PCB Design for Manufacturability
SYLLABUS

INSTRUCTOR INFORMATION

Instructor: Dana Korf

Email: dana.korf@korf.com

Phone: 408-643-1144

Best time to call: Generally available between 8am – 9pm pacific time USA.
Leave message anytime.

PROGRAM DESCRIPTION

Taught by an industry expert with more than 40 years of experience in the field, this 3-week online program is designed to provide the knowledge and skills necessary to reduce or eliminate design, documentation, and capability issues when transferring to manufacturing.

LEARNING AND PERFORMANCE OBJECTIVES

The IPC PCB Design for Manufacturing (DfM) course is designed to provide participants with the knowledge and skills necessary to understand and avoid common PCB questions that are raised when the completed PCB design documentation package is sent to the fabricator for production.

Upon completion, participants will be able to:

- Identify and correct documentation issues that stop a design from being manufactured
- Review participant real world issues that have been raised by their fabricators.
- Identify fabricator capabilities that impact layout features such as pad-to-hole relationships, solder mask and legend placement.
- Identify how conventional rigid, flexible, rigid-flexible and HDI board requirements differ
- Understand key factors and technical tradeoffs that can negatively affect the PCB cost, quality, and delivery time.
- Improve PCB acceptance specifications and how to create a company specific specification that will improve the PCB quality
- Improve how to use material specifications and manufacturing capability to create perfect material stackups, controlled impedance traces, and low loss traces.
- Identify proper fabrication notes content and use.
- Understand how to use an intelligent data format to reduce the amount of documentation that is commonly being generated and how this reduces documentation conflicts.

COURSE STRUCTURE

- Participants are encouraged to send example non-confidential DfM issues that have been noted in fabricator Deviation Requests or Technical Queries.
- Instructor and participants meet online twice per week from the comfort of their own home or office
- Participants can view recorded online sessions to review course content and class discussions
- Participants apply key concepts to resolve real-world issues each week
- 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 (INCLUDED)

- IPC-A-600 Acceptability of Printed Circuit Boards
- IPC-6012 Qualification and Performance Specification for Rigid Printed Boards
- IPC-6013 Qualification and Performance Specification for Flexible/Rigid-Flexible Printed Boards
- Rigid PCB Acceptance Specification - Dana Korf; Korf Consultancy LLC (provided with course)
- PCB Fabrication Design Guideline – Dana Korf; Korf Consultancy LLC (provided with course)

COURSE SCHEDULE

WEEK 1:

Lecture 1: DfM Course Introduction, Fabricator Capability and Materials
Lecture 2: Panelization, Stackups and Surface Finishes

We will begin Lecture 1 with an overview of the class schedule, options for accessing class material, and course assignments. Then, we will focus on the basic steps involved in the PCB manufacturing process. Lecture 2 will cover protocols for effective troubleshooting, the process of creating circuits within the layers of a printed circuit board (e.g., surface prep, etching, resist stripping), and the defects that can occur during this process.

Lecture 1:

- What is a DfM?
    - Design requirements
    - Manufacturing requirements
- PCB fabricator capability
    - Capability matrix versus design rules
    - How to merge multiple supplier requirements into a single set of rules
- Materials

- Rigid and HDI materials
- FPC and Rigid Flex materials
- How to interpret laminate and prepreg data sheets
- How to not over or under specify materials

Assignment:

- Review internal designs for material issues that have been identified by participant suppliers that were not addressed in the lecture.
- Submit issues for review at the next lecture for discussion.

Lecture 2:

- Panelization
  - How panel utilization affects cost
  - Relationship between PCB size/shape and PCB production panel
  - Difference between an assembly panel (array) and the PCB production panel
  - Coupon placement optimization
- Stackups
  - Prepreg styles and resin percentage
  - Core materials
  - PCB thickness changes during manufacturing
  - Thickness tolerances
  - Why Dk values are different between the data sheet and production
  - Construction types
  - Hybrid constructions
  - Documentation requirements
- Surface Finishes
  - Surface Finish specifications and use
  - Common multiple finish combinations
  - Documentation requirements

Assignment:

- Review internal designs for stackup, panelization and surface finish issues that have been identified by participant suppliers that were not addressed in the lecture.
- Submit issues for review at the next lecture for discussion.

WEEK 2:

Lecture 3: Conductive Features, Holes/Vias and Other Mechanical Features
Lecture 4: Masks and Inks, Impedance and Signal Loss, and Electrical Test

Lecture 3 will focus on laminates and issues that arise as a result of the lamination and drilling processes. In Lecture 4 we will explore defects that stem from the metallization of vias and circuit patterns.

Lecture 3:

- Conductive feature design rules
    - Outer layer conductor width and spacing
    - Inner layer conductor width and spacing
    - Copper thickness values and tolerances
    - Plating thickness impact on line width and spacing
    - Common DfM issues
- Mechanical and laser drill holes
    - Standard mechanical drill bit sizes
    - Aspect Ratio
    - Plated hole types, e.g., plater and non-plated holes
    - Buried and blind via types
    - Finished hole size tolerances
    - Design rules and impact on cost and reliability

Assignment:

- Review internal designs conductive features, vias and other mechanical feature issues that have been identified by participant suppliers that were not addressed in the lecture.
- Submit issues for review at the next lecture for discussion.

Lecture 4:

Lecture 4 will focus on two areas that tend to have the most DfM issues. They are solder mask and signal impacting properties such as impedance and signal loss.

- Masks and Inks
    - Solder mask design rules
    - Solder mask defined pads
    - Via tenting types and use
    - Via filling
    - Plating over vias (VIPPO)
    - Legend ink common DfM issues
- Impedance
    - Good documentation examples
    - Dk value determination
    - Impedance tolerance
    - Via impedance factors
- Signal Loss
    - Copper roughness impact
    - Back drilling
    - Signal skew mitigation techniques
- Electrical Test
    - Continuity and Isolation testing
    - Dielectric Withstanding Voltage (Hipot) testing

- Passive values testing
- Reliability testing

Assignment:

- Review internal design solder mask, impedance, signal loss and other electrical issues that have been identified by participant suppliers that were not addressed in the lecture.
- Submit issues for review at the next lecture for discussion.

WEEK 3:

Lecture 5: FPC, Documentation and Specifications

Lecture 5 will focus on solutions for common flexible (FPC) and rigid-flex PCB DfM issues. There are issues that are particular to FPC's and many that are common to rigid boards. We will also discuss how to properly document a design and how to effectively create and use acceptance specifications.

We will then wrap-up with an overall summary and open-up the discussion for final issues that the participant may have that have not been addressed.

Lecture 5:

- FPC design
  - FPC specific materials
  - FPC Construction types
  - Rigid-flex construction types
- Documentation
  - How stake holders affect the documentation quality
  - Intelligent data, e.g., IPC-2581, versus non-intelligent data
  - Complete documentation package content
  - Fabrication print notes
  - Order of Precedence
- Specifications
  - Regulatory
  - Company specific acceptance specification
  - IPC specifications
  - Global approved edit document
- DfM Feedback (Technical Query) format
- Course Wrap-up and Review