would also like to thank HOD sir Prof.(Dr.) S. V. Kulkarni for providing us all the facilities required.

I would like to express my gratitude to Director Prof.(Dr.) Vivek S. Deshpande.

I am extremely thankful to my colleagues and friends in Vishwakarma Institute of Information Technology who have helped me with their valuable suggestions and guidance has been helpful in various phases of completion of project.

Last but not least, my parents are also an important inspiration to me. so with due regards,I express my grattitude to them.

Sakshi R. Nimje
Roll No. 212035

Bansilal Ramnath Agarwal Charitable Trust's
Vishwakarma Institute Of Information Technology
(An Autonomous Institute affiliated to Savitribai Phule Pune University)
An ISO 9001-2008 Certified Institute Accredited with 'A' Grade By NAAC

Certificate

This to certify that the Problem Based Learning (PBL), entitled **"Unbiased Electronic Dice with LEDs"** submitted by **Sakshi R. Nimje** (GR. No. 21910310)(Roll No. 212035) in the partial fulfilment of the requirement for the PBL in Electronics Workshop course of **Second Year Bachelor Of Technology** in Department of Electronics and Telecommunication Engineering, Pune, during the session 2020-21, is a record of the work carried out by him under my supervision and guidance.

Prof. Ashwini Navghane
Guide
VIIT, Pune

Prof.(Dr.) S. V. Kulkarni
Head, Department of E&Tc Engineering
VIIT, Pune

Prof. (Dr.) Vivek S. Deshpande
Director
Vishwakarma Institute of Information Technology, Pune

Date: APRIL 2021

Place: Pune

Bansilal Ramnath Agarwal Charitable Trust's
Vishwakarma Institute Of Information Technology
(An Autonomous Institute affiliated to Savitribai Phule Pune University)
An ISO 9001-2008 Certified Institute Accredited with 'A' Grade By NAAC

Certificate of the Guide

Certified that the work incorporated in the report, **"Unbiased Electronic Dice with LEDs"** submitted by **Sakshi R. Nimje** (GR. No. 21910310)(Roll No. 212035), work carried out by the candidate for the PBL in Electronics Workshop course under my supervision and guidance.

Prof. Ashwini Navghane
Guide
Vishwakarma Institute of Information Technology,
Pune.

Date: APRIL 2021

Place: Pune

Contents

List of Tables

List of Figures

Chapter 1

Introduction

1.1 Circuit diagram of Unbiased Electronic Dice with LEDs

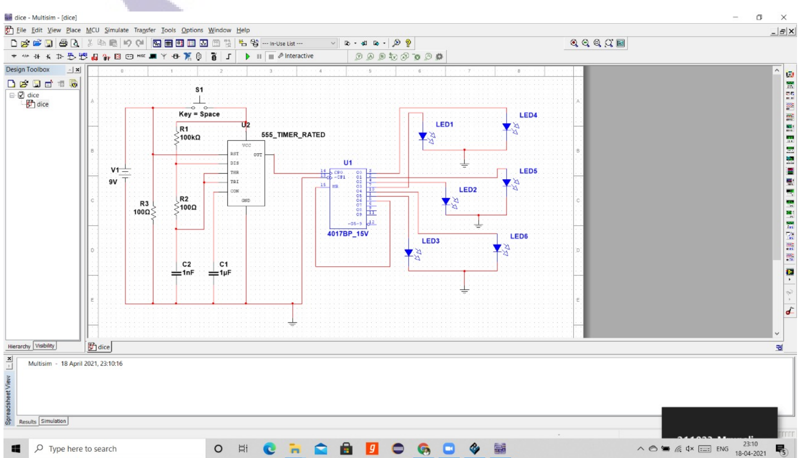


Figure 1.1: Unbiased Electronic Dice with LEDs [1]

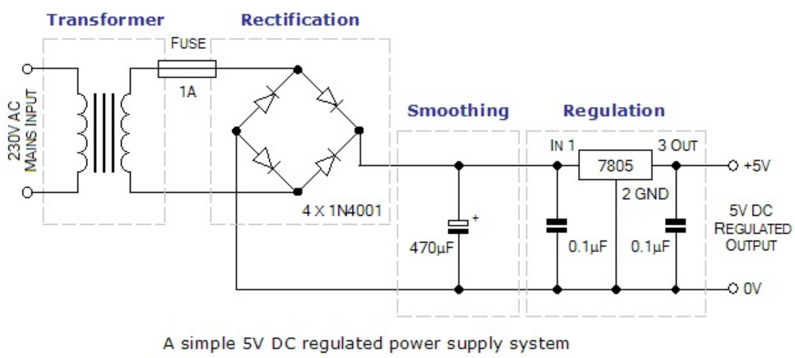


Figure 1.2: Circuit diagram of 9V power supply

1.1.1 Working / operation of Unbiased Electronic Dice with LEDs

This circuit uses 555 timer as an astable multivibrator. In this mode, the circuit isarranged with R2 = 100 K, R3 = 100 K and C2 = 0.1F.7 Chapter 3. Circuit diagram implementation and Simulation using EDA toolWith this configuration, the circuit operates as a pulse generator with a frequencyin order of kilo hertz. This means that the circuit produces a clock cycle of about0.000210 seconds which is imperceptible to the human eye. We cannot observe thevalues which change at that faster rate so there is hardly any possibility of getting the dice biased.**??**.

1.1.2 Working / operation of Power Supply

Power supplies have essential functions found in all models with additional operations added depending on the device type. Power supplies may need to change voltage up or down, convert power to direct current, or regulate power for smoother outcoming voltage. These functions will help you choose which supply you need for your electrical needs. Getting a device with too many features could cost you more money than you need to spend, but if you don't get the features you require, you could damage the devices you need to power.Basic power supplies will change the voltage and convert to DC power. These standard operations send unregulated voltage out of the power supply, but if you need regulated power, the devices have an additional step of regulating the voltage to smooth out waves . As represented in the figure 1.1.1 **??**

1.2 Electronic Component list and specifications

Following are the various electronics components are use as shown in table **??**

Table 1.1: Material Required for Application Design

A	Material Required for Application Design and Approximately cost				
Sr.No.	**Name of Component**	**Name / Specifications**	**Quantity**	**Cost / piece**	**Total**
1	capacitor	0.1uF	1	3	3
2	capacitor	1nF	1	4	4
3	resistors	2.2k	1	2.5	2.5
4	resistors	100k	2	4	2
5	Integrated circuit	555 , 4017	2	14,16	30
6	LED	blue	6	5	30
7	push button	PB NO	1	8	8
8	battery	9V	1	20	20
Total			**15**	**74.5**	**101.5**

Chapter 2

EDA tools for Simulation and PCB design

Electronic design automation (EDA), also referred to as electronic computer-aided design (ECAD), is a category of software tools for designing electronic systems such as integrated circuits and printed circuit boards [2].

The term Electronic Design Automation (EDA) refers to the tools that are used to design and verify integrated circuits (ICs), printed circuit boards (PCBs), and electronic systems, in general. Over time, these early computer-aided drafting tools evolved into interactive programs that performed integrated circuit layout. Other companies like Racal-Redac, SCI-Cards, and Telesis created equivalent layout programs for printed circuit boards. These integrated circuit and circuit board layout programs became known as Computer-Aided Design (CAD) tools. The companies promoting front-end tools for schematic capture and simulation classed them as Computer-Aided Engineering (CAE) [3].

2.1 Various types EDA Tools

Various EDA tools are

- **Altium Designer**
- **NI Multisim.**
- **KiCad EDA.3**
- **EasyEDA**
- **Multisim**

2.1.1 Altium Designer

With Altium Designer, you can quickly create and evaluate schematics for intricate electronics. Set the rules and constraints that you want for your design up front as you create the schematic, and not after it goes to layout. Learn more about the schematic design rules in Altium Designer [4].

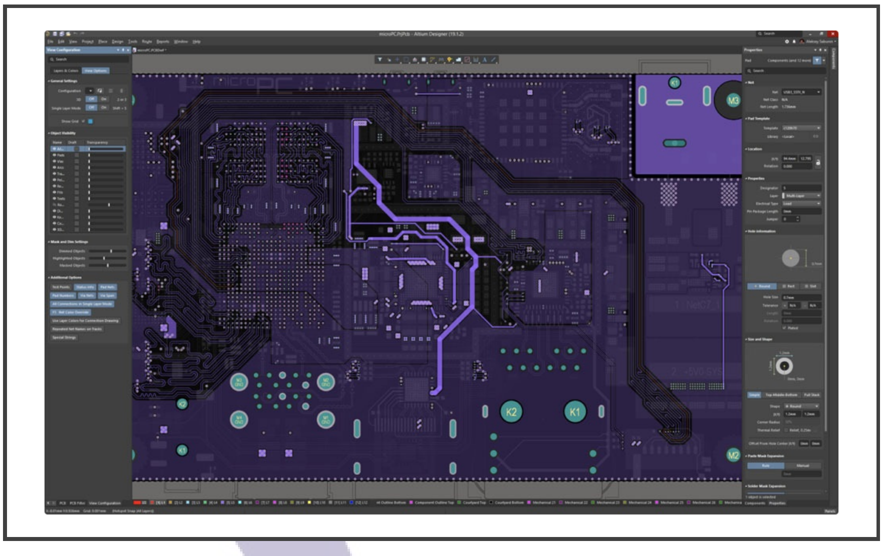


Figure 2.1: Altium [5]

2.1.2 Easy EDA

EasyEDA is a free, zero-install, Web and cloud-based EDA tool suite, integrating powerful schematic capture, mixed-mode circuit simulation and PCB layout. EasyEDA team tries to make our users happy. With this file format, you can create a schematic or PCB using some codes, even with Notepad

2.1.3 NI Multisim

NI Multisim (formerly MultiSIM) is an electronic schematic capture and simulation program which is part of a suite of circuit design programs, along with NI Ultiboard. ... Multisim is widely used in academia and industry for circuits education, electronic schematic design and SPICE simulation

2.1.4 Proteus

The Proteus Design Suite is a proprietary software tool suite used primarily for electronic design automation. The software is used mainly by electronic design engineers and technicians to create schematics and electronic prints for manufacturing printed circuit boards

We have used Multisim for implementation of circuit simulation and easy eda for development of PCB.

