

IPC-2231A

2021 - August

DFX Guidelines

Supersedes IPC-2231

April 2019

An international standard developed by IPC



BUILD ELECTRONICS BETTER



Documents

Document Sharing Hub

The Principles of Standardization

In May 1995 the IPC's Technical Activities Executive Committee (TAEC) adopted Principles of Standardization as a guiding principle of IPC's standardization efforts.

Standards Should:

- Show relationship to Design for Manufacturability (DFM) and Design for the Environment (DFE)
- Minimize time to market
- Contain simple (simplified) language
- Just include spec information
- Focus on end product performance
- Include a feedback system on use and problems for future improvement

Standards Should Not:

- Inhibit innovation
- Increase time-to-market
- Keep people out
- Increase cycle time
- Tell you how to make something
- Contain anything that cannot be defended with data

Notice

IPC Standards and Publications are designed to serve the public interest through eliminating misunderstandings between manufacturers and purchasers, facilitating interchangeability and improvement of products, and assisting the purchaser in selecting and obtaining with minimum delay the proper product for his particular need. Existence of such Standards and Publications shall not in any respect preclude any member or nonmember of IPC from manufacturing or selling products not conforming to such Standards and Publication, nor shall the existence of such Standards and Publications preclude their voluntary use by those other than IPC members, whether the standard is to be used either domestically or internationally.

Recommended Standards and Publications are adopted by IPC without regard to whether their adoption may involve patents on articles, materials, or processes. By such action, IPC does not assume any liability to any patent owner, nor do they assume any obligation whatever to parties adopting the Recommended Standard or Publication. Users are also wholly responsible for protecting themselves against all claims of liabilities for patent infringement.

IPC Position Statement on Specification Revision Change

It is the position of IPC's Technical Activities Executive Committee that the use and implementation of IPC publications is voluntary and is part of a relationship entered into by customer and supplier. When an IPC publication is updated and a new revision is published, it is the opinion of the TAEC that the use of the new revision as part of an existing relationship is not automatic unless required by the contract. The TAEC recommends the use of the latest revision. Adopted October 6, 1998

Why is there a charge for this document?

Your purchase of this document contributes to the ongoing development of new and updated industry standards and publications. Standards allow manufacturers, customers, and suppliers to understand one another better. Standards allow manufacturers greater efficiencies when they can set up their processes to meet industry standards, allowing them to offer their customers lower costs.

IPC spends hundreds of thousands of dollars annually to support IPC's volunteers in the standards and publications development process. There are many rounds of drafts sent out for review and the committees spend hundreds of hours in review and development. IPC's staff attends and participates in committee activities, typesets and circulates document drafts, and follows all necessary procedures to qualify for ANSI approval.

IPC's membership dues have been kept low to allow as many companies as possible to participate. Therefore, the standards and publications revenue is necessary to complement dues revenue. The price schedule offers a 50% discount to IPC members. If your company buys IPC standards and publications, why not take advantage of this and the many other benefits of IPC membership as well? For more information on membership in IPC, please visit www.ipc.org or call 847/597-2872.

Thank you for your continued support.



IPC-2231A

DFX Guidelines

Developed by the 1-14 DFX Subcommittee of the 1-10 Printed Board Design Committee of IPC

Supersedes:
IPC-2231, April 2019

Users of this publication are encouraged to participate in the development of future revisions.

Contact:

IPC
3000 Lakeside Drive, Suite 105N
Bannockburn, Illinois
60015-1249
Tel 847 615.7100
Fax 847 615.7105

This Page Intentionally Left Blank

Acknowledgment

Any document involving a complex technology draws material from a vast number of sources across many continents. While the principal members of the 1-14 DFX Subcommittee of the 1-10 Printed Board Design Committee are shown below, it is not possible to include all of those who assisted in the evolution of this standard. To each of them, the members of the IPC extend their gratitude.

1-10 Printed Board Design Committee

Chair:
Philip Henault
Raytheon

Vice-Chair:
Michelle Gleason
Plexus Corporation

1-14 DFX Subcommittee

Co-Chairs:
Karen McConnell
Northrop Grumman Corporation

Stephen Golemme
JITX Inc.

Dock Brown
ANSYS Inc.

Technical Liaison of the IPC Board of Directors

Bob Neves
Microtek (Changzhou) Laboratories

DFX Subcommittee

Edward Acheson
Cadence Design Systems Inc.

Benny Barbero
Allied Telesis Inc.

Jasbir Bath
Koki Solder America

Scott Bowles
Lockheed Martin Corporation

Randy Bremner
Northrop Grumman Corporation

Dock Brown
ANSYS Inc.

Jonathan Bruer
Creation Technologies Inc.

Vanja Bukva
Teledyne Dalsa

Zhiman Chen
Zhuzhu CRRC Times
Electric Co., Ltd.

Sean Clancy
HzO Inc.

Joe Clark
Lockheed Martin Missiles
& Fire Control

Michael Collier
Teledyne Leeman Labs

Michael Creedon
EPTAC Corporation

Don Dupriest
Lockheed Martin Missiles
& Fire Control

Peter Fernandez
Facebook

Gary Ferrari
EPTAC Corporation

Darrell Freiwald
Northrop Grumman Systems
Corporation

Sarah Friscano
L3Harris Communications

Daniel Gamota
Jabil Circuit, Inc.

Michelle Gleason
Plexus Corporation

Stephen Golemme
JITX Inc.

