MARRI LAXMAN REDDY
INSTITUTE OF TECHNOLOGY AND MANAGEMENT
(AN AUTONOMOUS INSTITUTION)
(Approved by AICTE, New Delhi & Affiliated to JNTUH, Hyderabad)
Accredited by NBA and NAAC with 'A' Grade & Recognized Under Section2(f) & 12(B)of the UGC act,1956

Department of Electronics & Communication Engineering

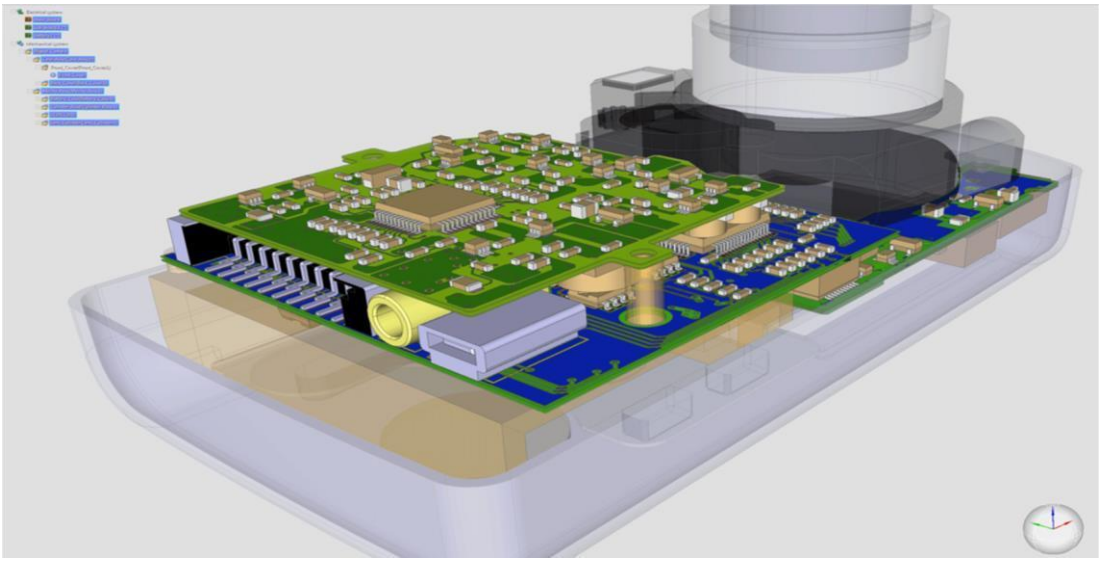
PRINTED CIRCUIT BOARD

Design Lab Manual

IIB.TECH-II SEMESTER

(ECE) R22 (MLRS)

REGULATION



A.Y2024-2025

CERTIFICATE

This is to certify that this manual is a Bonafide record of practical work in the ***Printed Circuit Board Design lab*** in **first Semester of II-year B. Tech (ECE) SEM-I Programme** during the academic year **2024-2025.** This Lab Manual is prepared Dr.R. Prabhakar **(Associate Professor)** Department of Electronics and Communication Engineering.

Head of the Department

PREFACE

This foundation for the Electronics and Communication Engineering students during Second year of their course.

In this course students will know how to design, simulate and layout design of Electronic Circuits and functional checks using EDA Design, Porteus and Kicad software tools.

By,

Dr.R. Prabhakar

ACKNOWLEDGEMENT

It was really a good experience, working with *Printed Circuit Board Design*. First, we would like to thank Dr. N. Srinivas Assoc. Professor, HOD of Department of Electronics and Communication Engineering, Marri Laxman Reddy Institute of technology & Management for his concern and giving the technical support in preparing the document.

We are deeply indebted and gratefully acknowledge the constant support and valuable patronage of Dr. Ravi Prasad, Dean, Marri Laxman Reddy Institute of technology & Management for giving us this wonderful opportunity for preparing the *Printed Circuit Board Design.*

We express our hearty thanks to Dr. R. Murali Prasad, Principal, Marri Laxman Reddy Institute of technology & Management, for timely corrections and scholarly guidance.

We express our hearty thanks to Dr. P. Sridhar, Principal, Marri Laxman Reddy Institute of technology & Management, for timely corrections and scholarly guidance.

At last, but not the least I would like to thanks the entire ECE Department faculty those who had inspired and helped us to achieve our goal.

By,

Dr.R. Prabhakar

GENERALINSTRUCTIONS

1. Students are instructured to come to *Printed Circuit Board Design*. Late comers are not entertained in the lab.

2. Students should be punctual to the lab. If not, then conducted experiments will not be repeated.

3. Students are expected to come prepared at home with the experiments which are going to be performed.

4. Students are instructed to display the identity cards before entering in to the lab.
5. Students are instructed not to bring mobile phones to the lab.
6. Any damage/loss of equipment like mouse, keyboard, CPU etc., during the lab session, it is student's responsibility and penalty or fine will be collected from the student.

7. Students should update the records and lab observation books session wise. Before leaving the lab, the student should get his lab observation book signed by the faculty.

8. Students should submit the lab records by the next lab to the concerned faculty members in the staffroom for their correction and return.

9. Students should not move around the lab during the lab session.
10. If any emergency arises, the student should take the permission from faculty member concerned in written format.

11. The faculty members may suspend any student from the lab session on disciplinary grounds.

12. Never copy the out put from other students. Write down your own out puts.

Printed Circuit Board Design

Dr.R. Prabhakar , Assoc. Professor, ECE

Course Objectives:

The students will try to

- Understand components and materials simulation tools
- Understand PCB simulation tools
- Develop PCB Designing Flow Chart and description
- Single layer and multilayer PCB
- Design of different circuits on PCB

Course Outcomes:

After successful completion of the course, students shall be able to

- Understand components and their categories
- Understand PCB simulation tools
- Develop PCB Designing Flow Chart and description
- Single layer and multilayer PCB
- Design of different circuits on PCB

List of Experiments:

1 Introduction to circuit creation and simulation using Easy EDA.

2 Introduction to Layout Tool, and creating Layout board using Proteus

3 Design a RLC circuit & verify it for different values of R, L &C.

4 Design a half wave rectifier using Proteus.

5 Design a full wave centre tapped rectifier using Proteus.

6 Design a clipper circuit using Proteus.

7 Design a clamper circuit using Proteus.

8 Convert the power supply circuit into PCB.

Materials required for the fabrication of simple PCB's

9.PCB copper clad boards

10.Ferrous chloride' for PCB etching

11.Hand drills with needles

12.Glossy photo paper (130gsm) hands on schematic implantation on board

13.Hand grouses, Soldering iron, Soldering paste flux, Soldering lead

14.Iron boxes as a heat sink

15. Development of different mini projects on PCB.

EXPERIMENT: 1

AIM: PCB Design of Process using Easy EDA Tool

.

TOOL: Easy EDA Tool.

THEORY:

Welcome to Easy EDA, a great web based EDA (Electronic Design Automation) tool for electronics engineers, educators, students, makers and enthusiasts. There is no need to install any software. Just open Easy EDA in any HTML5 capable, standards compliant web browser. Whether you are using Linux, Mac or Windows, it is highly recommended to use Chrome or Firefox as your browser. You can also download Easy EDA client. EasyEDA has all the features you expect and need to take your design rapidly and easily from conception through to production.

EasyEDA Editor:

https://easyeda.com/editor

Instruction:

- This tutorial document will be updated as changes are made to the EasyEDA editor.

Tutorial for PDF

EasyEDA-Tutorials.pdf

EasyEDA Provides:

- Simple, Easy, Friendly, and Powerful drawing capabilities
- Works Anywhere, Anytime, on Any Device
- Real-time Team Cooperation
- EasyEDA source file export(json)
- PCB Layout
- Design Rules Checking(DRC)
- EasyEDA source file export(json)
- Photo view
- 3D View

- Generate fabrication file(Gerber)
- PCB module
- Library management
- Symbol/Subpart create and edit
- Spice symbol/model create and edit
- Libraries management
- Footprint create and edit

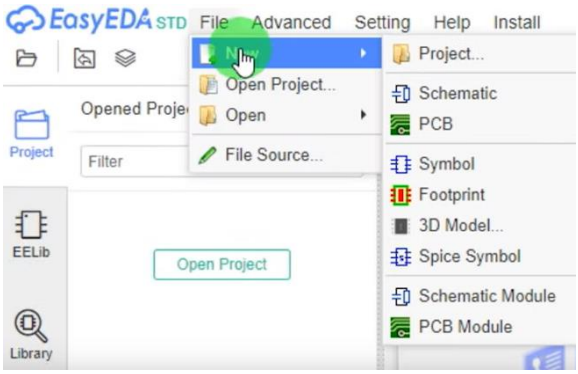
PCB Designing

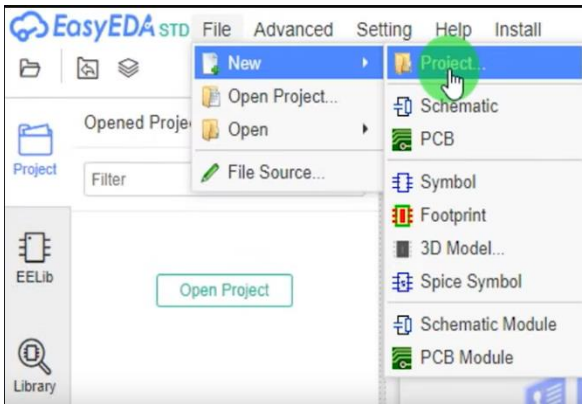
Printed circuit board (PCB) design **brings your electronic circuits to life in the physical form**. Using layout software, the PCB design process combines component placement and routing to define electrical connectivity on a manufactured circuit board.

