



ASSOCIATION CONNECTING
ELECTRONICS INDUSTRIES®

IPC-2221A

Generic Standard on Printed Board Design

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

Supersedes:

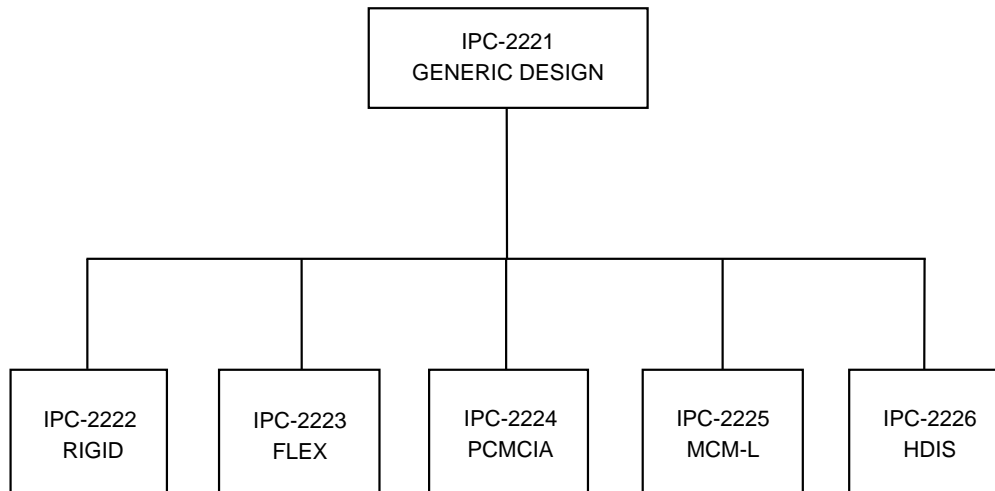
IPC-2221 - February 1998

Users of this publication are encouraged to participate in the development of future revisions.

Contact:

IPC
2215 Sanders Road
Northbrook, Illinois
60062-6135
Tel 847 509.9700
Fax 847 509.9798

HIERARCHY OF IPC DESIGN SPECIFICATIONS
(2220 SERIES)



FOREWORD

This standard is intended to provide information on the generic requirements for organic 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. It is crucial that a decision pertaining to the choice of product types be made as early as possible. Once a component mounting and interconnecting technology has been selected the user should obtain the sectional document that provides the specific focus on the chosen technology.

It may be more effective to consider alternative printed board construction types for the product being designed. As an example the application of a rigid-flex printed wiring board may be more cost or performance effective than using multiple printed wiring boards, connectors and cables.

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 user needs, as a minimum, the generic design document, 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	SCOPE	1	4.1.1	Material Selection for Structural Strength	17
1.1	Purpose	1	4.1.2	Material Selection for Electrical Properties	17
1.2	Documentation Hierarchy	1	4.1.3	Material Selection for Environmental Properties	17
1.3	Presentation	1	4.2	Dielectric Base Materials (Including Prepregs and Adhesives)	17
1.4	Interpretation	1	4.2.1	Preimpregnated Bonding Layer (Prepreg)	17
1.5	Definition of Terms	1	4.2.2	Adhesives	17
1.6	Classification of Products	1	4.2.3	Adhesive Films or Sheets	19
1.6.1	Board Type	1	4.2.4	Electrically Conductive Adhesives	19
1.6.2	Performance Classes	1	4.2.5	Thermally Conductive/Electrically Insulating Adhesives	19
1.6.3	Producibility Level	2	4.3	Laminate Materials	20
1.7	Revision Level Changes	2	4.3.1	Color Pigmentation	20
2	APPLICABLE DOCUMENTS	2	4.3.2	Dielectric Thickness/Spacing	20
2.1	IPC	2	4.4	Conductive Materials	20
2.2	Joint Industry Standards	3	4.4.1	Electroless Copper Plating	20
2.3	Society of Automotive Engineers	3	4.4.2	Semiconductive Coatings	20
2.4	American Society for Testing and Materials	3	4.4.3	Electrolytic Copper Plating	20
2.5	Underwriters Labs	3	4.4.4	Gold Plating	20
2.6	IEEE	3	4.4.5	Nickel Plating	22
2.7	ANSI	4	4.4.6	Tin/Lead Plating	22
3	GENERAL REQUIREMENTS	4	4.4.7	Solder Coating	22
3.1	Information Hierarchy	6	4.4.8	Other Metallic Coatings for Edgeboard Contacts	23
3.1.1	Order of Precedence	6	4.4.9	Metallic Foil/Film	23
3.2	Design Layout	6	4.4.10	Electronic Component Materials	23
3.2.1	End-Product Requirements	6	4.5	Organic Protective Coatings	24
3.2.2	Density Evaluation	6	4.5.1	Solder Resist (Solder Mask) Coatings	24
3.3	Schematic/Logic Diagram	6	4.5.2	Conformal Coatings	25
3.4	Parts List	6	4.5.3	Tarnish Protective Coatings	25
3.5	Test Requirement Considerations	7	4.6	Marking and Legends	25
3.5.1	Printed Board Assembly Testability	7	4.6.1	ESD Considerations	26
3.5.2	Boundary Scan Testing	8	5	MECHANICAL/PHYSICAL PROPERTIES	26
3.5.3	Functional Test Concern for Printed Board Assemblies	8	5.1	Fabrication Considerations	26
3.5.4	In-Circuit Test Concerns for Printed Board Assemblies	10	5.1.1	Bare Board Fabrication	26
3.5.5	Mechanical	12	5.2	Product/Board Configuration	26
3.5.6	Electrical	12	5.2.1	Board Type	26
3.6	Layout Evaluation	13	5.2.2	Board Size	26
3.6.1	Board Layout Design	13	5.2.3	Board Geometries (Size and Shape)	26
3.6.2	Feasibility Density Evaluation	13	5.2.4	Bow and Twist	27
3.7	Performance Requirements	15	5.2.5	Structural Strength	27
4	MATERIALS	17	5.2.6	Composite (Constraining-Core) Boards	27
4.1	Material Selection	17	5.2.7	Vibration Design	29

