ASSOCIATION CONNECTING
ELECTRONICS INDUSTRIES

IPC-2221

Generic Standard on Printed Board Design

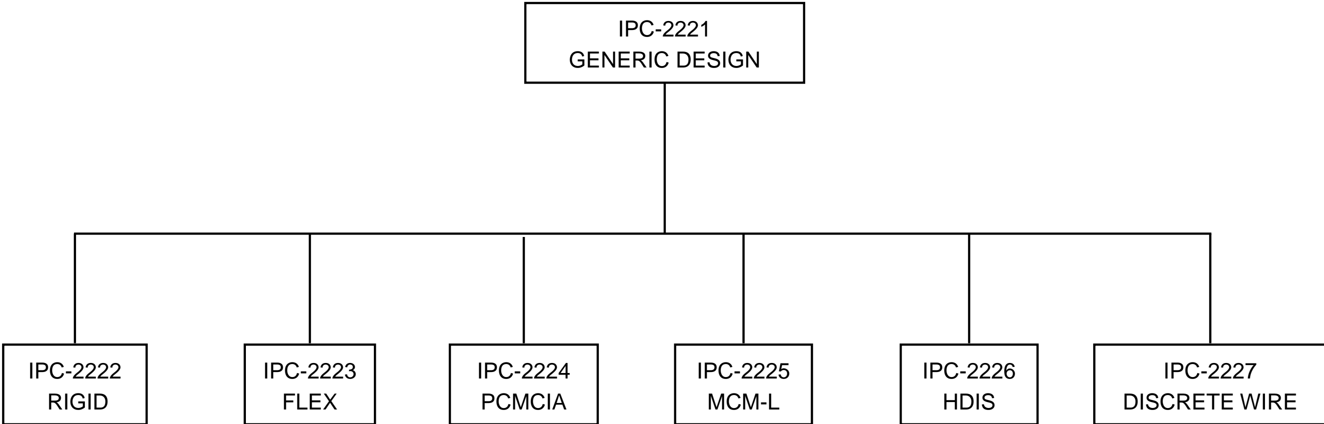
ANSI/IPC-2221
February 1998

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 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

1.0 SCOPE 1
1.1 Purpose 1
1.2 Documentation Hierarchy 1
1.3 Presentation 1
1.4 Interpretation 1
1.5 Definition of Terms 1
1.6 Classification of Products 1
1.6.1 Board Type 1
1.6.2 Performance Classes 1
1.6.3 Producibility Level 2

2.0 APPLICABLE DOCUMENTS 2
2.1 Institute for Interconnecting and Packaging Electronic Circuits (IPC) 2
2.2 Joint Industry Standards 3
2.3 Military 3
2.4 Federal 3
2.5 American Society for Testing and Materials 3
2.6 Underwriters Labs 3
2.7 IEEE 3
2.8 ANSI 3

3.0 GENERAL REQUIREMENTS 3
3.1 Information Hierarchy 4
3.1.1 Order of Precedence 4
3.2 Design Layout 4
3.2.1 End-Product Requirements 4
3.3 Schematic/Logic Diagram 4
3.4 Parts List 4
3.5 Test Requirement Considerations 4
3.5.1 Printed Board Assembly Testability 6
3.5.2 Boundary Scan Testing 7
3.5.3 Functional Test Concern for Printed Board Assemblies 7
3.5.4 In-Circuit Test Concerns for Printed Board Assemblies 9
3.5.5 Mechanical 11
3.5.6 Electrical 11
3.6 Layout Evaluation 12
3.6.1 Board Layout Design 12
3.6.2 Feasibility Density Evaluation 12
3.7 Performance Requirements 13

