PCB Design for Emerging Design Technologies
SYLLABUS

INSTRUCTOR INFORMATION

Instructor: Kris Moyer

Email: kristophermoyer@electronics.org

Phone: 916-674-2090

Best time to call: Usually available between 6pm – 9pm Pacific Time USA.

Leave message anytime.

PROGRAM DESCRIPTION

In the highly competitive electronics industry, the knowledge and skills of a workforce responsible for the design and layout of the Printed Circuit Board (PCB) and Printed Board Assembly (PBA) can have a direct impact on the success or failure of the product design and affect time to market. The PCB Design for Emerging Design Technologies course is designed to provide the skills necessary to create PCB/PBA designs that require cutting-edge emerging design technologies and comply with all necessary IPC standards, including new standards being developed in this area. Taught by a certified industry expert with 30+ years of experience in the field, the six-week program utilizes live instructor lead courses consisting of both theory and hand-on exercises to demonstrate the concepts and methodologies presented in class to facilitate mastery of the key concepts required by circuit board designers.

This course is intended for individuals who have completed or possess the equivalent skills or experience from PCB Fundamentals Parts 1 & 2 and who need further experience with design, manufacturing, packaging, and routing challenges for products that require advanced and emerging technologies to solve design challenges, including high-speed digital designs.

LEARNING AND PERFORMANCE OBJECTIVES

This program is designed to provide circuit board designers with a balanced foundation of theoretical knowledge and practical skills in printed circuit board design. Upon completion, participants will be able to:

- Understand basic Electrical Engineering (EE) concepts.

- Understand the trade-offs in materials used in these applications.
- Define a board stackup that implements structures that will meet the needs of specialty designs.
- Understand and mitigate signal integrity issues.
- Understand and implement design and routing techniques to mitigate HDI/UHDI issues.
- Understand and implement design and routing techniques for 3-D printed electronics.
- Understand and implement Power Distribution Network (PDN) design issues.

COURSE STRUCTURE

- Instructors and participants meet online twice per week from the comfort of their own home.
- Participants can view recorded online sessions to review course content and class discussions.
- Participants apply key concepts to create a real-world design from concept to completion.
- All required materials are included in the course. Participants may utilize any PCB design authoring software program of their choice. If participants do not have access to PCB design authoring software, participants will receive complimentary access to a program.
- Course materials are accessible 24/7 on the Edge Learning Management System.
- The course can be accessed on virtually any device with an Internet connection and major web browser, including Chrome, Firefox, Safari, Edge, and Internet Explorer

SUPPLEMENTAL MATERIALS

- Printed Circuit Handbook – *Clyde F. Coombs* McGraw-Hill
- Right the First Time – Lee W. Ritchey Speeding Edge
- Signal Integrity Issues and Printed Circuit Boards – *Douglas Brooks,* Prentice Hall

IPC STANDARDS COVERED (PROVIDED WITH COURSE)

- IPC-2152 STANDARD FOR DETERMINING CURRENT CARRYING CAPACITY IN PRINTED BOARD DESIGN
- IPC-2221 GENERIC STANDARD ON PRINTED BOARD DESIGN
- IPC-2222 SECTIONAL DESIGN STANDARD FOR RIGID ORGANIC PRINTED BOARDS
- IPC-2611 GENERIC REQUIREMENTS FOR ELECTRONIC PRODUCT DOCUMENTATION

