



ASSOCIATION CONNECTING
ELECTRONICS INDUSTRIES

IPC-2224

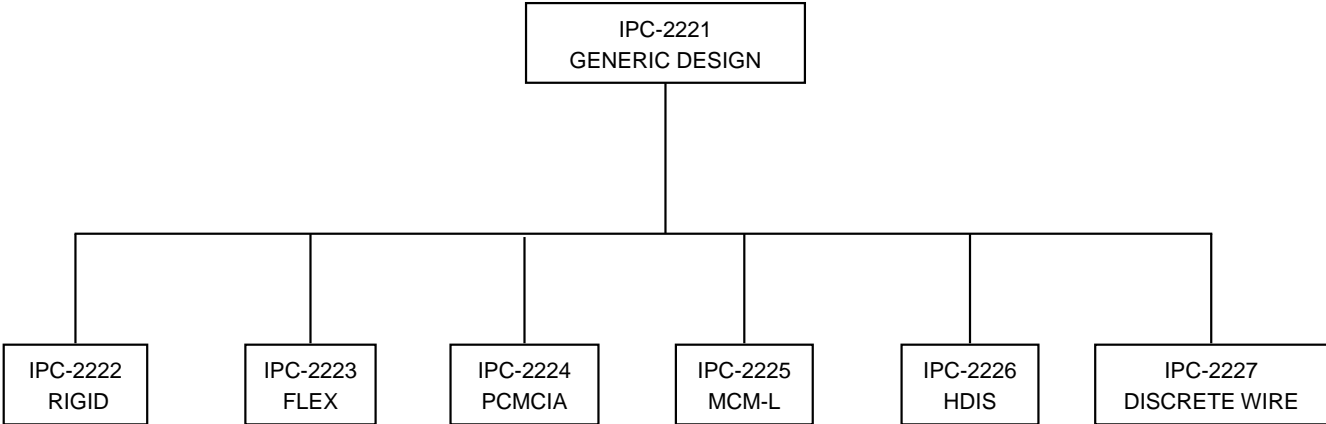
Sectional Standard for Design of PWBs for PC Cards

IPC-2224

January 1998

A standard developed by IPC

HIERARCHY OF IPC DESIGN SPECIFICATIONS
(2220 SERIES)



FOREWORD

This standard provides information on the generic requirements for PC Card printed board design. All aspects and details of the design requirements are addressed to the extent that they can be applied to the broad spectrum of those designs that use organic materials or organic materials in combination with inorganic materials (metal, glass, ceramic, etc.) to provide the structure for mounting and interconnecting electronic, electromechanical, and mechanical components.

The information contained herein is intended to be compared to the generic design requirements specified in IPC-2221. When coupled with the aforementioned inputs this comparison will provide a complete standard and facilitate the appropriate selection process of the materials, fabrication, and assembly technology necessary to meet the engineering design objectives.

IPC's documentation strategy is to provide distinct documents that focus on specific aspect of electronic packaging issues. In this regard document sets are used to provide the total information related to a particular electronic packaging topic. A document set is identified by a four digit number that ends in zero (0).

Included in the set is the generic information which is contained in the first document of the set and identified by the four digit set number. The generic standard is supplemented by one or multiple sectional documents, each of which provide specific focus on one aspect of the topic or the technology selected. The designer of the printed board, needs as a minimum, the generic, the sectional of the chosen technology and the engineering description of the final product.

As technology changes specific focus standards will be updated, or new focus standards added to the document set. The IPC invites input on the effectiveness of the documentation and encourages user response through completion of "Suggestions for Improvement" forms located at the end of each document.