4.0 MATERIALS 14
4.1 Material Selection 14
4.1.1 Material Selection for Structural Strength 15
4.1.2 Material Selection for Electrical Properties 16
4.1.3 Material Selection for Environmental Properties 16
4.2 Dielectric Base Materials (Including Prepregs and Adhesives) 16
4.2.1 Bonding Material 16
4.2.2 Adhesives 16
4.2.3 Adhesive Films or Sheets 18
4.2.4 Electrically Conductive Adhesives 18
4.2.5 Thermally Conductive/Electrically Insulating Adhesives 18
4.3 Laminate Materials 19
4.3.1 Color Pigmentation 19
4.3.2 Dielectric Thickness/Spacing 19
4.4 Conductive Materials 19
4.4.1 Electroless Copper Plating 19
4.4.2 Semiconductive Coatings 19
4.4.3 Electrolytic Copper Plating 19
4.4.4 Gold Plating 19
4.4.5 Nickel Plating 20
4.4.6 Tin/Lead Plating 21
4.4.7 Solder Coating 21
4.4.8 Other Metallic Coatings for Edgeboard Contacts 21
4.4.9 Metallic Foil/Film 21
4.4.10 Electronic Component Materials 21
4.5 Organic Protective Coatings 22
4.5.1 Solder Resist (Solder Mask) Coatings 22
4.5.2 Conformal Coatings 23
4.5.3 Tarnish Protective Coatings 23
4.6 Marking and Legends 23
4.6.1 ESD Considerations 24

5.0 MECHANICAL/PHYSICAL PROPERTIES 24
5.1 Fabrication Considerations 24
5.1.1 Bare Board Fabrication 24
5.2 Product/Board Configuration 24
5.2.1 Board Type 24
5.2.2 Board Size 24
5.2.3 Board Geometries (Size and Shape) 24
5.2.4 Bow and Twist 25
5.2.5 Structural Strength 25
5.2.6 Composite (Constraining-core) Boards 25
5.2.7 Vibration Design 27
5.3 Assembly Requirements 28
5.3.1 Mechanical Hardware Attachment 28
5.3.2 Part Support 28
5.3.3 Assembly and Test 28

5.4 Dimensioning Systems 29
5.4.1 Dimensions and Tolerances 29
5.4.2 Component and Feature Location 29
5.4.3 Datum Features 30

6.0 ELECTRICAL PROPERTIES 31
6.1 Electrical Considerations 31
6.1.1 Electrical Performance 31
6.1.2 Power Distribution Considerations 31
6.1.3 Circuit Type Considerations 33
6.2 Conductive Material Requirements 35
6.3 Electrical Clearance 39
6.3.1 B1-Internal Conductors 39
6.3.2 B2-External Conductors, Uncoated, Sea Level to 3050 m 40
6.3.3 B3-External Conductors, Uncoated, Over 3050 m 40
6.3.4 B4-External Conductors, with Permanent Polymer Coating (Any Elevation) 40
6.3.5 A5-External Conductors, with Conformal Coating Over Assembly (Any Elevation) 40
6.3.6 A6-External Component Lead/Termination, Uncoated 40
6.3.7 A7-External Component Lead/Termination, with Conformal Coating (Any Elevation) 40
6.4 Impedance Controls 40
6.4.1 Microstrip 40
6.4.2 Embedded Microstrip 41
6.4.3 Stripline Properties 41
6.4.4 Asymmetric Stripline Properties 41
6.4.5 Capacitance Considerations 43
6.4.6 Inductance Considerations 43

7.0 THERMAL MANAGEMENT 44
7.1 Cooling Mechanisms 44
7.1.1 Conduction 44
7.1.2 Radiation 45
7.1.3 Convection 46
7.1.4 Altitude Effects 46
7.2 Heat Dissipation Considerations 46
7.2.1 Individual Component Heat Dissipation 46
7.2.2 Thermal Management Considerations for Board Heatsinks 47
7.2.3 Assembly of Heatsinks to Boards 47
7.2.4 Special Design Considerations for SMT Board Heatsinks 49
7.3 Heat Transfer Techniques 49
7.3.1 Coefficient of Thermal Expansion (CTE) Characteristics 49
7.3.2 Thermal Transfer 49
7.3.3 Thermal Matching 49
7.4 Thermal Design Reliability 50

