ASSOCIATION CONNECTING
ELECTRONICS INDUSTRIES

IPC-2222

Sectional Design Standard for Rigid Organic Printed Boards

ANSI/IPC-2222

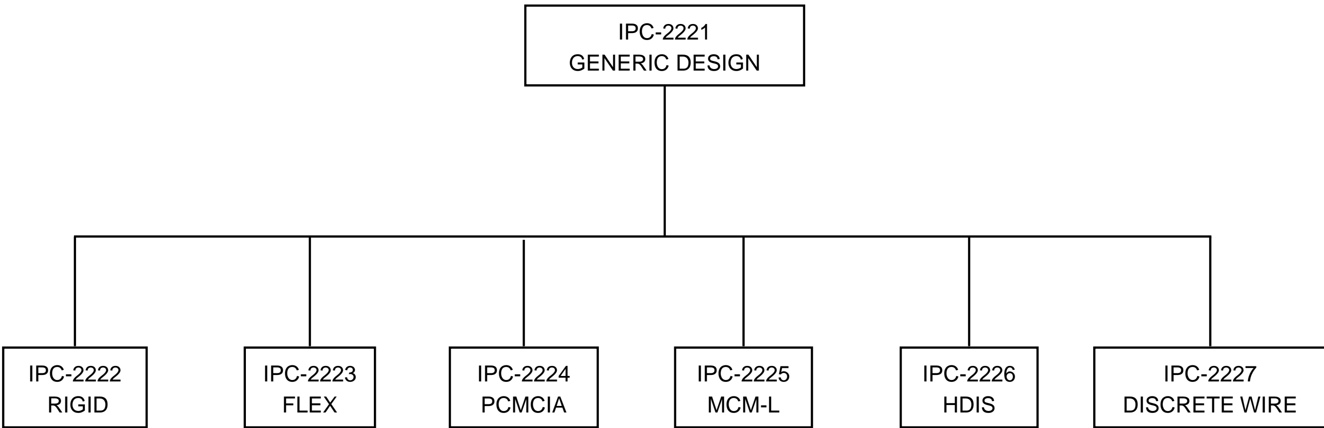
February 1998

A standard developed by IPC

Supersedes IPC-D-275
September 1991

2215 Sanders Road, Northbrook, IL 60062-6135
Tel. 847.509.9700 Fax 847.509.9798
www.ipc.org

HIERARCHY OF IPC DESIGN SPECIFICATIONS
(2220 SERIES)



FOREWORD

This standard is intended to provide information on the detailed requirements for organic rigid printed board design. All aspects and details of the design requirements are addressed to the extent that they can be applied to the unique requirements of those designs that use organic rigid (reinforced) 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 supplement generic engineering considerations and design requirements identified in IPC-2221. When coupled with the engineering design input, the complete disclosure should facilitate the appropriate selection process of the materials and the detailed organic rigid structure fabrication technology necessary to meet the engineering design objectives.

The selected component mounting and interconnecting technology for the printed board should be commensurate with the requirements provided and the specific focus of this sectional document.

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 many 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, the generic engineering considerations, and the engineering description of the final product.

Failure to have all information available prior to starting a design may result in a product that is difficult to manufacture or exceeds the cost predictions or expectations of the printed board.

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
1.1 Purpose ... 1
1.2 Document Hierarchy ... 1
1.3 Presentation ... 1
1.4 Interpretation ... 1
1.5 Classification of Products ... 1
1.5.1 Board Type ... 1
1.6 Assembly Types ... 1

2.0 APPLICABLE DOCUMENTS ... 1
2.1 Institute for Interconnecting and Packaging Electronic Circuits (IPC) ... 1
2.2 Underwriters Laboratories ... 3

3.0 GENERAL REQUIREMENTS ... 3
3.1 Performance Requirements ... 3

4.0 MATERIALS ... 3
4.1 Material Selection ... 3
4.2 Dielectric Base Materials (Including Prepregs and Adhesives) ... 3
4.2.1 Epoxy Laminates ... 3
4.2.2 High-Temperature Laminates ... 3
4.2.3 Special Clad Materials ... 3
4.2.4 Other Laminates ... 3
4.3 Laminate Materials ... 3
4.3.1 Measurement of Dielectric Thickness ... 3
4.3.2 Dielectric Thickness/Spacing ... 4
4.3.3 Laminate Properties ... 5
4.3.4 Prepreg ... 5
4.3.5 Single-Clad Laminates ... 5
4.3.6 Double-Clad Laminates ... 5
4.3.7 Laminate Material ... 5
4.4 Conductive Materials ... 13
4.5 Organic Protective Coatings ... 13
4.6 Markings and Legends ... 13

5.0 MECHANICAL/PHYSICAL PROPERTIES ... 13
5.1 Fabrication Requirements ... 13
5.2 Product/Board Configuration ... 13
5.2.1 Board Geometries ... 13
5.2.2 Support ... 13
5.3 Assembly Requirements ... 13
5.3.1 Assembly and Test ... 14
5.4 Dimensioning Systems ... 15
5.4.1 Grid Systems ... 15
5.4.2 Profiles, Cutouts and Notches ... 15

6.0 ELECTRICAL PROPERTIES ... 16

7.0 THERMAL MANAGEMENT ... 16

8.0 COMPONENT AND ASSEMBLY ISSUES ... 16
8.1 General Attachment Requirements ... 16
8.1.1 Attachment of Wires/Leads to Terminals ... 16
8.1.2 Board Extractors ... 16

