Sandip Foundation's
Sandip Institute of Technology & Research Centre
Mahiravani, Trimbak Road, Nashik 422213

Department of Electronics and Telecommunication Engineering

Activity Report

on

"One Day online workshop on Altium 365,a cutting-edge PCB design software"

Organized by

Sandip Foundation's

Sandip Institute of Technology and Research Centre, Nashik

And Department of

Electronics & Telecommunication Engineering Department

In

Association With

With IEEE student Branch, Engineers Without border (EWB)

Womens in Engineering(WIE)

Department of Electronics and Telecommunication Engineering

Name of Event: "One Day online workshop on Altium 365, a cutting-edge PCB design software"

Date of Event: 4th December 2024

Type of the Event: Workshop

Name of Event Coordinator: Prof. Mayur Ingale

Name of Resource Person: Mr. Surendra Narsimha, Sr. Designer, Altium, Bangalore

Mode of Event: Online

Audience: Student and Staff of E&TC department.

Objective:

1. To introduce the functionalities of Altium 365 for PCB design.
2. To explore real-time collaboration tools for engineers and teams.
3. To demonstrate cloud-based project management and sharing.
4. To provide hands-on experience in navigating Altium 365's interface.

Highlights from the event:

The Department of Electronics and Telecommunication, SITRC Nashik, successfully conducted a one-day online workshop on Altium 365, a cutting-edge PCB design software. Resource person, Mr. Surendra Narsimha, Sr. Designer, Altium, Bangalore shared his expertise with over 50 participants. The workshop covered topics like introduction to Altium 365, designing a PCB, and advanced PCB features.

- Introduction to Altium 365 : Altium 365 represents a significant leap forward in PCB design, offering a modern solution for the increasing complexity and demands of electronic product development. Whether for small startups or large engineering teams, Altium 365 empowers users to innovate faster, smarter, and more collaboratively.
- Designing a PCB, and advanced PCB features: Altium 365, in combination with Altium Designer, provides a comprehensive environment for designing high-quality printed circuit boards (PCBs). The platform offers an intuitive workflow, ensuring that every stage of the PCB design process is efficient and error-free.

Steps to Design a PCB in Altium 365

- **Project Creation**
  - Start by creating a new project in Altium Designer and linking it to Altium 365.

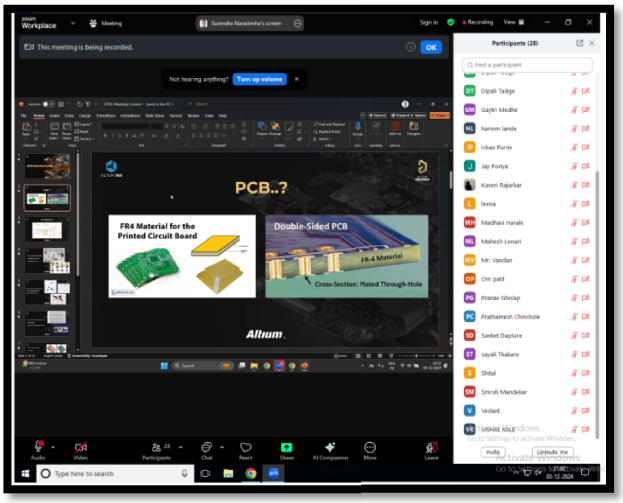
  - The cloud-based storage ensures centralized project management and easy sharing.
- **Schematic Design**
  - Use Altium Designer's schematic editor to create the circuit diagram.
  - Altium 365 ensures components are linked to verified libraries, minimizing errors.
- **Component Selection**
  - Access real-time component data from Altium's integrated libraries.
  - Ensure component availability with live supplier links and pricing updates.
- **PCB Layout Design**
  - Define the board shape, place components, and route traces.
  - Utilize design rules and constraints to optimize layout for manufacturability.
- **Simulation and Validation**
  - Run electrical simulations and DRC (Design Rule Check) to validate the design.
  - Altium Designer, integrated with Altium 365, highlights issues and suggests corrections.
- **Design Collaboration**
  - Share the design with team members for review directly via Altium 365.
  - Team members can comment, suggest changes, or approve the design in real-time.
- **Preparing for Manufacturing**
  - Use Altium 365's Manufacturing Package feature to generate production-ready files.
  - Share the design securely with fabricators and receive feedback directly through the platform.

Benefits of Using Altium 365 for PCB Design

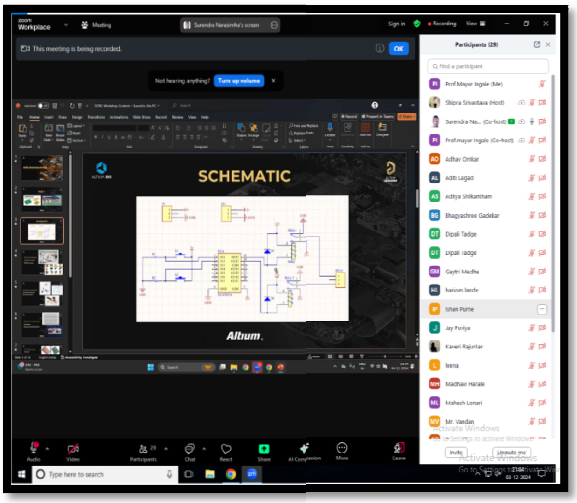
- **Enhanced Collaboration**: Breaks down silos between designers, teams, and stakeholders.
- **Faster Design Cycles**: Automation and cloud-based workflows reduce time-to-market.
- **Reduced Errors**: Integrated tools ensure that designs are optimized and error-free before production.
- **Scalable Solutions**: Supports everything from simple boards to complex, multi-layer designs.

Outcome:- The workshop successfully introduced participants to Altium 365 and empowered them with skills and knowledge essential for modern PCB design. It laid a strong foundation for their growth in electronics and design engineering fields.

Glimpse from the Event:-



Participants learn about the PCB Material



Participants learn to draw schematic using altium 365

Prof. Mayur Ingale
Event Co-ordinator

Dr. Gayatri M. Phade
HoD E&TC Department