Jay Gorajia
Siemens Digital Industries Software

Arnaud Grivon
Thales Global Service

David Hager
BAE Systems

Thomas Hausherr
PCB Libraries

Eddie Hofer
Collins Aerospace

Terry Hoffman
Cisco Systems Inc.

Happy Holden

Jennie Hwang
H-Technologies Group

Kyle Johnson
Lockheed Martin Missiles
& Fire Control

Kathy Johnson

Russell Kaunas
Collins Aerospace

Dana Korf
Korf Consultancy LLC

Kevin Kusiak
Lockheed Martin Corporation

Dale Lee
Plexus Corporation

Murilo Levy Casotti
Embraer SA

Howard Liu
Vayo (Shanghai)
Technology Co., Ltd.

Herry Magera
Motorola Solutions

Chris Mahanna
Robisan Laboratory Inc.

Karen McConnell
Northrop Grumman Corporation

Daniel McCormick
Naval Surface Warfare Center

Kristopher Moyer
California State University
Sacramento

Ron Nowak
Lockheed Martin Mission Systems
& Training



Documents

Document Sharing Hub

Russell Nowland Nokia	Jose Rios Raytheon	Rainer Taube Taube Electronic GmbH
Deogenes Papelera Fresenius Kabi	Thomas Romont IFTEC	Theodore Tontis Rockwell Automation
Jan Pedersen Elmatica	Robert Rowland Axiom Electronics, LLC	Cheryl Tulkoff
Binh Pham Viasat Inc.	Steven Roy Roy Design and Manufacturing Service	Louis Ungar Advanced Test Engineering Solutions, Inc.
Patrick Phillips Northrop Grumman Innovation Systems	Henry Sanftleben Aptiv	Jesse Vaughan ACDI – American Computer Development, Inc.
James Pierce Axiom Electronics LLC	Karl Sauter Oracle America, Inc.	Pietro Vergine Leading Edge
Micah Pledger	Hemant Shah	Steve Vetter Naval Surface Warfare Center
Ray Prasad Prasad Consultancy Group	Chris Shaw Fujitsu Network Communications	Scott Vorhies Space Exploration Technologies
Robert Priore Cisco Systems Inc.	Cameron Shearon Raytheon	Linda Woody LWC Consulting
Anthony Rafanelli Raytheon Company	Russell Steiner Casco Products	Paul Zutter U.S. Army Aviation & Missile Command
Bill Reed Ciena Corporation	Geok And Tan DSO National Laboratories	
Edward Rios Motorola Solutions		

Table of Contents

1	SCOPE	1	5.5.4	Outputs	9
1.1	Purpose	1	5.6	Product Validation	9
1.2	Goals of This Document	1	5.6.1	Overview	9
1.3	Limitations of This Document	1	5.6.2	Inputs	9
2	APPLICABLE DOCUMENTS	1	5.6.3	Tasks	9
2.1	IPC	1	5.6.4	Outputs	9
2.2	AE International	2	5.7	Manufacturing Validation	10
2.3	IEEE	2	5.7.1	Overview	10
2.4	JEDEC	2	5.7.2	Inputs	10
2.5	Government	2	5.7.3	Tasks	10
2.6	Industry Standards and Guidelines	2	5.7.4	Outputs	10
3	TERMS AND DEFINITIONS	3	5.8	Support	10
4	OVERVIEW OF DESIGN FOR EXCELLENCE (DFX) PRACTICES	3	5.8.1	Overview	10
4.1	Design for Manufacturing (DFM)	3	6	FABRICATION PROCESS	10
4.2	Printed Board Design for Fabrication (DFF)	4	6.1	Materials	10
4.3	Design for Assembly (DFA)	4	6.1.1	Solder Mask	10
4.4	Design for Test/Testability (DFT)	5	6.1.2	Cores / Pre-pregs	11
4.5	Design for Cost (DFC)	5	6.1.3	Legend	11
4.6	Design for Reliability (DFR)	5	6.1.4	Metal Foils	11
4.7	Design for Environment (DFE)	5	6.1.5	Surface Finishes	11
4.8	Design for Reuse	5	6.2	Fabrication Process	11
5	DESIGN PROCESS for PRINTED BOARD ASSEMBLY	6	6.2.1	Front-End Engineering	11
5.1	Overview	6	6.2.2	Registration	12
5.2	Concept Design and Analysis	6	6.2.3	Plating	12
5.2.1	Overview	6	6.2.4	Lamination	12
5.2.2	Inputs	7	6.2.5	Mechanical Drilling	12
5.2.3	Tasks	7	6.2.6	Laser Drilling	12
5.2.4	Outputs	7	6.2.7	Back Drilling	12
5.3	Detailed Design	7	6.2.8	Legend Application	13
5.3.1	Overview	7	6.2.9	Via Protection	13
5.3.2	Inputs	7	6.2.10	Copper Thieving	13
5.3.3	Tasks	8	6.3	Panel Configuration	13
5.3.4	Outputs	8	7	ASSEMBLY PROCESS	13
5.4	Design Release	8	7.1	Scope	13
5.4.1	Overview	8	7.2	Solder Mask	13
5.4.2	Inputs	8	7.3	Components	13
5.4.3	Tasks	8	7.4	Panel/Array	13
5.4.4	Outputs	9	7.5	Clearance Component	14
5.5	First Build Support	9	7.6	Component Placement/Mounting	14
5.5.1	Overview	9	7.7	Pad Outgassing	15
5.5.2	Inputs	9	7.8	Fiducials	15
5.5.3	Tasks	9	7.9	Tooling	15
			7.10	Pb-Free Solder Assembly Temperatures	15
			7.10.1	Copper Flooding and Thermal Relief	15