8.0 COMPONENT AND ASSEMBLY ISSUES 50
8.1 General Placement Requirements 52
8.1.1 Automatic Assembly 52
8.1.2 Component Placement 52
8.1.3 Orientation 53
8.1.4 Accessibility 53
8.1.5 Design Envelope 54
8.1.6 Component Body Centering 54
8.1.7 Mounting Over Conductive Areas 54
8.1.8 Clearances 54
8.1.9 Physical Support 55
8.1.10 Heat Dissipation 56
8.1.11 Stress Relief 56
8.2 General Attachment Requirements 58
8.2.1 Through-Hole 58
8.2.2 Surface Mounting 58
8.2.3 Mixed Assemblies 58
8.2.4 Soldering Considerations 58
8.2.5 Connectors and Interconnects 58
8.2.6 Fastening Hardware 59
8.2.7 Stiffeners 60
8.2.8 Lands for Flattened Round Leads 61
8.2.9 Solder Terminals 61
8.2.10 Eyelets 63
8.2.11 Special Wiring 63
8.2.12 Heat Shrinkable Devices 63
8.2.13 Bus Bar 64
8.2.14 Flexible Cable 64
8.3 Through-Hole Requirements 64
8.3.1 Leads Mounted in Through-Holes 64
8.4 Standard Surface Mount Requirements 68
8.4.1 Surface-Mounted Leaded Components 68
8.4.2 Flat-pack Components 68
8.4.3 Ribbon Lead Termination 68
8.4.4 Round Lead Termination 68
8.4.5 Component Lead Sockets 68
8.5 Fine Pitch SMT (Peripherals) 69
8.6 Bare Die 69
8.6.1 Wire Bond 69
8.6.2 Flip Chip 69
8.6.3 Chip Scale 69
8.7 Tape Automated Bonding 69
8.8 Solderball 69

9.0 HOLES/INTERCONNECTIONS 70
9.1 General Requirements for Lands with Holes 70
9.1.1 Land Requirements 70

9.1.2 Annular Ring Requirements 70
9.1.3 Thermal Relief in Conductor Planes 71
9.1.4 Lands for Flattened Round Leads 71
9.2 Holes 71
9.2.1 Location 71
9.2.2 Hole Location Tolerances 71
9.2.3 Quantity 71
9.2.4 Spacing of Adjacent Holes 71
9.2.5 Hole Pattern Variation 72
9.2.6 Aspect Ratio 72
9.2.7 Blind and Buried Vias 72

10.0 GENERAL CIRCUIT FEATURE REQUIREMENTS 73
10.1 Conductor Characteristics 73
10.1.1 Conductor Width and Thickness 73
10.1.2 Electrical Clearance 74
10.1.3 Conductor Routing 74
10.1.4 Conductor Spacing 74
10.1.5 Plating Thieves 74
10.2 Land Characteristics 74
10.2.1 Manufacturing Allowances 74
10.2.2 Lands for Surface Mounting 74
10.2.3 Test Points 74
10.2.4 Orientation Symbols 74
10.3 Large Conductive Areas 75

11.0 DOCUMENTATION 75
11.1 Special Tooling 75
11.2 Layout 75
11.2.1 Viewing 75
11.2.2 Accuracy and Scale 75
11.2.3 Layout Notes 78
11.2.4 Automated-Layout Techniques 78
11.3 Deviation Requirements 78
11.4 Phototool Considerations 78

