

# **ECE 39595 – Printed Circuit Board Design for Signal Integrity and EMC Compliance**

## **Type of Course**

Elective for the EE and CPE Technical Group 1

## **Catalog Description**

This course introduces the printed circuit board (PCB) design process, including component library creation, schematic capture, PCB layout, and design verification using industry-standard tools. Emphasis is placed on component placement, routing techniques, and design rule checks to produce manufacturable PCB layouts and documentation. The course also introduces practical considerations for signal integrity and electromagnetic compatibility (EMC), including layer stack-up design and power and ground plane implementation. Through hands-on projects, students develop the ability to design reliable multilayer PCBs that meet industry and IPC standards.

## **Credits**

3

## **Contact hours**

4

## **Prerequisite Courses**

ECE 20200, ECE20700, and ECE 27000

## **Corequisite Courses**

None

## **Prerequisites by Topics**

Students should have a foundational understanding of linear circuit analysis and digital logic design, including AC/DC circuit analysis, basic semiconductor devices, and combinational and sequential logic circuits. This knowledge is typically obtained through ECE 20200 (Linear Circuit Analysis II), ECE20700 (Electronic Measurement Techniques), ECE25500 (Introduction to Electronic Analysis and Design) and ECE 27000 (Digital Circuit Design).

## **Recommended Textbooks**

“Altium Designer 26.3 User Guide: Mastering Circuit Design, PCB Layout, Signal Integrity, Rigid Flex Architecture, Harness Systems, and Fabrication ...” , Matthew B. Chenoweth, Independently published, March 1, 2026, ISBN-13: 979-8250353779

“Signal Integrity, In Practice: A Practical Handbook for Hardware, SI, FPGA & Layout Engineers Revision 2”, Donald Telian, Independently published, 2023, ISBN-13: 979-78-904036-2

“EMC and the Printed Circuit Board”, Authors: Mark I. Montrose, Publisher: Wiley-IEEE Press, Publication Date: September 14, 1998, ISBN: 978-0780347038

## **Course Objectives**

This course aims to provide students with practical knowledge and hands-on experience in the design and layout of printed circuit boards (PCBs) for modern electronic systems. Students will learn the complete PCB design workflow, including schematic capture, component library management, PCB layout, and preparation of manufacturing documentation. Emphasis is placed on applying industry best practices to ensure signal integrity, power integrity, and electromagnetic compatibility (EMC) in multi-layer PCB designs.

## **Course Outcomes**

**Students who successfully complete this course will be able to:**

1. Explain the complete PCB design workflow, including component library creation, schematic capture, PCB layout, and preparation of fabrication and assembly documentation. (ABET Outcome 1)
2. Use industry-standard PCB design tools to create electronic schematics and implement multi-layer PCB layouts following professional design practices and constraints. (ABET Outcome 1)
3. Apply PCB layout techniques such as component placement, routing strategies, power and ground plane design, and design rule checks (DRC) to ensure manufacturability and reliable circuit operation. (ABET Outcome 2)
4. Analyze signal integrity and power integrity issues in high-speed PCB designs and apply appropriate routing, referencing, and termination strategies to improve system performance. (ABET Outcome 6)
5. Design printed circuit boards that meet electromagnetic compatibility (EMC) requirements, incorporating grounding, shielding, filtering, and noise-suppression techniques. (ABET Outcome 2)
6. Develop and document a complete PCB design project, including schematics, layout files, and manufacturing documentation, and communicate the design effectively in written or presentation form. (ABET Outcome 3)

## **Lecture Topics**

1. Introduction to Printed Circuit Board (PCB) Design and the complete design workflow
2. PCB components, libraries, and schematic capture techniques
3. PCB materials, layer stack-up design, and mechanical layer configuration
4. PCB design rules, constraints, and manufacturability considerations (DRC/DFM)
5. Component placement strategies and layout best practices

6. Routing techniques for analog, digital, and high-speed signals
7. Signal integrity fundamentals: transmission lines, differential pairs, and termination
8. Power integrity and power distribution networks (decoupling capacitor selection, grounding and return current paths, loop inductance effects and resonances)
9. Electromagnetic compatibility (EMC): noise mitigation, crosstalk, shielding, and filtering techniques
10. Linear and switching power supply design and layout best practices
11. PCB verification, simulation concepts, and generation of manufacturing documentation (Gerber, BOM – Bill of Materials, assembly drawings)

### **Computer Usage**

Medium

### **Laboratory Experience**

Medium

### **Design Experience**

High

### **Instructor:**

Ed Lyvers, Ph.D.

### **Coordinator**

Guoping Wang, Ph.D.

### **Date**

03/15/2026