PCB using EASY EDA software.

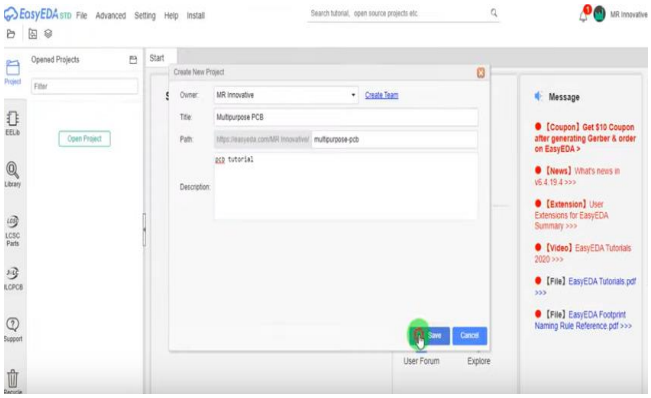
Steps for Designing the PCB

- First down load and Install the EASY EDA student version software.
- Now open the EASY EDA and click on new project and click on schematic.

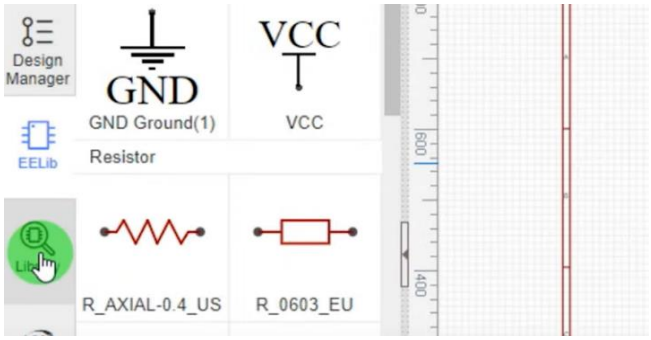




- Write File name and Save.



- Pick the components as required.
-



- Design the circuit as the per circuit diagram using wire tool.
- After the design is complete, verify the circuit and save the circuit.
- Now go to the top tool list, click on design and select convert schematic to PCB.
- Now place the components inside the boundary and arrange them properly in an order.
- Now route the components without over lapping at bottom layer, adjust the track width and hole diameter as per required.
- Save it now click on 3d to see the PCB 3d model.
- Now go to file, export it in pdf by selecting the layers.

Example

AIM:PCB Design of R,LED and Switch Circuit with DC Power Supply of Battery

.

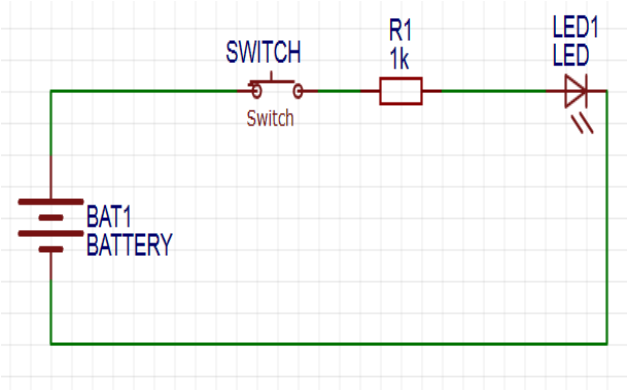
TOOL: Easy EDA Tool.

THEORY:

Easy EDA is a web-based EDA tool suite that enables hardware engineers to design, simulate, share - publicly and privately - and discuss schematics, simulations and printed circuit boards. Other features include the creation of a Bill of Materials, Gerber and pick and place files and documentary outputs in PDF, PNG and SVG formats. Easy EDA allows the creation and editing of schematic diagrams, SPICE simulation of mixed analogue and digital circuits and the creation and editing of printed circuit board layouts and, optionally, the manufacture of printed circuit boards.

Also check my other Two PCB Designing tutorials

Circuit Diagrams



Procedure

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 **ATTCON_20MM**

(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

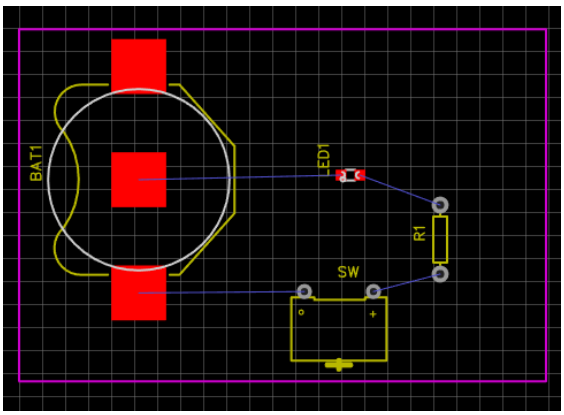
SW_08 and Click **Place**.

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

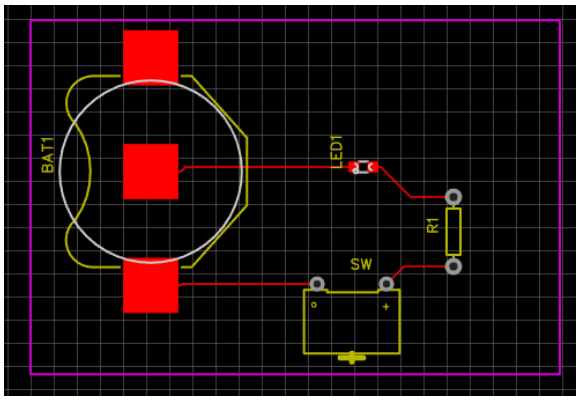
Press **Ctrl+S** to save and Click **convert project to PCB** (3rd opt after Zoom)

Step 2: Arrange Components



Arrange the components as per the image (Blue lines are called rats nest) or arrange keeping close to each other(not too close).

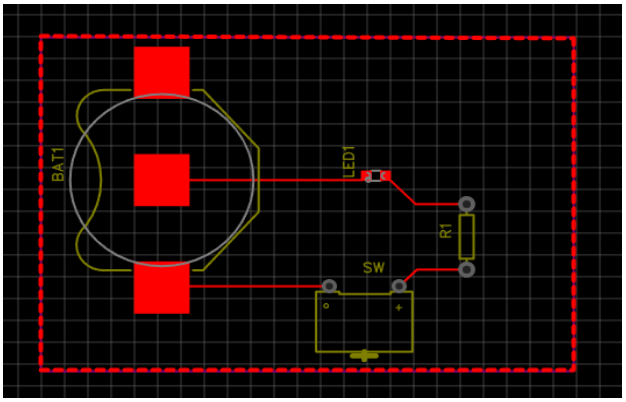
Step 3: Auto route



Click Auto router. (check image) or you can click track in PCB Tool and route of your own and Click Run (You can change the parameter as per requirement). See the image, the red lines connecting components are called Tracks

You can add Drill Holes (if you require), By Click Hole in PCB Tool

Step 4: Copper Area/Pour



Click Copper Area in PCB Tool and draw a border on the PCB Board

Save and click Fabrication Output and Download the Gerber Files and sent to your fabricator

RESULT: PCB Designed of R, LED and Switch Circuit with DC Power Supply of Battery

VIVA QUESTIONS:

1.What is Easy EDA?

2.Basics/Features of Easy EDA

3. Steps to Design Circuits in Easy EDA

4. Getting Started with Easy EDA for PCB Design

5. Design a Schematic Diagram

6. Checking the Connections

7. Simulating the Circuit

8. Conversion of PCB Layout through Easy EDA

EXPERIMENT: 2

AIM: PCB Design of Process using Proteus Software

.

Software : Proteus Software

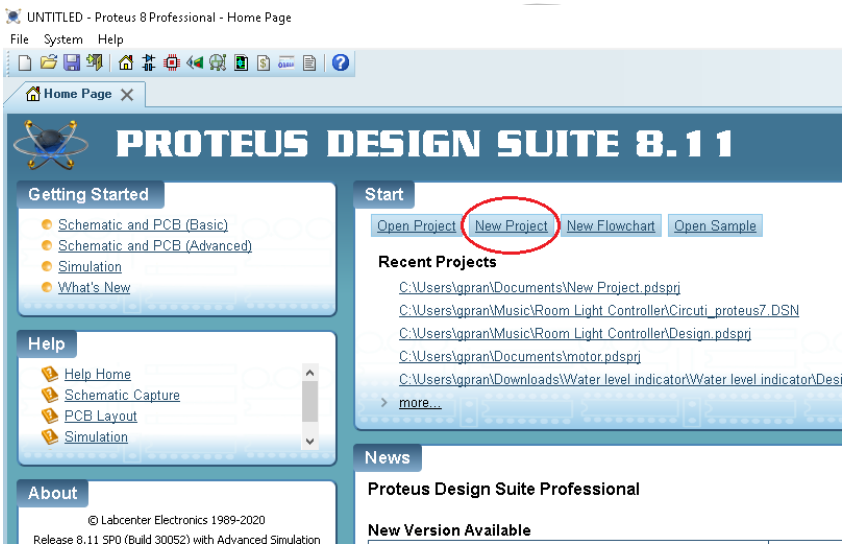
Theory:

Proteus is a simulation and design software tool developed by Lab center Electronics for Electrical and Electronic circuit design**.** It is a software suite containing schematic, simulation as well as PCB designing.

- **ISIS** is the software used to draw schematics and simulate the circuits in real time. The simulation allows human access during run time, thus providing real time simulation.
- **ARES** is used for PCB designing. It has the feature of viewing output in 3D view of the designed PCB along with components.
- The designer can also develop 2Ddrawingsfortheproduct.