5.3	Assembly Requirements	30	7.2.4	Special Design Considerations for SMT Board Heatsinks	52
5.3.1	Mechanical Hardware Attachment	30	7.3	Heat Transfer Techniques	52
5.3.2	Part Support	30	7.3.1	Coefficient of Thermal Expansion (CTE) Characteristics	52
5.3.3	Assembly and Test	30	7.3.2	Thermal Transfer	53
5.4	Dimensioning Systems	31	7.3.3	Thermal Matching	53
5.4.1	Dimensions and Tolerances	31	7.4	Thermal Design Reliability	53
5.4.2	Component and Feature Location	31			
5.4.3	Datum Features	31			
6	ELECTRICAL PROPERTIES	37	8	COMPONENT AND ASSEMBLY ISSUES	55
6.1	Electrical Considerations	37	8.1	General Placement Requirements	55
6.1.1	Electrical Performance	37	8.1.1	Automatic Assembly	55
6.1.2	Power Distribution Considerations	37	8.1.2	Component Placement	55
6.1.3	Circuit Type Considerations	39	8.1.3	Orientation	57
6.2	Conductive Material Requirements	40	8.1.4	Accessibility	57
6.3	Electrical Clearance	42	8.1.5	Design Envelope	57
6.3.1	B1–Internal Conductors	42	8.1.6	Component Body Centering	57
6.3.2	B2–External Conductors, Uncoated, Sea Level to 3050 m [10,007 feet]	42	8.1.7	Mounting Over Conductive Areas	57
6.3.3	B3–External Conductors, Uncoated, Over 3050 m [10,007 feet]	42	8.1.8	Clearances	58
6.3.4	B4–External Conductors, with Permanent Polymer Coating (Any Elevation)	42	8.1.9	Physical Support	58
6.3.5	A5–External Conductors, with Conformal Coating Over Assembly (Any Elevation)	43	8.1.10	Heat Dissipation	59
6.3.6	A6–External Component Lead/Termination, Uncoated, Sea Level to 3050 m [10,007 feet]	43	8.1.11	Stress Relief	60
6.3.7	A7–External Component Lead/Termination, with Conformal Coating (Any Elevation)	43	8.2	General Attachment Requirements	60
6.4	Impedance Controls	43	8.2.1	Through-Hole	60
6.4.1	Microstrip	44	8.2.2	Surface Mounting	60
6.4.2	Embedded Microstrip	44	8.2.3	Mixed Assemblies	61
6.4.3	Stripline Properties	44	8.2.4	Soldering Considerations	61
6.4.4	Asymmetric Stripline Properties	46	8.2.5	Connectors and Interconnects	62
6.4.5	Capacitance Considerations	46	8.2.6	Fastening Hardware	63
6.4.6	Inductance Considerations	47	8.2.7	Stiffeners	64
7	THERMAL MANAGEMENT	48	8.2.8	Lands for Flattened Round Leads	64
7.1	Cooling Mechanisms	48	8.2.9	Solder Terminals	64
7.1.1	Conduction	49	8.2.10	Eyelets	65
7.1.2	Radiation	49	8.2.11	Special Wiring	65
7.1.3	Convection	49	8.2.12	Heat Shrinkable Devices	67
7.1.4	Altitude Effects	49	8.2.13	Bus Bar	67
7.2	Heat Dissipation Considerations	49	8.2.14	Flexible Cable	67
7.2.1	Individual Component Heat Dissipation	50	8.3	Through-Hole Requirements	67
7.2.2	Thermal Management Considerations for Board Heatsinks	50	8.3.1	Leads Mounted in Through-Holes	67
7.2.3	Assembly of Heatsinks to Boards	50	8.4	Standard Surface Mount Requirements	71
			8.4.1	Surface-Mounted Leaded Components	71
			8.4.2	Flat-Pack Components	71
			8.4.3	Ribbon Lead Termination	72
			8.4.4	Round Lead Termination	72
			8.4.5	Component Lead Sockets	72
			8.5	Fine Pitch SMT (Peripherals)	72
			8.6	Bare Die	73

