

IPC-2222B

Sectional Design Standard for Rigid Organic Printed Boards

Developed by the IPC-2221/2222 Task Group (D-31b) of the Rigid Printed Board Committee (D-30) of IPC

Supersedes:

IPC-2222A - December 2010 IPC-2222 - February 1998 IPC-D-275 - September 1991 Users of this publication are encouraged to participate in the development of future revisions.

Contact:

IPC

Table of Contents

1 S	COPE	1	5.2.2	Me	chanical Support	10
1.1	Purpose	1	5.3		sign Requirements for Printed Board	
1.2	Document Hierarchy	1			sembly	
1.3	Presentation	1	5.3.1	Ass	sembly Array (or Pallet)	10
1.4	Interpretation	1	5.4		mensioning Systems	
1.5	Definition of Terms	1	5.4.1		ofiles, Cutouts and Notches	
1.5.1	As Agreed Between User and Supplier (AABUS)	1	5.5 6 E		rical Properties	
1.6	Classification of Products					
1.6.1	Printed Board Type	1	7 THERMAL MANAGEMENT 14			
1.7	Applicability		8 COMPONENT AND ASSEMBLY ISSUES 14			14
1.8	Revision Level Changes		8.1	Ge	neral Attachment Requirements	14
0 4			8.1.1	Att	achment of Wires/Leads to Terminals	14
	PPLICABLE DOCUMENTS		8.1.2	Pri	nted Board Extractors	14
2.1	IPC		9 H	IOLE/	INTERCONNECTIONS	1 5
2.2	Underwriters Laboratories	2	9.1		neral Requirements for Lands with Holes	
3 G	ENERAL REQUIREMENTS	2	9.1.1		nd Requirements	
3.1	Performance Requirements	2	9.1.2		ermal Relief in Conductor Planes	
4 N	IATERIALS	2	9.1.3		earance Area in Planes	
4.1	Material Selection	2	9.1.4		nfunctional Lands	
4.2	Dielectric Base Materials (Including Prepregs and Adhesives)		9.1.5	Co	nductive Pattern Feature Location erance	
4.2.1	Epoxy Laminates		9.2		les	
4.2.2	High T _g Laminates		9.2.1		supported Holes (Non-plated Holes)	
4.2.3	Special Clad Materials		9.2.2		ted-Through Holes (PTHs)	
4.2.4	Other Laminates		9.2.3		hback	
4.3	Laminate Materials	3				
4.3.1	UL Parameters	3	10	GENERAL CIRCUIT FEATURE REQUIREMENTS		21
4.3.2	Measurement of Dielectric Thickness		10.1		nductor Characteristics	
4.3.3	Dielectric Thickness/Spacing		10.1.1		nted Board Edge Spacing	
4.3.4	Laminate Properties		10.1.2		lanced Conductors	
4.3.5	Prepreg		10.1.3		sh Conductors for Rotating or Sliding	
4.3.6	Single-Clad Laminate				ntacts	21
4.3.7	Double-Clad Laminate		10.2	Laı	nd Characteristics	22
4.3.8	Laminate Material Code Designation		10.2.1	Off	set Lands	22
4.4	Conductive Materials		10.3	Laı	ge Conductive Areas	22
4.5	Electronic (Embedded) Component Materials		11	DOCI	JMENTATION	22
4.6	Organic Protective Coatings					
4.7	Markings and Legends		12	QUAL	ITY ASSURANCE	22
4.7	Warkings and Legends	/			Figures	
5 N	IECHANICAL/PHYSICAL PROPERTIES		Figure	4-1	Dielectric Layer Thickness Measurement	F
5.1	Fabrication Requirements	8	Figure		Designer/End User Materials Selection Map	
5.2	Product/Printed Board Configuration	8	Figure		Borders and Margins in Printed Board	
5.2.1	Printed Board Size and Shape	8			Panels, mm [in]	g