Table of Contents

1.0 SCOPE	1	5.1 Fabrication Requirements.....	6
1.1 Purpose	1	5.1.1 Panel Design.....	7
1.2 Documentation Hierarchy	1	5.1.2 Fabrication Notes	7
1.3 Presentation.....	1	5.2 Product/Board Configuration	7
1.4 Interpretation.....	1	5.2.1 Board Geometries.....	7
1.5 Classification of Products.....	1	5.2.2 Bow and Twist.....	9
1.5.1 Board Types.....	1	5.2.3 Constraining Core Boards.....	9
2.0 APPLICABLE DOCUMENTS	1	5.2.4 Support.....	9
2.1 Reference Documents.....	1	5.3 Assembly Requirements.....	9
2.1.1 Institute for Interconnecting and Packaging Electronic Circuits (IPC)	1	5.3.1 Tooling Rails	9
2.1.2 Underwriter's Labs.....	2	5.4 Dimensioning Systems	9
2.1.3 PCMCIA/JEIDA.....	2	5.4.1 Cutouts and Notches	9
2.1.4 Joint Industry Standards.....	2	5.5 Structural Strength.....	10
3.0 GENERAL REQUIREMENTS	2	6.0 ELECTRICAL PROPERTIES	10
3.1 Test Requirement Considerations	2	7.0 THERMAL MANAGEMENT	10
3.1.1 Mechanical.....	2	7.1 Convection.....	10
4.0 MATERIALS	2	7.2 Heat Dissipation Considerations.....	10
4.1 Material Selection.....	2	7.2.1 Enclosed Housing.....	10
4.2 Bonding Material.....	2	7.2.2 Ventilated Housing	11
4.2.1 Flexible PC Cards (bond ply).....	2	8.0 COMPONENT AND ASSEMBLY ISSUES	11
4.2.2 Adhesive Films.....	2	8.1 General Placement Requirements	11
4.3 Laminate Materials.....	2	8.2 Plated-Through Hole Requirements.....	11
4.3.1 Rigid Laminates	2	8.3 Mounting Methods for Connectors.....	11
4.3.2 Epoxy Laminates.....	4	8.4 Stiffeners	11
4.3.3 High Performance Laminates.....	4	8.5 Fine Pitch SMT (Peripherals)	11
4.3.4 High Temperature Laminates.....	4	8.6 Array SMT (BGA, μ BGA, etc.).....	11
4.3.5 High Speed and Low Loss Laminates.....	4	8.7 Bare Die.....	11
4.3.6 Double-Clad Laminates.....	4	8.7.1 Wire Bond.....	11
4.3.7 Special Clad Materials	5	8.7.2 Flip Chip.....	11
4.4 Flexible Substrates	5	8.7.3 Chip Scale.....	11
4.4.1 Flexible Laminate Materials (Flexible Metal Clad Dielectrics)	5	8.8 Tape Automated Bonding.....	11
4.4.2 Coverlayer.....	5	8.9 Castellations.....	11
4.5 Conductive Materials	5	8.10 Bottom Only Terminations.....	11
4.5.1 Copper Foil/Film	5	9.0 HOLES/INTERCONNECTIONS	11
4.5.2 Other Foils/Film	6	9.1 General Requirements for Lands with Holes.....	11
4.5.3 Metal Core Substrates	6	9.1.1 Clearance Areas in Planes.....	12
4.5.4 Electronic Component Materials (Buried Resistors and Capacitors).....	6	9.1.2 Conductive Pattern Feature Location Tolerance	13
4.5.5 Conductive Dielectric Composites.....	6	9.2 Holes	13
4.6 Organic Protective Coatings	6	9.2.1 Quantity	13
4.7 Marking and Legends.....	6	9.2.2 Spacing of Adjacent Holes.....	13
4.8 Thickness Class Tolerance	6	9.2.3 Hole Pattern Variation	13
5.0 MECHANICAL/PHYSICAL PROPERTIES	6	9.2.4 Blind/Buried Vias	13
		9.2.5 Unsupported Holes	13

9.2.6	Minimum Hole Sizes for Plated-Through Hole Vias	13
9.2.7	Etchback	13
9.3	Drill Size Recommendations for Printed Boards	13
10.0	GENERAL CIRCUIT FEATURE REQUIREMENTS	13
10.1	Conductor Characteristics	13
10.1.1	Conductor Width and Thickness	15
10.1.2	Edge Spacing	15
10.1.3	Large Conductive Areas	15
10.1.4	Balanced Conductors	15
10.2	Land Characteristics	15
10.2.1	Lands for Interfacial Connection Vias	15
10.2.2	Vias in Lands	15
11.0	DOCUMENTATION	15
11.1	Artwork	15
11.1.1	Component Pedestal Artwork	15
12.0	QUALITY ASSURANCE	15