12.0 QUALITY ASSURANCE 78
12.1 Conformance Test Specimen 79
12.2 Material Quality Assurance 79
12.3 Conformance Evaluations 79
12.3.1 Specimen Quantity and Location 79
12.3.2 Specimen Identification 79
12.3.3 General Specimen Requirements 80
12.4 Individual Specimen Design 81
12.4.1 Specimen A and B (Plated Hole Evaluation) 81
12.4.2 Specimen C (Plating Adhesion and Surface Solderability) 81
12.4.3 Specimen D (Interconnection Resistance and Continuity) 81
12.4.4 Specimen E and H (Insulation Resistance) 82
12.4.5 Registration Specimen 83
12.4.6 Specimen G (Solder Resist Adhesion) 88
12.4.7 Specimen M (Optional) 88
12.4.8 Specimen N (Optional) 88
12.4.9 Specimen S 88
12.4.10 Specimen T 88
12.4.11 Process Control Test Specimen 88

APPENDIX A 94

INDEX 95

Figures

Figure 3-1 Test Land Free Area for Parts and Other Intrusions 10
Figure 3-2 Test Land Free Area for Tall Parts 10
Figure 3-3 Probing Test Lands 10
Figure 3-4 Example of usable area calculation, mm 13
Figure 3-5 Printed board density evaluation 15
Figure 5-1 Example of printed board size standardization, mm 26
Figure 5-2 Typical asymmetrical constraining-core configuration 27
Figure 5-3A Multilayer Metal Core Board with Two Symmetrical Copper-Invar-Copper Constraining Cores 27
Figure 5-3B Symmetrical Constraining Core Board with a Copper-Invar-Copper Center Core 27
Figure 5-4 Advantages of positional tolerance over bilateral tolerance, mm 29
Figure 5-5A Example of location of a pattern of plated-through holes, mm 31
Figure 5-5B Example of a pattern of tooling/mounting holes, mm 31
Figure 5-5C Example of location of a conductor pattern using fiducials, mm 32
Figure 5-5D Example of printed board profile location and tolerance, mm 32
Figure 5-5E Example of a printed board drawing utilizing geometric dimensioning and tolerancing, mm 33
Figure 5-6 Fiducial clearance requirements 33
Figure 5-7 Fiducials, mm 34
Figure 5-8 Example of connector key slot location and tolerance, mm 35
Figure 6-1 Voltage/ground distribution concepts 36
Figure 6-2 Single reference edge routing 37
Figure 6-3 Circuit distribution 37
Figure 6-4 Conductor thickness and width for internal and external layers 38
Figure 6-5 Transmission line printed board construction 42

