PCB Designing

Description

The course covers modules in Capture CIS(Component Information System), understanding and working on different types of layouts, and much more, all with a Hands-On experience!
Avail this opportunity to enhance your skills in PCB Designing and get trained for the Industry along with contributing to Live Projects.
On successful completion of live project during the course, students will be awarded with Live project Completion Certificate.

Prerequisites

- Module CIS
- Basic Knowledge of Electronic Components

Key Course Contents

Basics of Android

- How to create a new project in CIS?
- All the file extensions of CIS
- Preferences for page setup & colors
- All the short cut keys
- Explanations about the various libraries
- Explanations about the different tools along with their significances
- How to edit any existing part?
- How to create the new part?
- How to create the homogeneous parts?
- Important precautions
- What are net alias & their proper use?
- Different types of Power Symbols
- What are off-page connectors & their use?
- What is annotation & reannotation?
- What is DRC?
- How to check DRC along with its various options?
- How to remove the DRC errors?
- How to summarize the complete schematic?
- How to create the Bill Of material?
- How to assign the footprints for all components?
- How to create the net list?

Layout

- Tools, sub icons & menus of layout
- Scales/ grids used in layout
- All the short cut keys
- Creating the new footprint?
- How to find a footprint from existing orcad
- How & why to modify the existing footprint?
- Assigning the footprints from layout to
- Creating the net list in schematic after assigning
- How to import net list from schematic to layout?
- How to start a new design in layout?

Advanced UI Programming

- How to place a component?
- Where & why to place the component?
- Various techniques of placement?
- What is the difference between copper pour & copper area?
- Routing techniques for power/digital/ analog pcb?
- Routing parameters for auto/ manual for single/ double side pcb.
- Track width parameters & their proper usage.
- How to generate the gerber or films after completion of a design?
- How to create the TEAR DROPS?