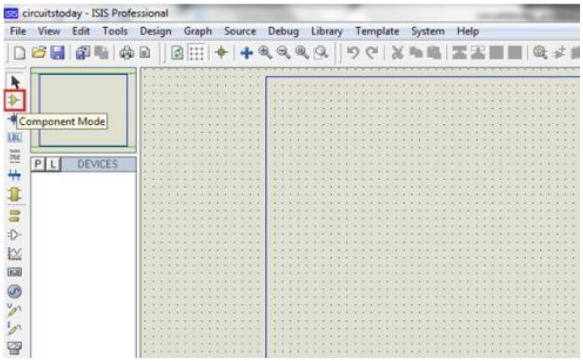
Steps to Design and Simulate

- Download proteus on pc.
- Open Proteus .At the home page ,click on New Project.

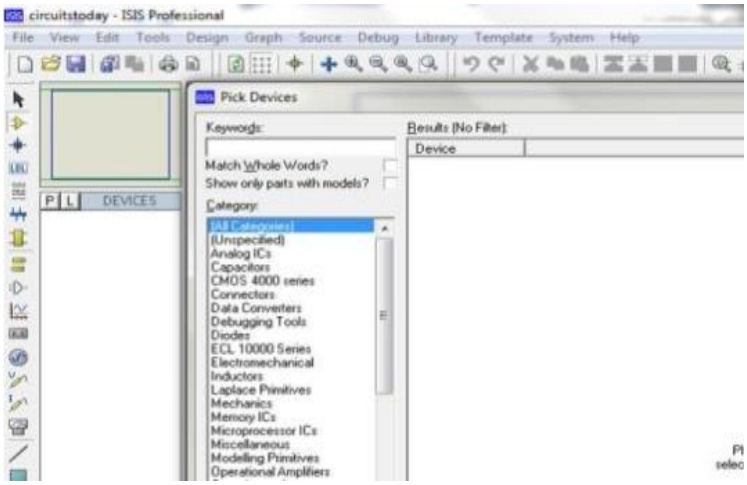


- After opening new project, we get an untitled design sheet will be opened, save it according to your wish, it is better to create a new folder for every layout as it generates other files supporting your design.

- To Select components, Click on the component mode button.

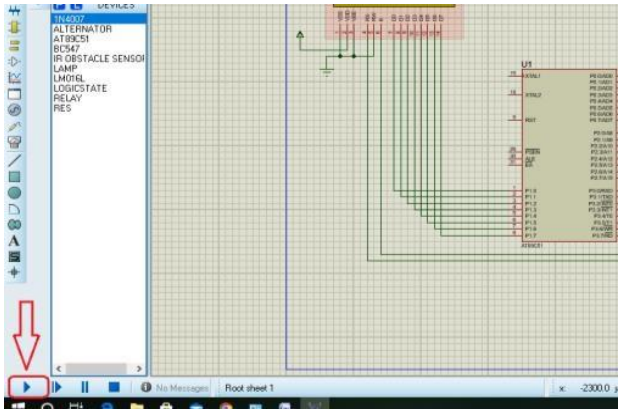


- Click On P(Pick from Libraries).
- Select the components from categories or type the part name in Keywords



text.

- Click ok
- Click on Play Button on the bottom left to start simulation
-



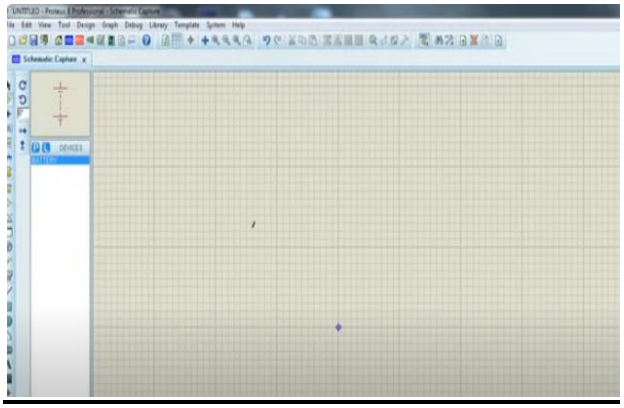
Example

Design Simple Circuit

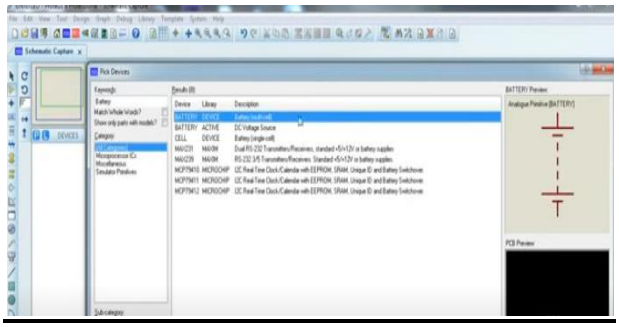
1.Battery
2.LED
3.330 ohm Resistor
54.Switch
5.Wires

- Open Proteus .At the home page ,click on New Project.
- To Select components, Click on the component mode button.
- Click On P(Pick from Libraries).
- Select the components from categories or type the part name in Keywords