9.0 HOLE/INTERCONNECTIONS ... 16
9.1 General Requirements for Lands with Holes ... 16
9.1.1 Land Requirements ... 16
9.1.2 Thermal Relief in Conductor Planes ... 16
9.1.3 Clearance Area in Planes ... 17
9.1.4 Nonfunctional Lands ... 18
9.1.5 Conductive Pattern Feature Location Tolerance ... 18
9.2 Holes ... 18
9.2.1 Unsupported Holes ... 18
9.2.2 Plated-Through Holes ... 19
9.2.3 Etchback ... 19
9.3 Drill Size Recommendations for Printed Boards ... 20

10.0 GENERAL CIRCUIT FEATURE REQUIREMENTS ... 20

INDEX ... 23
10.1 Conductor Characteristics ... 20
10.1.1 Edge Spacing ... 20
10.1.2 Balanced Conductors ... 21
10.1.3 Flush Conductors for Rotating or Sliding Contacts ... 21
10.1.4 Metallic Finishes for Flush Conductors ... 21
10.2 Land Characteristics ... 21
10.2.1 Lands for Interfacial Connection Vias ... 21
10.2.2 Offset Lands ... 21
10.2.3 Conductive Pattern Feature Location Tolerance ... 21
10.2.4 Nonfunctional Lands ... 21
10.3 Large Conductive Areas ... 21

11.0 DOCUMENTATION ... 22
11.1 Filled Holes ... 22
11.2 Nonfunctional Holes ... 22

12.0 QUALITY ASSURANCE ... 22

Figures

Figure 1-1 Electrical assembly types ... 2
Figure 4-1 Dielectric layer thickness measurement ... 4

Figure 4-2 Designer / end user materials selection map 11
Figure 5-1 Panel borders 14
Figure 5-2 Scoring parameters 14
Figure 5-3 Breakaway tabs 15
Figure 8-1 Permanent board extractor 16
Figure 8-2 External board extractor 16
Figure 9-1 Clearance area in planes, mm 17
Figure 9-2 Foil web size 18
Figure 9-3 Lead-to-hole clearance 19
Figure 10-1A Typical flush circuit 21
Figure 10-1B Surface flushness conditions 22
Figure 10-2 Cross-hatched large conductive layers with isothermal conductors 22

Tables

Table 4-1 Clad Laminate Maximum Operating Temperatures 4
Table 4-2 FR-4 Copper Clad Laminate Construction Selection Guide 6
Table 4-3 High T_G FR-4 Copper Clad Laminate Construction Selection Guide 7
Table 4-4 Cyanate Ester (170 to 250° T_G) Copper Clad Laminate Construction Selection Guide 8
Table 4-5 BT Copper Clad Laminate Construction Selection Guide 9
Table 4-6 Polyimide Copper Clad Laminate Construction Selection Guide 10
Table 5-1 Panel Size to Manufacturing Operation Relationships 14
Table 5-2 Standard Scoring Parameters 14
Table 5-3 Tolerance of Profiles, Cutouts, Notches, and Keying Slots, as Machined, mm 15
Table 9-1 Feature Location Tolerances (Lands, Conductor Pattern, etc.) (Diameter True Position) 18
Table 9-2 Minimum Unsupported Holes Tolerance Range (Difference between high and low hole size limits) 18
Table 9-3 Plated-Through Hole Diameter to Lead Diameter Relationships 20
Table 9-4 Plated-Through Hole Aspect Ratio 20
Table 9-5 Minimum Plated-Through Hole Diameter Tolerance Range, mm (Difference between high and low hole size limits) 20
Table 9-6 Minimum Drilled Hole Size for Plated-Through Hole Vias 20
Table 9-7 Drill Size Recommendations Related to Maximum Board Thickness 20
Table 10-1 Surface Flushness Requirements 21

Sectional Design Standard for Rigid Organic Printed Boards

1.0 SCOPE

This standard establishes the specific requirements for the design of rigid organic printed boards and other forms of component mounting and interconnecting structures. 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 specific design details that **shall** be used in conjunction with IPC-2221 (see 2.0) 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 function.

1.2 Document 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 Type This standard provides design information for different 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 Vias
Type 5 — Multilayer Metal-Core Board without Blind or Buried Vias
Type 6 — Multilayer Metal-Core Board with Blind and/or Buried Vias

1.6 Assembly Types A type designation signifies further sophistication describing whether components are mounted on one or both sides of the packaging and interconnecting structure. Type 1 defines an assembly that has components mounted on only one side; Type 2 is an assembly with components on both sides. Type 2, Class A is not recommended.

Figure 1-1 shows the relationship of two types of assemblies.

The need to apply certain design concepts should depend on the complexity and precision required to produce a particular land pattern or P&I structure. Any design class may be applied to any of the end-product equipment categories; therefore, a moderate complexity (Type 1B) would define components mounted on one side (all surface mounted) and when used in a Class 2 product (dedicated service electronics) is referred to as Type 1B, Class 2. The product described as a Type 1B, Class 2 might be used in any of the end-use applications; the selection of class being dependent on the requirements of the customers using the application.

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-2222 and those listed below, IPC-2222 takes precedence.

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

2.1 Institute for Interconnecting and Packaging Electronic Circuits (IPC)[1]

IPC-EG-140 Specification For Finished Fabric Woven From "E" Glass for Printed Board

IPC-MF-150 Metal Foil for Printed Wiring Applications

IPC-CF-152 Composite Metallic Materials Specification for Printed Wiring Boards

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

IPC-TM-650 Test Methods Manual

Method 2.1.1 Microsectioning

Method 2.1.6 Thickness of Glass Fabric

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

1. IPC, 2215 Sanders Road, Northbrook, IL 60062