Figure 5-3A	Multilayer Metal Core Board with Two Symmetrical Copper-Invar-Copper Constraining Cores (when the Copper-Invar-Copper planes are connected to the plated-through hole, use thermal relief per Figure 9-4)	29	Figure 8-7	Mounting with Feet or Standoffs	59
Figure 5-3B	Symmetrical Constraining Core Board with a Copper-Invar-Copper Center Core	29	Figure 8-8	Heat Dissipation Examples	60
Figure 5-4	Advantages of Positional Tolerance Over Bilateral Tolerance, mm [in]	32	Figure 8-9	Lead Bends	61
Figure 5-4A	Datum Reference Frame	32	Figure 8-10	Typical Lead Configurations	61
Figure 5-5A	Example of Location of a Pattern of Plated-Through Holes, mm [in]	33	Figure 8-11	Board Edge Tolerancing	63
Figure 5-5B	Example of a Pattern of Tooling/Mounting Holes, mm [in]	33	Figure 8-12	Lead-In Chamfer Configuration	63
Figure 5-5C	Example of Location of a Conductor Pattern Using Fiducials, mm [in]	34	Figure 8-13	Typical Keying Arrangement	63
Figure 5-5D	Example of Printed Board Profile Location and Tolerance, mm [in]	35	Figure 8-14	Two-Part Connector	64
Figure 5-5E	Example of a Printed Board Drawing Utilizing Geometric Dimensioning and Tolerancing, mm [in]	35	Figure 8-15	Edge-Board Adapter Connector	64
Figure 5-6	Fiducial Clearance Requirements	36	Figure 8-16	Round or Flattened (Coined) Lead Joint Description	65
Figure 5-7	Fiducials, mm	36	Figure 8-17	Standoff Terminal Mounting, mm [in]	66
Figure 5-8	Example of Connector Key Slot Location and Tolerance, mm [in]	37	Figure 8-18	Dual Hole Configuration for Interfacial and Interlayer Terminal Mountings	66
Figure 6-1	Voltage/Ground Distribution Concepts	38	Figure 8-19	Partially Clinched Through-Hole Leads	68
Figure 6-2	Single Reference Edge Routing	39	Figure 8-20	Dual In-Line Package (DIP) Lead Bends	68
Figure 6-3	Circuit Distribution	39	Figure 8-21	Solder in the Lead Bend Radius	69
Figure 6-4	Conductor Thickness and Width for Internal and External Layers	41	Figure 8-22	Two-Lead Radial-Leaded Components	69
Figure 6-5	Transmission Line Printed Board Construction	45	Figure 8-23	Radial Two-Lead Component Mounting, mm [in]	69
Figure 6-6	Capacitance vs. Conductor Width and Dielectric Thickness for Microstrip Lines, mm [in]	47	Figure 8-24	Meniscus Clearance, mm [in]	69
Figure 6-7	Capacitance vs. Conductor Width and Spacing for Striplines, mm [in]	48	Figure 8-25	“TO” Can Radial-Leaded Component, mm [in]	69
Figure 6-8	Single Conductor Crossover	48	Figure 8-26	Perpendicular Part Mounting, mm [in]	70
Figure 7-1	Component Clearance Requirements for Automatic Component Insertion on Through-Hole Technology Printed Board Assemblies [in]	51	Figure 8-27	Flat-Packs and Quad Flat-Packs	70
Figure 7-2	Relative Coefficient of Thermal Expansion (CTE) Comparison	54	Figure 8-28	Examples of Configuration of Ribbon Leads for Through-Hole Mounted Flat-Packs	70
Figure 8-1	Component Orientation for Boundaries and/or Wave Solder Applications	57	Figure 8-29	Metal Power Packages with Compliant Leads	70
Figure 8-2	Component Body Centering	58	Figure 8-30	Metal Power Package with Resilient Spacers	71
Figure 8-3	Axial-Leaded Component Mounted Over Conductors	58	Figure 8-31	Metal Power Package with Noncompliant Leads	71
Figure 8-4	Uncoated Board Clearance	59	Figure 8-32	Examples of Flat-Pack Surface Mounting	72
Figure 8-5	Clamp-Mounted Axial-Leaded Component	59	Figure 8-33	Round or Coined Lead	72
Figure 8-6	Adhesive-Bonded Axial-Leaded Component	59	Figure 8-34	Configuration of Ribbon Leads for Planar Mounted Flat-Packs	72
			Figure 8-35	Heel Mounting Requirements	72
			Figure 9-1	Examples of Modified Land Shapes	74
			Figure 9-2	External Annular Ring	74
			Figure 9-3	Internal Annular Ring	74
			Figure 9-4	Typical Thermal Relief in Planes	75
			Figure 10-1	Example of Conductor Beef-Up or Neck-Down	78
			Figure 10-2	Conductor Optimization Between Lands	79
			Figure 10-3	Etched Conductor Characteristics	80
			Figure 11-1	Flow Chart of Printed Board Design/ Fabrication Sequence	82
			Figure 11-2	Multilayer Board Viewing	83
			Figure 11-3	Solder Resist Windows	83
			Figure 12-1	Location of Test Circuitry	85

