The Bombay Salesian Society's

DON BOSCO INSTITUTE OF TECHNOLOGY

Premier Automobiles Road, Kurla West, Mumbai – 400070

IEEE-DBIT

IEEE-DBIT STUDENT CHAPTER

"REFLOW OVEN"

Title: Mastering PCB Fabrication and REFLOW OVEN Techniques.

Date: 2nd March, 2024

Time : 9am to 5:00 pm

Venue: EXTC Lab 2, RF LAB

Target Audience: EXTC Students

No. of Participants present: 12 IEEE members including and 10 non IEEE member

No. of girl participants:7

No. of boy participants:16

Resource Person: Prabhakar Ghawali, Suyog Vyawahare

Organization of Resource Person: Application Engineer- Quectel,

Faculty Coordinator: Dr. Ashwini Kotrashetti, Prof. Freda Carvalho

Objectives:

Develop comprehensive proficiency in PCB fabrication processes and reflow oven techniques to ensure the production of high-quality printed circuit boards with reliable solder joints. This mastery will encompass understanding PCB design, fabrication, assembly, and testing, as well as mastering the operation and optimization of reflow ovens to achieve precise temperature profiles for solder reflow. By mastering these techniques, one can ensure the quality, reliability, and efficiency of electronic device manufacturing processes.

Outcomes:

- Comprehensive understanding of the PCB manufacturing process, fabrication, assembly, and testing
- Proficiency in operating and optimizing reflow ovens to achieve precise temperature profiles for solder reflow
- Ability to ensure consistent solder joint quality and minimize defects by controlling the reflow process
- Knowledge of solder paste composition and selection to enable robust solder joints
- Capacity to efficiently assemble complex electronic devices with miniaturized components and surface mount devices
- Improved reliability and functionality of the final electronic products

Detailed Report:

The Session commenced from 10:30 am with a warm welcome by Mayuri Kadam to our esteemed guest lecturers Mr. Prabhakar Ghawali, Mr. Suyog Vyawahare.

Prabhakar sir introduced the students with the tools that are needed for PCB manufacturing. Firstly the meshes needed for printing the circuit board onto the copper plate. Two meshes with dimensions of 120 and 77 T are used for PCB manufacturing.

The meshes are developed in such a way that in such a way that for 120t mesh every centimetre contains 120 holes due to which the finer and smaller PCB can be manufactured with the atmost precision.

Here the 120t mesh is used for track printing and 77t is used for green masking. After explaining these things Prabhakar sir instructed students that before every use the mesh needs to cleaned with soap and dried so that pre screen coat can be applied.

Prabhakar sir explained all the steps necessary for manufacturing our own PCB:

- **Designing the PCB:** The first step is to design your PCB using CAD software or a free online tool. Make sure your design follows basic DFM (Design for Manufacturability) guidelines to ensure it can be fabricated manually. Export your design as a Gerber file.
- **Printing the Design:** Instead of using a plotter printer, we'll use a laser printer to print the PCB design on a transparency film. Print the design in mirror image. This film will act as a photomask for exposing the PCB.
- **Preparing the Copper Clad Laminate**: Cut a piece of single-sided copper clad laminate to the desired size using scissors or a utility knife. Clean the copper surface thoroughly with steel wool or sandpaper to remove any oxidation.
- **Applying Photoresist:** In a darkroom or under yellow light, apply a thin, even layer of positive photoresist onto the copper surface using a roller or squeegee. Allow it to dry completely.
- **Exposing the Photoresist:** Place the transparency film photomask over the coated copper laminate. Expose the assembly to UV light for 2-3 minutes. The exposed photoresist will become soluble.
- Developing the Photoresist: Immerse the exposed PCB in a developer solution until the exposed photoresist dissolves, revealing the copper traces. Rinse thoroughly with water and dry.
- **Etching the Copper:** Mix an etching solution (ferric chloride or ammonium persulfate) and immerse the developed PCB until the exposed copper is etched away. Rinse and dry the board.
- **Stripping the Photoresist**: Use acetone or a dedicated photoresist stripper to remove the remaining photoresist from the copper traces. Rinse and dry the board.
- **Drilling Holes**: Use a hand drill or Dremel tool to drill holes for vias and component leads at the desired locations. Be careful to align the holes accurately.