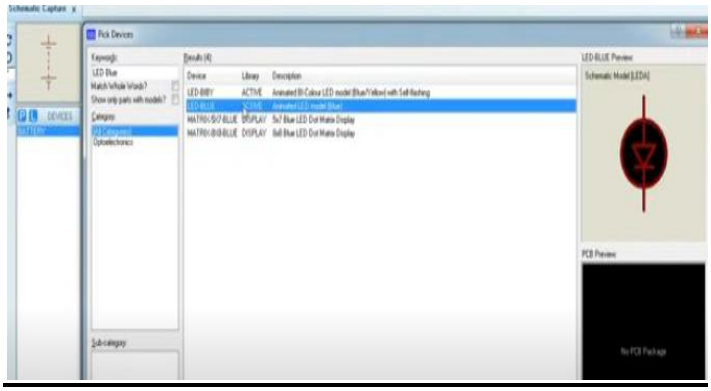
Circuit to be designed window



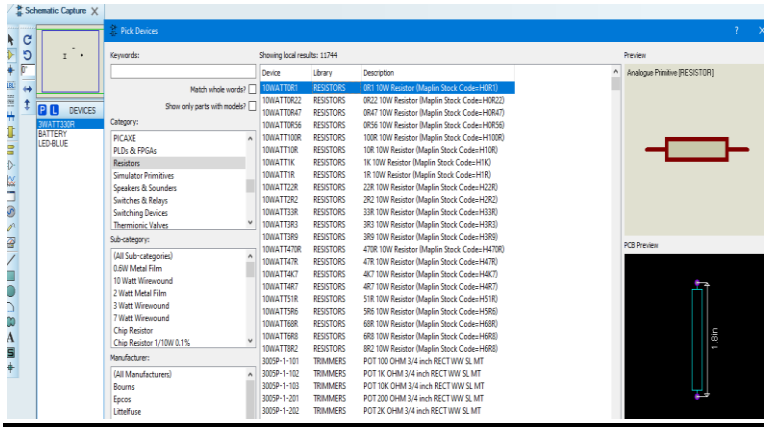
Battery Selection



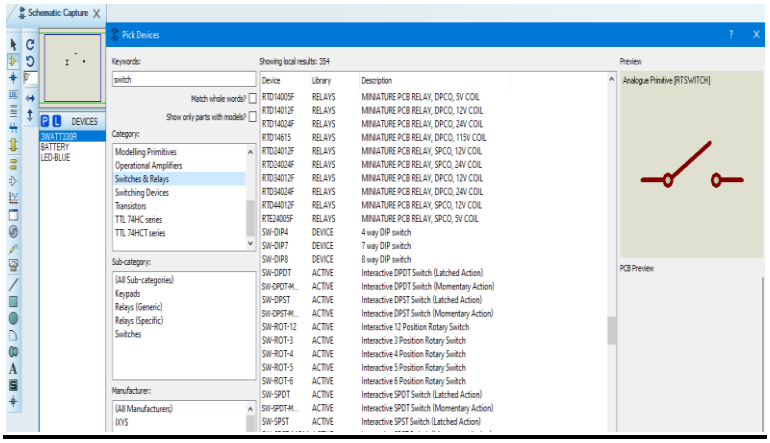
LED Selection



Resistor Selection

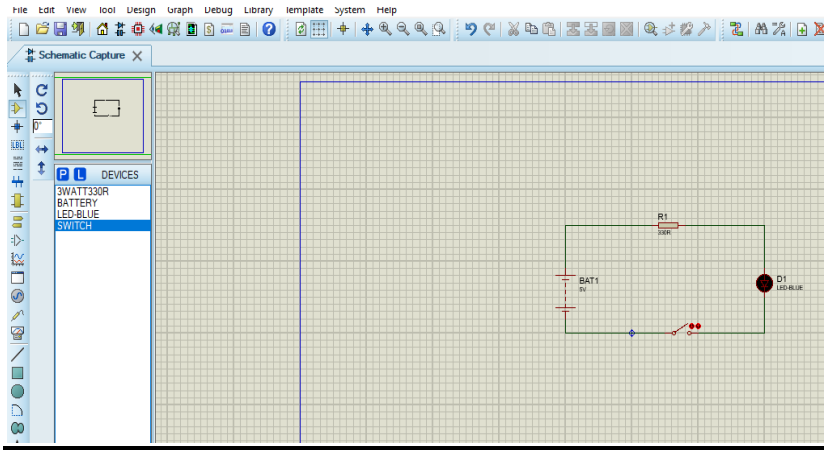


Switch

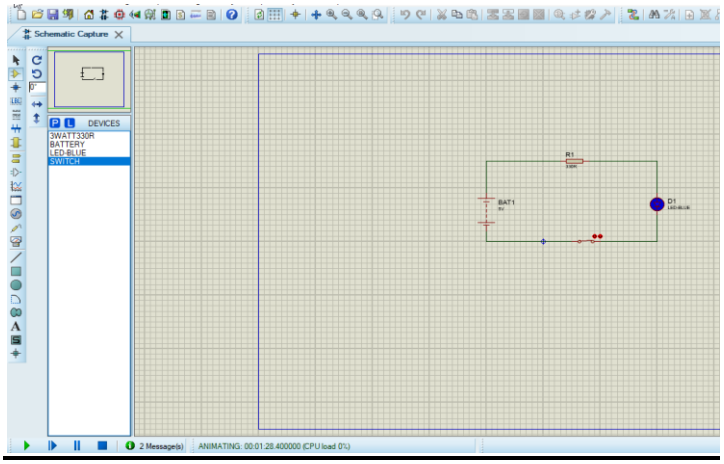


Simulation for

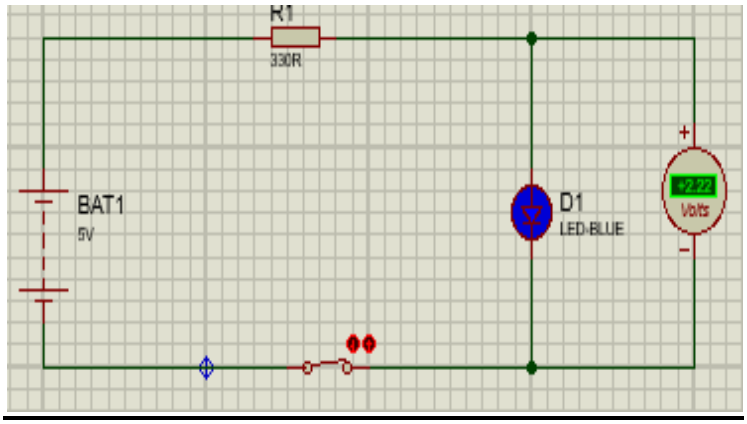
Open Switch



Simulation for Closed Switch



To calculate output dc voltage



Viva Questions

1. What is Arduino?

. How many types of Arduino do we have?

3. What language is a typical Arduino code based on?

4. Arduino shields are also called as _____

5. What language is the Arduino IDE built on?

6. How many analog pins are used in Arduino Mega board?

7. Arduino IDE consists of 2 functions. What are they?

8. Arduino Codes are referred to as _____ in the Arduino IDE.

9. What is the default bootloader of the Arduino UNO?

10. What does p refer to in ATmega328p?

11. What is the use of the Arduino.h header file?

12. What is the use of the Vin pin present on some Arduino Boards?

13. What is the correct execution process of an Arduino code?

14. What is the microcontroller used in Arduino UNO?

15. Which board is the first to use a microcontroller within the build USB?

16. Which Arduino Board contains an onboard joystick?

17. What is the function of the IOREF pin on the Arduino UNO?

18. Which processor supports the Arduino Zero?

19. Which software is used to upload the Arduino Sketches to the board?

20. What is the use for the 2 serial pins on the Arduino Decimal?

21. Which Arduino Boards use the Atmega2560?

22. What is the operating voltage of Atmega328?

23. Which Arduino Boards use the Atmega32U4?

a) Arduino Uno

24. Is the Arduino code an Object-Oriented programming language or a Procedural programming language?

25. What is the difference between an IDE and a compiler?

26. What will be the output of the following Arduino code?

27. Where does the Arduino IDE search if it needs to find out the Name of a type of Arduino Board?

28. How many times does the setup() function run on every startup of the Arduino

29. Which of the following statements is not true when dealing with the Fermata library?

30. What is the output of “pin1” if “pin2” is sent “1011” where 1 is 5V and 0 is 0V?

EXPERIMENT: 3

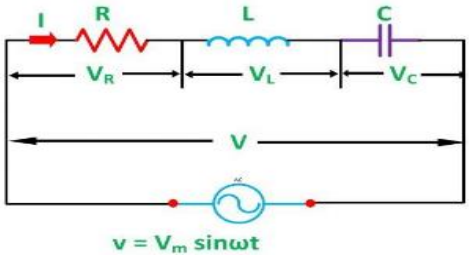
AIM:PCB Design of RLC Circuit using Proteus software

TOOL: Proteus software

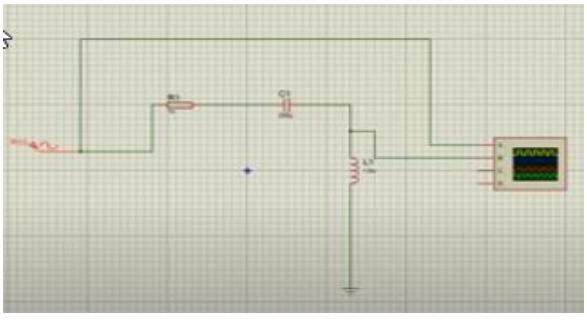
THEORY:

RLC circuit theory is the study of circuits that contain a resistor (R), an inductor (L), and a capacitor (C). RLC circuits are resonant circuits, meaning they oscillate and filter signals. Series RLC circuits. The impedance of a series RLC circuit depends on the angular frequency .The resonance of a series RLC circuit occurs when the inductive and capacitive reactances are equal in magnitude but cancel each other out .The phasor diagram for a series RLC circuit is created by combining the three individual phasor vector ally . Parallel RLC circuits. The R, L, and C components are all connected across the same voltage source . The voltage across each component is the same, while the current flowing through each component can vary. The total admittance of a parallel RLC circuit is calculated by adding the admittance values of individual components . RLC circuit applications: RLC circuits can be used to make variable analog tuners . The sharp minimum in impedance of a series RLC circuit can be used in tuning applications

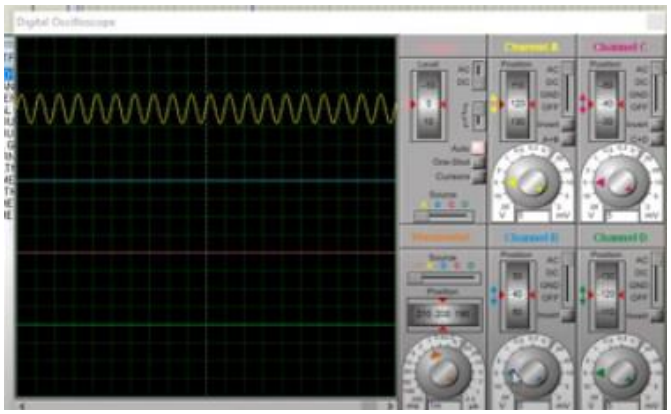
Circuit Diagram



Schematic Diagram



Simulation results



Viva Questions

1. In a series RLC circuit, the phase difference between the voltage across the capacitor and the voltage across the resistor is?
2. In a series RLC circuit, the phase difference between the voltage across the inductor and the voltage across the resistor is?
3. In a series RLC circuit, the phase difference between the voltage across the capacitor and the voltage across the inductor is?

4. In a series RLC circuit, the phase difference between the voltage across the resistor and the current in the circuit is?
5. In a series RLC circuit, the phase difference between the voltage across the capacitor and the current in the circuit is?
6. In a series RLC circuit, the phase difference between the voltage across the inductor and the current in the circuit is?
7. The current in the inductor lags the voltage in a series RLC circuit _____ resonant frequency.
8. The current in the capacitor leads the voltage in a series RLC circuit _____ resonant frequency.
9. The current in the inductor _____ the voltage in a series RLC circuit above the resonant frequency.
10. The current in the capacitor _____ the voltage in a series RLC circuit below the resonant frequency.

EXPERIMENT: 4

AIM: PCB Design of Half Wave Rectifier using Proteus Software

.

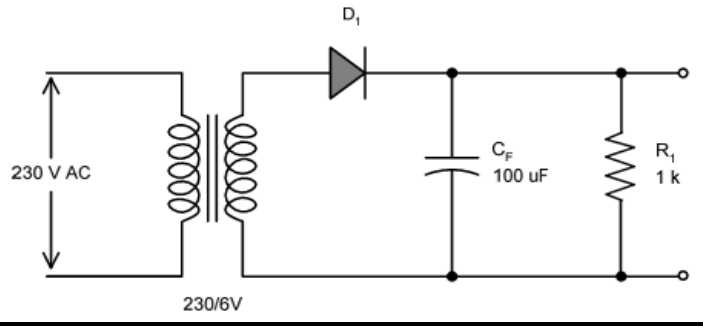
TOOL: Proteus Software

THEORY:

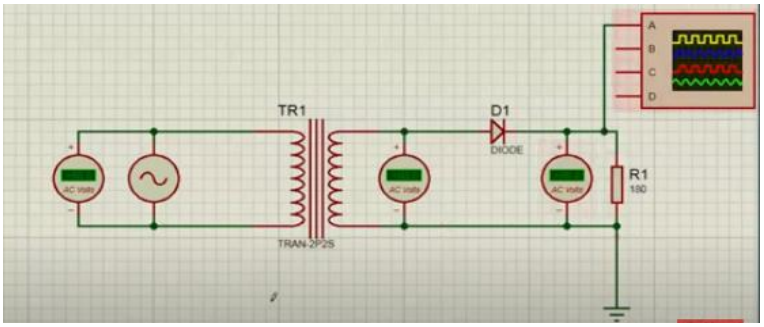
Rectifier changes ac to dc and it is an essential part of power supply. The unique property of a diode, permitting the current to flow in one direction, is utilized in rectifiers. Mains power supply is applied at the primary of the step-down transformer. All the positive half cycles of the stepped down ac supply pass through the diode and all the negative half cycles get eliminated.