Chapter 3

Circuit diagram implementation and Simulation using EDA tool

Multisim is used for Circuit implementation and simulation

The performance of a system texture richness .

3.1 Circuit diagram for Unbiased Electronic Dice with LEDs using EDA Tool

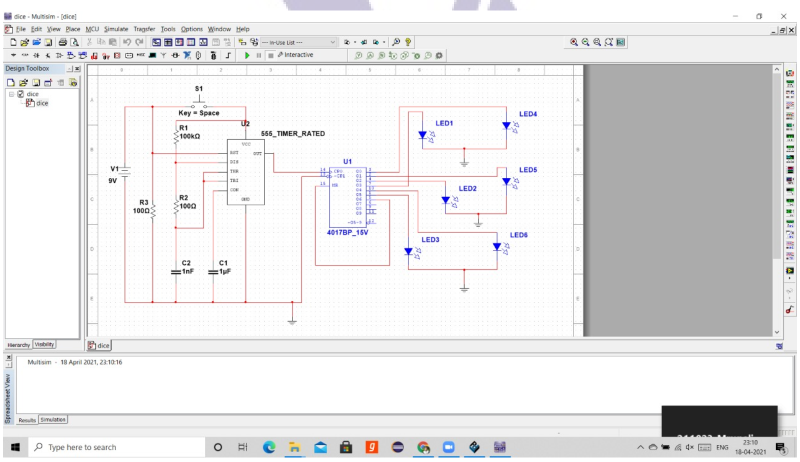


Figure 3.1: Schematic ofUnbiased Electronic Dice with LEDs

3.2 Circuit Simulation using EDA Tool

This circuit uses 555 timer as an astable multivibrator. In this mode, the circuit is arranged with R2 = 100 K, R3 = 100 K and C2 = 0.1 µF.

With this configuration, the circuit operates as a pulse generator with a frequency in order of kilo hertz. This means that the circuit produces a clock cycle of about 0.000210 seconds which is imperceptible to the human eye. We cannot observe the values which change at that faster rate so there is hardly any possibility of getting the dice biased.

The clock pulses are given to a counter cum decoder circuit IC 4017 with the seventh output given to reset. It has nine possible outputs out of which, the seventh is given to reset because we only need a count up to 6 as a dice has six faces only. The first six outputs are given respectively to the LEDs, so that the respective LED will glow for the corresponding count. If the count is 1, LED-1 will glow. If the count if 2, LED-2 will glow and so on until the sixth count. When the count is six, the sixth LED will glow and after that for the next clock pulse the counter will advance and the count increments to seven. In this count, the circuit resets itself as the seventh count is given to the reset pin which is PIN-15.
Display the simulation images which shows the results

This is how the circuit functions and you can increase the frequency of the circuit if you feel that you need more randomness so that it is very hard to perceive. This circuit can be implemented on a general purpose PCB with a 9V DC power supply.

3.3 Conclusion

Thus, after the simulation we can conclude that output or behavior of Unbiased Electronic Dice with LEDs works properly and hence it can be practically implemented on hardware. We can use it in various games like monopoly,ludo etc. Also here is no chance to cheat as the circuit operates at such a high speed that the circuit is almost imperceptible to the human eye. There is also little maintenance and there is hardly any impact on aging of the circuit.

Chapter 4

Printed Circuit Board Design

Multisim is used for PCB implementation

4.1 PCB Requirements specifications

The PCB layout design software is a complete design environment for PCB design – from initial concept through to product realization. With a unified suite of applications, engineers should seamlessly define, visualize, implement and verify their designs. Optimized for the typical small and mid-sized design team, it should appeals to individual all-rounder and corporations operating across multiple sites. 1. 25 micron nominal hole plating as per IPC class 3 2. No track welding or open circuit repair 3. Cleanliness requirements beyond those of IPC 4. Tight control on age of specific finishes 5. Internationally known base materials used – no 'local' or unknown brands allowed 6. Tolerance for copper clad laminate is IPC4101 class B/L 7. Defined soldermasks and ensuring accordance to IPC-SM-840 class T 8. Defined tolerances for profile, holes and other mechanical features 9. NCAB Group specifies soldermask thickness – IPC does not 10. NCAB Group defines cosmetic and repair requirements – IPC does not 11. Specific requirements of depth of via fill 12. Peters SD2955 peelable as standard 13. NCAB Group specific qualification and release process for every purchase order 14. No x-outs accepted

4.2 PCB Design procedure using EDA Tools

Go to easyeda.com,Create a Account and click new project Click new schematic Click Parts(In the Left Tab) and Enter Battery in search and select BATTCON20MM (You can choose Battery Holder based on your choice) and Click Place. Click Parts(In the Left Tab) and Enter Switch in search and select SW08 and Click Place. Click Parts(In the Left Tab) and Enter LED in search and select LED-805 and Click Place. Press 'W' as connect the component as per the image Press Ctrl+S to save and Click convert project to PCB

4.3 General PCB Design Principles

The following principles need to be followed in the determination of special components.
1. Shorten the length of the connection between FM components as far as possible. Vulnerable components can not be too close to each other, and input components and output components should be far away from each other.
2. There may be a high potential difference between certain components or wires, should increase the distance between them, to avoid accidental short-circuit caused an accident. The components with strong electric should be arranged in the position where body is not easy to reach.
3. Components of more than 15g weight shall be fixed by the bracket, and then welded. The components which large and heavy, and large calorific value are not fit for the PCB.
4. For the potentiometer, adjustable inductor coil, variable capacitor, micro switch, such as the layout of the components should be considered the structural requirements of the whole machine.
5. Should be set aside for the location of the positioning hole and the fixed bracket in PCB design.
The following principles need to be followed in the layout circuit and the functional unit.
1. According to the location of each functional unit in the circuit, the layout is convenient for the signal to circulate, and the signal can be kept in the same direction as possible.
2. To each functional unit of the core components as the center, around them to layout. Components should be arranged in PCB board uniformly, neatly, compactly. And try to reduce and shorten the connection between the various components of the lead.
3. For frequency modulation circuit, the distribution parameters of the components should be considered. General circuits should be made as far as possible to parallel arrangement of components. Thus, not only beautiful, but also easy to weld, and easy to mass production.
4. About the components located at the edge of PCB, the distance between them

should not less than 2mm. When the size of PCB board more than 200mm*150mm, should consider: the Mechanical Strength of PCB board.

4.4 Schematic Design

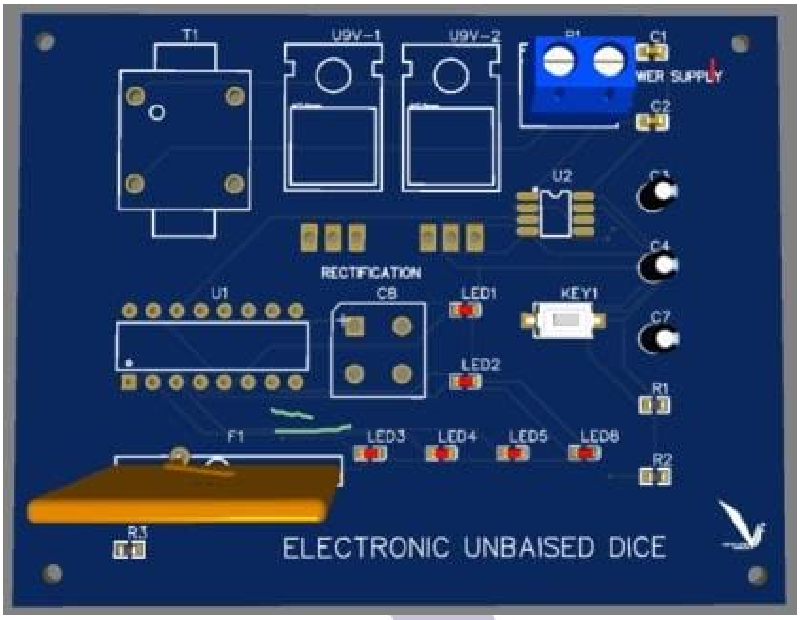
Before you start designing your PCB, it's a good idea to make a schematic of your circuit. The schematic will serve as a blueprint for laying out the traces and placing the components on the PCB. ... The PCB footprint will define the component's physical dimensions and placement of the copper pads or through holes.

