PCB DESIGN TRAINING COURSE

Prerequisite 👍

- Education : No limit , basic electronics knowledge is required
- Familiar with computer usage, having a dedicated PC for practice is recommended.

Module 1: Introduction - 2.5h

1. Electronics in logical and physical form.
2. Standard packages and visualization
3. Units (mils / mm) and Grid system
4. Types of components joint forming techniques (TH & SMT)
5. Why do we need printed designs ?
6. Types of PCBs and classification
7. Overview of PCB fabrication process, importance of design stage recognition
8. Overview of PCB Assembly process, importance of design stage recognition
9. Elements in PCB and formal identification
10. Visualization of PCB elements.
11. Structure of a typical CAD tool
12. Which PCB design tool to use (Online / Local Installed)

Module 2: Symbol Library - 2.5h

1. Types of symbols (Homogeneous & Heterogeneous)
2. Creating new symbol library
3. Creating new symbol
4. Elements in a logical symbol (Pin number, pin name, pin function, graphical)
5. Part searching and extracting information from datasheets
6. Validating the created symbol
7. Examples : Resistor, Capacitor , Inductor and IC

Module 3: Footprint Library -2.5h

1. Recall of components types
2. Elements in foot print
3. Extracting information from datasheets
4. Creating new footprint
5. 3D Symbol mapping
6. Validating the created footprint
7. Examples : Resistor, Capacitor , Inductor and IC -

Module 4: Part Library creation -45min

1. How is the integration working?
2. Adding new integrated library
3. Creating a new part
4. Adding attributes to the part
5. Editing existing part in library
6. Validation of created part

Module 5: Schematics Design and verification - 4h

1. Basic elements in schematics
2. Reading and understanding the created schematics
3. Basic tips to make a clear and readable schematics
4. Schematic drafting
    a. Setting up library
    b. Adding symbols
    c. Adding connections
    d. Assigning net names
    e. Adding text notes
    f. Adding attributes to symbols
5. Schematic validation -30min
    a. Electrical rule check (ERC)
    b. Single pin nets verification
6. Bill of material creation and validating the required information
7. Netlist generation
8. Design transfer to layout

Module 6: Layout Design 3.5h

1. Basic rules (Cu vs Thickness and spacing. Voltage vs spacing and current vs width) to make layout clear and right at first time
2. Setting up Layer stack up
3. Drafting board outline
    a. Manual drafting
    b. Importing from mechanical .dxf or other compatible format
4. Setting up Electrical design rules
5. Components placement
    a. Place components
    b. Tips and useful options for components placement
6. Routing task
    a. Basics of routing and elements
    b. Good and bad practices in routing

c. Useful commands wile routing a layout
d. Validating the 100% routing completion and correcting the errors

Module 7: Design Validation & Release set up -1.5h

1. Interacting with cross functional teams
2. DRC verification and validation
3. Adding text and documentation information
4. Adjusting silk screen information
5. Adjust Assembly text and drawings

Module 8: Design release & Verification -1h

1. What is required for manufacturing ?
2. Required output files set up
3. Generation of output files
4. Validating generated output files.

Module 9: KeyStone Projects

1. Lithium Ion Battery charger - Teaching Project
2. Arduino Micro board design - Practical1
3. IoT Enabled Arduino - Practical2
4. Lerner choice project with guidance - Practical3

What you Learn from this course 👏

Understand the manufacturing and design concept of electronics product design
You can draw schematics and design PCB
Generate manufacturable files
Bring imaginations to reality as your own product.

Thank You