



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

<b>1 SCOPE</b> .....	1	5.2.2 Mechanical Support .....	10
1.1 Purpose .....	1	5.3 Design Requirements for Printed Board Assembly .....	10
1.2 Document Hierarchy .....	1	5.3.1 Assembly Array (or Pallet) .....	10
1.3 Presentation .....	1	5.4 Dimensioning Systems .....	13
1.4 Interpretation .....	1	5.4.1 Profiles, Cutouts and Notches .....	13
1.5 Definition of Terms .....	1	5.5 Printed Board Thickness Tolerance .....	14
1.5.1 As Agreed Between User and Supplier (AABUS) .....	1	<b>6 ELECTRICAL PROPERTIES</b> .....	14
1.6 Classification of Products .....	1	<b>7 THERMAL MANAGEMENT</b> .....	14
1.6.1 Printed Board Type .....	1	<b>8 COMPONENT AND ASSEMBLY ISSUES</b> .....	14
1.7 Applicability .....	1	8.1 General Attachment Requirements .....	14
1.8 Revision Level Changes .....	2	8.1.1 Attachment of Wires/Leads to Terminals .....	14
<b>2 APPLICABLE DOCUMENTS</b> .....	2	8.1.2 Printed Board Extractors .....	14
2.1 IPC .....	2	<b>9 HOLE/INTERCONNECTIONS</b> .....	15
2.2 Underwriters Laboratories .....	2	9.1 General Requirements for Lands with Holes ..	15
<b>3 GENERAL REQUIREMENTS</b> .....	2	9.1.1 Land Requirements .....	15
3.1 Performance Requirements .....	2	9.1.2 Thermal Relief in Conductor Planes .....	16
<b>4 MATERIALS</b> .....	2	9.1.3 Clearance Area in Planes .....	17
4.1 Material Selection .....	2	9.1.4 Nonfunctional Lands .....	18
4.2 Dielectric Base Materials (Including Prepregs and Adhesives) .....	2	9.1.5 Conductive Pattern Feature Location Tolerance .....	18
4.2.1 Epoxy Laminates .....	2	9.2 Holes .....	19
4.2.2 High T <sub>g</sub> Laminates .....	3	9.2.1 Unsupported Holes (Non-plated Holes) .....	19
4.2.3 Special Clad Materials .....	3	9.2.2 Plated-Through Holes (PTHs) .....	19
4.2.4 Other Laminates .....	3	9.2.3 Etchback .....	20
4.3 Laminate Materials .....	3	<b>10 GENERAL CIRCUIT FEATURE REQUIREMENTS</b> .....	21
4.3.1 UL Parameters .....	3	10.1 Conductor Characteristics .....	21
4.3.2 Measurement of Dielectric Thickness .....	6	10.1.1 Printed Board Edge Spacing .....	21
4.3.3 Dielectric Thickness/Spacing .....	6	10.1.2 Balanced Conductors .....	21
4.3.4 Laminate Properties .....	6	10.1.3 Flush Conductors for Rotating or Sliding Contacts .....	21
4.3.5 Prepreg .....	6	10.2 Land Characteristics .....	22
4.3.6 Single-Clad Laminate .....	7	10.2.1 Offset Lands .....	22
4.3.7 Double-Clad Laminate .....	7	10.3 Large Conductive Areas .....	22
4.3.8 Laminate Material Code Designation .....	7	<b>11 DOCUMENTATION</b> .....	22
4.4 Conductive Materials .....	7	<b>12 QUALITY ASSURANCE</b> .....	22
4.5 Electronic (Embedded) Component Materials ..	7		
4.6 Organic Protective Coatings .....	7		
4.7 Markings and Legends .....	7		
<b>5 MECHANICAL/PHYSICAL PROPERTIES</b> .....	8		
5.1 Fabrication Requirements .....	8		
5.2 Product/Printed Board Configuration .....	8		
5.2.1 Printed Board Size and Shape .....	8		

### Figures

Figure 4-1 Dielectric Layer Thickness Measurement .....	6
Figure 4-2 Designer/End User Materials Selection Map ...	8
Figure 5-1 Borders and Margins in Printed Board Panels, mm [in].....	9

Figure 5-2	Scoring Parameters .....	11	Table 5-2	Tolerance of Profiles, Cutouts and Notches, as Machined, mm [in] .....	13
Figure 5-3	Conductor Clearance for V-Groove Scoring...	11	Table 5-3	Printed Board Overall Thickness Tolerance Levels, mm [in] .....	14
Figure 5-4	Breakaway Tabs .....	12	Table 9-1	Minimum Standard Fabrication Allowance for Interconnection Lands.....	15
Figure 5-5	Typical Low Stress Breakaway Tabs with Mouse Bites and Routed Slots.....	12	Table 9-2	Pad to Plane Clearance, mm [in] .....	17
Figure 5-6	Routed Slots and Break Line .....	13	Table 9-3	Nonfunctional Land Considerations .....	18
Figure 8-1	Permanent Printed Board Extractor .....	15	Table 9-4	Feature Location Tolerances (Lands, Conductor Pattern, etc.) (Diameter True Position) <sup>1</sup> , mm [in].....	18
Figure 8-2	External Printed Board Extractor.....	15	Table 9-5	Diameter Tolerance Range for Unsupported Holes, mm [in].....	19
Figure 9-1	Clearance Area in Planes.....	17	Table 9-6	PTH Diameter to Lead Diameter Relationships, mm [in].....	19
Figure 9-2	Foil (Copper) Web Size.....	17	Table 9-7	Plated Hole Aspect Ratio .....	20
Figure 9-3	Lead-to-hole Clearance .....	19	Table 9-8	Minimum PTH Diameter Tolerance Range, (LMC – MMC Hole Size Limits), mm [in] .....	20
Figure 10-1	Typical Flush Circuit .....	21	Table 9-9	Recommended Minimum Drill Size, mm [in].....	20
Figure 10-2	Surface Flushness Conditions.....	21	Table 10-1	Surface Flushness Requirements, mm [in].....	21
Figure 10-3	Cross-hatched Large Conductive Layers with Isothermal Conductors.....	22			
<b>Tables</b>					
Table 4-1	Typical Thermal Properties of Selected Dielectrics .....	4			
Table 4-2	Advantages and Disadvantages of some Common Laminate Materials .....	5			
Table 5-1	Standard Scoring Parameters, μm [in].....	10			

# 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.