
	<b>Akhil Bharatiya Maratha Shikshan Parishad's Anantrao Pawar College of Engineering &amp; Research</b>		
	<b>Record No.: ACA/D/021</b> <b>Revision: 00</b>	<b>DoI: 2/01/2023</b>	
<b>EVENT REPORT</b>			

Name of Event: Workshop on PCB Design at DST Prayas Shala.

Date of Event: 17/06/2023.

Time of event: 10:00 am to 4:00 pm

Name of Event Coordinator: Prof. Gopiraj Kavhekar

**Name of resource Person/ Speaker:** Mr. Millind Dhule, SMD Technologies.

Target Audience with count: EnTC Department, total 06 faculties.

**Brief Description of Event:**

Mr. Milind Dhule, from SMD Technologies, Pune who is also authorized dealer of ScienTech Technologies Pvt. Ltd. gives brief introduction on schematic preparation, uses of software, layouts of different circuits, thickness of PCB, PCB Machine software, extraction of code, PCB Designing Steps etc. He also explained about TINA. Toolkit for Interactive Network Analysis (TINA) is a SPICE-based electronics design and training software. TINA Design Suite is a powerful yet affordable circuit simulator, circuit designer and PCB design software package for analyzing, designing, and real time testing of analog, digital, IBIS, HDL, MCU, and mixed electronic circuits and their PCB layouts. With the TINA Cloud on-line circuit simulator, we can also edit and run schematic designs and their PCB layouts online on PCs, Macs, thin clients, tablets, smart phones, smart TVs and e-book readers without any installation.

He also explained details of different instruments present in Prayas Shala & gives demo on how to prepare PCB of Half wave rectifier using OPAMP. Innovation club members of APCOER, Mr. Sudarshan Natu & Dr. Padmakar Kelkar, were also present during the session.

**Expert Session Includes following Points:**

- 1) Drilling: PCB thickness should be 1.56mm.
- 2) Milling: Select the drill 0.5mm to prepare layout.
- 3) Give conductivity using wires: Use Si wire to give conductivity to drilling machine before PCB Designing.
- 4) Engraving: To remove excess copper after PCB designing.
- 5) Routing: To cut remaining portion.

PCB Designing is done using following steps:

- 1) Understand the electrical parameters.
- 2) Creating the schematic.



**Akhil Bharatiya Maratha Shikshan Parishad's  
Anantrao Pawar College of Engineering & Research**



**Record No.: ACA/D/021  
Revision: 00**

**DoI: 2/01/2023**

### **EVENT REPORT**

- 3) Use a schematic capture tool to create your PCB layout.
- 4) Design your PCB stack up.
- 5) Define design rules and requirements.
- 6) Place your components.
- 7) Insert drill holes.
- 8) Route the traces.
- 9) Add labels and identifiers
- 10) Generate design/layout files.

#### **Event Schedule:**

Sr. No.	Particulars	Time
1	Expert talk on PCB Designing	10:00 am-1:00 pm
2	Demo on tool	2:00 pm - 4:00 pm

#### **Photographs:**



**Mr. Millind Dhule, giving demo on PCB Designing.**



Akhil Bharatiya Maratha Shikshan Parishad's  
Anantrao Pawar College of Engineering & Research



Record No.: ACA/D/021  
Revision: 00

DoI: 2/01/2023

EVENT REPORT



EnTC staff with expert from SMD Technologies,Pune.

Date:

*Wahab*  
17/06/2023  
Event Coordinator

*Soojey*  
17/06/2023  
Head of Department

*[Signature]*  
Principal