Peak value of the output voltage is less than the peak value of the input voltage by 0.6V because of the voltage drop across the diode.For a half wave rectifier, $V_{rms} = V_m/2$ and $V_{dc} = V_m/\pi$: where V_{rms}= rms value of input, V_{dc}= Average value of input and V_m= peak value of output.nent.

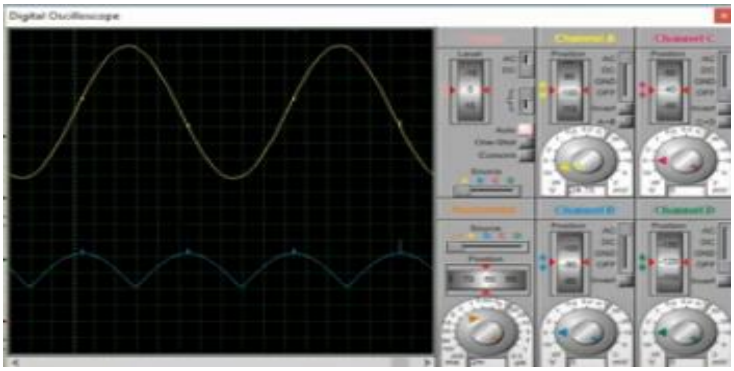
Circuit Diagram



Schematic Diagram



Simulation Results:



Viva Questions

1.What is a half wave rectifier?
2. Explain what is a rectifier?
3. Explain what is PIV of a diode in a rectifier circuit?
4. Explain what is the importance of peak inverse voltage?
5. Explain why diodes are not operated in the breakdown region in rectifiers?
6. Define ripple as referred to in a rectifier circuit.
7. How does a half wave rectifier work?
8. Explain what is transformer utilization factor?
9. The output of a 60Hz full-wave bridge rectifier has a 60 Hz ripple. It this circuit working properly?
10. Explain what is meant by filter?
11. Explain why series inductor and L-section filters cannot be used with half-wave rectifiers?
12. Explain why capacitor input filter is preferred to choke input filter?
13. Explain why π-filters are not suitable for varying loads?
14. Explain why R-C filters are suitable only for light loads?
15. Explain why is bleeder resistance employed in a filter circuit?
16. Explain what is the purpose of bleeder resistance in a rectifier circuit using L-C filter?
17. Explain what is meant by voltage regulation of a dc power supply?
18. Explain why is it necessary to include voltage stabilizer in a power supply?
19. Explain what are the commonly used filters?
20. Explain what is the difference between active and passive filters?
21. Explain why are active filters preferred
22. Explain what is a low-pass filter?
23. Explain what is a high-pass filter?
24.What is the output of a half wave rectifier
25.What is the difference between the input and output voltage of a half wave rectifier
26. Why is a half wave rectifier less efficient than a full wave rectifier
27.What are some applications of a half wave rectifier?
28.What is the purpose of a filter in a half wave rectifier?
29. Explain what is a dc power supply?

EXPERIMENT: 5

AIM: PCB Design a full wave centre tapped rectifier using Proteus.

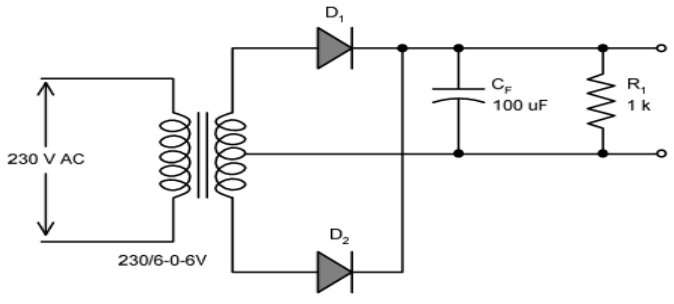
TOOL: Proteus Software

THEORY: A rectifier circuit whose transformer secondary is tapped to get the desired output voltage, using two diodes alternatively, to rectify the complete cycle is called as a Center-tapped Full wave rectifier circuit. The transformer is center tapped here unlike the other cases. The tapping is done by drawing a lead at the mid-point on the secondary winding. This winding is split into two equal halves by doing so. The voltage at the tapped mid-point is zero. This forms a neutral point. The center tapping provides two separate output voltages which are equal in magnitude but opposite in polarity to each other.

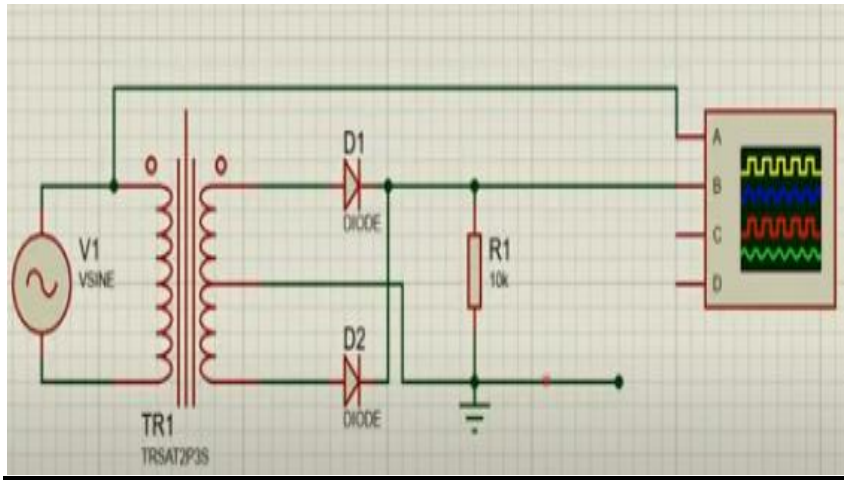
A number of tapings can be drawn out to obtain different levels of voltages. The center-tapped transformer with two rectifier diodes is used in the construction of a **Center-tapped full wave rectifier**. The working of a center-tapped full wave rectifier can be understood by the above figure. When the positive half cycle of the input voltage is applied, the point M at the transformer secondary becomes positive with respect to the point N. This makes the diode D1D1forward biased.

Hence current i1i1 flows through the load resistor from A to B. We now have the positive half cycles in the output. When the negative half cycle of the input voltage is applied, the point M at the transformer secondary becomes negative with respect to the point N.

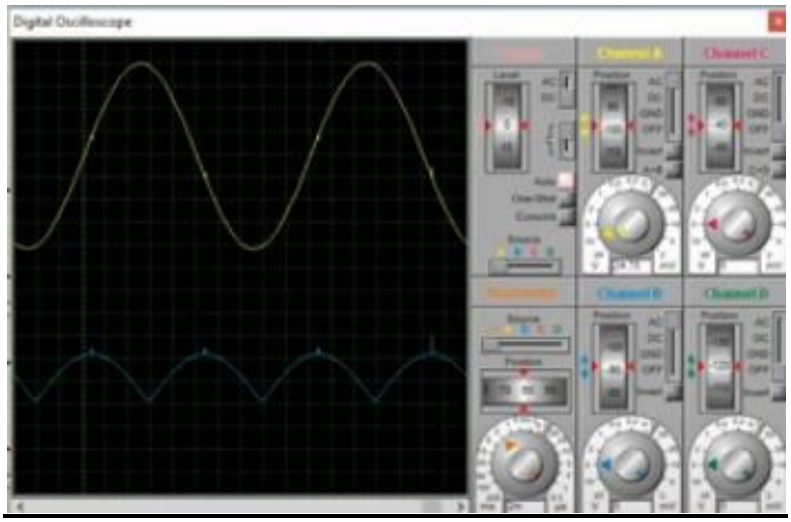
This makes the diode D2D2 forward biased. Hence current i2i2 flows through the load resistor from A to B. We now have the positive half cycles in the output, even during the negative half cycles of the input. The input and output waveforms of the center-tapped full wave rectifier are as follows.



Schematic diagram



Simulation Results



Viva Questions:

- **What is a full wave center-tapped rectifier?**
- Explain the working principle of a full wave center-tapped rectifier with a diagram.
- What are the key components of a full wave center-tapped rectifier circuit?
- How does a center-tapped transformer work in a full wave rectifier?
  Circuit Analysis:
- Derive the expression for the average DC output voltage of a full wave center-tapped rectifier.

- Explain how the output waveform of a full wave center-tapped rectifier looks like and why it has less ripple compared to a half-wave rectifier.
- Calculate the ripple factor for a full wave center-tapped rectifier.
- What is the peak inverse voltage (PIV) rating required for the diodes in a full wave center-tapped rectifier?

Comparison and Applications:

- Compare the advantages and disadvantages of a full wave center-tapped rectifier with a full wave bridge rectifier.
- What factors influence the choice of a full wave center-tapped rectifier over other rectifier circuits?
- Where are full wave center-tapped rectifiers commonly used in practical applications?

Advanced Topics:

- Explain the role of a filter capacitor in a full wave center-tapped rectifier circuit and how it reduces ripple.
- What is the transformer utilization factor (TUF) for a full wave center-tapped rectifier and how does it compare to other rectifier circuits?
- Discuss the effect of load resistance on the output voltage and ripple factor of a full wave center-tapped rectifier.

Potential Viva Questions:

- Explain how the diodes conduct during each half cycle in a full wave center-tapped rectifier.
- What is the relationship between the input AC voltage and the output DC voltage in a full wave center-tapped rectifier?
- How can you calculate the average DC current delivered to the load in a full wave center-tapped rectifier circuit?
- What happens if the center tap of the transformer is not properly connected in a full wave center-tapped rectifier?

EXPERIMENT: 6

AIM: Design a clipper circuit using Proteus.

.

