One Day Workshop on **"PCB Design Using KiCad"**

Date:- 27/07/2019

Time:- 10:00 AM to 4:00 PM

Venue:- Seminar Hall, First floor, A block & 208 Lab, A block, AIET

A one day workshop on "PCB Design Using Kicad" was organized by department of ECE on 27/07/2019 at Ashoka Institute of Engineering & Technology and the speaker for this workshop is one of the eminent and dynamic faculty member of department of ECE Mr. P. Abhishek, Assiatant Professor who has real time work experience in the field of PCB designing using different software like Xilinx, Multisim and MATLAB, he worked for Life Line Circuits Ltd, Tamil Nadu as Test Engineer.

The Session started at 10:00 Am in the morning with introduction speech of Dr. Hemachandran, ECE/HOD followed by address by Mr. Abhishek who started his session with basic concepts on PCB and designing tools.

The KiCad started its journey in 1992 which was introduced by Jean Pierre. He explained various circuit arrangement techniques using KiCad software for EDA (Electronic Design Automation) which helps in lay out designing in 3D. Also analyzing the artwork, Gerber files, and 3D views of the PCB and its components were explained with examples.

Latest techniques of exploring different resources that are beneficial in Eeschema, Pcbnew concepts for CSC (Custom Symbol Creation) and ERC (Electrical Rules Check) makes designing part easy and compilation of errors with adjustments in the circuit is also flexible. Initially the designing part faced lot of challenges when exposed to a software development, which led to enhancement and development of KiCad and other software tools during 20^{th} centuary.

The Session with theoretical explanation completed in forenoon. Afternoon session carried by the practical explanation and hand on practice to the students which helps to improve knowledge on graphical representation as this software has friendly platform with C++ , wxWidgets to run on FreeBSD, Linux, Microsoft Windows and Mac OS X and abstraction layer with OpenGL and Cairo back ends which create awareness on different libraries and language of programming portable for KiCad such as Java and VRML.

The development of KiCad has replaced the gap which was found in EAGLE software in EDA tool, KiCad has clear vision not only in electronics but has influenced the work in field of mechanical and electrical as the designing part comprises heavy machine designs in mechanical and the development of software in circuit designing makes save lot of efforts and generation of error free outputs.

Students seen enjoying the practical session with lot of passion to learn new concepts which helps them for real time designing and tools testing which was horrible task in a hardware where circuit cannot explain the faulty tolerance values and generation of proper output. The session concluded at 4:00 PM with a brief query and feedback session.