Figure 6-6 Capacitance vs. conductor width and dielectric thickness for microstrip lines, mm. 44
Figure 6-7 Capacitance vs. conductor width and spacing for striplines, mm. 45
Figure 6-8 Single conductor crossover 45
Figure 7-1 Component clearance requirements for automatic component insertion on through hole technology printed board assemblies. (in.) 48
Figure 7-2 Relative coefficient of thermal expansion (CTE) comparison 51
Figure 8-1 Component orientation for boundaries and/or wave solder applications 54
Figure 8-2 Component body centering 54
Figure 8-3 Axial-leaded component mounted over conductors 54
Figure 8-4 Uncoated board clearance 55
Figure 8-5 Clamp-mounted axial-leaded component 55
Figure 8-6 Adhesive-bonded axial-leaded component 55
Figure 8-7 Mounting with feet or standoffs 56
Figure 8-8 Heat dissipation examples 57
Figure 8-9 Lead bends 57
Figure 8-10 Typical Lead configurations 58
Figure 8-11 Board edge tolerancing 60
Figure 8-12 Lead-in chamfer configuration 60
Figure 8-13 Typical keying arrangement 60
Figure 8-14 Two-part connector 61
Figure 8-15 Edge-board adapter connector 61
Figure 8-16 Round or flattened (coined) lead joint description 62
Figure 8-17 Standoff terminal mounting, mm 62
Figure 8-18 Dual hole configuration for interfacial and interlayer terminal mountings 63
Figure 8-19 Partially clinched through-hole leads 64
Figure 8-20 Dual in-line package (DIP) lead bends 65
Figure 8-21 Solder in the lead bend radius 66
Figure 8-22 Two-lead radial-leaded components 66
Figure 8-23 Radial two-lead component mounting, mm 66
Figure 8-24 Meniscus clearance, mm 66
Figure 8-25 “TO” can radial-leaded component, mm 66
Figure 8-26 Perpendicular part mounting, mm 67
Figure 8-27 Flat-packs and Quad Flat-packs 67
Figure 8-28 Examples of configuration of ribbon leads for through-hole mounted flat-packs 67
Figure 8-29 Metal power packages with compliant leads 67
Figure 8-30 Metal power package with resilient spacers 67
Figure 8-31 Metal power package with non-compliant leads 67
Figure 8-32 Examples of flat-pack surface mounting 69
Figure 8-33 Round or coined lead 69
Figure 8-34 Configuration of ribbon leads for planar mounted flat-packs 69
Figure 8-35 Heel mounting requirements 69
Figure 9-1 Examples of modified land shapes 70
Figure 9-2 External annular ring 71
Figure 9-3 Internal annular ring 71
Figure 9-4 Typical thermal relief in planes 72
Figure 10-1 Example of conductor beef-up or neck-down 74
Figure 10-2 Conductor optimization between lands 75
Figure 10-3 Etched Conductor Characteristics 76
Figure 11-1 Flow Chart of Printed Board Design/ Fabrication Sequence 77
Figure 11-2 Multilayer Board Viewing 78
Figure 11-3 Solder resist windows 78
Figure 12-1 Location of test circuitry 80
Figure 12-2 Test Specimen A and B, mm 81
Figure 12-3 Test Specimen A and B (conductor detail) 82
Figure 12-4 Specimen C, external layers only, mm 82
Figure 12-5 Test Specimen D, mm 83
Figure 12-5 cont. 10 Layer Example 84
Figure 12-6 Example of a 10 layer specimen D, modified to include blind and buried vias 85
Figure 12-7 Test Specimen D for process control of 4 layer boards 86
Figure 12-8 Specimen E, mm 86
Figure 12-9 Optional Specimen H, mm 87
Figure 12-10 Comb pattern examples 87
Figure 12-11 “Y” pattern for chip component cleanliness test pattern 88
Figure 12-12 Test Specimen F, mm 89
Figure 12-13 Test Specimen R, mm 90
Figure 12-14 Worst-case hole/land relationship 90
Figure 12-15 Test Specimen G, mm 91
Figure 12-16 Test Specimen M, surface mounting solderability testing, mm 91
Figure 12-17 Test Specimen N, surface mounting bond strength and peel strength, mm 92
Figure 12-18 Test Specimen S, mm 92
Figure 12-19 Systematic path for implementation of statistical process control (SPC) 93

Tables

Table 3-1 PWB Design/Performance Tradeoff Checklist Considerations 5
Table 3-2 Component Grid Areas 14
Table 4-1 Typical Properties of Common Dielectric Materials 16
Table 4-2 Environmental Properties of Common Dielectric Materials 17

Table 4-3 Final Finish, Surface Plating Coating Requirements 20

Table 4-4 Gold Plating Uses 20

Table 4-5 Copper Foil/Film Requirements 21

Table 4-6 Metal Core Substrates 22

Table 4-7 Conformal Coating Functionality 24

Table 5-1 Fabrication Considerations 25

Table 5-2 Normal Assembly Equipment Limits 29

Table 6-1 Electrical Conductor Spacing 39

Table 6-2 Typical Relative Bulk Dielectric Constant of Board Materials 42

Table 7-1 Effects of Material Type on Conduction 46

Table 7-2 Emissivity Ratings for Certain Materials 46

Table 7-3 Board Heatsink Assembly Preferences 49

Table 7-4 Comparative Reliability Matrix Component Lead/Termination Attachment 50

Table 9-1 Minimum Standard Fabrication Allowance for Interconnection Lands 70

Table 9-2 Annular Rings (Minimum) 71

Table 9-3 Minimum Hole Location Tolerance, dtp 72

Table 9-4 Minimum Drilled Hole Size for Buried Vias 73

Table 9-5 Minimum Drilled Hole Size for Blind Vias 73

Table 10-1 Internal Layer Foil Thickness After Processing 73

Table 10-2 External Conductor Thickness After Plating 73

Table 10-3 Conductor Width Tolerances for 46 µm Copper 73

Table 12-1 Specimen Frequency Requirements 80

Generic Standard on Printed Board Design

1.0 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.