7.10.2	Heat Sensitive Component	15	8.8.3	Printed Board Layout	22
7.10.3	Assembly Effects on Printed Board Assembly Materials	16	8.9	DFT Guidelines and Assessment	22
7.10.4	Moisture Sensitivity	16	8.9.1	Design for Testability Metrics	22
7.11	Surface Mount Paste Printing	16	8.10	References	23
7.12	Machine Soldering	16	8.10.1	IEEE Standards for DFT	23
7.12.1	Single Point Automated Soldering Systems	16	8.10.2	Industry Standards and Guidelines	23
7.12.2	Reflow Soldering	16	8.10.3	Military Standards and Guidelines	23
7.12.3	Convection Reflow	16	8.10.4	Commercially Available DFT Guidelines	23
7.12.4	Vapor Phase Reflow	17	9	COST	24
7.12.5	Pin in Paste/Paste in Hole/Intrusive Soldering/Solder Preforms	17	9.1	Bill of Material (BOM) Costs	24
7.12.6	Wave Soldering	17	9.2	Printed Board and Assembly Costs	24
7.12.7	Selective Soldering/Point-to-Point Soldering	17	9.3	Sustaining Costs	24
7.12.8	Point-to-Point Soldering	17	9.4	Test Costs	25
7.12.9	Selective Dip Soldering	17	9.5	Rework and Repair Costs	25
7.13	Laser Soldering	17	10	DESIGN FOR RELIABILITY	25
7.14	Manual/Rework Soldering	17	10.1	Reliability Physics /Physics of Failure	25
7.14.1	Soldering Iron	17	10.2	Solder Joint Reliability	26
7.14.2	Vacuum Tipped Soldering Iron	18	10.3	Tin Whisker Risk Considerations	26
7.14.3	Hand Hot Air/Gas Soldering	18	10.4	CTE of Common Materials (CTE)	26
7.14.4	BGA/Area Array Rework	18	10.5	Comparative Reliability Matrix	27
7.14.5	Mini-Pot (Solder Fountain) Wave Rework	18	11	DESIGN FOR THE ENVIRONMENT	29
7.15	Cleaning	18	11.1	Sustainability	29
7.16	Underfill	18	11.2	Materials	29
7.17	Conformal Coating	19	11.3	Regulatory	29
7.18	Design Guidelines for System Enclosures/Box Builds	19	11.3.1	End-of-Life	29
8	TESTABILITY (DFT)	20	11.3.2	Life Cycle Assessments	29
8.1	Overview	20	12	DESIGN FOR REUSE	30
8.2	Concept Design and Analysis	20	12.1	Designing for Reuse within an Organization	30
8.2.1	Activities	20	12.2	Designing for Reuse by Third Parties	30
8.2.2	Description	20	13	CHECK LISTS COMPARISONS	31
8.3	Detailed Design	20	14	ACRONYMS	48
8.3.1	Activities	20			
8.3.2	Description	20			
8.4	Design Release	21			
8.4.1	Activities	21			
8.5	First Build Support	21			
8.5.1	Activities	21			
8.5.2	Description	21			
8.6	Validation	21			
8.6.1	Activities	21			
8.7	Support	21			
8.7.1	Activities	21			
8.7.2	Description	21			
8.8	DFT Considerations	21			
8.8.1	Parts Selection	22			
8.8.2	Test Strategy Formulation	22			

Figures

Figure 3-1	Plated-Through-Hole (PTH)	3
Figure 5-1	Process Flow	6
Table 10-1	Cyclic Services Environment over Design Life	27
Figure 10-1	Through-Hole Solder Joint	28
Figure 10-2	Surface Mount Discrete Solder Joint	28
Figure 10-3	Surface Mount J-Lead Joints	28
Figure 10-4	Surface Mount BGA Solder Joints	28

Table

Table 10-1	Cyclic Services Environment over Design Life	27
------------	--	----

Design for Excellence (DFX) Guidelines

1 SCOPE

This document provides guidelines for establishing a best practice methodology for use in developing a formal DFX (Design for Manufacturing, Fabrication, Assembly, Testability, Cost, Reliability, Environment, Reuse) process for layout of printed board assemblies that utilize surface mount and through hole devices.

1.1 Purpose The document provides a DFX process framework to establish a discipline of design review necessary to perform a detailed analysis of manufacturability attributes commonly found in electronics hardware for fabrication and around which to model a printed board assembly.

1.2 Goals of This Document The goals of this document are to:

- Use a multi-discipline engineering assessment tactic on elements influencing DFX.
- Allow the user to establish standardized DFX checklist(s) for major design elements such as bare printed board fabrication, printed board assembly manufacturing, electrical testability, and elements influencing product reliability, reuse, and impact on environment.

1.3 Limitations of This Document Electronics hardware defined under this DFX review process is limited to features of influence on DFX for bare printed board and printed board assembly.

