

Subject Code: 01EV0106

Subject Name: Computer Aided PCB Design

MTech. Year – 1 (Semester – 1)

**Objective:**

.

**Credits Earned:** 5 Credits**Course Outcomes:** After completion of this course, student will be able to

- Distinguish the various terms of PCB design
- Select, place and connect circuit components in different layer.
- Develop different layer required for manufacturing process.
- Fabricate Printed circuit board for the given circuit.
- Design of PCB with compliance to EMI and EMC concepts

**Pre-requisite of course:****Teaching and Examination Scheme**

| Teaching Scheme (Hours) |          |           | Credits | Theory Marks |             |              | Tutorial/ Practical Marks |                | Total Marks |
|-------------------------|----------|-----------|---------|--------------|-------------|--------------|---------------------------|----------------|-------------|
| Theory                  | Tutorial | Practical |         | ESE (E)      | Mid Sem (M) | Internal (I) | Viva (V)                  | Term work (TW) |             |
| 3                       | 0        | 2         | 4       | 50           | 30          | 20           | 25                        | 25             | 150         |

**Contents:**

| <b>Unit</b>        | <b>Topics</b>   | <b>Contact Hours</b> |
|--------------------|---|----------------------|
| 1                  | <b>Introduction to PCB Design</b><br>The old days, PCB packages, standard , The schematic , Imperial and metric , working to grids, working from the top , tracks ,pads , via ,polygons , clearances  | 8                    |
| 2                  | <b>Component and placement &amp; Design</b><br>Create new project, Drawing the Schematic, Add the Components, Join Them Together, electrical rule check, laying rule check, Basic Routing, Finishing touches, Single sided design, Double sided design.   | 10                   |
| 3                  | <b>Layers</b><br>Silkscreen, solder mask, mechanical layers, keep out, Layer alignment , Netlists, rats net, design rule checking, forward and backward annotation , multi layer design, power planes , good grounding, good bypassing , high frequency design techniques, Double sided loading, auto routing ,auto placement | 8                    |
| 4.                 | <b>Introduction to PCB Fabrication &amp; Assembly</b><br>Steps involved in fabrication of PCB, PCB Fabrication techniques-single, double sided and multilayer, Etching: chemical principles and mechanisms, Post operations- stripping, black oxide coating and solder masking component assembly processes                   | 8                    |
| 5                  | <b>EMI EMC Design for Printed Circuit Boards</b><br>Introduction to EMI EMC , EMI Emissions Sources , Inductance, Ground, Shielding   | 8                    |
| <b>Total Hours</b> |   | 42                   |

**References:**

1. Shawn Wallace, Jumpstarting Your Own PCBs Learn How to Design and Make Customized Circuit Boards, Maker Media, Inc San Francisco
2. Simon Monak, Make your own PCBs with EAGLE from schematic Designs to Finished Boards, Mc Graw Hill Education.
3. Bruce R. Archambault, IBM corporation, PCB Design for real world EMI control ,Springer science + business Media LLC.
4. David L. Jones, PCB design Tutorial, alternatezone.com
5. Kraig Mitzner, Complete PCB design using ORCAD capture and PCB editor , Elsevier Inc.

**Suggested Theory distribution:**

The suggested theory distribution as per Bloom's taxonomy is as per follows. This distribution serves as guidelines for teachers and students to achieve effective teaching-learning process

| Distribution of Theory for course delivery and evaluation |            |       |         |          |        |
|---|------------|-------|---------|----------|--------|
| Remember  | Understand | Apply | Analyze | Evaluate | Create |
| 5%  | 10%        | 15%   | 30%     | 20%      | 30%    |

**Suggested List of Experiments:**

1. Schematic Entry using Eagle for given circuit diagram of power electronics related circuit.
2. PCB Layout Design using Eagle for given schematic and specifications of power electronics related circuit.
3. Schematic Entry using Eagle for given circuit diagram of Embedded systems related circuit.
4. PCB Layout Design using Eagle for given schematic and specifications of Embedded related circuit.
5. Use of contributed component library in Eagle.
6. Creating symbol of component using Eagle from given component datasheet.
7. Creating footprint of component using Eagle from given component datasheet.
8. Creating and distributing component library using Eagle.
9. Generate Gerber file for given Eagle project as per manufacturer specifications.