Figure 12-2 Test Coupons A and B, mm [in] 87

Figure 12-3 Test Coupons A and B (Conductor Detail) mm, [in] 88

Figure 12-4 Test Coupon A/B, mm [in] 89

Figure 12-5 Test Coupon A/B (Conductor Detail), mm [in] 90

Figure 12-6 Coupon C, External Layers Only, mm [in] 90

Figure 12-7 Test Coupon D, mm [in] 91

Figure 12-8 Example of a 10 Layer Coupon D, Modified to Include Blind and Buried Vias 93

Figure 12-9 Test Coupon D for Process Control of 4 Layer Boards 94

Figure 12-10 Coupon E, mm 94

Figure 12-11 Optional Coupon H, mm [in] 95

Figure 12-12 Comb Pattern Examples 95

Figure 12-13 “Y” Pattern for Chip Component Cleanliness Test Pattern 96

Figure 12-14 Test Coupon F, mm [in] 97

Figure 12-15 Test Coupon R, mm [in] 98

Figure 12-16 Worst-Case Hole/Land Relationship 98

Figure 12-17 Test Coupon G, Solder Resist Adhesive, mm [in] 99

Figure 12-18 Test Coupon M, Surface Mounting Solderability Testing, mm [in] 99

Figure 12-19 Test Coupon N, Surface Mounting Bond Strength and Peel Strength, mm [in] 100

Figure 12-20 Test Coupon S, mm [in] 100

Figure 12-21 Systematic Path for Implementation of Statistical Process Control (SPC) 101

Figure 12-22 Test Coupon X, mm [in] 102

Figure 12-23 Bending Test 102

Figure B-1 Original Design Chart 104

Figure B-2 IPC 2221A External Conductor Chart 106

Figure B-3 Board Thickness 106

Figure B-4 Board Material 107

Figure B-5 Air/Vacuum Environment 107

Tables

Table 3-1 PCB Design/Performance Tradeoff Checklist 4

Table 3-2 Component Grid Areas 15

Table 4-1 Typical Properties of Common Dielectric Materials 18

Table 4-2 Environmental Properties of Common Dielectric Materials 18

Table 4-3 Final Finish, Surface Plating Coating Thickness Requirements 21

Table 4-4 Gold Plating Uses 22

Table 4-5 Copper Foil/Film Requirements 23

Table 4-6 Metal Core Substrates 23

Table 4-7 Conformal Coating Functionality 26

Table 5-1 Fabrication Considerations 27

Table 5-2 Typical Assembly Equipment Limits 31

Table 6-1 Electrical Conductor Spacing 43

Table 6-2 Typical Relative Bulk Dielectric Constant of Board Material 45

Table 7-1 Effects of Material Type on Conduction 49

Table 7-2 Emissivity Ratings for Certain Materials 49

Table 7-3 Board Heatsink Assembly Preferences 52

Table 7-4 Comparative Reliability Matrix Component Lead/Termination Attachment 53