2 APPLICABLE DOCUMENTS

2.1 IPC¹

J-STD-001 Requirements for Soldered Electrical and Electronic Assemblies

IPC-T-50 Terms and Definitions for Interconnecting and Packaging Electronic Circuits

IPC-CH-65 Guidelines for Cleaning of Printed Boards and Assemblies

IPC-D-279 Design Guidelines for Reliable Surface Mount Technology Printed Board Assemblies

IPC-D-325 Documentation Requirements for Printed Boards, Assemblies and Support Drawings

IPC-A-610 Acceptability of Electronic Assemblies

IPC-A-630 Acceptability Standard for Manufacture, Inspection, and Testing of Electronic Enclosures

IPC-SM-785 Guidelines for Accelerated Reliability Test of Surface Mount Solder Attachments

IPC-CC-830 Qualification and Performance of Electrical Insulating Compound for Printed Wiring Assemblies

IPC-2221 Generic Standard on Printed Board Design

IPC-2222 Sectional Design Standard for Rigid Organic Printed Boards

IPC-2223 Sectional Design Standard for Flexible Printed Boards

IPC-2224 Sectional Standard for Design of PWBs for PC Cards

IPC-2225 Sectional Design Standard for Organic Multichip Modules (MCM-L) and MCM-L Assemblies

IPC-2226 Sectional Design Standard for High Density Interconnect (HDI) Printed Boards

IPC-2581 Generic Requirements for Printed Board Assembly Products Manufacturing Description Data and Transfer Methodology

IPC-2615 Printed Board Dimensions and Tolerances

IPC-4761 Design Guide for Protection of Printed Board Via Structures

IPC-6012 Qualification and Performance Specification for Rigid Printed Boards

IPC-7351 Generic Requirements for Surface Mount Design and Land Pattern Standard

IPC-7525 Guidelines for Stencil Design

¹ www.ipc.org

IPC-7711/7721 Rework, Modification and Repair of Electronic Assemblies

IPC-7801 Reflow Oven Process Control Standard

IPC-9701 Performance Test Methods and Qualification Requirements for Surface Mount Solder Attachments

2.2 SAE International²

GEIA-STD-0005-1 Performance Standard for Aerospace and High Performance Electronic Systems Containing Lead-free Solder

GEIA-STD-0005-2 Standard for Mitigating the Effects of Tin Whiskers in Aerospace and High Performance Electronic Systems

2.3 IEEE³

IEEE Std 1500(TM)/IEC 62528 Standard Testability Method for Embedded Core-based Integrated Circuits

IEEE 1149.1 IEEE Standard for Test Access Port and Boundary-Scan Architecture

IEEE 1149.4 IEEE Standard for a Mixed-Signal Test Bus

IEEE 1149.5 IEEE Standard for Module Test and Maintenance Bus (MTM-Bus) Protocol

IEEE 1149.6 IEEE Standard for Boundary-Scan Testing of Advanced Digital Networks

IEEE 1149.7 IEEE Standard for Reduced-Pin and Enhanced-Functionality Test Access Port and Boundary-Scan Architecture

IEEE 1149.8.1 IEEE Standard for Boundary-Scan-Based Stimulus of Interconnections to Passive and/or Active Components

IEEE 1232 IEEE Standard for Artificial Intelligence Exchange and Service Tie to All Test Environments (AI-ESTATE)

IEEE 1450– IEEE Standard Test Interface Language (STIL) for Digital Test Vector Data

IEEE 1500 IEEE Standard Testability Method for Embedded Core-based Integrated Circuits

IEEE 1522 IEEE Standard for Testability and Diagnosability Characteristics and Metrics

IEEE 1532 IEEE Standard for In-System Configuration of Programmable Devices

IEEE 1581 IEEE Standard for Static Component Interconnection Test Protocol and Architecture

IEEE 1687 IEEE Standard for Access and Control of Instrumentation Embedded within a Semiconductor Device

IEEE 62525 (IEEE Std 1450) International Standard Test Interface Language (STIL) for Digital Test Vector Data

2.4 JEDEC⁴

JESD-B106-D Test Method B106C Resistance to Soldering Temperature For Through-Hole Mounted Devices

JESD22-A111 Evaluation Procedure for Determining Capability to Bottom Side Board Attach by Full Body Solder Immersion of Small Surface Mount Solid State Devices

2.5 Government

MIL-A-28870 General Specification for Assemblies, Electrical Backplane, Printed Wiring

MIL-P-50884 General Specification for Printed Wiring, Flexible and Rigid Flex

MIL-DTL-31000 Technical Data Packages

MIL-HDBK-454 General Guidelines for Electronic Equipment

MIL-STD-2119 Design Requirements for Printed-Wiring Electrical Backplane Assemblies

MIL-HDBK-2175 Department of Defense Handbook: Testability Handbook for Systems and Equipments (31-Jul-1995)
[Supersedes MIL-STD-2175A]

2.6 Industry Standards and Guidelines

SMTA/TMAG Testability Guidelines 101E

² www.sae.org

³ www.ieee.org

⁴ www.jedec.org

3 TERMS AND DEFINITIONS

Terms and definitions used herein are in accordance with IPC-T-50 except as otherwise specified.

Note: Any definition denoted with an asterisk (*) is a reprint of the term defined in IPC-T-50.

***Assembly** A number of parts, subassemblies or combinations thereof joined together. (Note: This term can be used in conjunction with other terms listed herein, e.g., “Printed Board Assembly.”)

***Automatic Test Equipment** Equipment that automatically analyzes functional or static parameters in order to evaluate performance.

Contract Data Requirements List A list in the RFP (Request for Proposal) specifying the amount and kind of deliverable data to be submitted to the Customer. This list covers a variety of data, such as schedules, test reports, drawings and associated lists, contract work breakdown structure (WBS), and various Program plans (e.g., PA, configuration management, safety, and reliability). The Customer may require that some data be submitted with the proposal; the rest of the data is delivered at specified times during the execution of the Program.

