IPC-2224

Sectional Standard for Design of PWBs for PC Cards
**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

1.1 Purpose .......................................................... 1
1.2 Documentation Hierarchy ...................................... 1
1.3 Presentation .................................................. 1
1.4 Interpretation .................................................. 1
1.5 Classification of Products .................................. 1
1.5.1 Board Types ............................................... 1

## 2.0 APPLICABLE DOCUMENTS

2.1 Reference Documents ......................................... 1
2.1.1 Institute for Interconnecting and Packaging Electronic Circuits (IPC) ........................................ 1
2.1.2 Underwriter’s Labs .......................................... 2
2.1.3 PC/IMCA/JEIDA ............................................ 2
2.1.4 Joint Industry Standards ................................... 2

## 3.0 GENERAL REQUIREMENTS

3.1 Test Requirement Considerations ............................. 2
3.1.1 Mechanical .................................................. 2

## 4.0 MATERIALS

4.1 Material Selection .............................................. 2
4.2 Bonding Material ............................................... 2
4.2.1 Flexible PC Cards (bond ply) ........................... 2
4.2.2 Adhesive Films .......................................... 2
4.3 Laminate Materials ............................................. 2
4.3.1 Rigid Laminates .......................................... 2
4.3.2 Epoxy Laminates ......................................... 4
4.3.3 High Performance Laminates ............................. 4
4.3.4 High Temperature Laminates ............................. 4
4.3.5 High Speed and Low Loss Laminates .................. 4
4.3.6 Double-Clad Laminates .................................. 4
4.3.7 Special Clad Materials .................................. 5
4.4 Flexible Substrates ............................................ 5
4.4.1 Flexible Laminate Materials (Flexible Metal Clad Dielectrics) .............................................. 5
4.4.2 Coverlayer .................................................. 5
4.5 Conductive Materials .......................................... 5
4.5.1 Copper Foil/Film ......................................... 5
4.5.2 Other Foils/Film .......................................... 6
4.5.3 Metal Core Substrates .................................... 6
4.5.4 Electronic Component Materials (Buried Resistors and Capacitors) ........................................ 6
4.5.5 Conductive Dielectric Composites ....................... 6
4.6 Organic Protective Coatings ................................ 6
4.7 Marking and Legends ......................................... 6
4.8 Thickness Class Tolerance .................................... 6

## 5.0 MECHANICAL/PHYSICAL PROPERTIES

5.1 Fabrication Requirements .................................... 6
5.1.1 Panel Design ............................................... 7
5.1.2 Fabrication Notes ......................................... 7
5.2 Product/Board Configuration ................................ 7
5.2.1 Board Geometrics .......................................... 7
5.2.2 Bow and Twist ............................................. 9
5.2.3 Confining Core Boards ................................... 9
5.2.4 Support .................................................... 9
5.3 Assembly Requirements ....................................... 9
5.3.1 Tooling Rails .............................................. 9
5.4 Dimensioning Systems ....................................... 9
5.4.1 Cutouts and Notches ..................................... 9
5.5 Structural Strength ........................................... 10

## 6.0 ELECTRICAL PROPERTIES

## 7.0 THERMAL MANAGEMENT

7.1 Convection .................................................... 10
7.2 Heat Dissipation Considerations ............................ 10
7.2.1 Enclosed Housing ......................................... 10
7.2.2 Ventilated Housing ........................................ 11

## 8.0 COMPONENT AND ASSEMBLY ISSUES

8.1 General Placement Requirements ........................... 11
8.2 Plated-Through Hole Requirements ........................ 11
8.3 Mounting Methods for Connectors .......................... 11
8.4 Stiffeners ..................................................... 11
8.5 Fine Pitch SMT (Peripherals) ................................ 11
8.6 Array SMT (BGA, μBGA, etc.) ............................... 11
8.7 Bare Die ........................................................ 11
8.7.1 Wire Bond .................................................. 11
8.7.2 Flip Chip ................................................... 11
8.7.3 Chip Scale .................................................. 11
8.8 Tape Automated Bonding .................................... 11
8.9 Castellations ................................................... 11
8.10 Bottom Only Terminations .................................. 11

## 9.0 HOLES/INTERCONNECTIONS

9.1 General Requirements for Lands with Holes ............. 11
9.1.1 Clearance Areas in Planes ................................ 12
9.1.2 Conductive Pattern Feature Location Tolerance ...... 13
9.2 Holes .......................................................... 13
9.2.1 Quantity .................................................... 13
9.2.2 Spacing of Adjacent Holes ............................... 13
9.2.3 Hole Pattern Variation ................................... 13
9.2.4 Blind/Buried Vias ......................................... 13
9.2.5 Unsupported Holes ........................................ 13

January 1998  IPC-2224
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 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 Reference Documents

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-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