Display the image of schematic design

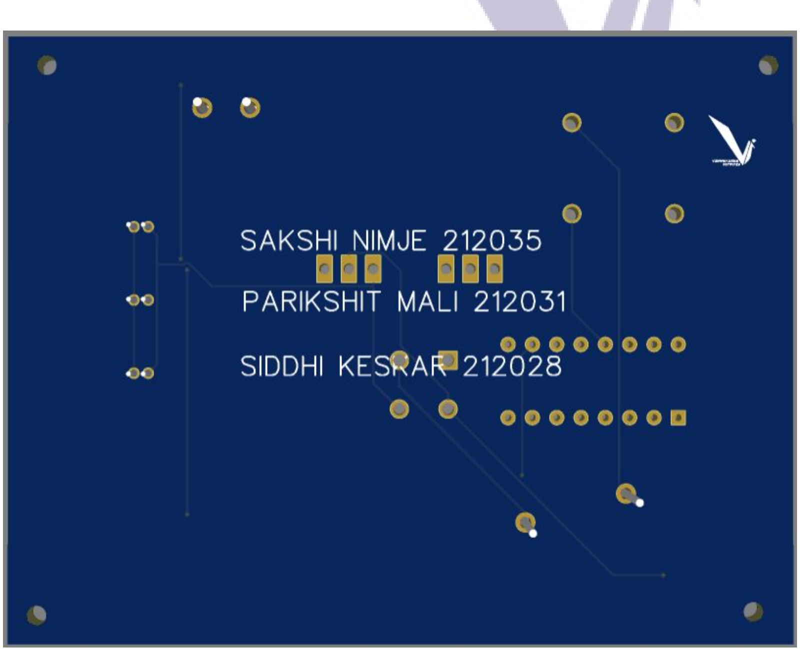
4.4.1 PCB Design rules

Printed Circuit Board Layout Guideline 1. Your board frame should be created on a grid of 0.05 inches with the lower left corner starting at the reference 0.0.
2. Different shapes such as polygons can only be used in specific cases. Normally rectangular board frame is utilized.
3. A 0.05 inch grid is to be utilized for sticking parts. This rule has to be strictly followed unless it's absolutely necessary to be broken.
4. LEDs used on the PCB should be marked for identification. (For example: Status, Lock, D1, D2, Fault)
5. Connectors should be marked as well, such as Vin, Input Port, Vout, etc.
6. Pins on the board are to be labeled, such as, RX, Power, +5V, -5V, etc.
7. Labels for switched should be used, such as, Test, USB, Off, On, etc.
8. You should try to avoid Vias going through silkscreen when labels are being added.
9. It is advised to place components in groups. For example capacitors and resistors around an IC in the schematics should be placed near to the IC on the PCB during layout.
10. 15mil is the minimum size for drill

4.4.2 3D view (Top view components side)



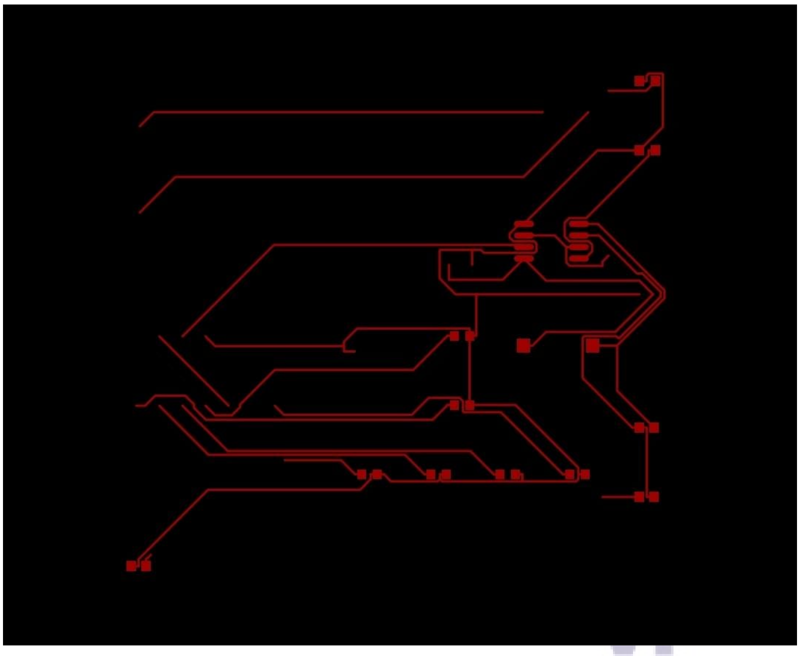
4.4.3 3D view (Bottom view copper side)



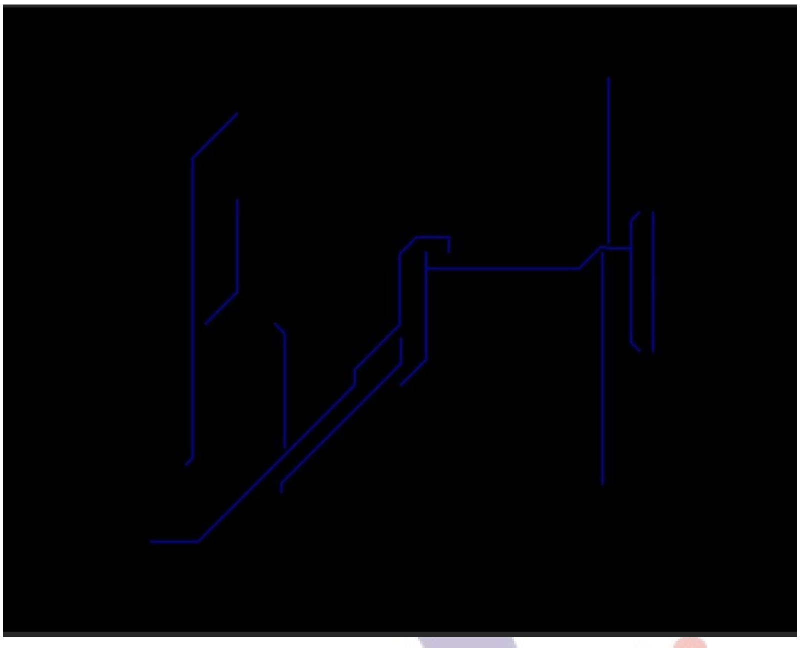
4.5 3D view of PCB design

It is extremely useful in many electronic applications because circuits on one side of the board can be connected on the other side of the board with the help of holes drilled on the board.

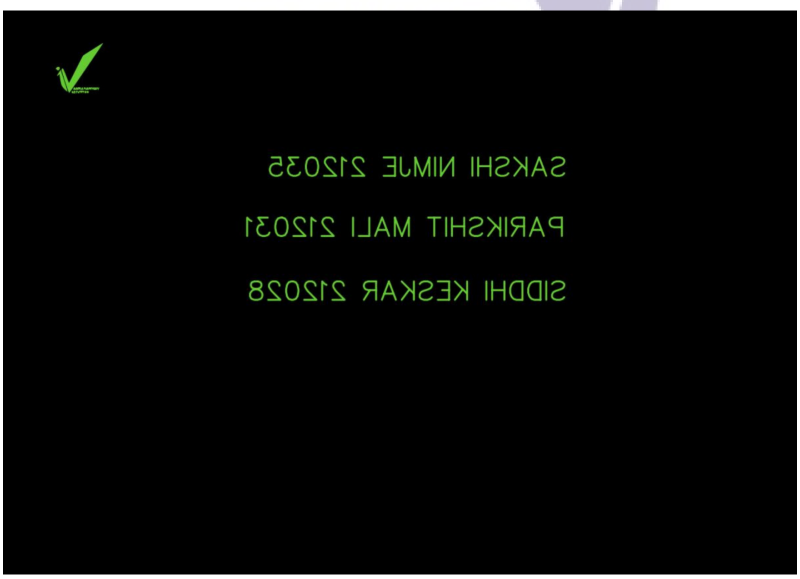
4.6 PCB Design: Top layer (If double sided PCB)



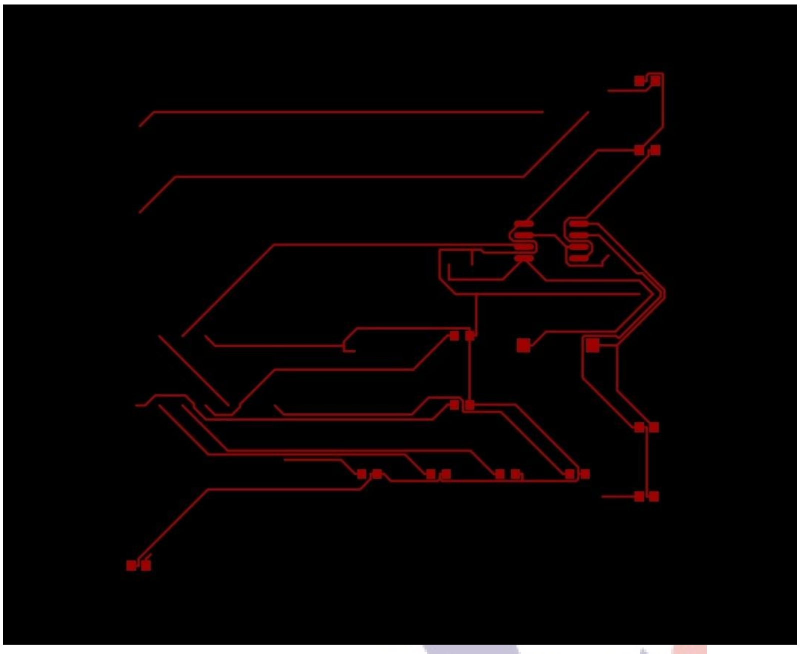
4.7 PCB Design: Bottom layer



4.8 PCB Design: Bottom Silk layer



4.9 PCB Design: Top Silk layer



4.10 PCB Information

A printed circuit board (PCB) mechanically supports and electrically connects electrical or electronic components using conductive tracks, pads and other features etched from one or more sheet layers of copper laminated onto and/or between sheet layers of a non-conductive substrate.