Nodal A point in a network or diagram at which lines or pathways intersect or branch. (from Google Search - Dictionary)

***Plated-Through Hole (PTH)/Plated Hole** A hole with plating on its walls that makes an electrical connection between conductive patterns on internal layers, external layer, or both, of a printed board. (See Figure 1-3).

***Printed Board** A composite structure incorporating point-to-point interconnections for electronic circuits. It may include embedded components. (This includes single-sided, double-sided, multilayer, rigid, rigid-flex, and flex constructions.)

***Printed Board Assembly** The assembly that uses a printed board for component mounting and interconnecting purposes.

***Registration** The degree of conformity of the position of a pattern (or portion thereof), a layer, a hole, or other feature to its intended position on a product.

Reliability Physics/Physics of Failure A technique under the practice of Design for Reliability that leverages the knowledge and understanding of the processes and mechanisms that induce failure to predict reliability and improve product performance.

Statement of Work A formal document that captures and defines the work activities, deliverables, and timeline a vendor must execute in performance of specified work for a client.

Validation Confirmation by examination and provision of objective evidence that the particular requirements for a specific intended use can be consistently fulfilled.

Verification Confirmation by examination and provision of objective evidence that specified requirements have been fulfilled.

4 OVERVIEW OF DESIGN FOR EXCELLENCE (DFX) PRACTICES

Design for Excellence (DFX) includes many disciplines. Modifications to accommodate DFX needs are less costly when addressed early in the design phase.

This document expands on the existing definition of performing DFX.

DFX is more than just checking for errors. It is a series of decisions about potential downstream problems made during the design phase with full knowledge of impact, cost, quality needs, and schedule requirements. The earlier in the design cycle problems or trade-offs are found, the least costly modifications are to implement.

4.1 Design for Manufacturing (DFM) The DFM scope covers the Design for Fabrication (DFF) and Design for Assembly (DFA). Many DFM guides use the word manufacturing interchangeably with their specific function (fabrication, assembly).

Design for manufacturing is a process of optimizing a design to ensure that it can be built with highest quality at the best cost. Decisions made during every stage of design effect the manufacturability of a product. Manufacturing checks should be done on the CAM data files, such as BOM, approved manufacturer list (AML), approved vendor list (AVL), netlist, IPC-2581, ODB++, or Gerber / Drill and Route / Pick and Place, in the Documentation Package and not on the CAD data.

For more detailed information, refer to IPC-D-325 and IPC-261x series.

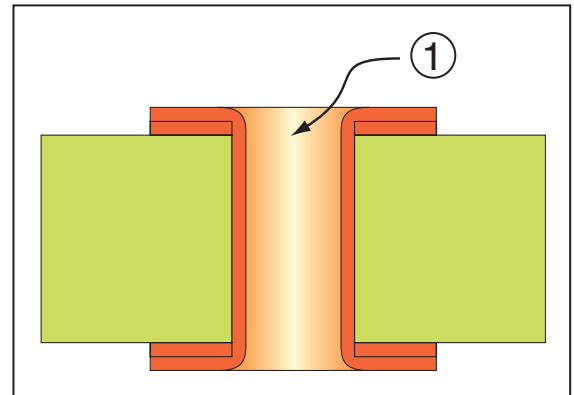


Figure 3-1 Plated-Through-Hole (PTH)

1. Plated-Through-Hole

In addition to checking the data files, all drawings in the Documentation Package should be reviewed for accuracy and completeness. A best practice is to generate notes for each board via a program or copied from a repository of approved notes.

Checks performed by the ECAD design tool are not necessarily the same as the checks used to check the manufacturability. DFM tools check post processed data for errors specific to the manufacturing process.

Make sure all documents supporting the product are reviewed by the manufacturer performing the DFM prior to release of the design for manufacturing. Notify the manufacturer performing the DFM to use the supplied documentation package and notes and to modify the DFM output based on any issues already resolved by the supplied documents. The manufacturer performing the DFM should provide the results in an easy to read and understand format as agreed between user and supplier. Defects should be prioritized from Critical to Preferred.

4.2 Printed Board Design for Fabrication (DFF) Due to the existence of published design for fabrication guides, only an overview section is provided in this document. Please refer to IPC-222x series of standards for detailed design guides. IPC-2221 Generic Standard on Printed Board Design covers all fabrications. Specific types of fabrications are covered in IPC-2222 (rigid), IPC-2223 (flex), IPC-2224 (PCMCIA), IPC-2225 (MCM-L), IPC-2226 (HDI). For inorganic substrates, and board types not covered in the detailed design guides above, consult the fabricator's design guide and expertise.

Work with your fabricator(s) and manufacturer(s) to match DFF guidelines to supplier capability and project requirements. Be sure to consider material availability and process differences when establishing design rules for printed boards that may be manufactured in different regions (Europe, Asia, America, other regions). Some of the bigger items to consider are:

- Board type
- Laminate material
- Symmetry of stackup
- Copper density balance
- Trace width and spacing
- Complexity of via connections
- Incorporation of embedded components
- Incorporation of printed components
- Mixed Dielectrics
- Signal Integrity
- Thermal Management
- Panel Utilization
- Regional material and process differences
- Make sure fab meets other DFX requirements

4.3 Design for Assembly (DFA) When DFM guides discuss placement clearances, they are defining DFA criteria. It is important to establish the requirements of assembly as part of the board requirements. Often component spacing rules are reduced to fit more packages on the printed board. These rules define the space needed by automated component insertion machine. Violation of the rules may force a costlier manual assembly of the board. The decision on placement rules needs to be made before the printed board is released for fabrication. Work with your assembler to establish good DFA guidelines.

