PCB Design Module

10 to 15days - 3hr/day

Basics of Industrial Electronics

- ➢ Introduction of Electronics
- ➢ Basic Components
- ➢ Circuit Calculations

ORCAD

Chapter – 1

- ➢ Introduction to ORCAD Software.
- ➢ Getting started with OrCAD Capture
- ➢ Learn about different tools of OrCAD Capture
- ➢ Learn about Design Preparation.
- ➢ Building simple Schematics.
- ➢ Building multiple page schematics.

Chapter - 2

- ➢ Design Rule checking (DRC error)
- ➢ Creation Bill Of Materials for design
- ➢ Creation of Pdf Documentation of Design

Chapter - 3

- ➢ Introduction to parts/symbol library
- ➢ Creating Parts & Symbol
- ➢ Preparing the design for layout (Net listing)

PCB Editor

Chapter 1: Library Development

- ➢ Introduction to various DIP and Surface mount Component.
- ➢ Learn about design preparation (libraries/Footprints).
- ➢ PAD stacks Designing.
- ➢ Creation of Footprints for DIP and SMD Devices.

Chapter 2: Getting Started

- ➢ Getting started with PCB Editor.
- ➢ Preparation of board for layout using board wizard.
- ➢ Learn about user interface.
- ➢ Defining parameter of board (Constraint Manager).

Chapter 3: Logic Import, Design Rules and Component Placement

- ➢ Net listing/logic import from schematic to PCB
- ➢ Component Placement

Chapter 4: Routing, Glossing and Copper Areas

- ➢ Introduction to manual routing
- ➢ Introduction to different parameter of perfect routing
- ➢ Finishing of routing
- ➢ Checking the status of routing.

Chapter 5: Post Processing and Manufacturing Output

- ➢ Assigning specific text (silkscreen) to design
- ➢ Creating Pdf documentation of design.

OrCAD PSPICE SIMULATOR

- ➢ PSPICE introduction
- ➢ Modifying schematic for simulation
- ➢ Transit analysis
- ➢ Creation of simulation profile for AC and DC sweep analysis