

# Institute of Petrochemical Engineering, Lonere-402103

## Department of Electrical Engineering

### A Report on “PCB Designing and Fabrication Workshop”

Academic Year 2019-20

The Department of Electrical Engineering organized the “**PCB Designing and Fabrication Workshop**” on, 30<sup>th</sup> and 31<sup>st</sup> August, 2019 for the students of Electrical Engineering in collaboration with Proto Electronics Pvt. Ltd. Badlapur, Mubai.

Dr. N. S. Lingayat, Mr. U. A. Kharat, Mrs. D. S. Shet, Ms. M. G. Kalsekar and Ms. P.P. Chopade were present for the inaugural function. Dr. M. A. Dabhade welcomed the resource persons. Dr. N. S. Lingayat conveyed the objectives of the Work-Shop.

**Resource Persons:** Mr. Kaustubh Koshe, Mr.Vedant Manjrekar, Mr.Yashwant Mhaiskar, Ms. Saloni Sonawane Proto Electronics Pvt. Ltd. Mumbai.

#### **Contact Details:**

**Ms. Saloni Sonawane**

**Mobile:** +917276828856



**Ms. Saloni Sonawane being welcomed by Dr. M. A. Dabhade**



The objective of the workshop was to make every student an expert in designing their own **PCB board** which would be very useful for developing their own projects. The workshop progressed in the following sequence:

### 1. **Basic PCB Concepts**

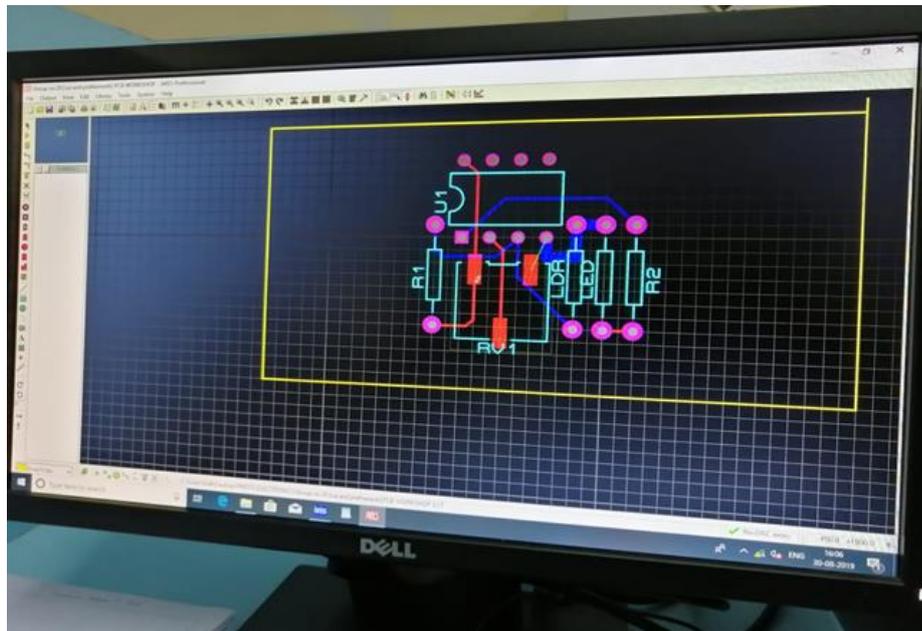
First of all Mr. Kaustubh Koshe taught the concepts which will be very helpful for designing the PCB practically, using some power point presentations. In this theoretical explanation part he explained about the **Express PCB** and **Proteus** software and the use of software for further practical implementation in designing the PCB. He also gave a briefing about active and passive electronic components with the packaging details which the students will be using in a PCB.



**Mr. Kaustubh Koshe while interacting with the students...**

## 2. Designing the PCB

Designing the PCB is the basic step and it is one of the important steps for designing a PCB. Editing and Routing gives the circuit layout from one component to the other components.