- Elimination of hand soldering or wave soldering when possible
- Component spacing
- What are the size of components and complexity of packaging limitations, if any
- Orientation of components to wave solder
- Shadowing during wave solder
- Solder Pallet Clearance
- Appropriate dimensions and spacing for PTHs and bond pads
- Attachment methods
- Moisture sensitivity level (MSL)
- Cleanliness
- Inspection
- Component thermal mass and placement on assembly
- Spacing under component to allow cleaning if required

Make sure all documents supporting the product are submitted to the manufacturer performing the DFA. Notify the manufacturer to review the drawings and notes on the Assembly drawing, Fabrication Drawing, and/or Fabrication text file. Notify the manufacturer performing the DFA to use the supplied drawing and notes and modify the DFA output based on any issues already resolved by the supplied documents.

Not having this performed leads to duplication of effort and reporting of errors that do not actually exist. Pictures of the potential defect should be listed with an x/y coordinate of the defect based on the zero coordinate of the supplied data file.

Any defects should be prioritized from Critical to Preferred. A header must include at a minimum the part number of the board, date performed, Company name and contact information, and company that requested the data.

4.4 Design for Test/Testability (DFT) DFT is the name given to design techniques that added testability features to an electronic hardware product design. The premise of the added features is that they make it easier to develop and apply manufacturing tests for the designed hardware. The purpose of manufacturing tests is to validate that the product hardware contains no defects that could adversely affect the product's correct functioning.

Make sure all documents supporting the product are submitted to the manufacturer performing the DFT. Notify the manufacturer to review the drawings and notes on the Assembly drawing, Fabrication Drawing, and/or Fabrication text file. If required, Boundary scan data should be provided prior to having the DFT performed. Notify the manufacturer performing the DFT to use the supplied drawing and notes and modify the DFT output based on any issues already resolved by the supplied documents. Not having this performed leads to duplication of effort and reporting of errors that do not actually exist. Pictures of a potential defect should be listed with an x/y coordinate of the defect based on the zero coordinate of the supplied data file. At a minimum, a header must include the part number of the board, date performed, Company name and contact information, and company that requested the data.

Raw data should never be submitted as results because it is un-readable by the majority of the design team.

4.5 Design for Cost (DFC) Cost of a product is determined by either marketplace demands or bids for specific design tasks. Cost goals can be realized by treating cost as an independent design requirement that must be met during all phases of product development. Many military/aerospace companies refer to this practice as CAIV (Cost As an Independent Variable) and include cost analysis at all design reviews.

4.6 Design for Reliability (DFR) Design for Reliability (DFR) is the process for weaving the reliability engineering into the total development cycle. DFR takes the entire life cycle of the product into consideration when making decisions during the design phase. Some of the questions that DFR answers are:

- Where is the product to be used?
- How long must the product work without a failure?
- What are the consequences of failure?
- How is the product to be used?
- What environment is the product intended to withstand?

4.7 Design for Environment (DFE) Design for Environment was developed in a concerted effort to deal with legislation concerning the hazardous materials used in producing electronic products and the disposal of the product at end of its lifecycle.

Some of the legislation that Design for Environment deals with:

- European Union: The Restriction of Hazardous Substances Directive (ROHS)
- European Union: Waste Electrical and Electronic Equipment Directive (WEEE)
- United States: Conflict Minerals Law

In addition, the United States Environmental Protection Agency (EPA) created the Design for the Environment Program (DFE) to prevent pollution and its risks to humans and the environment. The effects of material and processes must be considered in all phases of a circuit card assembly's life cycle.

4.8 Design for Reuse Reuse can take two forms: within an organization and by third parties. Reuse has the benefits of getting more people to use and improve the design for cost and quality. Design elements that can be reused include specifications, block diagrams, interfaces, test plans, test setups, firmware, software, and routing.

5 DESIGN PROCESS for PRINTED BOARD ASSEMBLY

5.1 Overview The Printed Board Assembly process is intended to guide the engineering development of Printed Board Assemblies by identifying inputs, tasks, and outputs required to perform design. It represents the collection of design best practices. It applies to development, prototype and production programs and is applicable to all electronic assemblies from consumer level to industrial to medical to avionics/military to space applications.

The documentation of this process may be tailored as appropriate for each individual project or company with engineering management concurrence. The level of tailoring and associated risks should be defined at the start of the program.

Critical to the success of the design process is an emphasis on generating an accurate specification, implementing best practices, design for excellence (DFX), and conducting thorough design reviews at the conclusion of each design phase.

The process is divided into six major phases:

- Concept Design and Analysis
- Detailed Design
- Design Release
- First Build Support
- Product Validation
- Manufacturing Validation