4.11 PCB Bill of material

Table 4.1: Material Required for Designing Power Supply and Approximately cost

B	**Material Required for Designing Power Supply and Approximately cost**				
Sr. No. Total	**Sr.No.**	**Name of Component**	**Name / Specifications**	**Quantity**	**Cost / piece**
1	capacitor	0.1uF	1	3	3
2	capacitor	1nF	1	4	4
3	resistors	2.2k	1	2.5	2.5
4	resistors	100k	2	4	2
5	Integrated circuit	555 , 4017	2	14,16	30
6	LED	blue	6	5	30
7	push button	PB NO	1	8	8
8	battery	9V	1	20	20
Total			**15**	**74.5**	**101.5**

4.12 Inferences

In this section we designed the PCB for out circuit on easyEDA which is an online PCB designing software. We designed the PCB by following all the general rules and Principles. The schematic was drawn by placing all the components individually and then by connecting each component with one another using wires. The 3D view of top and bottom part of our PCB has also been shown along with various layers of PCB.

Chapter 5

PCB Development Process

Kindly refer Expt 6

5.1 PCB Manufacturing Process

Step 1: Design and Output
Step 2: From File to Film
Step 3: Printing the Inner layers: Where Will the Copper Go?
Step 4: Removing the Unwanted Copper
Step 5: Layer Alignment and Optical Inspection
Step 6: Layer-up and Bond
Step 7: Drill
Step 8: Plating and Copper Deposition
Step 9: Outer Layer Imaging
Step 10: Plating
Step 11: Final Etching
Step 12: Solder Mask Application
Step 13: Surface Finish
Step 14: Silkscreen
Step 15: Electrical Test
PCB's are often classified on the basis of frequency, a number of layers and substrate used. Some popular types are discussed below.
Single Sided PCBs:
Single sided PCBs are the basic type of circuit boards, which contain only one layer of substrate or base material. The layer is covered with a thin layer of metal, i.e.copper- which is a good conductor of electricity. These PCBs also contain a protective solder mask, which is applied on the top of the copper layer along with a silk screen coat.
Some advantages offered by single sided PCB's are:

1. Single sided PCB's are utilized for volume production and are low in cost.
2. These PCBs are utilized for simple circuits such as power sensors, relays, sensors and electronic toys.

Double Sided PCBs:

Double sided PCBs have both the sides of the substrate featuring metal conductive layer. Holes in the circuit board allow the metal parts to be attached from one side to the other. These PCBs connect the circuits on the either side by either of the two mounting schemes, namely through-hole technology and surface mount technology. The through-hole technology involves inserting of lead components through the predrilled holes on the circuit board, which are soldered to the pads on the opposite sides. The surface mount technology involves electrical components to be directly placed on the surface of the circuit boards.

Advantages offered by double sided PCBs are:

Surface mounting allows more circuits to be attached to the board in comparison to the through-hole mounting. These PCB's are utilized in a wide range of applications, including mobile phone system, power monitoring, test equipment, amplifiers, and many others.

Multi-layer PCBs:

Multi-layer PCBs are printed circuit boards, which comprise more than two copper layers like 4L, 6L, 8L, etc. These PCBs expand the technology used in double sided PCB's. Various layers of a substrate board and insulating materials separate the layers in multi-layer PCBs. The PCBs are compact sized, and offer benefits of weight and space.

Some advantages offered by multi-layer PCBs are:

1. Multi-layer PCBs offer a high level of design flexibility.
2. These PCBs play an important role in high speed circuits. They provide more space for conductor pattern and power. Rigid PCBs
3. Rigid PCBs refer to those types of PCBs whose base material is fabricated from a solid material and which cannot be bent.

Some salient advantage offered by them:

These PCBs are compact, which ensures the creation of variety of complex circuitry around it. Rigid PCBs offer easy repair and maintenance, as all the components are clearly marked. Also, the signal paths are well organized.

Flexible PCBs:

Flexible PCBs are constructed on a flexible base material. These PCBs come in

single sided, double-sided and multilayer formats. This helps in reducing the complexity within the device assembly.
Some advantages offered by these PCBs are:
1.These PCBs help save a lot of space, along with reducing the overall board weight.
2. Flexible PCBs helps in decreasing the board size, which makes it ideal for various applications where high signal trace density is needed.
3. These PCBs are designed for working conditions, where temperature and density is a main concern.
Rigid-Flex-PCBs:
Rigid flex PCBs are the combination of rigid and flexible circuit boards. They comprise of multiple layers of flexible circuits attached to more than one rigid board. These PCBs are precision built. Hence, it is used in various medical and military applications. Being light-weight, these PCB offer 60 percent of weight and space savings.
High-Frequency PCBs:
High-frequency PCBs are used in the frequency range of 500MHz – 2GHz. These PCBs are used in various frequency critical applications like communication systems, microwave PCBs, microstrip PCBs, etc.
Aluminum backed PCBs:
These PCBs are used in high power applications, as the aluminum construction helps in heat dissipation. Aluminum backed PCBs are known to offer high level of rigidity and low level of thermal expansion, which makes them ideal for applications having high mechanical tolerance. The PCBs are used for LEDs and power supplies.

5.2 Parts of a PCB

U can Display the image also
Some common PCB components include:
1. Battery: provides the voltage to the circuit.
2. Resistors: control the electric current as it passes through them.
3. LEDs: Light emitting diode.
4. Transistor: amplifies charge.
5. Capacitators: these are components which can harbour electrical charge.

5.3 PCB Manufacturing Process Steps

5.3.1 Step 1: Design and Output

Circuit boards should be rigorously compatible with, a PCB layout created by the designer using PCB design software. Commonly-used PCB design software includes Altium Designer, OrCAD, Pads, KiCad, Eagle etc. NOTE: Before PCB fabrication,designers should inform their contract manufacturer about the PCB design software version used to design the circuit since it helps avoid issues caused by discrepancies. Once the PCB design is approved for production, designers export the design into format their manufacturers support. The most frequently used program is called extended Gerber. The 1980's baby food ad campaign sought beautiful babies, and this software creates some beautifully designed offspring. Gerber also goes by the name IX274X.

5.3.2 Step 2: Printing the PCB Design- From file to film

PCB printing begins after designers output the PCB schematic files and manufacturers conduct a DFM check. Manufacturers use a special printer called a plotter, which makes photo films of the PCBs, to print circuit boards. Manufacturers will use the films to image the PCBs. Although it's a laser printer, it isn't a standard laser jet printer. Plotters use incredibly precise printing technology to provide a highly detailed film of the PCB design

5.3.3 Step 3: Printing the Copper for the Interior Layers

The creation of films in previous step aims to map out a figure of copper path. Now it's time to print the figure on the film onto a copper foil. This step in PCB manufacturing prepares to make actual PCB. The basic form of PCB comprises a laminate board whose core material is epoxy resin and glass fiber that are also called substrate material. Laminate serves as an ideal body for receiving the copper that structures the PCB. Substrate material provides a sturdy and dust-resistant starting point for the PCB. Copper is pre-bonded on both sides. The process involves whittling away the copper to reveal the design from the films.

5.3.4 Step 4: Removing the Unwanted Copper

With the photo resist removed and the hardened resist covering the copper we wish to keep, the board proceeds to the next stage: unwanted copper removal. Just as the alkaline solution removed the resist, a more powerful chemical preparation eats away the excess copper. The copper solvent solution bath removes all of the exposed copper. Meanwhile, the desired copper remains fully protected beneath the hardened layer of photo resist.

- **PCB Etching**
  While making PCBs, a layer of copper is added on the substrate. Sometimes, both the sides of the substrate are covered with copper layers. PCB etching process is carried out to get rid of the excessive copper, and only the required copper traces are left behind after the etching process.

- **Material required for PCB etching**
  Currently used for print circuit board etching of solvents are iron (Ferric Chloride), cupric chloride (Cupric Chloride), alkaline ammonia (Alkaline Ammonia), sulfuric acid hydrogen peroxide (Sulfuric Acid + Hydrogen Peroxide) etching fluid, ammonium persulfate, sulfuric acid – chromic acid etching solution.

- **Process for PCB etching**
  Step 1: The very first step of the etching process is designing the circuit, using the software of your choice. Once the design is ready, flip it, and then get it printed.

  Step 2:On the transfer paper, print the circuit design. Make sure that the design is printed on the shiny side of the paper.

  Step 3: Now, take the copper plate, and rub sand paper on it. This will make the surface of copper rough, and thus helps it to hold the design efficiently. There are certain points to remember step 3 till the last step:

  Use safety gloves, while handling copper plate and etching solution. This will prevent the oil from hands getting transferred to copper plate, and will also protect your hands from the solution or chemicals. When you are sanding the copper plate, make sure you do it properly especially at the edges of the plate. Step 4: Now, wash the plate by some rubbing alcohol and water, so that any small particles of copper that get removed from the surface during sanding are washed off. Allow the plate to dry after washing.

