A Report on 3- Day "Train the Trainer" Program

Date: 27th Jan – 29th Jan 2025

Time: 9 am– 6 pm

Venue: Aryabhata Seminar Hall, Dept. of ECE, KSSEM, Bangalore

Target Audiences: Faculty Members

No. of Participants: 21

Objective: The 3-day "Train the Trainer" Program is organized with the objective of training faculty members from KSIT, KSSEM and KS Polytechnic in the efficient usage of Altium Designer tool for PCB Design. 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.

Day 1: 27th Jan 2025

Department of Electronics and Communication Engineering, KSSEM, in association KSRIF and the IEEE Student Branch, organized a 3-Day "Train the Trainer" workshop using Altium Designer tool, for the faculty members of KS Group of Institutions, from 27th to 29th January 2025**.** The workshop was inaugurated by Dr. KVA Balaji, CEO, KSGI, Dr. K. Rama Narasimha, Principal and Director, KSSEM, and Mr. Vikram Poojary, Senior University Relations Manager, Altium, and hosted by Dr. Renuka V Tali. The dignitaries emphasized on the impact of technology and AI on the industrial scenario, and how students and faculty need to keep their skillsets updated with the latest industry-standard technologies, in order to stay relevant and employed.

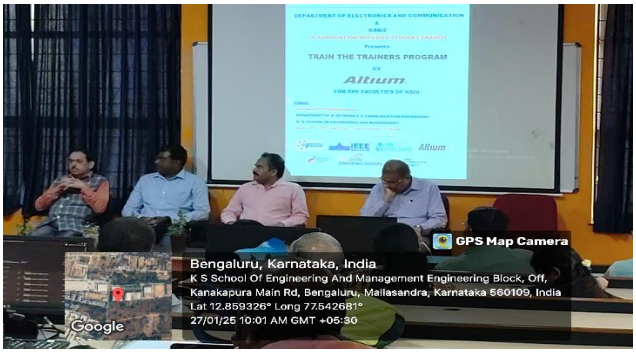

Fig No. 1: Inauguration Session with dignitaries on Stage. Fig No.2: HoD welcoming Dignitaries on stage

The inauguration was followed by the hands-on training session, led by Mr. Madhesh Perumal, Certified Trainer and Content Developer for Altium, accompanied by Mr. Jefferson Moses and Mr. Surendra Narasimha, Certified Trainers.

The training session started with Mr. Perumal providing the participants with an introduction to the 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 3: Trainer Mr. Madhesh Perumal explaining the fundamentals of PCB Design
Fig 4: Participants learning about the schematic capture process in Altium

Day 2: 28th Jan 2025

On the second day, the training resumed with the participants learning about efficient schematic capture using signal ports and power ports, followed by a session on setting up Electrical rules, and running Electrical Rules check on the implemented schematics, and the various types of errors and resolution strategies. This stage was followed by lessons on transferring the schematics to PCB layouts, and the rest of the day was spent learning about configuring the PCB design rules, learning about the various PCB layers, designing and customizing the board outlines and industry standards used in PCB design.

Figure 5: Participants involved in hands-on learning of PCB design
Figure 6: Trainer Mr. Madhesh explaining Design Rules Checking options in Altium Designer

Day 3: 29th Jan 2025

On the Third day, the participants began with the creation of PCB design rules and constraints, and learned about Altium's options available for PCB routing, especially Quick Routing and Interactive Routing, which make this critical task easy, with various options available. This was followed by a session on Design Rules Check (DRC), the critical step that follows PCB layout design, and strategies to rectify the common types of errors and violations encountered during PCB Design. The day concluded with the participants learning about Altium's ease in generating various job files, be it the drill files, assembly files or the design files, which would be shared by the designers, to the fabrication and assembly teams.

Figure 7: Trainer Mr. Jefferson explaining PCB routing to participants
Figure 8: Participants learning about Output file generation capabilities of Altium Designer

Outcomes:

- 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 gained experience navigating the Altium Designer application as well as the cloud and collaborative working features provided by Altium 365 service.
- Participants learned how to import custom libraries, select and place components and wire them to capture schematics of projects.
- Participants learned to 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 got hands-on experience generating various types of output job files, and documentation, using Altium's built in tools.
- Overall, the participants got a very comprehensive in-depth training and experience working with Altium's industry-grade ECAD tool, Altium Designer, and were able to gain a good insight into the various stages of PCB design process, and the knowhow to use Altium Designer to do the same.

FDP Co-Ordinators:

1. Mr. Ravikiran B A

HoD,ECE

Dr. K Senthil Babu
Professor & Head
Dept. of Electronics & Communication Engineering
K.S. School of Engineering & Management
Bangalore - 560 109