- **Applying Solder Mask (Optional):** For added protection, you can apply a solder mask by screen printing or manually painting a UV-curable solder mask ink onto the PCB. Cure it under UV light.
- **Silk Screening (Optional):** Use a small screen-printing setup or stencils to manually apply silkscreen legends, labels, and logos onto the PCB surface.
- **Applying Surface Finish (Optional):** For better solderability, you can apply a surface finish like tin-lead plating or hot air solder leveling (HASL) using manual dipping or brushing techniques.
- **Cleaning and Inspection:** Thoroughly clean the finished PCB using isopropyl alcohol or a dedicated PCB cleaner. Visually inspect the board for any defects, shorts, or missing traces.
- **Assembly and Testing:** Once the PCB is fabricated, you can manually assemble the components and solder them in place. Test the assembled PCB to ensure it functions as intended.

These are steps to be followed while using the Reflow Oven:

- **Preparing the PCB:** Before using a reflow oven, ensure your PCB is fabricated and ready for assembly. Follow the DIY PCB manufacturing guide to create your PCB.
- **Applying Solder Paste:** Use a stencil or manually apply solder paste to the PCB pads using a syringe or stencil. Ensure the solder paste is evenly distributed and not too excessive.
- **Placing Components**: Manually place the components onto the PCB, aligning them with the pads. Use tweezers or a vacuum pickup tool to handle small components.
- **Reflow Oven Setup:** Use a DIY reflow oven or a modified toaster oven with a temperature controller. Set the oven to a temperature profile suitable for the solder paste and components used.
- **Reflow Process:** Place the PCB in the reflow oven and start the temperature cycle. The oven will heat the PCB to the peak temperature, causing the solder paste to melt and form connections. The cooling phase will solidify the solder joints.
- **Inspecting and Reworking**: Visually inspect the assembled PCB for any defects or cold solder joints. Rework any defective joints using a soldering iron and desoldering wick.
- **Cleaning and Testing**: Clean the assembled PCB using isopropyl alcohol or a dedicated PCB cleaner. Test the PCB to ensure it functions as intended.

Mr. Suyog explained the students about the drill bit sizes necessary for PCB manufacturing:

Sr. No.	Size
1	0.8mm
2	0.9mm
3	1.1mm
4	1.3mm
5	1.5mm
6	2.5mm
7	3mm
8	3.5mm
9	13.5mm

Mr. Suyog also emphasized on the material composition of these drill bits as they have to be made of carbide as they tend to precise.

He explained the students about the copper PCB material sizes necessary for PCB manufacturing:

Sr. No.	Size
1	35macros
2	17.5macros
3	50macros
4	70macros

Generally 35macros 1.5mm of thickness are used for manufacturing small scale PCB.

Based on these Processes 2000 PCB can be created by a single person with appropriate equipment at their disposal.

Conclusion: Mayuri Kadam extended heartfelt gratitude to Mr. Prabhakar Ghawali and Mr. Suyog Vyawahare for delivering a remarkable lecture and demonstration on the PCB manufacturing process. The comprehensive overview provided valuable insights into the intricacies of PCB fabrication, from design to production, captivating the audience with its informative and engaging content. The DIY approach showcased during the session shed light on the potential to create functional PCBs using minimal machinery and manual techniques.

The lecture and demonstration significantly enhanced the understanding of the PCB manufacturing process, fostering a deep appreciation for the precision and complexity involved in producing high-quality printed circuit boards. The opportunity to learn from Mr. Ghawali's expertise and experience was truly invaluable, enriching the knowledge and skills of all participants.