Step 5: Cut the printed design properly, and place them on the copper plate facing down.

Step 6: The copper plate is now passed through the laminator several times until the plate gets heated.

Step 7: Take the plate out from the laminator, after it is hot, and hold it in a cold bath. Agitate the plate so that all the paper comes off and floats on the water. You will see a traced circuit in black on the copper plate.

Step 8: Now take the board out of the bath, and place it in the etching solution. Agitate the copper plate for around 30 minutes. Ensure that all the copper around the design is dissolved.

Step 9: Take out the copper plate and wash it in the water bath again. Keep it to dry. Once it has dried completely, you can use rubbing alcohol to remove the ink transferred to the printed circuit board.

Step 10: This completes the etching process of a printed circuit board. You can now drill the holes using proper tools with the required drill bit size. Display the image of etching process.

5.3.5 Step 5: Layer Alignment and Optical Inspection

With all the layers clean and ready, the layers require alignment punches to ensure they all line up. The registration holes align the inner layers to the outer ones. The technician places the layers into a machine called the optical punch, which permits an exact correspondence so the registration holes are accurately punched.

5.3.6 Step 6: Laminating the PCB Layers

In this stage, the circuit board takes shape. All the separate layers await their union. With the layers ready and confirmed, they simply need to fuse together. Outer layers must join with the substrate. The process happens in two steps: layer-up and bonding.

5.3.7 Step 7: PCB Drilling

Finally, holes are bored into the stack board. All components slated to come later, such as copper-linking via holes and leaded aspects, rely on the exactness of precision drill holes. The holes are drilled to a hairs-width - the drill achieves 100 microns in diameter, while hair averages at 150 microns.

- **PCB Drilling**
  The basic materials forming a board are an epoxy glass mix, made by impregnating rolls of woven glass cloth with resin, and sheets of copper foil which are used to make the electrical connections . A typical board may have multiple alternate layers of epoxy glass and copper foil, manufactured by squeezing in a large hydraulic press. The board will not become a circuit until connections are made between the different copper layers and when the discrete electrical components are soldered or welded in place. Thus, through hole drilling is one of the fundamental processes required in the manufacture to produce an electrical connection between the circuits on the board.

- **Material required for PCB Drilling**
  High-Speed Steel (HSS) and tungsten carbide (WC) are commonly used tool materials for drilling of composites. Carbide tools offer better tool wear and tool life during the machining of GFRP. The cemented carbide drills are generally used in PCB drilling U can Display the image also

- **Process for PCB Drilling**
  Step 1: Design and Output. Step 2: From File to Film. Step 3: Printing the

Inner layers: Where Will the Copper Go? Step 4: Removing the Unwanted Copper. Step 5: Layer Alignment and Optical Inspection. Step 6: Layer-up and Bond. Step 7: Drill. Step 8: Plating and Copper Deposition

5.3.8 Step 8: PCB Plating

After drilling, the panel moves onto plating. The process fuses the different layers together using chemical deposition. After a thorough cleaning, the panel undergoes a series of chemical baths. During the baths, a chemical deposition process deposits a thin layer - about one micron thick - of copper over the surface of the panel. The copper goes into the recently drilled holes.

5.3.9 Step 9: Imaging and Plating the Outer Layer

In Step 3, we applied photo resist to the panel. In this step, we do it again - except this time, we image the outer layers of the panel with PCB design. We begin with the layers in a sterile room to prevent any contaminants from sticking to the layer surface, then apply a layer of photo resist to the panel. The prepped panel passes into the yellow room. UV lights affect photo resist. Yellow light wavelengths don't carry UV levels sufficient to affect the photo resist.

5.3.10 Step 10: The Last Etching

We return to the plating room. As we did in Step 8, we electroplate the panel with a thin layer of copper. The exposed sections of the panel from the outer layer photo resist stage receive the copper electro-plating. Following the initial copper plating baths, the panel usually receives tin plating, which permits the removal of all the copper left on the board slated for removal. The tin guards the section of the panel meant to remain covered with copper during the next etching stage. Etching removes the unwanted copper foil from the panel.

5.3.11 Step 11: Applying the Solder Mask

The tin protects the desired copper during this stage. The unwanted exposed copper and copper beneath the remaining resist layer undergo removal. Again, chemical solutions are applied to remove the excess copper. Meanwhile, the tin protects the valued copper during this stage. The conducting areas and connections are now properly established

5.3.12 Step 12: Finishing the PCB and Silkscreening

Before the solder mask is applied to both sides of the board, the panels are cleaned and covered with an epoxy solder mask ink. The boards receive a blast of UV light, which passes through a solder mask photo film. The covered portions remain unhardened and will undergo removal.

5.3.13 Step 13: Electrical Reliability Testing

To add extra solder-ability to the PCB, we chemically plate them with gold or silver. Some PCBs also receive hot air-leveled pads during this stage. The hot air leveling results in uniform pads. That process leads to the generation of surface finish. PCBCart can process multiple types of surface finish according to customers' specific demands.

5.3.14 Step 14: Cutting and Profiling

The nearly completed board receives ink-jet writing on its surface, used to indicate all vital information pertaining to the PCB. The PCB finally passes onto the last coating and curing stage.

5.3.15 Step 15: PCB Cleaning

s a final precaution, a technician performs electrical tests on the PCB. The automated procedure confirms the functionality of the PCB and its conformity to the original design.

- **Material required for PCB Cleaning**
  Most grime can be removed with a cleaning agent, such as isopropyl alcohol (IPA) and a Q-tip, small brush or clean cotton cloth. Cleaning a PCB with

a solvent like IPA should only be done in a well vented environment, ideally under a fume hood. Demineralized water can be used as an alternative.

- **Process for PCB Cleaning**
  PCB board is soaked into acetone solvent for ten minutes. Then, it is placed into quartz container specialized for absolute ethyl alcohol in which circuit board is soaked. Next, ultrasonic cleaning is implemented after placing quartz container into ultrasonic cleaning tank.

5.4 Verification of PCB (Track Continuity checking)

A PCB functional test verifies a PCB's behavior in the product's end-use environment. The requirements of a functional test, its development, and procedures can vary greatly by PCB and end product.

5.5 Verification procedure

Step:1 Understanding the requirements and the block diagram:
Get the required features document from appropriate source. Understand the block. PCB Development Process diagram, and look if they are properly translating to required feature set. In case any feature is not implemented contact to designer for clarification update and restart from Step 1. Get familier with the signal flow directions across blocks. Note down the block, major(active) components in block and related pagenembers. Check for the race condition at block level.
Step 2. Collecting the required data set:Generate/Receive(If provided by designer) BOM for the given schematic and verify if the value, description and footprint name matches in the BOM. Make sure all datasheet for all components in BOM is available with you beforehand, else download them. Make sure that best device is chosen for given operating condition(high efficiency, small footprint or low cost).
Step 3. Power supply verification:
Ask for the power tree from the designer, or create one based on the given schematic to understand the power flow. Power tree should contain the information regarding voltage at each node, which block/components it is supplying, and which ground

reference is used(if the design has multiple grounds). Update the Power tree with interconnect between grounds. Make sure that the ground return path is available everywhere. Ask for current calculation sheet from the designer, or create one. To create such a sheet, all datasheet of components used in the schematic should be handy. Update the power tree with current in each branch. Verify the power supply is correct for every component and the components in the power supply path (e.g. resistor/inductor) are capable of current carrying capacity mentioned in power tree. If any irregularity is found, report to the designer for clarification and update and restart from step 3 if no components are changed, else start from step 2. Check if necessary filter is provided at power inlet so that it should not polute the power supply itself. Inform the Designer

Step 4. Active Component wise verification :

Match the pinout of Schematic symbol with the datasheet. Check if Thermal PAD PCB Development Process is also given a pin or not. If any irregularity is found, report to the designer for clarification and update and restart from step 4. Generally, each component datasheet contains the optimum value of peripheral components and a recomended design pattern, match this against the provided schematic with considerable design tolarance. If any irregularity is found. report to designer for clarification and update and restart from step 4 if no components are changed, else start from step 2. Step 5. Isolation Verification : If the design uses Isolated parts make sure to note them down separately. Make sure "no signal" even Ground is passing from one isolated block to another without Isolation(Capacitive/Magnetic/Optics). If any irregularity is found, report to designer for clarification and update and restart from step 5 if no components are changed, else start from step 2.

Step 6. Net-list Verification:

Generate/Receive netlist in following format : Sr.No., NetName, Level(Power,Analog,LVCMOS3.3) Frequency, Components with pin(e.g. U1.1 means component referenceU1 pin 1). Also, keep a PDF or print of schematic ready. Pick each net and verify if their connection is okay, in terms of voltage level, power handling capacity of the component connected to it and if it is not mistakenly connected to an unwanted component. Create a report and if a mismatch is found add as an error. Check for single nets. Add to report as Warning. Check for termination is the net is a high-speed signal. Add to report as an error. Check for un-necessary branching in any net if the speed of net is greater than 9KHz. All such branches should be either too small (<1 inch) or one on one connection should be made. Add to report as a warning. Check for the race condition on the net level. If violations are there Add to report as an error. With every node being checked, keep marking on the PDF/Printed sheet. Once all net are checked, find if there are any unmarked node left in schematic and report them as error/warning/no error based on engineering judgement. Share the report with designer and verify the corrections w.r.t the shared report before approving it for final approval.

