www.armacademy.in
ramani@armacademy.in

Hardware and PCB Design

Course Fee	**Rs 8,260/-** (inclusive of taxes)	Duration	**15 sessions** (2 hours per session)

Summary: This course is designed for those who need to be proficient in Hardware Design using a Schematics and Capture package. The course teaches the student to create production level PCB from schematics that are common in power electronics, control and instrumentation.

Prerequisites: Basic knowledge of integrated circuits, analog and digital systems.

Audience: Hardware design engineers who need to build a product from concept will need to understand the process of creating a PCB. Fresher engineers and diploma students in Electrical and Electronics need to undergo this course necessarily.

Course Contents

1. Introduction to Protel system Design - Use of different types of PCB tools, Description of Protel design software
2. Introduction to Schematics -Creating a New Project, Draw a schematic using various tools from Schematic Library.
3. Library Creation-New component creation, Creating many parts in multiple packages.
4. Creating Netlist, Design Rule Check, Generating Netlist from the schematics, Generating Bill Of Materials (BOM).
5. Introduction to Printed Circuit Board-Different PCB layers. -Pad Stacks, Vias, Track, Drill sizes.
6. Introduction to PCB Layout-Various tools in PCB layout, Description of various menu items, Generate the PCB file for the schematic.
7. Introduction to Footprint creation-Description of Library Manager and menu items, Different types of footprints.
8. Importing Netlist and Placement-Creating a board outline, Importing netlist and technology files, Place all the components on the board
9. Introduction to Manual/Auto Routing -Changing track angles and widths, Routing the board, changing track widths.
10. Introduction to Gerber generation -Different gerber files.

Course Director: Dr. Ramani Kalpathi (94440 34160)