Table 9-1 Minimum Standard Fabrication Allowance for Interconnection Lands 74

Table 9-2 Annular Rings (Minimum) 74

Table 9-3 Minimum Drilled Hole Size for Buried Vias 76

Table 9-4 Minimum Drilled Hole Size for Blind Vias 76

Table 9-5 Minimum Hole Location Tolerance, dtp 76

Table 10-1 Internal Layer Foil Thickness After Processing 77

Table 10-2 External Conductor Thickness After Plating 78

Table 10-3 Conductor Width Tolerances for 0.046 mm [0.00181 in] Copper 78

Table 12-1 Coupon Frequency Requirements 85

Table B-1 Test Samples 106

Generic Standard on Printed Board Design

1 SCOPE

This standard establishes the generic requirements for the design of organic printed boards and other forms of component mounting or 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 design principles and recommendations that **shall** be used in conjunction with the detailed requirements of a specific interconnecting structure sectional standard (see 1.2) to produce detailed designs intended to mount and attach passive and active components. This standard is not intended for use as a performance specification for finished boards nor as an acceptance document for electronic assemblies. For acceptability requirements of electronic assemblies, see IPC/EIA-J-STD-001 and IPC-A-610.

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 Documentation Hierarchy This standard identifies the generic physical design principles, and is supplemented by various sectional documents that provide details and sharper focus on specific aspects of printed board technology. Examples are:

IPC-2222	Rigid organic printed board structure design
IPC-2223	Flexible printed board structure design
IPC-2224	Organic, PC card format, printed board structure design
IPC-2225	Organic, MCM-L, printed board structure design
IPC-2226	High Density Interconnect (HDI) structure design
IPC-2227	Embedded Passive Devices printed board design (In Process)

The list is a partial summary and is not inherently a part of this generic standard. The documents are a part of the PCB Design Document Set which is identified as IPC-2220. The number IPC-2220 is for ordering purposes only and will include all documents which are a part of the set, whether released or in-process proposal format at the time the order is placed.

1.3 Presentation All dimensions and tolerances in this standard are expressed in hard SI (metric) units and paren-

thetical soft imperial (inch) units. Users of this and the corresponding performance and qualification specifications are expected to use metric dimensions.

1.4 Interpretation “**Shall**,” the imperative form of the verb, is used throughout this standard whenever a requirement is intended to express a provision that is mandatory. Deviation from a “**shall**” requirement may be considered if sufficient data is supplied to justify the exception.

The words “should” and “may” are used whenever it is necessary to express nonmandatory provisions. “Will” is used to express a declaration of purpose.

To assist the reader, the word “**shall**” is presented in bold characters.

1.5 Definition of Terms The definition of all terms used herein **shall** be as specified in IPC-T-50.

1.6 Classification of Products This standard recognizes that rigid printed boards and printed board assemblies are subject to classifications by intended end item use. Classification of producibility is related to complexity of the design and the precision required to produce the particular printed board or printed board assembly.

Any producibility level or producibility design characteristic may be applied to any end-product equipment category. Therefore, a high-reliability product designated as Class “3” (see 1.6.2), could require level “A” design complexity (preferred producibility) for many of the attributes of the printed board or printed board assembly (see 1.6.3).

1.6.1 Board Type This standard provides design information for different board types. Board types vary per technology and are thus classified in the design sectionals.

1.6.2 Performance Classes Three general end-product classes have been established to reflect progressive increases in sophistication, functional performance requirements and testing/inspection frequency. It should be recognized that there may be an overlap of equipment between classes. The printed board user has the responsibility to determine the class to which his product belongs. The contract **shall** specify the performance class required and indicate any exceptions to specific parameters, where appropriate.

Class 1 General Electronic Products Includes consumer products, some computer and computer peripherals, as well as general military hardware suitable for applications where cosmetic imperfections are not important and the