5.6 Inferences

In this section we learnt the PCB development Process and also came to know various types and sides of PCBs along with their application and advantages.also, We came to know the various steps involved in the manufacture of PCB from scratch till the final product. Lastly, We also learnt the the verification procedure to verify each and every component weather it be Power supply components or active components in the circuit.

Chapter 6

Soldering

6.1 Introduction

Soldering is a joining process used to join different types of metals together by melting solder. Solder is a metal alloy usually made of tin and lead which is melted using a hot iron. The iron is heated to temperatures above 600 degrees fahrenheit which then cools to create a strong electrical bond.

6.2 Types of PCB

Single-Sided Printed Circuit Boards This single-sided printed circuit board includes just one layer of base material or substrate. One end of the substrate is coated with a thin layer of metal, usually copper because it is a good electrical conductor. Generally, a protecting solder mask is seated on the peak of the copper layer, and the last silkscreen coat may be applied to the top to mark elements of the board. Double-Sided PCBs This type of PCBs is much more familiar than single-sided boards. Both sides of the board's substrate include metal conductive layers, and elements attach to both sides as well. Holes in the PCB let circuits on a single side to attach to circuits on the other side. Multilayer PCBs These PCBs further enlarges the density and complexity of PCB designs by adding extra layers beyond the top bottom layers seen in a configuration of double-sided. With the accessibility of over many layers in a multilayer printed circuit board configurations, multilayer PCBs let designers make very thick and highly compound designs. Rigid PCBs In addition to having different numbers of layers sides, Printed circuit boards can also come in changing inflexibilities. Most customers usually think of inflexible PCBs when they image a circuit board. Flex PCBs Generally, the substrate is a flexible board is a flexible plastic. This fundamental material permits the board to fit into forms that inflexible boards cannot to turn or shift during use without harmful the circuits on the printed circuit board. Rigid-Flex Printed Circuit Boards Rigidflex boards merge technology from both flexible and rigid circuit boards. An easy rigid-flex board comprises of a rigid circuit board that joints to a flex circuit board.

6.2.1 Through-Hole Components

Through-hole technology, refers to the mounting scheme used for electronic components that involves the use of leads on the components that are inserted into holes drilled in printed circuit boards (PCB) and soldered to pads on the opposite side either by manual assembly (hand placement) or by the use of automated insertion mount machines

6.2.2 Surface Mount Components

Surface-mount technology (SMT) is a method in which the electrical components are mounted directly onto the surface of a printed circuit board (PCB). An electrical component mounted in this manner is referred to as a surface-mount device (SMD). In industry, this approach has largely replaced the through-hole technology construction method of fitting components, in large part because SMT allows for increased manufacturing automation which reduces cost and improves quality - It also allows for more components to fit on a given area of substrate. Both technologies can be used on the same board, with the through-hole technology often used for components not suitable for surface mounting such as large transformers and heat-sinked power semiconductors.

6.3 Soldering Tools and Accessories

1. Soldering Iron.
2. Wire Cutter.
3. Tweezers.
4. USB Microscope.
5. Wet Sponge.
6. Soldering Stand.
7. Safety Glasses

6.4 Solder

There are so many different kinds of solder available on the market today that it can be daunting picking the right one for your project. Fortunately, there are really only three main categories of solder which you can use to narrow down your search: Lead based solder was what kicked of the electronics revolution. The most common mixture is a 60/40 (tin/lead) blend with a melting point around 180-190°C. Known colloquially as soft solder, tin is selected for its lower melting point while lead is used to inhibit the growth of tin whiskers. The higher the tin concentration, the better the tensile and shear strengths. Lead free solder started taking off when the EU started restricting the inclusion of lead in consumer electronics. In the US, manufacturers could receive tax benefits for using lead-free solders. Tin whiskers can be mitigated by using newer annealing techniques, incorporating additives such as nickel, and using conformal coatings. Lead free solders generally have a higher melting point than conventional solder. Flux core solder is sold as a spool of "wire" with a reducing agent at the core. The flux is released during soldering and reduces (reverses oxidation of) metal at the point of contact to give you a cleaner electrical connection. It also improves the wetting properties of the solder. In electronics, flux is usually rosin. Acid cores are for metal mending and plumbing, and should not be used on electronics.

6.5 Flux

a flux is a chemical cleaning agent, flowing agent, or purifying agent. Fluxes may have more than one function at a time. They are used in both extractive metallurgy and metal joining.Some of the earliest known fluxes were sodium carbonate, potash, charcoal, coke, borax,[1] lime,[2] lead sulfide[3] and certain minerals containing phosphorus. Iron ore was also used as a flux in the smelting of copper. These agents served various functions, the simplest being a reducing agent, which prevented oxides from forming on the surface of the molten metal, while others absorbed impurities into the slag, which could be scraped off the molten metal. Fluxes are also used in foundries for removing impurities from molten nonferrous metals such as aluminium, or for adding desirable trace elements such as titanium.

6.6 Soldering Irons

A soldering iron is a hand tool used to heat solder, usually from an electrical supply at high temperatures above the melting point of the metal alloy. This allows for the solder to flow between the workpieces needing to be joined.

6.6.1 Wattage

The formula for calculating wattage is: W (joules per second) = V (joules per coulomb) x A (coulombs per second) where W is watts, V is volts, and A is amperes of current. In practical terms, wattage is the power produced or used per second. For example, a 60-watt light bulb uses 60 joules per second

6.6.2 Temperature

Temperature is a physical quantity that expresses hot and cold. It is the manifestation of thermal energy, present in all matter, which is the source of the occurrence of heat, a flow of energy, when a body is in contact with another that is colder or hotter. Temperature is measured with a thermometer. Thermometers are calibrated in various temperature scales that historically have used various reference points and thermometric substances for definition. The most common scales are the Celsius scale (formerly called centigrade, denoted as °C), the Fahrenheit scale (denoted as °F), and the Kelvin scale

6.6.3 Power

Power is the amount of energy transferred or converted per unit time. In the International System of Units, the unit of power is the watt, equal to one joule per second. In older works, power is sometimes called activity.[1][2][3] Power is a scalar quantity. The output power of a motor is the product of the torque that the motor generates and the angular velocity of its output shaft. The power involved in moving a ground vehicle is the product of the traction force on the wheels and the velocity of the vehicle.

6.7 Tips

Power is the amount of energy transferred or converted per unit time. In the International System of Units, the unit of power is the watt, equal to one joule per second. In older works, power is sometimes called activity.[1][2][3] Power is a scalar quantity. The output power of a motor is the product of the torque that the motor generates and the angular velocity of its output shaft. The power involved in moving a ground vehicle is the product of the traction force on the wheels and the velocity of the vehicle.

6.8 Tip Contamination and Cleaning

Power is the amount of energy transferred or converted per unit time. In the International System of Units, the unit of power is the watt, equal to one joule per second. In older works, power is sometimes called activity.[1][2][3] Power is a scalar quantity. The output power of a motor is the product of the torque that the motor generates and the angular velocity of its output shaft. The power involved in moving a ground vehicle is the product of the traction force on the wheels and the velocity of the vehicle.

6.9 Desoldering

desoldering is the removal of solder and components from a circuit board for troubleshooting, repair, replacement, and salvage.Anything with a base unit with provision to maintain a stable temperature, pump air in either direction, etc., is often called a "station" (preceded by rework, soldering, desoldering, hot air); one, or sometimes more, tools may be connected to a station, e.g., a rework station may accommodate a soldering iron and hot air head. A soldering iron with a hollow tip and a spring-, bulb-, or electrically-operated suction pump may be called a desoldering iron

6.10 Hazards involved in soldering

Soldering with lead (or other metals used in soldering) can produce dust and fumes that are hazardous. In addition, using flux containing rosin produces solder fumes that, if inhaled, can result in occupational asthma or worsen existing asthmatic conditions; as well as cause eye and upper respiratory tract irritation

6.11 Steps for soldering

Step 1 – Melting the Solder
This is the very first step in the entire wave soldering process. It is the basic requirement of the process to melt the solder. The wave soldering machine has solder contained in a tank. The tank is heated to melt the solder. Appropriate temperature is reached to meet the right consistency, so that the process of soldering can be carried out further.
Step 2 – Cleaning the Components
This is a very crucial step to be carried out. The components to be soldered are cleaning thoroughly in this step. If any oxide layers are formed on the components, then they are removed. This is done by the process called fluxing. There are two main types of fluxing – corrosive (high acidity) and noncorrosive (high acidity).
Step 3 – Placement of the PCB
After melting the solder and cleaning the components to be soldered, the printed circuit board is placed on the melted solder. The board is held with the metal clasps of the machine, which ensure the firm positioning and placement of the PCB.
Step 4 – Application of Solder
Now that the PCB is placed properly, molten solder is applied, and is allowed to settle. Sufficient time is given to this step to allow the solder to settle into the joints completely, and ensure no bumps are formed.

6.12 Inferences

In this section we came to the soldering process in detail along with the types of drill bits and type of drilling for various types of components. Later we collected more information on various types of PCB soldering tools and Accessories. We got to know various terms used in soldering such as Flux, solder etc. Lastly we learnt about the steps of soldering our components along with the hazards involved in it and along the desoldering if we soldered any component incorrectly.