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 Organic board design using discrete wiring

The list is a partial summary and is not inherently a part of this generic standard. The documents are a part of the PWB 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 SI (metric) 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 non-mandatory 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 major requirement is function of the completed printed board or printed board assembly.

Class 2 Dedicated Service Electronic Products Includes communications equipment, sophisticated business machines, instruments and military equipment where high

performance and extended life is required, and for which uninterrupted service is desired but is not critical. Certain cosmetic imperfections are allowed.

Class 3 High Reliability Electronic Products Includes the equipment for commercial and military products where continued performance or performance on demand is critical. Equipment downtime cannot be tolerated, and must function when required such as for life support items, or critical weapons systems. Printed boards and printed board assemblies in this class are suitable for applications where high levels of assurance are required and service is essential.

1.6.3 Producibility Level When appropriate this standard will provide three design complexity levels of features, tolerances, measurements, assembly, testing of completion or verification of the manufacturing process that reflect progressive increases in sophistication of tooling, materials or processing and, therefore progressive increases in fabrication cost. These levels are:

Level A General Design Complexity—Preferred
Level B Moderate Design Complexity—Standard
Level C High Design Complexity—Reduced

The producibility levels are not to be interpreted as a design requirement, but a method of communicating the degree of difficulty of a feature between design and fabrication/assembly facilities. The use of one level for a specific feature does not mean that other features must be of the same level. Selection should always be based on the minimum need, while recognizing that the precision, performance, conductive pattern density, equipment, assembly and testing requirements determine the design producibility level. The numbers listed within the numerous tables are to be used as a guide in determining what the level of producibility will be for any feature. The specific requirement for any feature that must be controlled on the end item **shall** be specified on the master drawing of the printed board or the printed board assembly drawing.

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-2221 and those listed below, IPC-2221 takes precedence.

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

IPC-A-22 UL Recognition Test Pattern

IPC-T-50 Terms and Definitions for Interconnecting and Packaging Electronic Circuits

IPC-L-109 Specification for Resin Preimpregnated Fabric (Prepreg) for Multilayer Printed Boards

IPC-MF-150 Metal Foil for Printed Wiring Applications

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

IPC-FC-232 Adhesive Coated Dielectric Films for Use as Cover Sheets for Flexible Printed Wiring

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

IPC-D-310 Guidelines for Phototool Generation and Measurement Techniques

IPC-D-317 Design Guidelines for Electronic Packaging Utilizing High-speed Techniques

IPC-D-322 Guidelines for Selecting Printed Wiring Board Sizes Using Standard Panel Sizes

IPC-D-325 Documentation Requirements for Printed Boards

IPC-D-330 Design Guide Manual

IPC-D-350 Printed Board Description in Digital Form

IPC-D-356 Bare Substrate Electrical Test Data Format

IPC-D-422 Design Guide for Press Fit Rigid Printed Board Backplanes

IPC-TM-650 Test Methods Manual

Method 2.4.22 Bow and Twist

IPC-ET-652 Guidelines and Requirements for Electrical Testing of Unpopulated Printed Boards

IPC-CM-770 Printed Board Component Mounting

IPC-SM-780 Component Packaging and Interconnecting with Emphasis on Surface Mounting

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

IPC-SM-785 Guidelines for Accelerated Reliability Testing of Surface Mount Solder Attachments