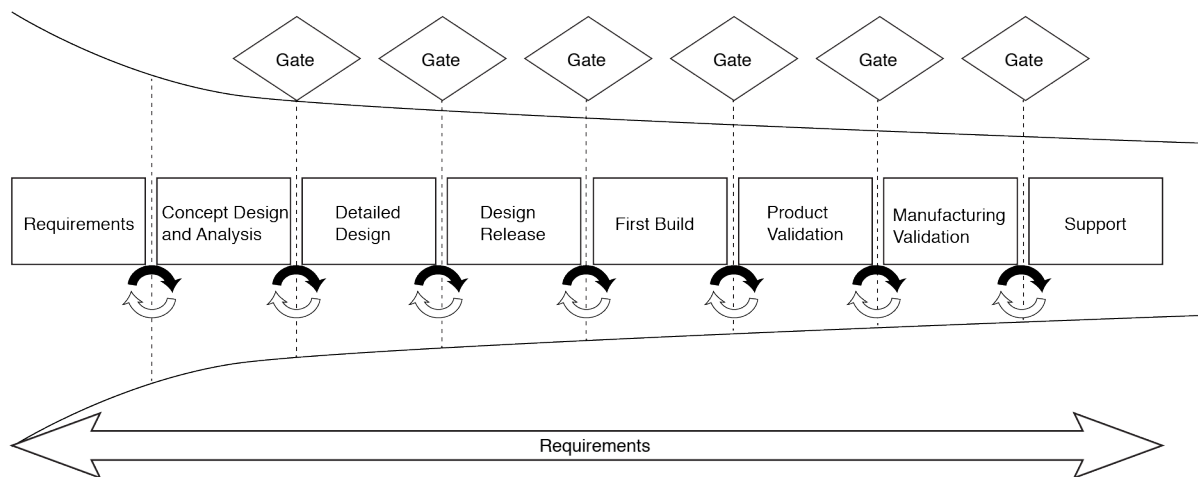


Figure 5-1 Process Flow

Note 1: Requirements development are outside the scope of this document and assumed to be completed at the beginning of this process.

5.2 Concept Design and Analysis

5.2.1 Overview

The Concept Development Phase evaluates the proposed concept against the requirements to determine the feasibility based on the established design for excellence (DFX) principles.

The Concept Development Phase confirms appropriate resources, looks at ways to compress the schedule, and establishes the requirements that will drive the design effort. This phase should be completed prior to commencing the time consuming design phases to assure that all of the major requirements are analyzed, documented, reviewed and that bottlenecks are reduced or eliminated.

The key outputs of the Concept Development Phase is completing preliminary documentation of program-unique requirements of the printed board assembly level in the requirements specification, the schedule, and appropriate resource requirements. Detail should be sufficient to convey all of the requirements at the level of understanding by the design engineer. Successful completion of the requirements document in adequate detail defining the requirements analysis and functional allocation of the design should be accomplished by a disciplined and timely concept design review.

5.2.2 Inputs

- Product size
- Product function
- Product environment
- Product regulatory requirements
- Product max cost per unit at a quantity
- Product lifecycle
- Product Budget
- Product Schedule

5.2.3 Tasks

- Review requirements and all applicable regulations and standards
- Review all hardware systems requirements
- Determine component requirements including the use of COTS or custom devices
- Determine the expected life cycle of the product and determine what steps in design and component choice are needed to assure the product meets life cycle requirements
- Assess additional resource requirements and ways to compress the schedule
- Build and iterate mockups with placement of key components
- Discuss and establish preliminary design verification plan, qualification test plan, manufacturing validation plan
- Start to draft an engineering specification that notes special technologies, hardware requirements for software, and system constraints

5.2.4 Outputs

- Block diagram with critical components
- Consensus on critical materials, components, and packages
- Placement strategy and preliminary manufacturing strategy
- Allowable board outline area, mechanical dimensions, and interconnects
- Draft design verification plan
- Draft qualification test plan
- Draft manufacturing validation plan (fabrication, assembly, test, etc.)
- Identification of resources needed and when to produce, verify, and qualify the printed board
- Identification of supply chains, logistics, and supply chain options.
- Dependency identification
- Risk Info/Mitigation Plans
- Support Area Agreements

5.3 Detailed Design

5.3.1 Overview

The Detailed Design Phase completes the detailed test plans and procedures, completes part selection and schematic design, establishes the printed board requirements, and accomplishes the place and route efforts for printed board assembly. Tools, artwork, and data files are developed for printed board assembly. A Pre-Release Design Review should be conducted, and the documentation package prepared for use by design, manufacturing and test.

The documentation package consists of items needed to produce and assemble the PCB as well as any additional information required by the design verification, qualification test, manufacturing, validation, and lifecycle plans.

For further information on what should be in the documentation, refer to the IPC-2610 series, the ASME Y14 series, and relevant internal company standards.

5.3.2 Inputs

- Block diagram with critical components
- Consensus on critical materials, components, and packages
- Placement strategy and preliminary manufacturing strategy
- Allowable board outline area, mechanical dimensions, and interconnects
- Draft design verification plan
- Draft qualification test plan
- Draft manufacturing validation plan (fabrication, assembly, test, etc.)
- Identification of resources needed and when to produce, verify, and qualify board
- Dependency identification
- Risk Info/Mitigation Plans
- Support Area Agreements

5.3.3 Tasks

- Define stackup with impedance requirements
- Complete component selection and schematic design with any needed test elements
- Perform calculations, tolerance analysis, software investigations, and simulations on schematic and layout
- Develop sourcing agreements & contracts with key suppliers
- Hold in process reviews as needed, including a schematic review
- Finalize board profile, requirements, materials, placement, engineering bill of materials, layout
- Generate manufacturing data including any special process instructions and special tooling
- Run checks.
- Investigate substituting equivalent components to reduce the number of component placement technologies.