Chapter 7

In Circuit Testing and Troubleshooting

7.1 Introduction

1. Software Testing Saves Money Testing has many benefits and one of the most important ones is cost-effectiveness. Having testing in your project can save money in the long run. Software development consists of many stages and if bugs are caught in the earlier stages it costs much less to fix them. That is why it's important to get testing done as soon as possible. Getting testers or QA's who are technically educated and experienced for a software project is just like an investment and your project will benefit budget-wise. 2. Security Another important point to add is security. This is probably the most sensitive and yet most vulnerable part. 3. Product Quality In order to make your product vision come to life, it has to work as planned. Following product requirements is imperative, to an extent, because it helps you get the wanted end results. 4. Customer Satisfaction The ultimate goal for a product owner is to give the best customer satisfaction. Reasons why apps and software should be tested is to bring the best user experience possible. Being the best product in this saturated market will help you gain trustworthy clients which will have great long-term effects.

7.2 In Circuit Testing for the respective Application

Firstly starting with visual Inception, We checked the whole circuit thoroughly with all the connections where we can find an error or any wiring or placing issue. If there is any mis-happening while soldering the PCB we can see burn spots or telltale signs on the PCB . Secondly, By checking the Power Module, by powering up the PCB, we measured the Voltage rails with Digital MultiMeter (both the input and output of the Voltage Regulator need to show the expected values).Checking of fuse is necessary if the input voltage regulator shows 0V reading. After replacement of fuse if it immediately breaks after power-up, it means other components are shorted and draining a huge amount of current. Also, getting a voltage of 0V, or below Vcc, at the output often means that the regulator or a component along the voltage rail has a short circuit. If that's the case, the damaged component will heat up quickly. We can check it by gently placing our hand over the component so that we can feel the excessive Heat. Likewise checking all the components that are being heated up, we checked our circuit. Lastly, we checked the Input and Output ports of the circuit, as they are the common points of failure.Damage on Input and output ports seldom shuts down the whole circuit, but it usually results in anomalies in the system.

7.3 Troubleshooting Procedure for the respective Application

The troubleshooting process steps are as follows:
Step 1. Identify the problem.
Step 2. Establish a theory of probable cause.
Step 3. Test the theory to determine the cause.
Step 4. Establish a plan of action to resolve the problem and implement the solution.
Step 5. Verify full system functionality and, if applicable, implement preventive measures.
Step 6. Document findings, actions, and outcomes.

Chapter 8

Electronics Enclosure Design

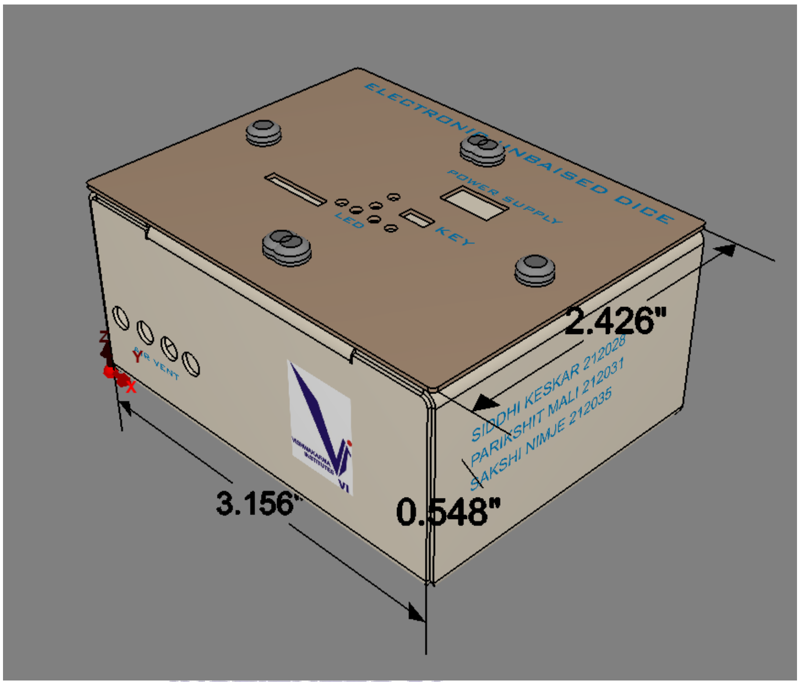


Figure 8.1: 3D View

This is the 3D view of our enclosure which will be used to place PCB we designed

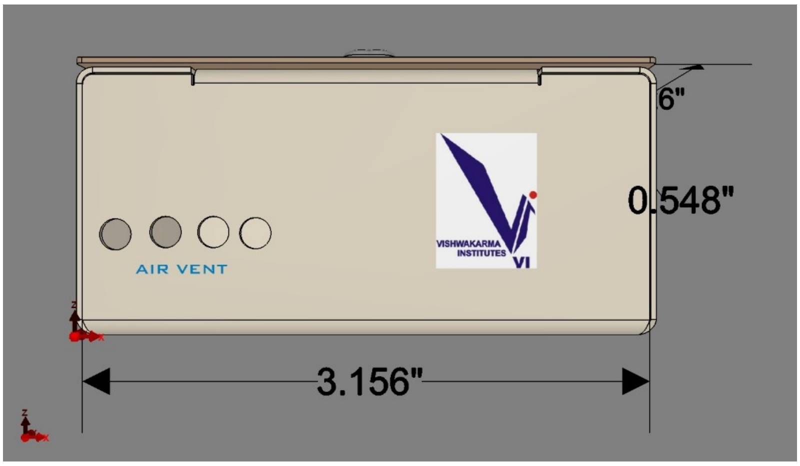


Figure 8.2: Front View

This is the Front view of our enclosure which contains holes for air ventilation so that our PCB will not heat. It also displays the image of our college logo.

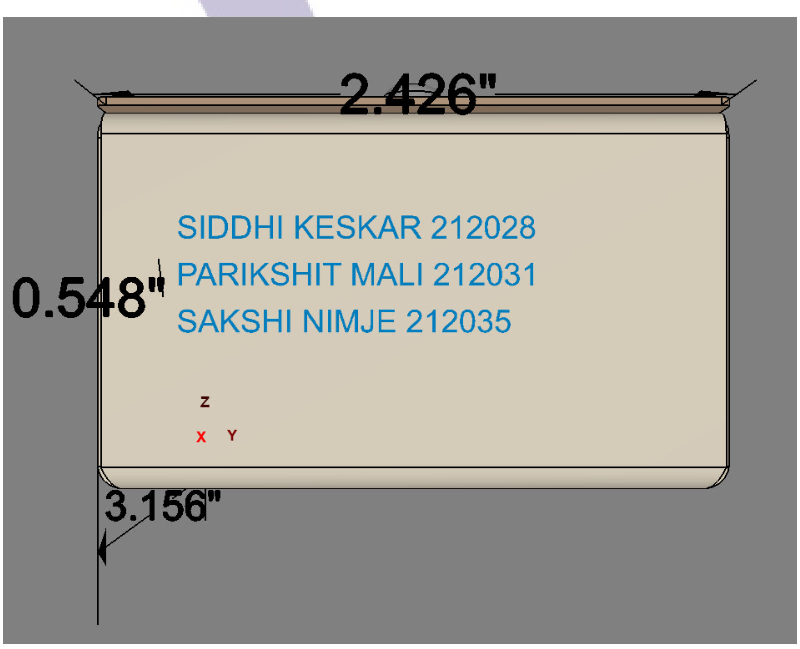


Figure 8.3: Right View

This is the right view of our enclosure which displays the name of the students who contributed in this project.

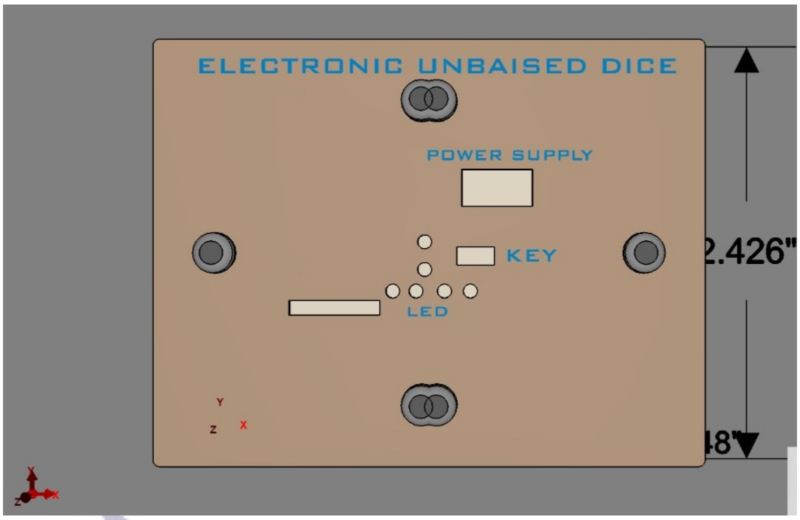


Figure 8.4: Top View

This is the Top View of our enclosure which has the 1. Power Supply Hole from where we can give the Power supply to the PCB. 2. Key to initiate the process 3. Led space where we can see a red Led so we get to know that our circuit is functioning correctly.