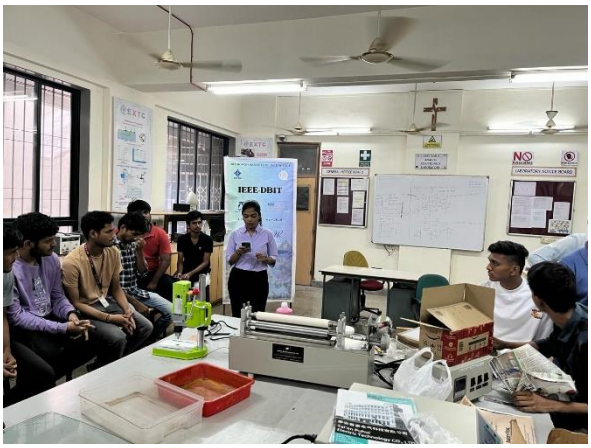
The Bombay Salesian Society's

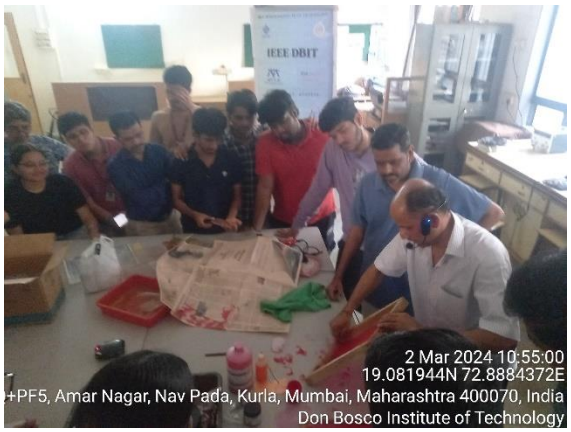
DON BOSCO INSTITUTE OF TECHNOLOGY

Premier Automobiles Road, Kurla West, Mumbai – 400070

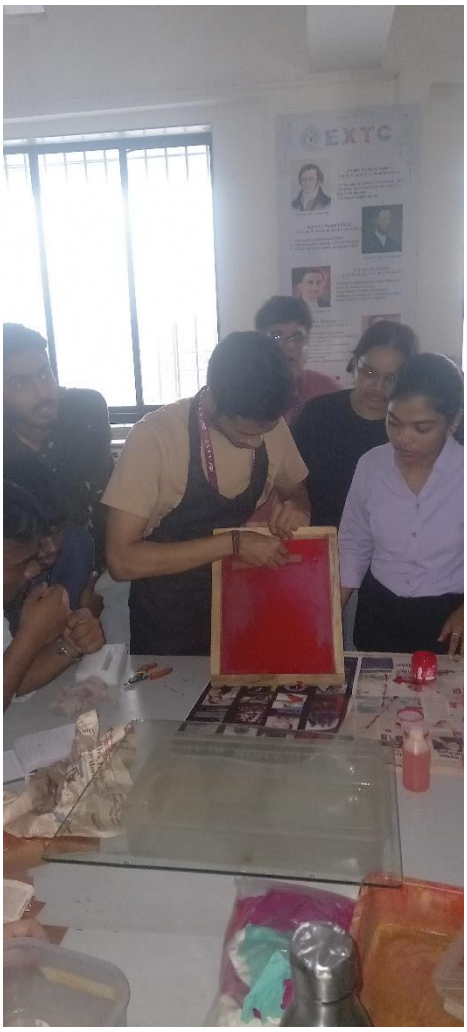
IEEE - DBIT

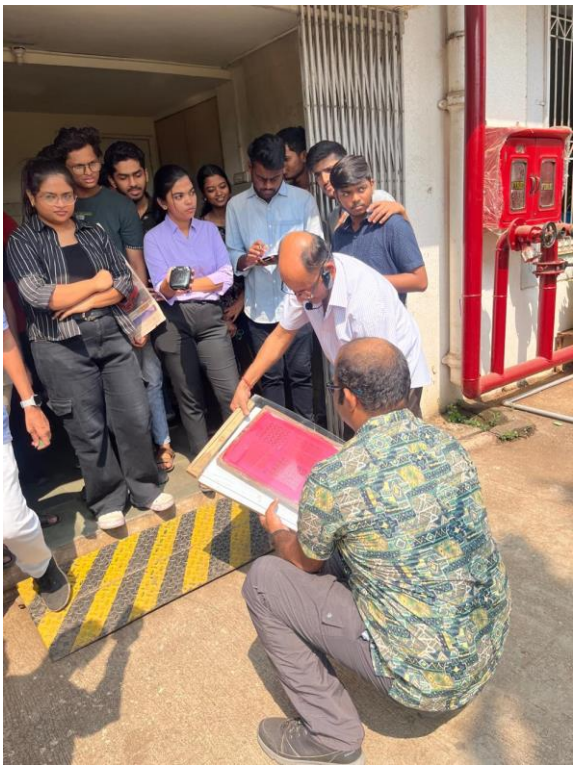
Pictures of the event:



