

## A Report on 1- Day “Altium Designer Essentials: Mastering the Fundamentals” Program

Date: 3<sup>rd</sup> Feb 2025

Time: 10 am– 5 pm

Venue: A-D Seminar Hall, Dept. of AI&DS, KSSEM, Bangalore

Target Audiences: Students

No. of Participants: 22

**Objective:** The 8-hour workshop on “Mastering the Fundamentals of Altium Designer” is intended to provide the students with a strong foundation of PCB design using Altium Tools. The hands-on workshop is expected to familiarize the participants with the PCB Design process, and to give them a strong foundational knowledge of implementing the same workflow in Altium Designer tool.

The Department of Electronics and Communication Engineering, KSSEM, in association with Altium, organized a 1-Day hands-on workshop on “Altium Designer Essentials: Mastering the Fundamentals”, for the final year students of Electronics and Communication Engineering department of KS School of Engineering and Management, on 3<sup>rd</sup> February 2025.

The workshop was conducted by Mr. Surendra Narasimha, Certified Trainer and Content Creator, Altium India, and was conducted over Zoom Meetings. The registered students accessed the training lectures through Zoom, and worked on the Altium Designer tool on local computers, with resources accessed on Altium 365 Cloud.



Fig 1: Students attending the Online Workshop conducted via Zoom Meetings

The training session started with Mr. Surendra providing the participants with an introduction to the fundamentals of PCBs and the associated terminologies, followed by PCB Design workflow, with emphasis on the various stages of the process, and how Altium Designer tools can make the entire process efficient and economical. This was followed by an introduction to the various features of Altium Designer, where the participants explored the options for schematic design, component selection, placement, wiring and schematic capture using Altium Designer. The participants learned about the shortcuts and tricks available in Altium, which simplify the process greatly.

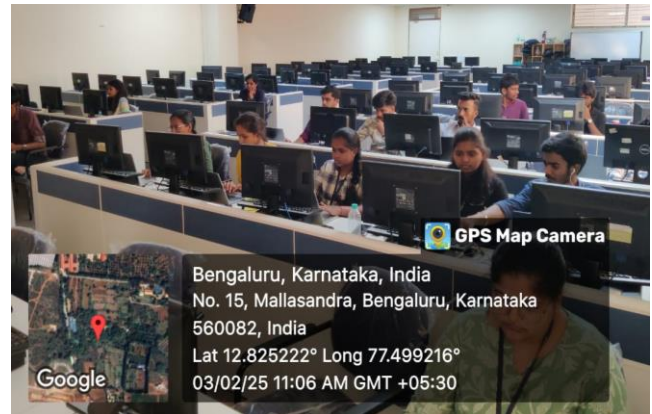
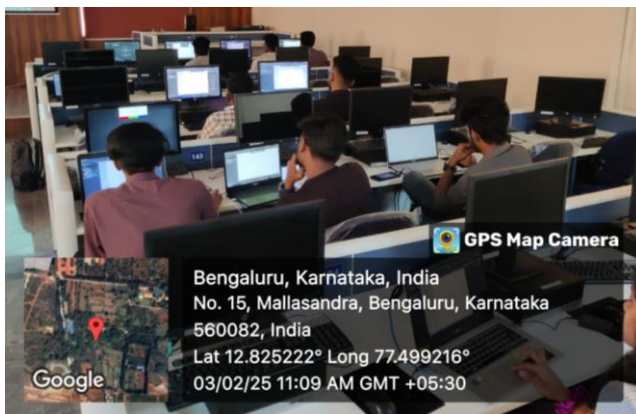


Fig 2: Participants practicing Schematic Capture . Fig 3: Students involved in hands-on training

### Outcome:

- Participants gained knowledge about the PCB design process, beginning with the draft design to Schematic Capture, PCB design and generation of Output Job Files.
- Participants learned how to import custom libraries, select and place components and wire them to capture schematics of projects, run Electrical Rules Check on the designed schematics and resolve violations.
- Participants also got experience converting the Schematics to PCB layouts, and learned about the various routing options available in Altium, with their advantages and limitations.
- The participants received extensive, in-depth training and hands-on experience with Altium Designer, Altium's industry-grade ECAD tool. They gained valuable insights into the various stages of the PCB design process and acquired the skills needed to use Altium Designer effectively.

FDP Co-Ordinators

1. Mr. Ravikiran B A

*BA*

*[Signature]*  
HOD, ECE

Dr. K Senthil Babu  
Professor & Head

Dept. of Electronics & Communication Engineering  
K.S. School of Engineering & Management  
Bangalore - 560 109