



Introduction to PCB Design II

SYLLABUS

PROGRAM DESCRIPTION

In the highly competitive electronics industry, the knowledge, and skills of staff directly 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 impact time to market. The IPC PCB Fundamentals Course II is designed to provide the skills necessary to create PCB/PBA design that accurately implement the design intent, create all necessary design rules, and comply with all necessary IPC standards. Taught by an IPC-certified industry expert with 25+ years of experience in the field, the eight-week program utilizes interactive webinars, on-demand recorded class sessions, job-specific exercises, and team projects to facilitate mastery of the key concepts required by circuit board designers.

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:

- Define and create DRC rules to the appropriate IPC Standards
- Create Rigid and Rigid-Flex boards to meet a wide variety of applications
- Implement industry best practices for:
 - Manufacturability
 - Reliability
 - Documentation
 - Manufacturing file generation
- Recognize the trade-offs between the different layout types and when to use each type
- Assess different component types and attachment methods
 - Differentiate when to use each type of component
- Define standard PCB and CCA notes
 - Apply best practices in negotiating these terms with customers

COURSE STRUCTURE

- Instructor 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 a PCB design authoring software program of their choice. If participants do not have access to PCB design authoring software, IPC will provide complimentary access to a select choice of programs.
- Course materials are accessible 24/7 on the new IPC 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-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-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-SM-840 QUALIFICATION AND PERFORMANCE SPECIFICATION OF PERMANENT SOLDER MASK AND FLEXIBLE COVER MATERIALS
- IPC-CC-830 QUALIFICATION AND PERFORMANCE OF ELECTRICAL INSULATING COMPOUND FOR PRINTED WIRING ASSEMBLIES
- IPC-D-325 DOCUMENTATION REQUIREMENTS FOR PRINTED BOARDS

COURSE SCHEDULE

WEEK 1 – BASIC PCB LAYOUT

Program overview outlining class schedule and options for accessing class material and assignments. Session 1 will focus on the basics of PCB layout, including rigid through-hole boards and analog vs. digital designs. Session 2 will focus on basic placement and routing strategies for single and double-sided designs.

ASSIGNMENT:

- No assignments in week 1

WEEK 2 – ADVANCED PCB LAYOUT

Advanced concepts in PCB layout, including multi-layer routing, stackup design, impedance control, planar capacitance, sheet resistance, and design rules.

Introduction of the project that will be created over the duration of the course.

INDIVIDUAL ASSIGNMENT:

- Use project from IPC PCB Fundamentals 1 to define net classes, import parts into PCB file, and define DRC rules.
 - Complete by Week 3 Session 2

WEEK 3 – PCB MATERIALS

Materials used to build a PCB. Key concepts include:

- Mechanical and electrical properties
- Resins and foils
- Solder mask
- Legend
- IPC standards



INDIVIDUAL ASSIGNMENT:

- Place key components in the board and pre-route rooms for repeated circuits.
 - Complete by Week 4, Session 2

WEEK 4 – SIGNAL INTEGRITY

Signal integrity concepts, including:

- Overshoot / undershoot
- Cross-talk
- Rise time / fall time
- Transmission lines - introduction
- Signal propagation

INDIVIDUAL ASSIGNMENT:

- Place all remaining circuits / repeated channels on board.
 - Complete by Week 5, Session 2

WEEK 5 – TRANSMISSION LINES

Transmission lines, EM fields, and EMI/RFI reduction techniques. Key concepts include:

- What flows in the trace
- How to define a transmission line
- Distributed model of a Transmission line
- Emi / RFI and reduction techniques

INDIVIDUAL ASSIGNMENT:

- None

WEEK 6 – FLEX CIRCUITS

Flex-Circuit design. Session 1 will be a review of the differences between the requirements for rigid boards vs flex boards. Session 2 will be a simple project on how to define a flex design.

INDIVIDUAL ASSIGNMENTS:

- Complete design project package
 - Submit by Week 7, Session 2

WEEK 7 – DOCUMENTATION

Proper documentation of PCB's and CCA's. Review application of IPC-26xx standards. Key concepts include:

- Use of standard title blocks.
- Standard page sequencing.
- Readability
- Standard notes
- ECO and revision history
- Annotation methodology
- Design rules
- Separate Parts lists
- Assembly drawing
- PCB drawing
- Dimensioning
- Call-out / flag-notes

INDIVIDUAL ASSIGNMENT:

- Format and generate documentation and manufacturing packages for both PCB and CCA
 - Complete by Week 8, Session 2

WEEK 8 – CONTENT REVIEW AND FINAL EXAM

Class session will focus on content review, submission of final project, and final exam. Session 1 will be review. Session 2 will be final exam.

INDIVIDUAL ASSIGNMENTS:

- Complete design project package and documentation
 - Submit by Week 8, Session 2

FINAL EXAM:

- Complete final exam during Session 2
- 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.