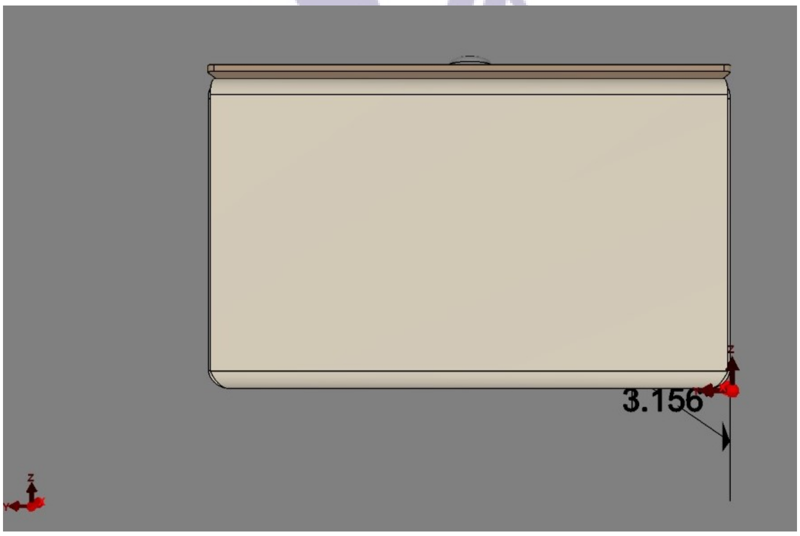


Figure 8.5: Left View

This is the Left view of our enclosure.

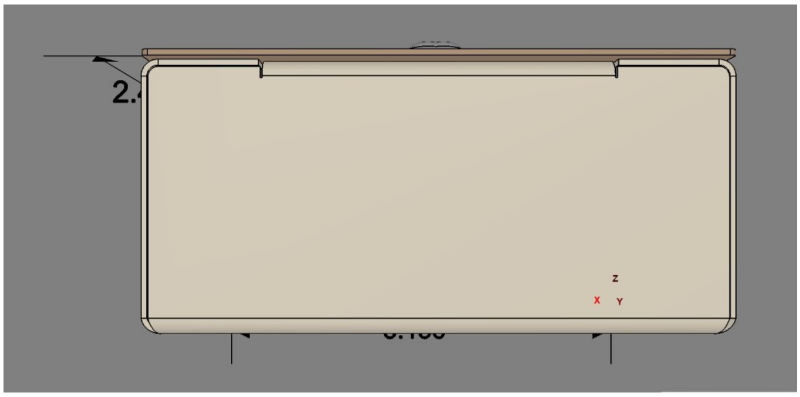


Figure 8.6: Back View

This is the Back view of our enclosure.

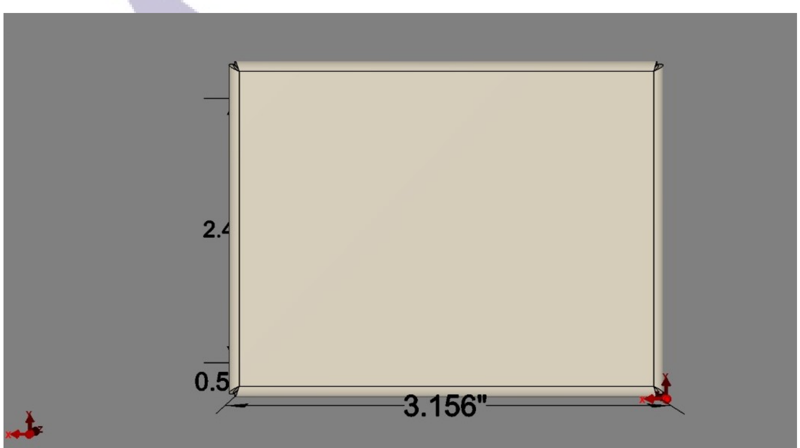


Figure 8.7: Bottom View

This is the Bottom view of our Enclosure which will be facing to the surface where we will be placing it.

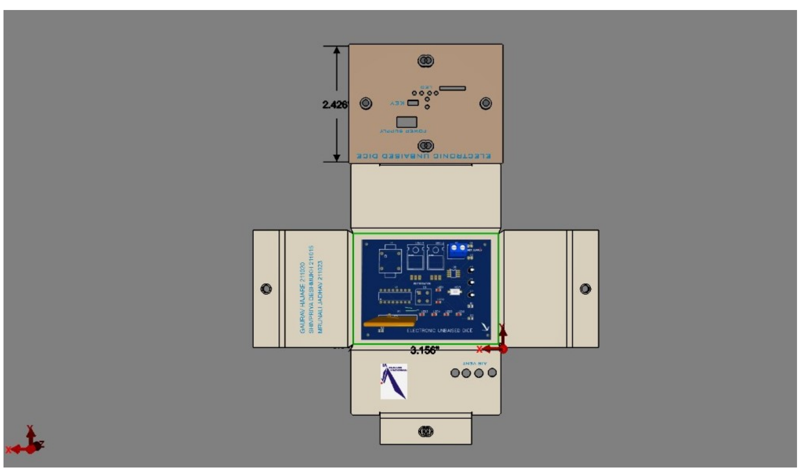


Figure 8.8: Flat View with PCB

Chapter 9

Budget

9.1 Approximate Budget of PBL based Project

For example ; Following table shows approximate budget of the project

Table 9.1: Material Required for Application Design

A	**Material Required for Application Design and Approximately cost**				
Sr.No.	**Name of Component**	**Name / Specifications**	**Quantity**	**Cost / piece**	**Total**
1	capacitor	0.1uF	1	3	3
2	capacitor	1nF	1	4	4
3	resistors	2.2k	1	2.5	2.5
4	resistors	100k	2	4	2
5	Integrated circuit	555 , 4017	2	14,16	30
6	LED	blue	6	5	30
7	push button	PB NO	1	8	8
8	battery	9V	1	20	20
Total			**15**	**74.5**	**101.5**

Table 9.2: Material Required for Designing Power Supply and Approximately cost

B	Material Required for Designing Power Supply and Approximately cost				
Sr. No.	**Name of material**	**Specification**	**Quantity**	**Cost / piece**	**Total**
1	Transformer	230V, 50Hz, Step Down	1	80	80
2	Diode	IN4007	4	2	8
3	Capacitor	470uF	1	1	1
4	Capacitor	0.01uF	2	1	2
5	Voltage Regulator	9V DC output	1	10	10
6	Terminal block	2 terminals	2	5	10
7	Pcb clad	3.175cm x3.937cm	1	10	10
		Total	**12**	**109**	**121**

Table 9.3: Material/Tools Required Developent of PCB and Cabinet and Approx. cost

C	Material/Tools Required Developent of PCB and Cabinet and Approx. cost				
Sr. No.	**Name of material/ Tools**	**Specification**	**Quantity**	**Cost / piece**	**Total**
1	FeCl3	FeCl3	50 ml	20	8
2	Tray	15cm X15 c	1	25	10
3	Drill machine	Hand drill	1	50	50
4	Drill Bit	1mm	1	15	15
5	Drill Bit	.8 mm	1	15	15
6	Soldring Gun	50 Watt	1	150	150
7	Solder	1mm Diameter Tin	50 gm	20	20
8	Multimeter	DMM	1	300	300
9	Wire stripper	stripper/cutter	1	30	30
N	Cabinet (Aluminium)	9.1cm X 7.3cm X 3cm	1	80	80
		Total	**8**	**705**	**678**

Chapter 10

Overall Conclusion

10.1 Overview of Objectives and Achievements

Chapter wise understanding...
(Students are expected to write overall conclusion from each chapter for the respective application)

Chapter 1: Thus, after the simulation we can conclude that output or behavior of Unbiased Electronic Dice with LEDs works properly and hence it can be practically implemented on hardware. We can use it in various games like monopoly,ludo etc. Also here is no chance to cheat as the circuit operates at such a high speed that the circuit is almost imperceptible to the human eye. There is also little maintenance and there is hardly any impact on aging of the circuit.
Chapter 2: EDA tool is used to simulate our project we understood various attributes of different EDA tools and choose Multisim for simulation of our project design.
Chapter 3: After choosing EDA Tool as multisim we simulated our project and then we simulated our project schematic on multisim and observed and understood working of circuit .
Chapter 4: After simulating and ensuring proper working of circuit it was time tocreate pcb so we used multisim as our tool and observed 3d view which helped us in accurate placing of components.
Chapter 5: When we create PCB we need to assure Quality of pcb.How to maintain quality of pcb how it should be developed and how to manufacture PCB.
Chapter 6: To place components its very essential of proper soldering of components. Soldering of components only decides the working and to maintain continuity. We understood proper soldering techniques and information about solder guns and other soldering components and their properties in this chapter.Desoldering is another topic we understood.
Chapter 7: If after proper soldering if any error occures we should know proper

troubleshooting techniques and we should know about in circuit testing technique which we understood from this chapter.
Chapter 8: As our PCB is ready and as our project is Ready we need enclosure to make our circuit safe.Which material to use for enclosure and all other techniques are explained in chapter 8.
Chapter 9: Chapter 9 shows us overall budget of the project.

Bibliography

[1] [Online]. Available: https://www.electronics-tutorials.ws/wp-content/uploads/2018/05/waveforms-tim20.gif.

[2] [Online]. Available: https://en.wikipedia.org/wiki/Electronic_design_automation#:~:text=Electronic.

[3] [Online]. Available: https://www.sciencedirect.com/topics/engineering/electronic-design-automation.

[4] [Online]. Available: https://www.sciencedirect.com/topics/engineering/electronic-design-automation.

[5] [Online]. Available: https://play.vidyard.com/s4L9MaH2itrsS6DwKBL18w.jpg.