PCB Design

Session 1 - Computerized capture of electronic designs

- Creation of symbolic representation of circuits (schematic diagram)
- Searching, loading and placing of components
- Wiring connections between components
- Updating component information in PCB layout (Auto packaging).
- Entering Page Notes.
- Saving the Project.

Session 2 - Introduction to PCB Layout

- PCB Structure and Standards.
- Tracks, Pads and Vias.
- Ratsnest and Netlist.
- Design of PCB layout of a circuit.
- Define the Board Outline.
- Place the components.

- Routing of the components - Manual and Automatic
- Detection and repair of Design rules violation, clearance errors and missing or incomplete connections.
- 3D View of the designed board.

Description:

- The workshop will be for two days .
- All practical, from Schematic diagram drawing to Fabrication output files generation, will be based upon EDWinXP software.
- Student should carry laptop equipped with EDWinXP 1.90 evaluation version installed. We will provide the evaluation version of the software.
- PCB boards shall be present in the lab during the workshop for demonstration purposes. (Optional)
- PCB fabrication involving hardware will be present for this workshop.
- will be provided with participation certificates

Session 4 – Fabrication

- Gerber & Photo-tooling.
- Fabrication Setup.
- Generation of Artwork.
- NC-Drill Generation – Drilling of holes (NCD files)