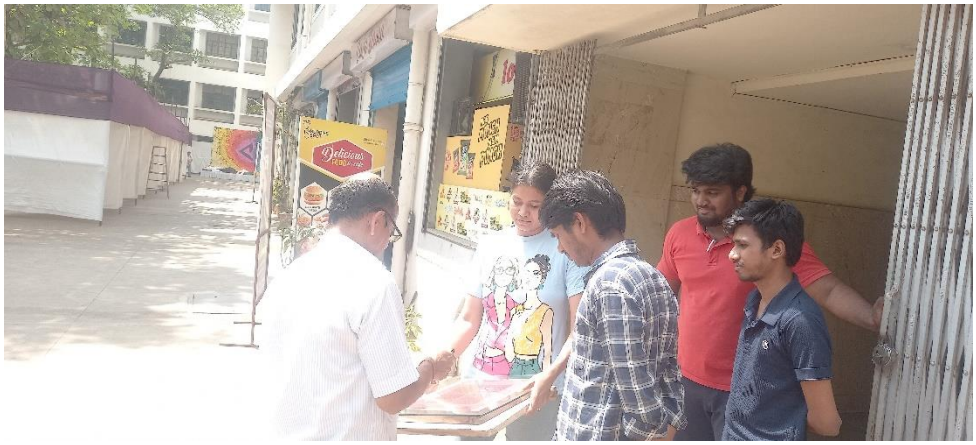


Mayuri Kadam welcoming our guest Lecturers





Mr. Prabhakar demonstrating Photoresist technique

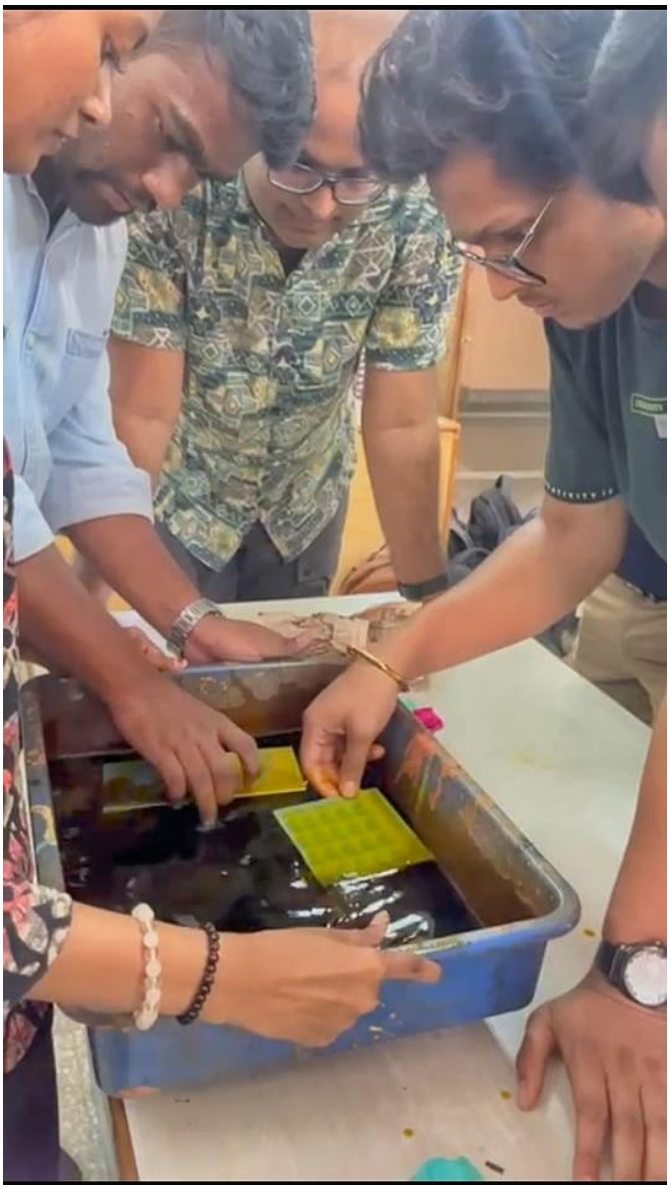


The Bombay Salesian Society's

DON BOSCO INSTITUTE OF TECHNOLOGY

Premier Automobiles Road, Kurla West, Mumbai – 400070

IEEE - DBIT

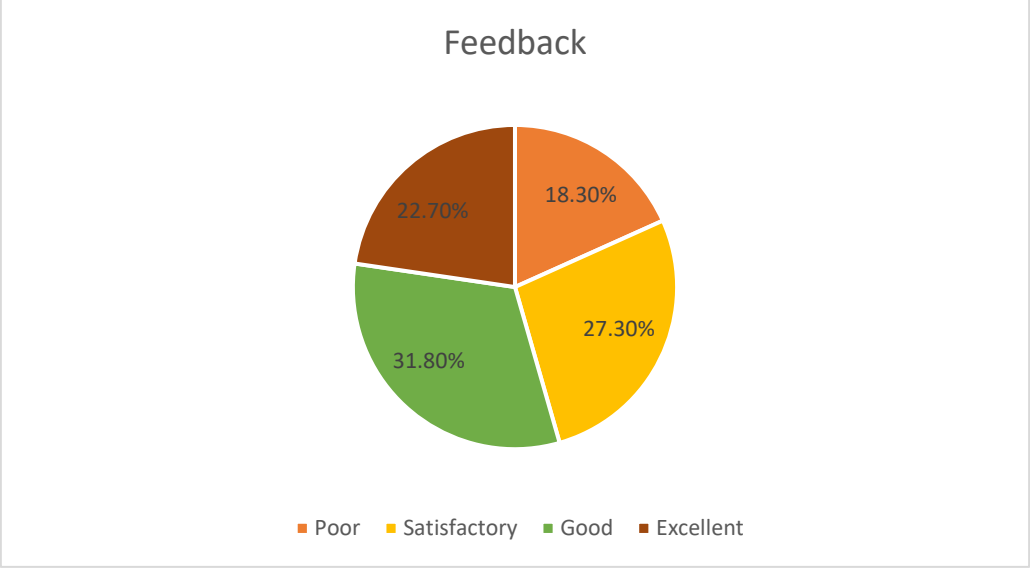


Copper Etching done by the students

Sr.No.	Name	Branch

1	Mohd Raza Ansari	TE EXTC
2	Aditya Ajay Jadhav	TE EXTC
3	Mayuri Sanjay Kadam	TE EXTC
4	Girish Vikas Sangare	TE EXTC
5	Dibyarupa Pradhan	TE EXTC
6	Rutvik Kiran Patil	TE EXTC
7	Premkumar Pradeep Singh	TE EXTC
8	Khushi Shetty	TE EXTC
9	Annanya Zadbuke	TE EXTC
10	Neel Pujari	SE EXTC
11	Prathamesh Kurudekar	SE EXTC
12	Jasmit Ganvir	SE EXTC
13	Kartik Dandeliya	SE EXTC
14	Aryan Arde	SE EXTC
15	Aftab Shaikh	SE EXTC
16	Aditya Punekar	SE EXTC
17	Gopal Jha	SE EXTC
18	Sarvambh Desai	SE EXTC
19	Kajal Sathe	BE EXTC
20	Darshan Kadlag	BE EXTC
21	Anto Ridon	BE EXTC
22	Vaishnav Nadar	BE EXTC

Feedback analysis:



The Bombay Salesian Society's

DON BOSCO INSTITUTE OF TECHNOLOGY

Premier Automobiles Road, Kurla West, Mumbai – 400070

Report Prepared By: Aditya Ajay Jadhav **Name of the Student:** Aditya Ajay Jadhav **Post of the Student:** Reporting Head IEEE-DBIT	**Report Approved By:** **Name of the Faculty: Freda Carvalho** **Post of the Faculty: IEEE DBIT SB Counselor**