Workshop on - **PCB Designing**

*Organized by IEEE Student Branch*

*association with DIT University On 2nd to 5th February’17*

***[Coordinated by: ]***

***Overview:***

The *Workshop* was dedicated in training the students on the design and manufacturing of a **Printed Circuit Board**. This workshop was an invaluable resource for those who are learning PCB design. Designing a PCB is not something you will do in a couple of hours. It is a highly technical learned skill that will take years to master.

The Workshop provided the basics of PCB design as well as more advanced topics. It was four day workshop which focuses on advanced topic of PCB design along Fabrication process. Participant ere involved in etching process on a laminated copper sheet provided to them, they also performed drilling process using a Hand driller and finally assembling and soldering the electrical components onto board.

*Impressions:*

The duration of this workshop was for four days, with three hours session, properly divided into theory and hand on practical sessions. Certificate of participation will be provided by **IEEE Student Branch** in association with **DIT University.**

Objectives:

* Introduction to Circuit Diagram Editor
* Simulation, PCB Design Structure
* Component Placement and Design
* Testing, Board Analysis
* Fabrication output generation.

Components Set:

* PCB boards were present in the lab during the workshop for demonstration purposes.
* PCB fabrication involving hardware were available for this workshop.
* Skills Gained:
  + - Appreciate how a PCB is made and understand the variations in construction.
    - Evaluate the key aspects of a PCB design software.
    - Gain hands-on experience of different PCB routing and placement techniques.
    - Establish the important requirements of PCB manufacture.