TOOL: Proteus Software

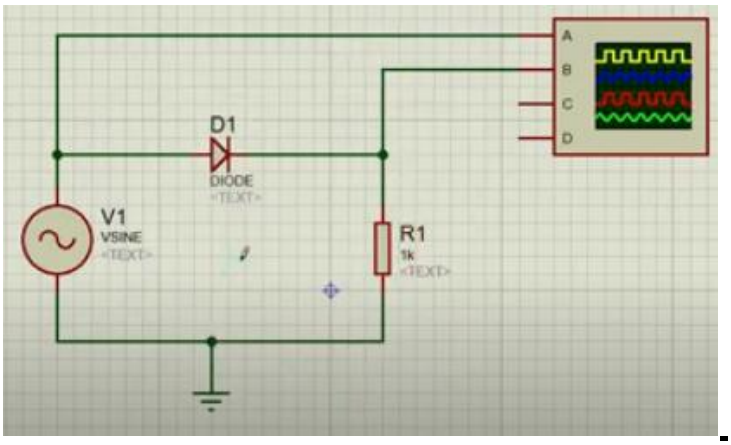
THEORY:

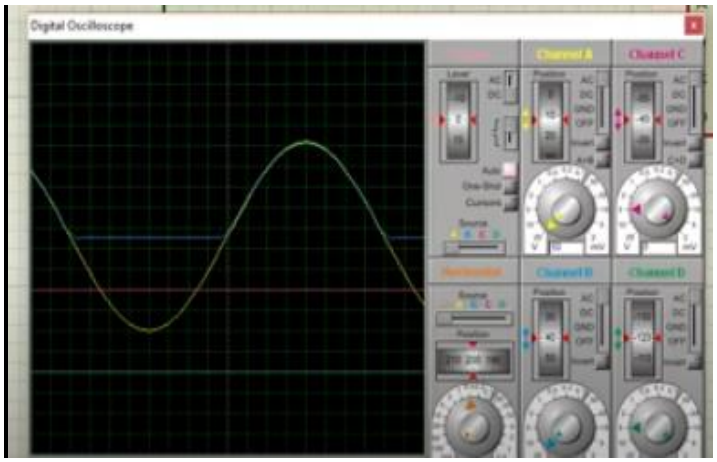
Clippers Clipper circuits are wave shaping circuits that has the ability to "clip" off a portion of the input signal. without distorting the remaining part of the alternating waveform. The half wave rectifier is an example of the simplest form of diode clipper. Depending on the orientation of the diode, the positive or negative region of the input signal is "clipped" off.

There are two general categories of clippers: series and parallel. The series configuration is defined as one where the diode is in series with the load, while the parallel variety has the diode in branch parallel to the load. The limiter sets a "limit" in amplitude of the signal. Zener diode is used in limiter circuits

Positive Clipper Schematic Diagram

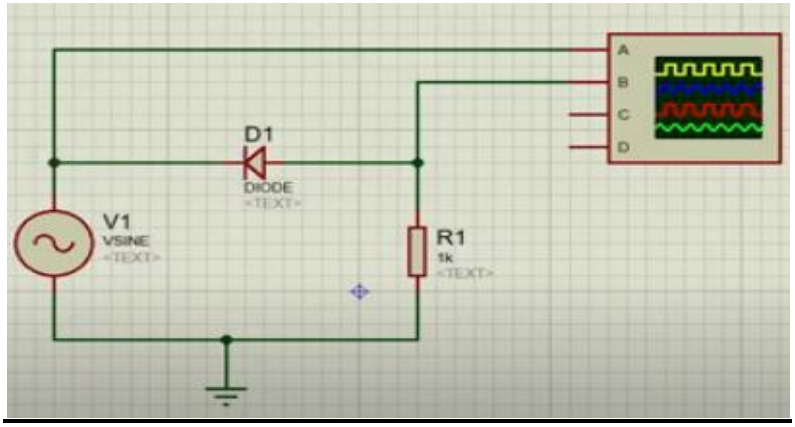
.

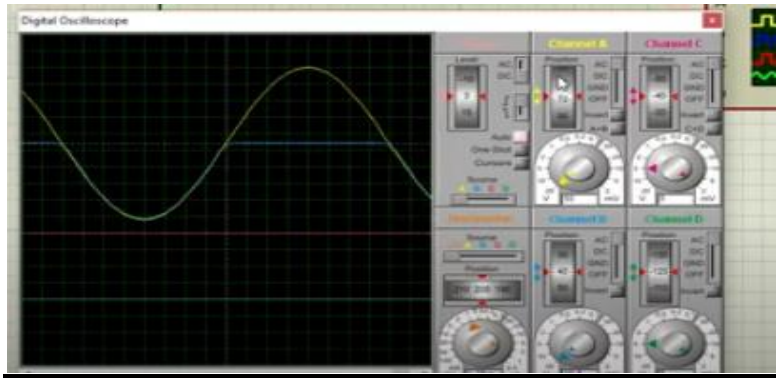




Simulation Results

Negative Clipper Schematic Diagram





Simulation Results

EXPERIMENT: 7

AIM: Design of Clamper Circuits

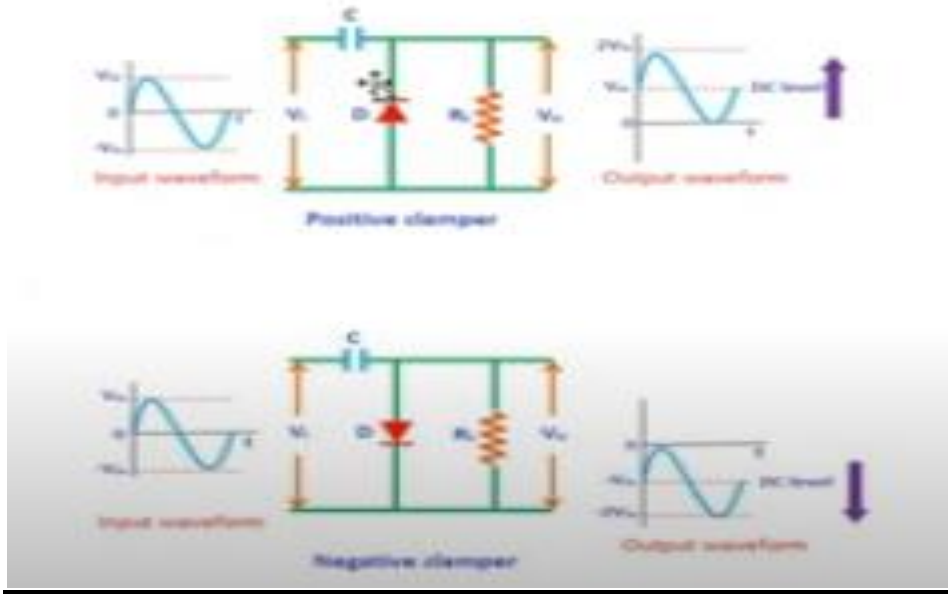
TOOL: using Proteus 8 Software

THEORY:

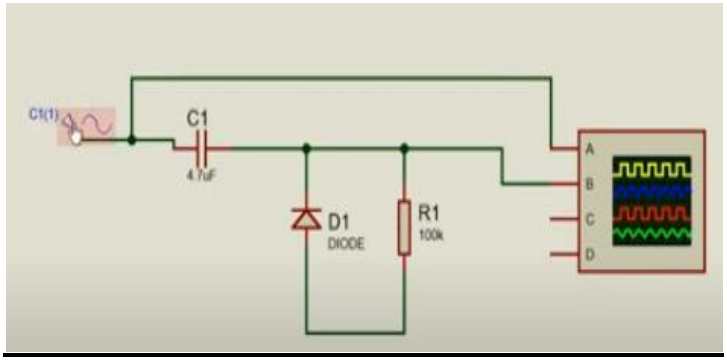
Clamper circuit theory explains how a clamper circuit adds or subtracts a DC voltage to an AC signal. This shifts the waveform vertically without distorting its shape. A clamper circuit uses a diode, capacitor, and resistor. The diode allows current to flow during one half-cycle, charging the capacitor. During the next half-cycle, the stored charge shifts the output waveform.

A positive clamper circuit outputs a positive waveform, while a negative clamper circuit outputs a negative waveform. Clamper circuits are used in TV receivers, oscilloscopes, and signal processing. They are an essential component of electrical engineering, especially in waveform analysis and manipulation. Clamping circuit theorem The clamping circuit theorem states that the ratio of the area under the output voltage curve in the forward direction to that in the reverse direction is equal to the ratio R f/R.

