

IPC-2222

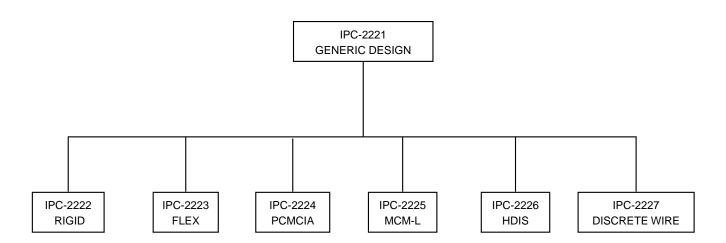
Sectional Design Standard for Rigid Organic Printed Boards

ANSI/IPC-2222

February 1998

A standard developed by IPC

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

February 1998 IPC-2222

Table of Contents

1.0	SCOPE	1	7.0	THERMAL MANAGEMENT	16
1.1	Purpose	1	8.0	COMPONENT AND ASSEMBLY ISSUES	16
1.2	Document Hierarchy	1	8.1	General Attachment Requirements	
1.3	Presentation	1	8.1.1	Attachment of Wires/Leads to Terminals	
1.4	Interpretation	1	8.1.2	Board Extractors	
1.5	Classification of Products	1			
1.5.1	Board Type	1	9.0	HOLE/INTERCONNECTIONS	
1.6	Assembly Types	1	9.1	General Requirements for Lands with Holes.	16
2.0	APPLICABLE DOCUMENTS	1	9.1.1	Land Requirements	
2.1	Institute for Interconnecting and Packaging	1	9.1.2	Thermal Relief in Conductor Planes	16
2.1	Electronic Circuits (IPC)	1	9.1.3	Clearance Area in Planes	
2.2	Underwriters Laboratories		9.1.4	Nonfunctional Lands	18
			9.1.5	Conductive Pattern Feature Location	
3.0	GENERAL REQUIREMENTS			Tolerance	
3.1	Performance Requirements	3	9.2	Holes	
4.0	MATERIALS	3	9.2.1	Unsupported Holes	
4.1	Material Selection	3	9.2.2	Plated-Through Holes	
4.2	Dielectric Base Materials (Including		9.2.3	Etchback	19
	Prepregs and Adhesives)	3	9.3	Drill Size Recommendations for	2.
4.2.1	Epoxy Laminates	3		Printed Boards	20
4.2.2	High-Temperature Laminates	3	10.0	GENERAL CIRCUIT FEATURE	
4.2.3	Special Clad Materials	3		REQUIREMENTS	20
4.2.4			INDEX	· · · · · · · · · · · · · · · · · · ·	23
4.3	Laminate Materials		10.1	Conductor Characteristics	
4.3.1	Measurement of Dielectric Thickness	3	10.1.1		
4.3.2	Dielectric Thickness/Spacing	4	10.1.1		
4.3.3	Laminate Properties	5	10.1.2		∠1
4.3.4	Prepreg	5	10.1.5	Contacts	21
4.3.5	Single-Clad Laminates	5	10.1.4		
4.3.6	Double-Clad Laminates	5	10.2	Land Characteristics	
4.3.7	Laminate Material	5	10.2.1		
4.4	Conductive Materials	. 13	10.2.2		
4.5	Organic Protective Coatings	. 13	10.2.3		21
4.6	Markings and Legends		10.2.3	Tolerance	21
			10.2.4		
5.0	MECHANICAL/PHYSICAL PROPERTIES		10.3	Large Conductive Areas	
5.1	Fabrication Requirements				
5.2	Product/Board Configuration		11.0	DOCUMENTATION	
5.2.1	Board Geometries		11.1	Filled Holes	
5.2.2	11		11.2	Nonfunctional Holes	22
5.3	Assembly Requirements		12.0	QUALITY ASSURANCE	22
5.3.1	Assembly and Test		-		
5.4	Dimensioning Systems				
5.4.1	Grid Systems	. 15		Figures	
5.4.2	Profiles, Cutouts and Notches		Figure	_	3
6.0	ELECTRICAL PROPERTIES	16	Figure		
5.0	ELECTRICAL FROF ERTIES	. 10	riguie	The Dielectric layer trilonless measurement	4

Figure 4-2	Designer / end user materials selection map1		Table 4-5	BT Copper Clad Laminate Construction Selection Guide	. 9
Figure 5-1	Panel borders		Table 4-6	Polyimide Copper Clad Laminate Construction Selection Guide	10
Figure 5-2	Scoring parameters 1		T.I. 5.4		10
Figure 5-3	Breakaway tabs1	5	Table 5-1	Panel Size to Manufacturing Operation Relationships	14
Figure 8-1	Permanent board extractor 10	6	Table 5-2	Standard Scoring Parameters	14
Figure 8-2	External board extractor1	6	Table 5-3	Tolerance of Profiles, Cutouts, Notches, and	
Figure 9-1	Clearance area in planes, mm1		10010 0 0	Keying Slots, as Machined, mm	15
Figure 9-2	Foil web size1	8	Table 9-1	Feature Location Tolerances (Lands, Conductor Pattern, etc.) (Diameter True Position)	
Figure 9-3	Lead-to-hole clearance1	9			
Figure 10-1/	A Typical flush circuit2	1	Table 9-2		18
Figure 10-1	Surface flushness conditions	2			
Figure 10-2	Cross-hatched large conductive layers with isothermal conductors				18
		2	Table 9-3	Plated-Through Hole Diameter to Lead Diameter Relationships	20
	Tables		Table 9-4	Plated-Through Hole Aspect Ratio	20
	Tables		Table 9-5	Minimum Plated-Through Hole Diameter	
Table 4-1	Clad Laminate Maximum Operating			Tolerance Range, mm (Difference between high and low hole	
	Temperatures4	4			
Table 4-2	FR-4 Copper Clad Laminate Construction		Table 9-6	Size limits)	20
	Selection Guide6	6			20
Table 4-3	High T _G FR-4 Copper Clad Laminate		T.I. 0 =	Hole Vias	20
	Construction Selection Guide	1	Table 9-7	Drill Size Recommendations Related to Maximum Board Thickness	
Table 4-4	Cyanate Ester (170 to 250° T _G) Copper Clad Laminate Construction Selection Guide		Table 10.1		
	Ciao Faminale Construction Selection Guide 👚	0	12016 1U-1	Surface Flushness Requirements	21

February 1998 IPC-2222

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