Figure 5-2	Scoring Parameters 11	Table 5-2	Tolerance of Profiles, Cutouts and Notches as Machined, mm [in]	
Figure 5-3	Conductor Clearance for V-Groove Scoring 11			13
Figure 5-4	Breakaway Tabs 12	Table 5-3	Printed Board Overall Thickness Tolerance Levels, mm [in]	14
Figure 5-5	Typical Low Stress Breakaway Tabs with Mouse Bites and Routed Slots 12	Table 9-1	Minimum Standard Fabrication Allowance	
Figure 5-6	Routed Slots and Break Line 13		for Interconnection Lands	
Figure 8-1	Permanent Printed Board Extractor 15	Table 9-2	Pad to Plane Clearance, mm [in]	17
Figure 8-2	External Printed Board Extractor	Table 9-3	Nonfunctional Land Considerations	18
Figure 9-1	Clearance Area in Planes 17	Table 9-4	Feature Location Tolerances (Lands, Conductor Pattern, etc.) (Diameter True Position) ¹ , mm [in]	
Figure 9-2	Foil (Copper) Web Size			18
Figure 9-3	Lead-to-hole Clearance	Table 9-5	Diameter Tolerance Range for	
Figure 10-1	Typical Flush Circuit		Unsupported Holes, mm [in]	19
Figure 10-2	Surface Flushness Conditions	Table 9-6	PTH Diameter to Lead Diameter	
Figure 10-3	Cross-hatched Large Conductive Layers with Isothermal Conductors		Relationships, mm [in]	19
		Table 9-7	Plated Hole Aspect Ratio	20
	Tables	Table 9-8	Minimum PTH Diameter Tolerance Range, (LMC – MMC Hole Size Limits), mm [in]	20
Table 4-1	Typical Thermal Properties of Selected Dielectrics	Table 9-9	Recommended Minimum Drill Size, mm [in]	20
Table 4-2	Advantages and Disadvantages of some Common Laminate Materials 5	Table 10-1	Surface Flushness Requirements, mm [in]	
Table 5-1	Standard Scoring Parameters Jum [in] 10			

October 2020 IPC-2222B

Sectional Design Standard for Rigid Organic Printed Boards

1 SCOPE

This standard establishes the specific requirements for the design of rigid organic printed boards.

The following overview describes what are the core knowledge and competencies to best serve in the role of *Printed Board Design Layout* as a stand-alone professional, or as the engineer performing this responsibility. Today's printed board designer needs to address numerous perspectives for success within a given schedule, with the goal of making the first design iteration work as intended, summarized as:

- Layout Solvability Complex Packaging Skillset
- Electrical Integrity Signal & Power Performance on all Layers
- Manufacturability Design for Excellence (DfX) Considerations for High Yield and Lower Cost
- Application considerations Environmental, Performance, Shelf life, etc.

The result provides for optimal component placement, routing density and electrical performance to achieve an efficient design with high yield and defect-free manufacturability.

1.1 Purpose The requirements contained herein are intended to establish specific design details that **shall** be used in conjunction with IPC-2221 to produce printed boards that perform as an integral part of functional electronic hardware.

The organic materials may be homogeneous, reinforced, or used in combination with inorganic materials; the interconnections may be single, double, or multilayered. They 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 Definition of Terms** The definition of all terms used herein **shall** be in accordance with IPC-T-50 and as defined in 1.5.1.
- **1.5.1** As Agreed Between User and Supplier (AABUS) A bilateral agreement which indicates additional or alternate requirements to be decided between the user and the supplier in the procurement documentation. Examples include contractual requirements, modifications to purchase documentation and information on the drawing. Agreements can be used to define test methods, conditions, frequencies, categories or acceptance criteria within a test, if not already established.
- **1.6 Classification of Products** Classification of products **shall** be in accordance with the generic standard IPC-2221 and as defined in 1.6.1.
- **1.6.1 Printed Board Type** This standard provides design information for different printed board types. Printed board types are classified as:
- Type 1 Single-Sided Printed Board
- Type 2 Double-Sided Printed Board
- Type 3 Multilayer Printed Board without blind or buried vias
- Type 4 Multilayer Printed Board with blind and/or buried vias
- Type 5 Multilayer Metal Core Printed Board without blind or buried vias
- Type 6 Multilayer Metal Core Printed Board with blind and/or buried vias
- **1.7 Applicability** The contents of this standard may not apply to certain leading-edge technologies. Refer to IPC-2221 for additional information.