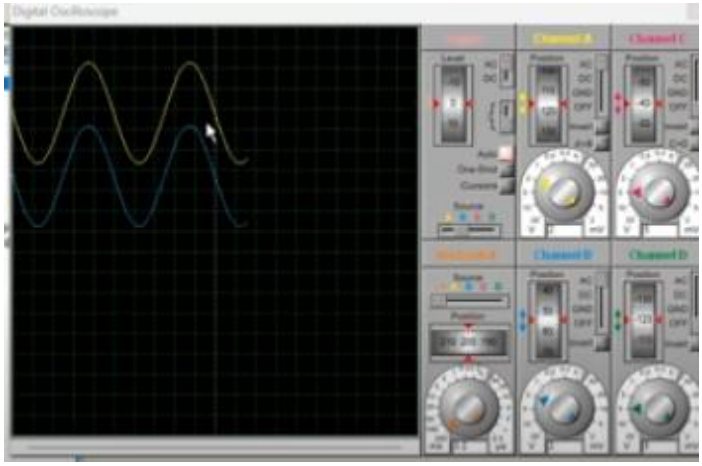
Clamper Circuits



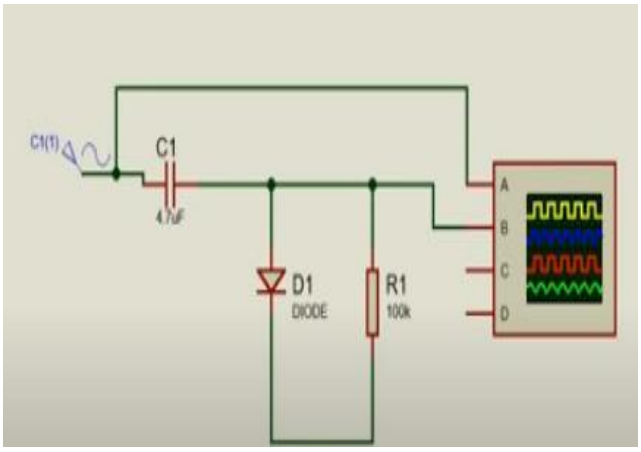
Schematic diagram of Positive Clamper



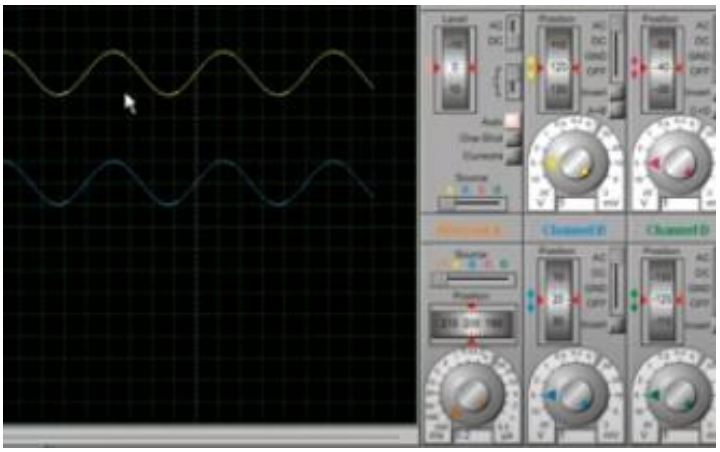
Simulation Results



Schematic diagram of Negative Clamper



Simulation Results



Viva Questions

1.What is clamper?

2. What is clamping circuit?

3.What is the necessity for establish the clamping?

4. Explain various types of clamper circuit?

5. What is positive and negative clamp circuit?

6. Why we do clamping for input protection?

7. What is the principles of operation of clamper?

8. Give some example of clamper?

9. What values of R and C should be taken for good clamping?

10. Define voltage multiplier

11. List different types of voltage multipliers

12. Define voltage doubler

13. Define voltage tripler

14. Define voltage quadrupler

EXPERIMENT: 8

AIM: Design5V dc Power Supply

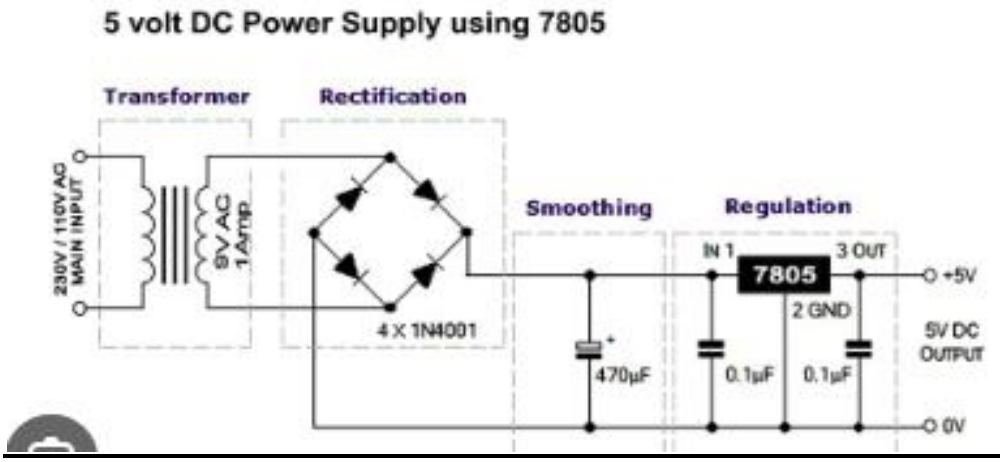
TOOL: Proteus.

THEORY:

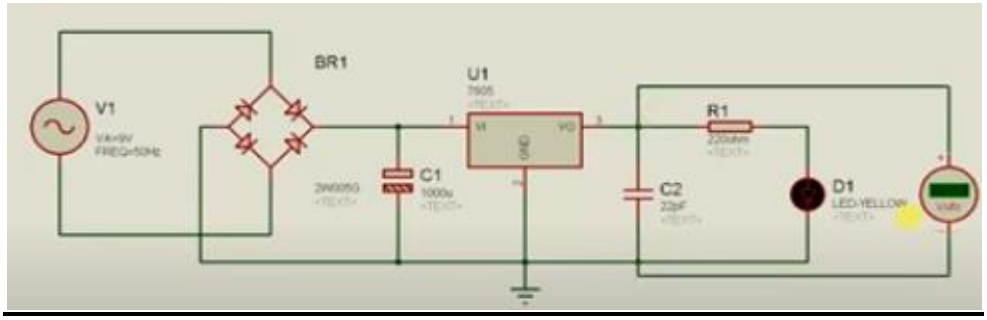
How does the 5V Power Supply work?

First, we are using a step-down transformer [Secondary rating 9Volt & 1 Amp] to step down 230V/ 110V AC supply to 9-Volt AC. Then we rectify the 9V AC to 9 V DC using a diode bridge rectifier [Full wave rectifier]. After the rectifier, we have used Capacitors to filter the ripple from the circuit and fed it to the input of the 7805 voltage regulator. 7805 regulates the 9 volt DC to 5 Volt DC and at the output of 7805 ic, we get constant 5 Volt DC output.

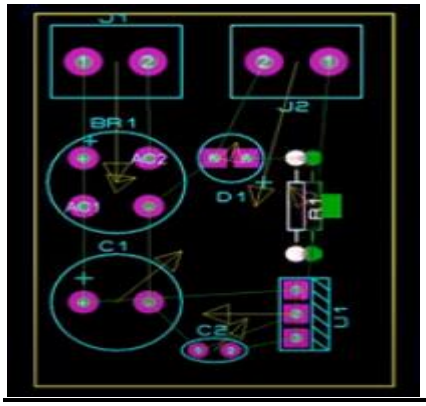
Circuit Digram

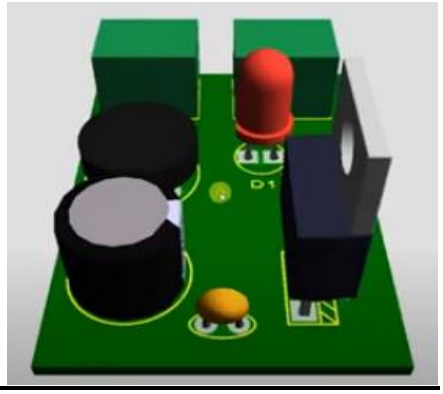


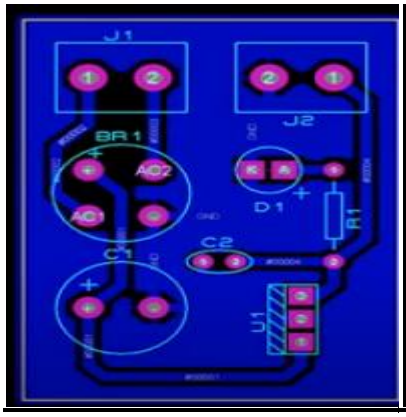
Schematic Diagram

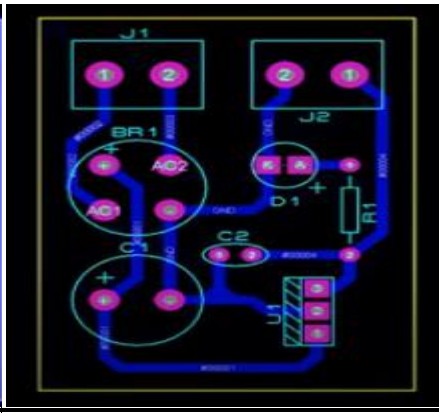


Layout Designs and 3D View

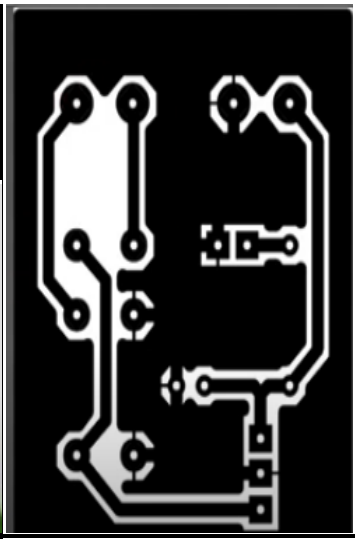








PCB Design



Viva Questions

- What is a 5V power supply and what does "5V" represent?
- Explain the difference between AC and DC power, and why a 5V power supply provides DC.
- What are the key components of a basic 5V power supply circuit?

Components and Functions:

- What is the role of a transformer in a power supply?
- How does a rectifier convert AC to DC?
- Explain the function of a filter capacitor in a power supply circuit.
- What is a voltage regulator and why is it important in a 5V power supply?

Specific to 5V Regulation:

- Which common voltage regulator IC is often used to create a 5V power supply? (e.g., 7805)
- Describe the internal workings of a voltage regulator like the 7805
- What is the significance of "dropout voltage" in a voltage regulator?

Applications and Considerations:

- Why is 5V a widely used voltage in electronics?
- What kind of devices typically operate on a 5V power supply?
- What are the potential issues with a 5V power supply, such as current limitations or ripple voltage?