5.3.4 Outputs

- Schematics drawings
- Bill of Materials (BOM) with alternates and approved vendor list (AVL) with available quantities, lead times, second sourcing, and risks
- Mechanical drawings with critical dimensions and 3D CAD model
- Simulation results
- Printed board and printed board assembly manufacturing data (CAD manufacturing outputs)
- Documentation package including stackup and copper thickness, BOM, schematic drawings, pick and place files, fabrication and assembly test point files, digital model, and any other files required to support the manufacturing verification, qualification test, manufacturing, or lifecycle plans
- Sourcing agreements as needed
- Design verification
- Design verification plan
- Qualification test plan
- Manufacturing validation plan (fabrication, assembly, test, test requirements, etc.)
- Lifecycle plan, as required

5.4 Design Release

5.4.1 Overview The design release process ensures all the design steps were taken and the manufacturing documentation is clear and complete at the time the Printed Board Assembly arrives.

5.4.2 Inputs

- Schematics
- Bill of Materials (BOM)
- Mechanical drawings with critical dimensions and 3D CAD
- Printed board and printed board assembly manufacturing data
- Documentation package
- Design verification plan
- Qualification test plan
- Manufacturing validation plan (fabrication, assembly, test, etc.)
- Lifecycle plan, as required

5.4.3 Tasks

- Perform a review of documentation package and iterate design based on input of reviews
- Perform reviews of the design with respect to test plans
- Review disposition of results from automated and manual checks
- Release the data in a version control system
- Send documentation, any consigned materials, and purchase order(s) to suppliers
- Review and approve any proposed modifications from suppliers
- Verify resources needed to execute design manufacturing, manufacturing verification, and qualification test plans are available when needed

5.4.4 Outputs

- Released complete documentation package including manufacturing, verification, qualification production test, and lifecycle plans
- Consigned materials and purchase order(s) sent to suppliers

5.5 First Build Support

5.5.1 Overview The first build will uncover issues not caught prior to release and to get the boards on schedule. The goal of the support is to get the printed board assemblies on schedule, document and address each issue, improve the design and manufacturing processes, and provide feedback to the designers. These changes may be implemented by modifying printed board model Including potential updates to bill of material and netlist.

5.5.2 Inputs

- Release documentation package and plans for manufacturing, verification, qualification, and production test
- Consigned materials and purchase order sent to suppliers
- Supplier contacts

5.5.3 Tasks

- Execute the manufacturing validation plan aspects related to first build
- Resolve any manufacturability issues found during first build
- Track design through the manufacturing process. Record and address any issues that arise
- Implement corrective actions as required
- Perform a first article inspection on the printed board assemblies when they arrive to ensure they match production documentation. The first article inspection report should include feedback to the design team and feedback to the suppliers.
- Oversee fabrication of any special tools and fixtures

5.5.4 Outputs

- Engineering and manufacturing change orders
- Corrective actions for design and manufacturing
- First Article Inspection Report (FAIR)
- Special tools and fixtures

5.6 Product Validation

5.6.1 Overview This procedure defines the steps necessary to validate the design. This involves execution of the validation plan and qualification test plan. Implement corrective actions for each issue uncovered.

5.6.2 Inputs

- First Article Hardware
- Test Equipment and Software
- Design verification plan
- Qualification test plan

5.6.3 Tasks

- Execute the design verification plan
- Execute the qualification test plan
- Record findings and propose corrective actions
- Execute corrective actions, including design revisions if necessary

5.6.4 Outputs

- Engineering and manufacturing changes
- Tested Hardware (Chip, Card, Unit)
- Test Reports
- Lessons Learned

5.7 Manufacturing Validation

5.7.1 Overview This procedure defines the steps necessary to validate manufacturability and the manufacturing process. This involves execution of the manufacturing validation plan. Implement corrective actions for each issue uncovered.

5.7.2 Inputs

- First Article Hardware
- Test Equipment and Software
- Manufacturing validation plan
- Special tools and fixtures

5.7.3 Tasks

- Execute the manufacturing validation plan
- Record findings and propose corrective actions
- Execute corrective actions, including design revisions if necessary
- Implement the test plan

5.7.4 Outputs

- Manufacturing and engineering changes
- Test Reports
- Lessons Learned

5.8 Support

5.8.1 Overview Support begins after the product and manufacturing process is verified and extends through product life. The following are examples of the most commonly performed support:

- Implement manufacturing test plan
- Training and demonstrations
- Support to systems integration
- Identifying product enhancements
- Field installation and test
- Material Review Board (MRB) and hardware configuration
- Production problem analysis and resolution
- Change board activity
- Lifecycle plan, including End-of-Life (EOL)/Diminishing Source
- Component issues
- Trouble-shooting at locations
- Develop and execute cost reduction plans

Requests for support typically come from product test, manufacturing, field operations, process engineering, program management and systems engineering. Though the support activities vary from program to program, the task steps below are performed on every program.

6 FABRICATION PROCESS

This section details the processes in electronic board fabrication and provides guidelines as to how these processes are impacted by design choices. Designers should be familiar with these process steps as they impact directly on cost and reliability.

6.1 Materials

6.1.1 Solder Mask Solder mask aperture size and shape and solder mask opening distance to nearest trace along with soldermask application type (LPI vs. Dry film vs wet film) all impact fabrication. DFM checks of printed board assemblies containing fine pitch and very small components should include checking notes to allow solder mask modification AABUS.

The solder mask shapes, size, Thickness (both overall and under SMT components), and distance to nearest trace should include shift tolerances. Please refer to the appropriate standards in the IPC-6010 series for requirements. Note that various different vendors have different levels of capability to compensate for these effects. When working on designs that have outgassing limitations such as space applications care must be given to the selection of materials that meet the appropriate requirements. Some assembly processes need to be considered when designing the solder mask layer. The thickness of the solder mask needs to ensure that SMT parts can make contact with the solder pads during the solder process. Over thick soldermask can lead to