IPC-MC-790 Guidelines for Multichip Module Technology Utilization

IPC-CC-830 Qualification and Performance of Electrical Insulating Compound for Printed Board

1. The Institute for Interconnecting and Packaging Electronic Circuits, 2215 Sanders Road, Northbrook, IL 60062-6135

IPC-SM-840 Qualification and Performance of Permanent Polymer Coating (Solder Mask) for Printed Boards

IPC-2510 Series

IPC-2511 Generic Requirements for Implementation of Product Manufacturing Description Data and Transfer Methodology

IPC-2513 Drawing Methods for Manufacturing Data Description (formerly IPC-D-351)

IPC-2514 Printed Board Manufacturing Data Description (formerly IPC-D-350)

IPC-2515 Bare Board Product Electrical Testing Data Description (formerly IPC-D-356)

IPC-2516 Assembled Board Product Manufacturing (formerly IPC-D-355)

IPC-2518 Parts List Product Data Description (formerly IPC-D-354)

IPC-2615 Printed Board Dimensions and Tolerances

IPC-4101 Laminate/Prepreg Materials Standard for Printed Boards

IPC-6011 Generic Performance Specification for Printed Boards

IPC-6012 Qualification and Performance Specification for Rigid Printed Boards

IPC-100002 Universal Drilling & Profile Master Drawing

IPC-100047 Composite Test Pattern Basic Dimension Drawing - Ten Layer

IPC-100103 Master Drawing for Capability Test Board (Ten Layer Multilayer Board without Blind or Buried Vias)

SMC-TR-001 An Introduction to Tape Automated Bonding Fine Pitch Technology

2.2 Joint Industry Standards[1]

J-STD-001 Requirements for Soldered Electrical and Electronic Assemblies

J-STD-003 Solderability Tests for Printed Boards

J-STD-005 Requirements for Soldering Pastes

J-STD-006 Requirements for Electronic Grade Solder Alloys and Fluxed and Non-Fluxed Solid Solders for Electronic Soldering Applications

J-STD-012 Implementation of Flip Chip and Chip Scale Technology

J-STD-013 Implementation of Ball Grid Array and Other High Density Technology

2.3 Military[2]

MIL-G-45204 Gold Plating (Electrodeposited)

2.4 Federal[2]

QQ-N-290 Nickel Plating (Electrodeposited)

QQ-A-250 Aluminum Alloy, Plate and Sheet

QQ-S-635 Steel

2.5 American Society for Testing and Materials[3]

ASTM-B-152 Copper Sheet, Strip and Rolled Bar

ASTM-B-579 Standard Specification for Electrodeposited Coating of Tin-Lead Alloy (Solder Plate)

2.6 Underwriters Labs[4]

UL-746E Standard Polymeric Materials, Material used in Printed Wiring Boards

2.7 IEEE[5]

IEEE 1149.1 Standard Test Access Port and Boundary-Scan Architecture

2.8 ANSI[6]

ANSI/EIA 471 Symbol and Label for Electrostatic Sensitive Devices

3.0 GENERAL REQUIREMENTS

The information contained in this section describes the general parameters to be considered by all disciplines prior to and during the design cycle.

Designing the physical features and selecting the materials for a printed wiring board involves balancing the electrical, mechanical and thermal performance as well as the reliability, manufacturing and cost of the board. The tradeoff checklist (see Table 3-1) identifies the probable effect of

2. Application for copies should be addressed to Standardization Documents Order Desk, Building 4D, 700 Robbins Avenue, Philadelphia, PA 19111-5094
3. American Society for Testing and Materials, 100 Barr Habor Drive, West Conshohocken, PA 19428-2959
4. Underwriters Labs, 333 Pfngsten Road, Northbrook, IL 60062-2002
5. IEEE, 445 Hoes Lane, P.O. Box 1331, Piscataway, NJ 08855-1331
6. ANSI, 655 15th Street N.W., Suite 300, Washington, DC 20005-5794