Advanced Topics:

- Explain the difference between a linear regulator and a switching regulator
- What are the advantages and disadvantages of using a switching regulator for a 5V power supply?
- How would you design a 5V power supply with a specific current rating for a particular application?

HARDWARE SPECIFICATIONS

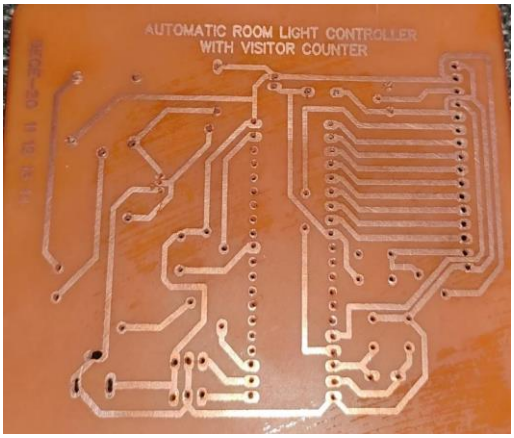
PCB MAKING

A printed circuit board (PCB) mechanically supports and electrically connects electronic components using conductive tracks, pads, and other features etched from copper sheets laminated onto a non-conductive substrate. A printed circuit board has pre-designed copper tracks on a conducting sheet. The pre-defined tracks reduce the wiring, thereby reducing the faults arising due to lose connections. One simply needs to place the components on the PCB and solder them.

STEPSFOR MAKINGTHE PCB

- Print the layout of PCB on a glossy paper or photo paper using laser printer.
- Now cut the copper board according to the size of the layout.
- Clean the board properly so that layout is printed exactly.
- Transfer the PCB print on to the copper board using iron.
- Put the glossy paper on copper board exactly. Now iron the glossy paper all along using the tip while applying a little pressure for about 5 to 15 mins.
- The heat from the iron transfers the ink printed on the glossy paper to the copper plate.
- After ironing, place the printed plate lukewarm water for about 10 minutes. The paper will dissolve, then you can remove the paper gently. Remove the paper by peeling it from a low angle.
- Now it's time for etching.
- Take a plastic box and fill it up with some water.
- Dissolve2-3 tea spoons off erricchloride powder in the water.

- Dip the PCB into the etching solution (Ferric chloride solution, FeCl3) for approximately 30 mins.
- The FeCl3 reacts with the unmasked copper and removes the unwanted copper from the PCB.
- This process is called Etching. Use pliers to take out the PCB and check if the entire unmasked area has been etched or not. In case it is not etched, leave it in the solution for some more time.
- Clean it by using thinner to remove print on the copper board.
- Drill holes using a component to the board.



Soldering:

Like this PCB driller B and solder all your cool

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.

Solder is melted by using heat from an iron connected to a temperature controller.

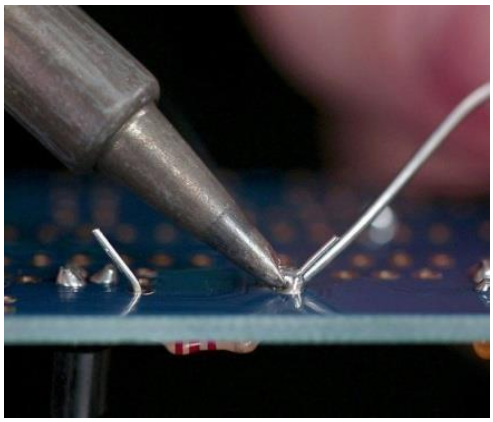
It is heated up to temperatures beyond its melting point at around 600 degrees Fahrenheit which then causes it to melt, which then cools creating the soldered joint.

As well as creating strong electrical joints solder can also be removed using a desoldering tool.

Solder is a metal alloy used to create strong permanent bonds; such as copper joining in circuit boards and copper pipe joints.

It can also be supplied in two different types and diameters, lead and lead free and also can be between .032” and .062”.

Inside the solder core is the flux, a material used to strengthen and improve its mechanical properties



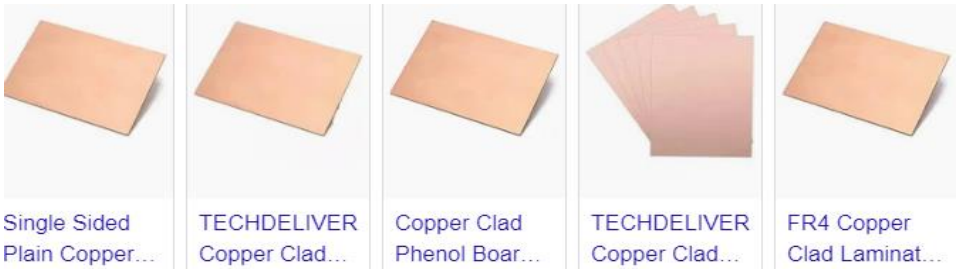
STEPSFORSOLDERINGTHEPCB

- Place all the components on PCB.
- Clean the soldering tip properly. Apply the flux on PCB so that it is properly soldered.
- Heat the soldering iron and soldered the components to PCB using the lead.

Now load the hex file of program into microcontroller and put it in the circuit.

Materials required for the fabrication of simple PCB's

. 9. **PCB copper clad boards**



Copper clad boards are used in many industries, including aerospace, automotive, and electronics. They are used to make printed circuit boards (PCBs).

Applications

Aerospace: Heavy copper PCBs are used in harsh environments and extreme temperatures.

Automotive: Copper clad laminates are used to make electronic components like infotainment systems, engine control units, and advanced driver assistance systems.

Electronics: Copper clad boards help prevent overheating in electronic devices. They are also used in DIY PCB fabrication.

Benefits

Durability: Copper clad boards are durable and resistant to bending and fracturing.

Chemical resistance: Copper clad boards can withstand chemicals used in electronic manufacturing processes.

Thermal conductivity: Copper clad boards help dissipate heat from electronic components.

Electrical conductivity: Copper clad boards are good conductors of electricity.

Mechanical strength: Copper clad boards have high mechanical strength.

Manufacturing

Copper clad boards are made of fiberglass-reinforced epoxy laminate with a thin layer of copper foil bonded to one or both sides.

10.**Ferrous chloride' for PCB etching**

Why ferric chloride is used in PCB etching? Ferric chloride is a corrosive, acidic chemical compound that will eat away all copper on the board that is not protected by the marker's ink.

Pour a modest amount of ferric chloride into a plastic container with a lid; just enough to cover the board completely.

11.Hand drills with needles

A hand needle is a long, thin tool with a pointed tip and an eye for thread that is used to sew by hand.

Parts of a hand needle

- **Point**: The sharp tip that pierces the fabric
- Eye: The hole through which the thread passes
- Shaft: The body of the needle that determines its length and thickness

  Types of hand needles
- Sharp needles: The most common type of hand sewing needle, used for general sewing on most fabrics
- Beading needles: Very fine needles with a narrow eye for small beads
- Quilting needles: Used for thick fabrics

  Needle materials
- Early needles were made of bone or wood

- Modern needles are made of high carbon steel wire
- Some needles are plated with nickel, 18K gold, platinum, or titanium to resist corrosion Choosing a needle
- The right needle for a project depends on the fabric, thread, and the desired stitch.

12.**Glossy photo paper (130gsm) hands on schematic implantation on board**

Glossy Paper refers to any coated papers designed to present an ultra-smooth to shiny appearance. Applications include brochures, advertising, flyers, one sheets, photographic printing and other presentation documents.

LRS A4 Glossy Photo Paper (130 Gsm- Pack Of 200)

13.Hand grouses, Soldering iron, Soldering paste flux, Soldering lead

Soldering Printed Circuit Boards

The soldering of components onto a printed circuit board (PCB) is also commonly done with soldering irons.

- Begin with the tallest components, soldering interconnected wiring last.
- Place through-hole components in correct holes, making sure they sit flush on the board.
- Bend component marginally to keep it stationary.
- Heat soldering iron and, once it reaches appropriate temperature, touch the pad in order to heat both the component's metal and the pad. Correct temperatures are vital, as temperatures that are too high may damage the board and components, while temperatures that are too low won't create a joint sufficient for making an electrical connection.
- Apply solder so that it flows around the component in liquid form, using enough to make solid connections without gaps though not so much that there's excess solder.
- Draw soldering iron directly upwards from the part being soldered, which should result in a solder joint shaped like a cone.
- Check the joint, ascertaining that it's shiny in appearance, not too much solder was used and there aren't any gaps.
- Should the solder joint be satisfactory, cut any excess from the component above the joint.

14.Iron boxes as a heat sink

While an iron box could technically be used as a heat sink in a pinch, it's not an ideal choice due to its relatively poor thermal conductivity compared to materials like aluminum or copper, which are typically used for heat sinks; meaning it would not effectively dissipate heat from a heat source.

Why iron boxes aren't great as heat sinks:

- **Low thermal conductivity:**

  Iron has a lower ability to transfer heat compared to other metals commonly used in heat sinks, resulting in less efficient cooling.

- **Design limitations:**

  A typical iron box lacks the fins or extended surface area that most heat sinks have, which are crucial for maximizing heat dissipation.

- **Potential for overheating:**

  If the heat source is generating a significant amount of heat, an iron box might not be able to handle it effectively, leading to overheating issues.

  Better options for heat sinks:

- **Aluminum heat sinks:**

  Widely available, lightweight, and have good thermal conductivity, making them suitable for most applications.

- **Copper heat sinks:**

  Offer even better thermal conductivity than aluminum but are typically more expensive.

15. **Development of different mini projects on PCB.**