IPC PCB Design for Radio Frequency Boards
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 Design for RF Boards Course is designed to provide the skills necessary to create PCB/PBA designs that will be used in high-speed analog, RF, and microwave frequencies to comply with all necessary IPC standards. Taught by an IPC-certified industry expert with 25+ years of experience in the field, the six-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.

This course is ideal for designers, engineers, technicians, and other individuals who want to acquire or increase their ability to meet the design, manufacturing, and routing challenges posed by designs that employ high-speed analog, radio (RF), and microwave frequencies. It is recommended that participants complete the PCB Fundamentals courses (I and II), OR be familiar with:

- Schematic symbol creation in accordance with (IAW) IPC-2612-1
- Schematic Generation IAW IPC-2612
- Documentation and Dimensioning IAW IPC-2614, IPC-2615, & IPC-D-325
- Standard Rigid Printed Board Design IAW IPC-2221 & IPC-2222
- Printed Board manufacturing IAW IPC-6011 & IPC-6012
- Printed Board Assembly IAW IPC-J-STD-001
- Basics of Signal Integrity

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:

- Design boards for high-speed analog and RF / microwave frequencies.
- Understand the trade-offs in materials used in these applications.
- Define a board stackup that implements structures that will meet needs of these designs.
- Understand and mitigate signal integrity issues for these designs.
- Understand and define effects of mechanical retention needs for these applications.
- Define and implement the tighter manufacturing tolerances needed for these designs.
- Understand the use of mixed material stackups in these designs.
- Understand the documentation requirements for these designs.

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-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-CC-830 QUALIFICATION AND PERFORMANCE OF ELECTRICAL INSULATING COMPOUND FOR PRINTED WIRING ASSEMBLIES

- IPC-SM-840 QUALIFICATION AND PERFORMANCE SPECIFICATION OF PERMANENT SOLDER MASK AND FLEXIBLE COVER MATERIALS

- IPC J-STD-003 SOLDERABILITY TESTS FOR PRINTED 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

- COMPONENTS

- IPC-6018 QUALIFICATION AND PERFORMANCE SPECIFICATION FOR PRINTED BOARDS CONTAINING EMBEDDED ACTIVE AND PASSIVE CIRCUITRY

- IPC-2141 DESIGN GUIDE FOR HIGH-SPEED CONTROLLED IMPEDANCE CIRCUIT BOARDS

- IPC-2251 DESIGN GUIDE FOR THE PACKAGING OF HIGH-SPEED ELECTRONIC CIRCUIT

- IPC-2252 DESIGN GUIDE FOR RF/MICROWAVE CIRCUIT BOARDS

- IPC-CM-770 COMPONENT MOUNTING GUIDELINES FOR PRINTED BOARDS

COURSE SCHEDULE

WEEK 1 – BASIC CIRCUIT DESIGN

Program overview outlining class schedule and options for accessing class material and assignments. Class session will focus on basics of RF design.

Key concepts include:

- RF Signals
- Stackup management
- Basic routing

Resins and foils **ASSIGNMENT:**

- Design stackup for simple board
    - Complete by Week 2 Session 2

WEEK 2 – ADVANCED CIRCUIT DESIGN

Advanced RF design.

Key concepts include:

- Advanced RF signal concepts
    - Insertion loss
    - Skin effect
    - Wave guides
- Advanced Routing concepts
    - Guard bands / picket fences
    - Antennae
- IPC standards

INDIVIDUAL ASSIGNMENT:

- Design a board for an antenna
- Define stackup / stackup zones.
    - Complete by Week 3 Session 2

WEEK 3 – MATERIALS

Materials used in RF designs. Physical properties. Types of structures, trade-offs

Key concepts include:

- What materials are used in these designs
- How to define
- Physical and mechanical properties
- Trade-offs in different material types
- IPC standards

INDIVIDUAL ASSIGNMENT:

- Define board multiple RF layers for a communications application.
    o Complete by Week 4, Session 2

WEEK 4 – MANUFACTURING PROCESS

RF manufacturing process. Effects on design.

Key Concepts include:

- Understand the manufacturing process used in these designs
- Understand special fabrication allowances for space designs
- Understand assembly challenges and mitigation with these designs
- Extended qualifications
- Tailored test and certification

INDIVIDUAL ASSIGNMENT:

- Design board for an RF application using specialty material requiring tailored assembly process.
    o Complete by Week 5, Session 2

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
- Documentation methodology
- Special feature call outs

INDIVIDUAL ASSIGNMENT:

- Create documentation package
- Define all special requirements described in class session
    - Submit by Week 6, Session 2

WEEK 6 – CONTENT REVIEW AND FINAL EXAM

Class session will focus on content review and final exam. Session 1 will be review. Session 2 will be final exam.

INDIVIDUAL ASSIGNMENTS:

- none

FINAL EXAM:

- Complete final exam during Session 2 or a 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.