



ASSOCIATION CONNECTING  
ELECTRONICS INDUSTRIES

---

# IPC-2224

## Sectional Standard for Design of PWBs for PC Cards

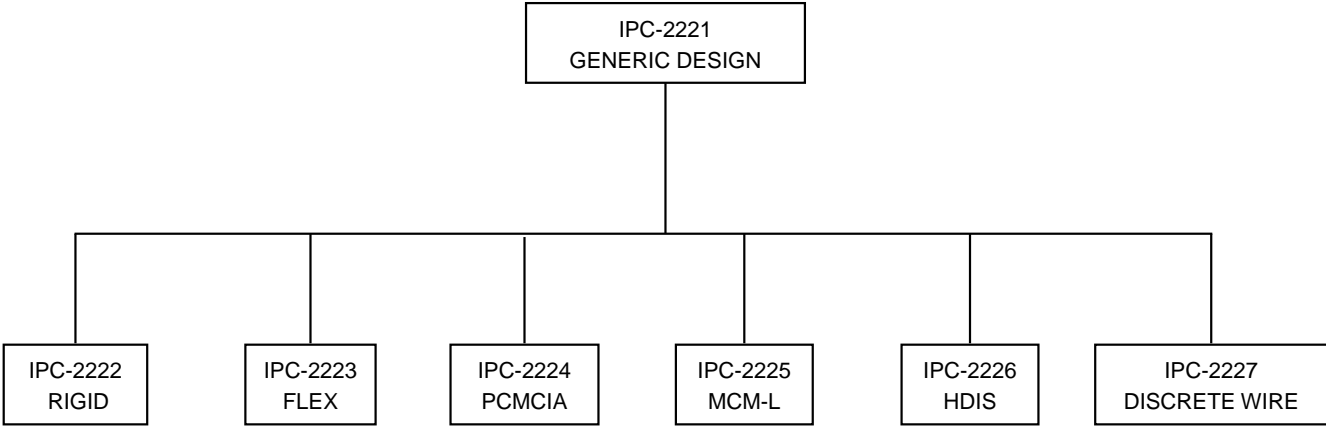
**IPC-2224**

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

<b>1.0 SCOPE</b> .....	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
<b>2.0 APPLICABLE DOCUMENTS</b> .....	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
<b>3.0 GENERAL REQUIREMENTS</b> .....	2	<b>6.0 ELECTRICAL PROPERTIES</b> .....	10	
3.1 Test Requirement Considerations .....	2	<b>7.0 THERMAL MANAGEMENT</b> .....	10	
3.1.1 Mechanical.....	2	7.1 Convection.....	10	
<b>4.0 MATERIALS</b> .....	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	<b>8.0 COMPONENT AND ASSEMBLY ISSUES</b> .....	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, $\mu$ 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	<b>9.0 HOLES/INTERCONNECTIONS</b> .....	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	
<b>5.0 MECHANICAL/PHYSICAL PROPERTIES</b> .....	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
<b>10.0</b>	<b>GENERAL CIRCUIT FEATURE REQUIREMENTS .....</b>	<b>13</b>
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
<b>11.0</b>	<b>DOCUMENTATION .....</b>	<b>15</b>
11.1	Artwork .....	15
11.1.1	Component Pedestal Artwork .....	15
<b>12.0</b>	<b>QUALITY ASSURANCE .....</b>	<b>15</b>

## 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)<sup>1</sup>

**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