CID-Certified Interconnect Designer

What you will learn

Design is the heart of the electronics manufacturing process. A vision is defined, and the designer makes that vision a reality. Design phase establishes true product cost, reliability, and performance characteristics.
The IPC Designer Certication program can help students meet their goals: designs that can be consistently manufactured in the designated supply chain with the minimum number of defects, and operate according to their intended use.

Chapter 1 – Design Considerations

1.1 Considerations for Design

- Develop a design checklist, build a design team, identify stakeholders
- Three performance class designations
- Producibility levels
- Rigid board classifications

1.2 Placement and Routing Techniques

- Component placement and routing factors
- Electrical functionality effect on placement
- Analog and digital placement
- Mechanical constraints on electrical functionality, placement, and routability
- Return paths in layer stack-ups Conductor geometries and planes

1.3 Functional Electrical Characteristics

- Ohm’s law
- Logic pins
- Electromagnetic interference

1.4 Copper Clad Laminates

- Resin types
- Types of reinforcement
- Conductor formation, core, foil, and stackup
- Bow and twist

1.5 Holes in Printed Boards

- Creating a plated through hole
- Annular ring
- Calculate land size
- Land-to-hole relationship
- Supported and unsupported holes
- Through holes, through vias, blind vias, micro vias and buried vias

1.6 Drilling and Hole Locations

- Explain the need for tolerances on size and location of a hole pattern
- Bilateral tolerance
- MMC and LMC

1.7 Features Formed in Copper

- Subtractive process of copper
- Functional and non-functional lands and land shapes
- Holes for lands and thermal relief concepts

Chapter 2 – Thermal, Reliability and Testing

2.1 Thermal Management for Boards

- Cooling methods at the bare board level
- Using thermal planes
- Managing thermal characteristics with thermal vias
- High heat components and board stress

2.2 Thermal Management for Assemblies

- CTE and its impact on reliability
- Components with high thermal ratings
- Methods used to dissipate component heat
- Thermal cycling: bare board and assembly reliability

2.3 Reliability - Terms and Design Issues

- Mean Time Between Failure (MTBF)
- HAST
- Coupons

2.4 Testing

- Lands and vias as test points
- UUT (Unit Under Test)
- Functional testing: in-circuit test, clam shell, flying probe, BIST, boundary scan

Chapter 3 – Physical Board Principles

3.1 Printed Board and Assembly Viewing Principles

- Primary side, secondary side and datum planes on printed boards
- Layer assignment concepts on printed boards
- Conductive and non-conductive layers on printed boards
- Legends

3.2 Datum Dimensioning

- Datum features
- Primary, secondary and tertiary datum planes
- True positioning techniques

3.3 Grid Systems

- Grid system
- Grid relationship to component placement and conductor routing
- Grid choice impact on test
- Grid location
- Metric vs. inch grid system

3.4 Tooling Holes and Fiducials

- Tooling hole use - on or off the board
- Tooling hole dimensioning
- Relationship of board to panel tooling holes
- MMC and LMC as applied to plated or unsupported holes
- Multiple levels of fiducials on an assembly panel

3.5 Board and Assembly Panelisation

- Common panel size configuration

3.6 Panel Separation Methods

- Common panel separation techniques

Chapter 4 – Component Types

4.1 Component Overview

- Electrical and physical representation of a component
- Identify the six types of components
- Reference designators
- Discrete and integrated components
- Through hole and surface mount packaging
- Right angle connectors
- Schematic/Logic Symbols

Define the symbol for a three input NAND gate
When is a connector a “P” or “J” designator?

- Legend and Polarity Markings

Differences between cathode and anode
Capacitor marking for polarity distinction

4.2 Edge board Connectors

- Describe edge board connectors
- Through-hole vs. surface mount connectors
- Right-angle connector attachment

4.3 Stiffeners

- Physical and electrical clearance requirements on and off the board
- Benefits of a stiffener

4.4 Bus Bars

- Primary use and characteristics of a bus bar

4.5 Sockets

- Identify the differences between a DIP device and a chip carrier device
- Component profile impact on routing, assembly, and CTE

4.6 Jumpers and Terminals

- Jumper wires
- Differences between jumpers and haywires

4.7 MELF

- MELF technology – pros and cons
- MELF packages and electrical functions
- Attachment techniques

4.8 Eyelets

- Use of eyelets

Chapter 5 – Component & Assembly Issues

5.1 Designing for Assembly

- Identify joining systems for through-hole and surface mount components
- Cost advantages of uniform part orientation for pick-and-place
- Types of assemblies
- Component clearance and placement issues
- Clearance requirements for attachment techniques
- Preferred part orientation

5.2 Mounting Through-Hole Components

- Axial leaded through-hole devices
- Radial leaded devices

5.3 Mounting Surface Mount Components

- Features of a healthy solder joint

5.4 Component Modifications

- Coining
- Modifying an SMT to a TH (through-hole) assembly installation, and vice versa

5.5 Component Insertion and Attachment Techniques

- Describe hole size differences for manual vs. auto-insertion

5.6 Solder Processes

- Different solder methods used for SMT and through-hole
- Pin in Paste process
- Understand the importance of component orientation on the solder process

5.7 Clinched and Un-clinched Leads

- When to use un-clinched vs. clinched leads

Chapter 6 – Board Surface Treatments

6.1 Solder Mask

- Documentation requirements of solder mask used on bare boards
- Types and uses of solder mask
- Solder mask impact on surface mount assembly
- SMOBC
- Dendritic growth
- Via tenting
- Clearance considerations for bare board coating

6.2 Conformal Coating

- Understand the purpose of conformal coating

6.3 Protective Coatings/Surface Finishes

- Documentation requirements for protective coating of bare boards
- Describe the physical aspects of the different tarnish coatings
- Identify the different surface finishes
- Pros and cons of each surface finish

6.4 Legend

- Relationship of the legend to components on the PCB
- Polarity and pin markings on ICs, capacitors, diodes, and connectors
- Cost implications of legend

Chapter 7 – Documentation and Dimensioning

7.1 Documentation and classifications

- Documentation used to define a printed board assembly
- Two sets of classification
- Three grades of documentation
- Levels of completeness
- Use of the Documentation Package

7.2 Basic Drawing Formats

- Common drawing size formats used in USA and Europe
- Drawing size ratio
- Format zones and their uses

Chapter 8 – Schematic and Logic Diagrams

- **Schematic and logic diagrams**
- Hierarchical and flat schematics - the “flow” of a schematic or logic diagram
- Identify and describe the use of different line weights
- Define a phantom line on a schematic

Chapter 9 – Fabrication and Tolerancing Requirements

9.1 Board Fabrication Documentation

- Notes to be included on the fabrication (master) drawing
- Describe the final fabrication data set
- Relationship to datum reference for hole pattern

9.2 Dimensioning and Tolerancing

- Advantages of geometric tolerancing

Chapter 10 – Assembly Documentation and BOMs

- Use of bar codes on the PCB
- Documents that define clearance and attachment requirements
- Describe the BOM or Parts List
- Firmware - when it should be included in the assembly documentation
- Documentation requirements for coating of bare boards

For more information about IPC Training please contact us on ipc-cid@quadrasol.co.uk or 01254 301 888

quadra
solutions