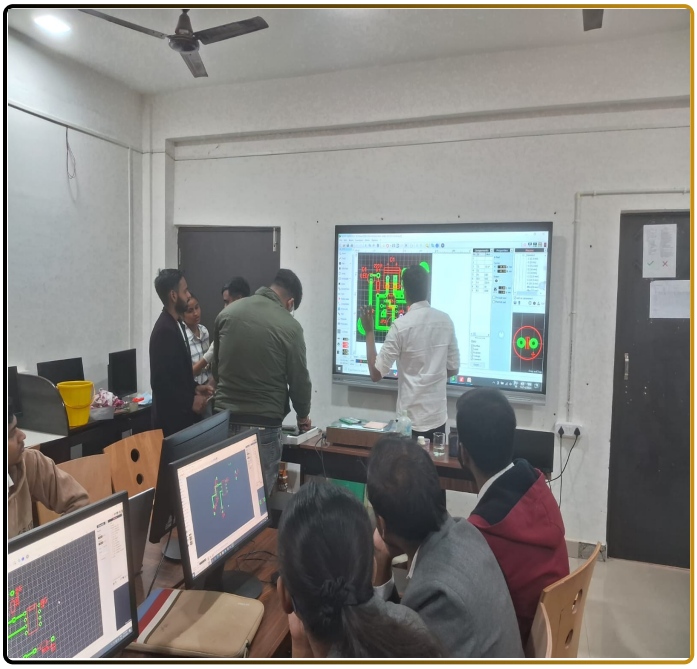
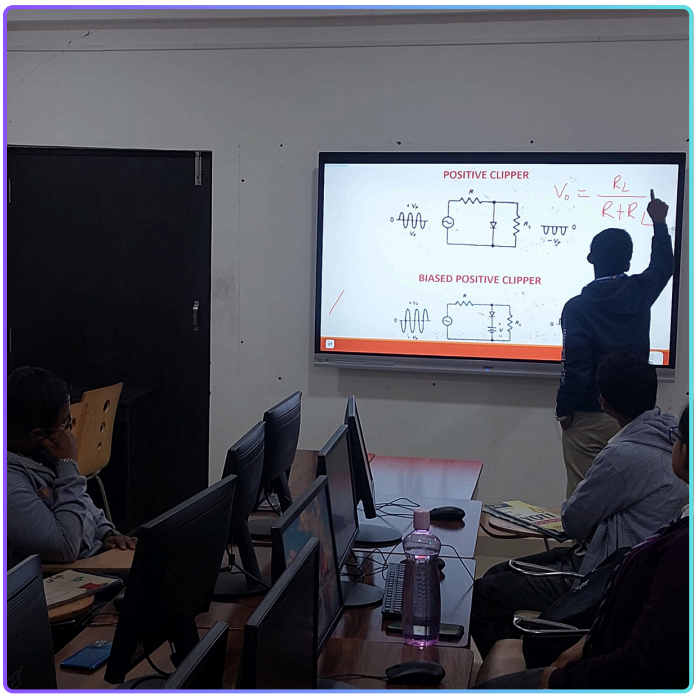
SWAMI VIVEKANANDA UNIVERSITY

ELECTRONICS AND COMMUNICATION ENGINEERING

March, 2025

Two Weeks Cadence based PCB Design Workshop



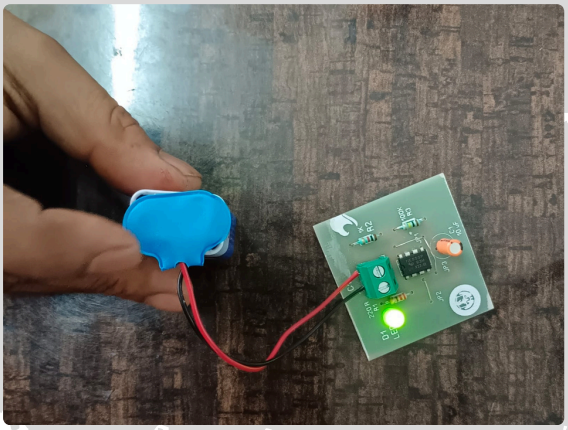


Students attending the Two Weeks Cadence Workshop

“Over ten immersive days, our ECE students embarked on a comprehensive journey through the Cadence PCB design environment, transforming theoretical circuit concepts into tangible, manufacturable boards. The workshop began with an introduction to the Cadence PCB interface—covering the purpose of each copper, silkscreen, solder mask, and drill layer—followed by hands-on sessions in which participants learned to create and validate schematics before ever placing a single component footprint. Early lectures focused on navigating the toolbars, managing schematic libraries, and generating accurate netlists, so that students could seamlessly translate their circuit diagrams into preliminary board outlines.

Midway through the program, emphasis shifted toward advanced design considerations: signal integrity, high-speed interconnects, and miniaturization strategies were demonstrated to illustrate how modern PCBs must accommodate ever-increasing frequency requirements. Parallel sessions introduced powerful simulation techniques, enabling participants to predict voltage drops, thermal hotspots, and potential crosstalk issues long before committing to a layout. By the end of the first week, students had mastered design-rule checking, layer stack-up decisions, and component footprint creation—laying a robust foundation for the hands-on fabrication phase.

In the second week, participants took their finalized, rule-checked designs and generated production-ready Gerber files, verifying drill and solder-paste layers to ensure manufacturability. The lab remained open for extended training and peer-guided projects, where students refined multi-layer layouts, resolved design rule violations, and executed test soldering runs on prototype boards. By the workshop’s conclusion, each student had not only designed but also observed the fabrication process of their own PCB, gaining a holistic understanding of the CAD-to-fabrication pipeline and forging the skills necessary to tackle future challenges in electronics manufacturing.”

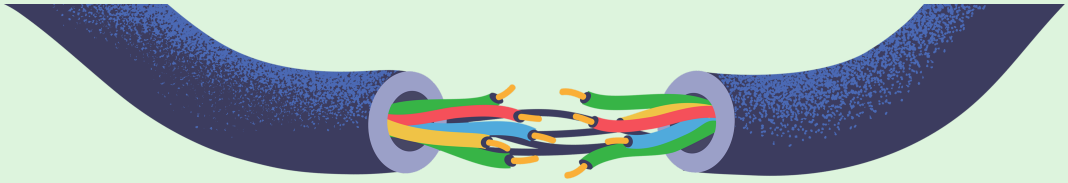


Live hands-on PCB design demonstration in Cadence, with students observing layout and routing techniques in real time

Closing Ceremony and Certificate Distribution

Feedback and Assessment

Over the course of the workshop, students mastered the end-to-end PCB design flow—moving from schematic creation and simulation to layout, rule-checking, and Gerber file generation—gaining hands-on proficiency in Cadence tools and fabrication fundamentals. Feedback was overwhelmingly positive: participants praised the real-time demonstrations and practical labs for deepening their confidence in PCB design.



In post-workshop assessments, students demonstrated solid understanding across all modules, with average scores exceeding 80%, reflecting both strong engagement and technical growth.