II Year II Semester   **L   T   P   C**

Code:20EC4202   **1   0   2   2**

PCB DESIGN USING ORCAD

Course Objectives:
1. To discuss the basics of PCB designing
2. To familiarize design and fabrication techniques using modern tools
3. To study various PCB production techniques
4. To create/design libraries and footprints using ORCAD software
5. To apply and implement routing and post processing using ORCAD software.

UNIT–I: Basics of Printed Circuit Board designing

Fundamental of electronic components, basic electronic circuits, Basic terminology of PCB design, Layout planning, general rules and parameters, ground conductor considerations, thermal issues, check and inspection of artwork. Design rules for Digital circuit PCBs.

UNIT–II: Introduction to Electronic design automation (EDA) tools for PCB designing

Brief Introduction of various simulators, Selecting the Components Footprints as per design, Making New Footprints, Assigning Footprint to components, Net listing, PCB Layout Designing, Auto routing and manual routing.

UNIT–III: Introduction PCB production techniques

Photo printing, film master production, reprographic camera, basic process for double sided PCBs photo resists, Screen printing process, plating, relative performance and quality control, etching machines, Solders alloys, fluxes, soldering techniques, Mechanical operations. Multilayer PCBs.

UNIT–IV: Getting Started & Library Development using ORCAD

Getting started with PCB Editor, Preparation of board for layout using board wizard, User interface, defining parameter of board (Constraint Manager), Different tools of Orcad/Allegro PCB Editor, Introduction to various DIP and Surface mount Component, preparing libraries/ Footprints), PAD stack Designing, Creating footprints for DIP and SMD Devices.

UNIT–V: Design Rules, Routing and Post Processing using ORCAD

Net listing/logic import from schematic to PCB, Component Placement, Manual routing, Different parameters for perfect routing, Finishing of routing (Glossing, unmitter), Introduction to copper pour for power plane, Checking the status of routing, Routing power track (VCC & GND), Reducing Crosstalk effect, Reducing EMI effect, Assigning specific text (silkscreen), Creating reports, Creating manufacturing data (GERBER) for design, Plotting of layout design for home production, Creating Pdf documentation.

Course Outcomes:

A student who successfully fulfils this course requirement will be able to:

S.No	Course Outcome	BTL
1.	Remember the basics of PCB designing	L1
2.	Familiarize with the advance techniques and study modern tools for designing and fabrication of PCBs	L2
3.	Understand about various PCB production techniques	L2
4.	Apply ORCAD software to create/design libraries and footprints	L3
5.	Implement the logic Import, Design Rules, Component Placement, Routing and Post Processing using ORCAD software	L3

Correlation of COs with POs& PSOs:

CO	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12	PSO1	PSO2
CO1	1	1	2	-	2	-	-	-	3	-	-	-	2	3
CO2	1	2	2	-	3	-	-	-	3	-	-	-	2	3
CO3	1	2	2	-	3	-	-	-	3	-	-	-	2	3
CO4	2	3	3	2	3	-	-	-	3	-	-	-	3	3
CO5	2	3	3	3	3	-	-	-	3	-	-	-	3	3

Text Books:

1. Printed circuit board design, fabrication assembly and testing – R. S. Khandpur, Tata McGraw Hill, 2006.
2. Complete PCB Design using OrCad Capture and PCB Editor – Kraig Mitzner, Bob Doe Alexander, Akulin Anton, Suponin Dirk Müller, Newnes (Elsevier), 2009.

Reference Books:

1. Printed Circuit Board Design and technology– Walter C. Bosshart, McGraw Hill, 2000.
2. Printed Circuits Handbook–Clyde F. Coombs Jr., Happy T. Holden, McGraw Hill, Sixth Edition, 2016.