Maharaja Surajmal Institute of Technology, New Delhi

Electrical and Electronics Engineering Department

A Report on

Hands-On Workshop on "**PCB Design using KiCAD**"

Department of Electrical and Electronics Engineering organized a comprehensive Hands-On Workshop on "**PCB Design using KiCAD**" for EEE IV and VI semester. The workshop was conducted by Mr. Vaibhav Bhardwaj and Mr. Shayak Choudhari, Ph.D. Prime Minister Research Fellow Scholar, IIT Delhi. The workshop was aimed at providing EEE students with practical skills and knowledge in the field of Printed Circuit Board (PCB) design. The sessions were conducted on March 6 and March 15, 2024, April 19, 2024, April 24, 204, 2024 and April 26, 2024. The workshop was initiated with a detailed overview of PCBs, elucidating their significance, usage, and diverse applications across industries. Through engaging presentations and real-world examples, participants understood how PCBs are vital components in electronic devices, facilitating their functionality and connectivity. Furthermore, the speaker meticulously expounded on the structure and layers of PCBs, unraveling the complex architecture underlying these essential components. Participants were introduced to the multilayered composition of PCBs, comprising substrates, conductive traces, vias, and solder masks. Speaker A's lucid explanations and illustrative diagrams demystified the intricate layers of PCBs, fostering a comprehensive comprehension among attendees. An interactive environment by actively engaging with participants was created. He encouraged students to ask questions and took time to address doubts, ensuring clarity and understanding. This interactive

approach enriched the learning experience, allowing participants to delve deeper into PCB designing concepts and applications. An insightful discussion, centered on the intricacies of the circuits participants would undertake for design throughout the workshop was done. Speakers provided comprehensive insights into the complexity and functionality of various circuits, emphasizing key design considerations and potential challenges.

Subsequently, Speakers introduced KiCAD, a widely acclaimed open-source PCB design software, essential for efficient circuit designing. Through live demonstrations and hands-on exercises, participants were acquainted with KiCAD's user interface, tools, and functionalities. The benefits of employing KiCAD, including its versatility, extensive community support, and cost-effectiveness compared to proprietary software alternatives are underscored.

Throughout the session, participants actively engaged with speakers, seeking guidance on harnessing KiCAD for their upcoming projects. Speakers patiently addressed queries, offering invaluable tips and strategies to optimize the design process and enhance productivity within the software.

The session culminated with participants acquiring a solid foundation in circuit design principles and gaining practical proficiency in utilizing KiCAD for PCB designing purposes.