Designing PCB circuit on Express PCB Software

## 3. Printing the PCB Design

After all the checks are complete, the PCB design can be printed. Unlike other plans, like architectural drawings, PCB plans don't print out on a regular 8.5 x 11 sheet of paper. Instead, a special kind of printer, known as a plotter printer, is used. A plotter printer makes a "film" of the PCB. The final product of this "film" looks much like the transparencies that used to be used in schools – it's essentially a photo negative of the board itself.

The inside layers of the PCB are represented in two ink colors:

**Black Ink:** Used for the copper traces and circuits of the PCB

**Clear Ink:** Denotes the non-conductive areas of the PCB, like the fiberglass base

On the outer layers of the PCB design, this trend is reversed – clear ink refers to the line of copper pathways, but black ink also refers to areas where copper will be removed.

After the film is printed, they're lined up and a hole, known as a registration hole, is punched through them. The registration hole is used as guide to align the films later on in the process.

## 4. Toner Transfer Method

This is the first step in the actual manufacturing process where the PCB making starts. After the PCB design is printed onto a piece of laminate, copper is then pre-bonded to that same piece of laminate, which serves as the structure for the PCB. This will be passed through a temperature of about 160 to 180 degrees so that the tracks of the circuit will be remained on the wafer.

## 5. Getting Rid of the Unneeded Copper

The next stage in the process is that of removing the unwanted copper. Much like the alkaline solution from earlier, another powerful chemical is used to eat away at the copper that is not covered by photo resist. Once the unprotected copper is removed, the hardened photo resist from earlier needs to be removed, as well. Another solvent is used, leaving only the copper necessary for the PCB.

Note that when it comes to removing the unwanted copper from your PCB, heavier boards may require more copper solvent or more exposure to the solvent



**Students participating in Etching Process**

## 6. Drilling Technique

The board will be drilled with holes where the components have to be placed; the holes will be drilled in the board depending on the terminals available for the components in the design. The hole should be in the size so that the terminal has to be freely placed in the hole.



**Students participating in Drilling Process**

#### **7. Mounting Components**

This process done with a pair of tweezers, in which assemblers had to pick and place components by hand.

#### **8. Soldering Technique**

The components that are placed in the board should be soldered to the track so that the circuit is connected as per the design. After this step the engraved PCB will be ready to use. Once the solder paste and surface mount components are all in place, they need to remain there. This means the solder paste needs to solidify, adhering components to the board. PCB assembly accomplishes this through a process called "reflow".



**Students participating in soldering Process**

## 9. Electrical Reliability Testing

After the PCB has been coated and cured (if necessary), a technician performs a battery of electrical tests on the different areas of the PCB to ensure functionality. The main tests that are performed are the circuit continuity and isolation tests. The circuit continuity test checks for any disconnections in the PCB, known as “opens.” The circuit isolation test, on the other hand, checks the isolation values of the PCB’s various parts in order to check if there are any shorts. While the electrical tests mainly exist to ensure functionality, they also work as test of how well the initial PCB design stood up to the manufacturing process.



## 10. Functional Test

Functional test (FCT) is used as a final manufacturing step. It provides a pass/fail determination on finished PCBs before they are shipped. An FCT’s purpose in manufacturing is to validate that product hardware is free of defects that could, otherwise, adversely affect the product’s correct functioning in a system application.

In short, FCT verifies a PCB’s functionality and its behavior. It is important to emphasize that the requirements of a functional test, its development, and procedures vary widely from PCB to PCB and system to system.

### Student Response

Almost 90 students from second year and third year had taken part in the workshop. All the students responded that they have learned and had hands on experience in designing a PCB. They were very excited to participate in the workshop and requested for more workshops on similar way so that they can simultaneously gain the practical knowledge.

The program came to the end with the vote of thanks proposed by Dr N. S. Lingayat.



**Certificate Distribution**