- IPC-2612 SECTIONAL REQUIREMENTS FOR ELECTRONIC DIAGRAMMING DOCUMENTATION (SCHEMATIC AND LOGIC DESCRIPTIONS)
- IPC-2612-1 SECTIONAL REQUIREMENTS FOR ELECTRONIC DIAGRAMMING SYMBOL GENERATION METHODOLOGY
- IPC-2614 SECTIONAL REQUIREMENTS FOR BOARD FABRICATION DOCUMENTATION
- IPC-2615 PRINTED BOARD DIMENSIONS AND TOLERANCES
- IPC-4101 SPECIFICATION FOR BASE MATERIALS FOR RIGID AND MULTILAYER PRINTED BOARDS
- IPC-4104 SPECIFICATION FOR HIGH DENSITY INTERCONNECT (HDI) AND MICROVIA MATERIALS
- IPC-6011 GENERIC PERFORMANCE SPECIFICATION FOR PRINTED BOARDS
- IPC-6012 QUALIFICATION AND PERFORMANCE SPECIFICATION FOR RIGID PRINTED BOARDS
- IPC-7351 GENERIC REQUIREMENTS FOR SURFACE MOUNT DESIGN AND LAND PATTERN STANDARD
- IPC J-STD-001 REQUIREMENTS FOR SOLDERED ELECTRICAL AND ELECTRONIC ASSEMBLIES
- IPC-2226 SECTIONAL DESIGN STANDARD FOR HIGH DENSITY INTERCONNECT (HDI) PRINTED BOARDS
- IPC-6016 QUALIFICATION AND PERFORMANCE SPECIFICATION FOR HIGH DENSITY INTERCONNECT (HDI) LAYERS OR BOARDS
- IPC J-STD-004 REQUIREMENTS FOR SOLDERING FLUXES
- IPC J-STD-005 REQUIREMENTS FOR SOLDERING PASTES
- IPC J-STD-006 REQUIREMENTS FOR ELECTRONIC GRADE SOLDER ALLOYS AND FLUXED AND NON-FLUXED SOLID SOLDERS FOR ELECTRONIC SOLDERING APPLICATIONS
- IPC-T-50 TERMS AND DEFINITIONS FOR INTERCONNECTING AND PACKAGING ELECTRONIC CIRCUITS

COURSE SCHEDULE

WEEK 1: DESIGN – HDI / UHDI

Program overview outlining class schedule and options for accessing class material and assignments. The lecture will focus on HDI/UHDI design.

Key concepts include:

- HDI

- Micro-Vias
- Sequential Lamination
- Circuit Reduction / Compression
- Difference from Std HDI
- Special design requirements for UHDI
- Additive vs. Subtractive process
- Challenges and pitfalls with UHDI
- IPC standards

DEMONSTRATION:

- Stackup and layout using UHDI techniques

WEEK 2: DESIGN – 3D PRINTING

Key concepts include:

- Difference in design
- Special design tools
- Difference in routing
- Structural issues
- Size issues
- Thermal issues
- IPC standards

DEMONSTRATION:

- Demonstration of design and routing techniques for 3D printing

WEEK 3: MATERIALS

Materials and HDI/UHDI & 3D printing. Physical properties. Types of structures, trade-offs

Key concepts include:

- Std materials
- HDI / UHDI materials
- 3D-Printing materials
- Signal Integrity
- IPC standards

DEMONSTRATION:

- Demonstrate stackup and PDN design

WEEK 4: MANUFACTURING

Manufacturing techniques and trade-offs.

Key Concepts include:

- HDI Manufacturing
- 3D printing manufacturing
- Trade-offs
- IPC Standards

DEMONSTRATION:

- Create manufacturing files

WEEK 5 – DOCUMENTATION

Produce proper documentation in compliance with IPC standards for these designs

Key concepts include:

- IPC-2610 series.
- IPC-D-325.
- IPC-J-STD-001
- Space addendum
- Documentation methodology
- Special feature call-outs

INDIVIDUAL ASSIGNMENT:

- Demonstrate key technology documentation requirements

WEEK 6: CONTENT REVIEW AND FINAL EXAM

The class session will focus on content review and the final exam. Session 1 will be a review, there will be no live session 2. The final exam will be available for the remainder of the week.

FINAL EXAM:

- Complete final exam during the defined exam time during the last week of the course.
- Completion of the program with a score of 70% or higher on the final exam and/or final project is required to earn a certificate of completion.
- Attempts allowed: 2. Grading method: Highest grade.