Program : **Diploma in Electronics Engineering/ Electronics and Communication Engineering / Biomedical Engineering**	
Course Code : **4048**	Course Title: **PCB and Prototyping Workshop**
Semester : **4 / 4 / 3**	Credits: **1.5**
Course Category: **Program Core**	
Periods per week: **3 (L:0, T:0, P:3)**	Periods per semester: **45**

Course Objectives:

- To develop the competency required to prepare PCBs up to industrial grade.
- To impart the skill in the preparation of schematics, PCB layout, custom component design, custom footprint design, and design rules.

General Instructions:

The course is intended to provide students exposure to industry practices in the Printed Circuit Board (PCB) design with drafting software. It is the primary responsibility of the institution to ensure that copyright or licensing terms applicable to the drafting software are followed. The course recommends usage of free/open license/ limited license tools like Free EDA, Autodesk Eagle (limited license), KiCAD, etc. However, usage of industry-specific software like Altium Designer, ORCAD, Proteus are encouraged if licensed versions of such software are available.

Course Prerequisites:

Topic	Course code	Course name	Semester
Basic concepts of dimensions and projections		Engineering Graphics	1
Interpretation of component data sheets		Electronic circuits	3

Course Outcomes:

On completion of the course, the student will be able to:

COn	Description	Duration (Hours)	Cognitive level
CO1	Develop schematic diagram using suitable software.	6	Applying

CO2	Develop layout for double layer and single layer PCBs	12	Applying
CO3	Develop custom component and custom foot print.	12	Applying
CO4	Apply design rule check for PCB design.	12	Applying
	Lab Exam	3	

CO – PO Mapping:

Course Outcomes	PO1	PO2	PO3	PO4	PO5	PO6	PO7
CO1	3		3	3			
CO2	3	3	3				
CO3	3						
CO4	3	3	3	3			

3-Strongly mapped, 2-Moderately mapped, 1-Weakly mapped

Course Outline:

Module Outcomes	Description	Duration (Hours)	Cognitive Level
CO1	**Make use of suitable software tools to prepare schematic diagram.**		
M1.01	Develop schematic diagram of various analog, digital and microcontroller applications.	6	Applying
CO2	**Develop layout for double layer and single layer PCBs.**		
M2.01	Demonstrate single layer, multilayer PCBs, through hole and SMD packages, Net listing, PCB design rules, layers foot prints, tracks, vias, PTH, PCB border, materials, IPC standards, usage of connectors terminals etc.	3	Understanding
M2.03	Develop double layer PCBs for various analog, digital and microcontroller applications using auto and manual routing. (Double layer PCBs are included purposely before single layer due to its simplicity in implementation through auto routing)	3	Applying

M2.04	Develop single layer PCBs for various analog, digital and microcontroller applications using auto and manual routing.	6	Applying
	Lab Exam – I	1.5	
CO3	**Develop custom component and custom foot print.**		
M3.01	Develop custom components for schematics following dimensional and symbolic representation as provided in the datasheet.	6	Applying
M3.02	Develop custom foot prints for new components following schematics following dimensional specifications provided in the datasheet and link the foot print to schematics.	6	Applying
CO4	**Apply design rule check for PCB design**		
M4.01	Apply PCB design rules for analog circuits, power supplies, sensor modules, electronic projects and embedded projects.	3	Applying
M4.02	Make use of post manufacturing procedures in printing and design, etching, drilling, Interconnecting and Package electronic Circuits (IPC) standards, Gerber generation, pre-production tests, costing, Masking, artworks etc.	3	Applying
M4.03	Apply design rules in developing PCBs for dimensional compatibility with enclosures and casings.	6	Applying
	Open Ended Experiments		Applying
	Lab Exam – II	1.5	

**** - Suggested Open Ended Projects**

Open ended experiments are recommended for compulsory inclusion in Continuous Internal Evaluation. Students shall perform open-ended experiments individually.

- Follow modular design concepts (usage of proper connectors and proper mountings) to design Printed Circuit Boards for Project work, prepare costing, use online/offline methods to get PCB manufactured.

Text / Reference:

T/R	Book Title/Author
T1	Data sheets and instruction manuals for the software tools used.
R1	Christopher T. Robertson, Prentice Hall, Printed Circuit Board Designer's Reference: Basics.
R2	Walter C Bosshart, Tata McGraw-Hill, Printed Circuit Boards: Design and Technology
R3	Matthew Scarpino, Prentice Hall, Designing Circuit Boards with EAGLE: Make High-Quality PCBs at Low Cost

Online Resources:

Sl.No	Website Link
1	https://learn.sparkfun.com/tutorials/pcb-basics/all
2	https://www.circuitbasics.com/make-custom-pcb/.
3	https://www.pcbway.com/blog/?tag=PCB%20Design%20Tutorial