Tables

Table 4-1	Clad Laminate Maximum Operating Temperatures	3
Table 4-2	Guide to Laminate Thickness, mm	3
Table 5-1	Fabrication Capability Assessment	6
Table 5-2	Panel Size to Manufacturing Operation Relationships	6
Table 5-3	Substrate Dimensions, mm	7
Table 5-4	Maximum Interconnect Warpage Area	9
Table 5-5	Maximum PC Card Warpage (by type)	9
Table 5-6	Tolerance Cutouts, Notches, and Keying Slots, as Machined, mm	10
Table 9-1	Feature Location Tolerances (Lands, Conductor Pattern, etc.), mm	13
Table 9-2	Minimum Unsupported Holes Tolerance Range	13
Table 10-1	Typical Values to be Added or Subtracted for the Desired Nominal Conductor Width in Order to Arrive at the Nominal Width on the Production Master	16

Figures

Figure 4-1	Dielectric layer thickness measurement	4
Figure 5-1	Panel borders	8
Figure 8-1	Horizontal part mouting	12
Figure 8-2	Lead-in chamfer configurations	12
Figure 9-1	Clearance area in planes, mm	12
Figure 10-1	Etched Conductor Characteristics	14

Sectional Standard for Design of PWBs for PC Cards

1.0 SCOPE

This standard establishes the requirements for the design of printed boards for PC card form factors. The organic materials may be homogeneous, reinforced, or used in combination with inorganic materials; the interconnections may be single, double, or multilayered.

1.1 Purpose The requirements contained herein are intended to establish design principles and recommendations that **shall** be used in conjunction with IPC-2221 (see 1.2) to produce detailed designs intended to mount and attach passive and active components.

The components may be through-hole, surface mount, fine pitch, ultra-fine pitch, array mounting or unpackaged bare die. The materials may be any combination able to perform the physical, thermal, environmental, and electronic functions.

1.2 Documentation Hierarchy Document hierarchy **shall** be in accordance with the generic standard IPC-2221.

1.3 Presentation Presentation **shall** be in accordance with the generic standard IPC-2221.

1.4 Interpretation Interpretation **shall** be in accordance with the generic standard IPC-2221.

1.5 Classification of Products Classification of Products **shall** be in accordance with the generic standard IPC-2221 and as follows:

1.5.1 Board Types Board types are classified as:

- Type 1 — Single-sided printed board
- Type 2 — Double-sided printed board
- Type 3 — Multilayer board without blind or buried vias
- Type 4 — Multilayer board with blind and/or buried bias
- Type 5 — Multilayer metal core board without blind or buried vias
- Type 6 — Multilayer metal core board with blind and/or buried vias

2.0 APPLICABLE DOCUMENTS

The following documents form a mandatory part of this standard and all requirements stated therein apply, unless modified in the section in which they are invoked.

IPC-2221 Generic Standard on Printed Board Design

2.1 Reference Documents The following documents form a part of this document to the extent specified herein. If a conflict of requirements exist between IPC-2224 and those listed below, IPC-2224 takes precedence.

The revision of the document in effect at the time of solicitation **shall** take precedence.

2.1.1 Institute for Interconnecting and Packaging Electronic Circuits (IPC)¹

IPC-DD-135 Qualification for Deposited Organic Interlayer Dielectric Materials for Multichip Modules

IPC-MF-150 Metal Foil for Printed Wiring Applications

IPC-L-125 Specification for Plastic Substrates, Clad or Unclad, for High Speed/High Frequency Interconnections

IPC-FC-231 Flexible Bare Dielectrics for Use in Flexible Printed Wiring

IPC-FC-232 Adhesive Coated Dielectric Films for Use as Cover Sheets for Flexible Printed Wiring

IPC-FC-241 Flexible Metal-Clad Dielectrics for Use in Fabrication of Flexible Printed Wiring

IPC-D-300 Printed Board Dimensions and Tolerances

IPC-D-310 Guidelines for Phototool Generation and Measurement Techniques

IPC-D-322 Guidelines for Selecting Printed Wiring Board Sizes Using Standard Panel Sizes

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

IPC-TM-650 Test Methods Manual

Method 2.1.1 Microsectioning

Method 2.1.6 Thickness, Glass Fabric

IPC-SM-782 Surface Mount Design and Land Pattern Standard

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

IPC-MC-790 Guidelines for Multichip Module Technology Utilization

1. Institute for Interconnecting and Packaging Electronic Circuits, 2215 Sanders Road, Northbrook, IL 60062