ANSI/IPC-2221
February 1998  A standard developed by IPC
Supersedes IPC-D-275
September 1991

IPA/2221
Generic Standard on
Printed Board Design

ANSI/IPC-2221
February 1998  A standard developed by IPC
Supersedes IPC-D-275
September 1991

IPA/2221
Generic Standard on
Printed Board Design

ANSI/IPC-2221
February 1998  A standard developed by IPC
Supersedes IPC-D-275
September 1991

IPA/2221
Generic Standard on
Printed Board Design

ANSI/IPC-2221
February 1998  A standard developed by IPC
Supersedes IPC-D-275
September 1991

IPA/2221
Generic Standard on
Printed Board Design

ANSI/IPC-2221
February 1998  A standard developed by IPC
Supersedes IPC-D-275
September 1991

IPA/2221
Generic Standard on
Printed Board Design

ANSI/IPC-2221
February 1998  A standard developed by IPC
Supersedes IPC-D-275
September 1991

IPA/2221
Generic Standard on
Printed Board Design

ANSI/IPC-2221
February 1998  A standard developed by IPC
Supersedes IPC-D-275
September 1991

IPA/2221
Generic Standard on
Printed Board Design

ANSI/IPC-2221
February 1998  A standard developed by IPC
Supersedes IPC-D-275
September 1991

IPA/2221
Generic Standard on
Printed Board Design
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 Productivity 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 .............. 5
  2.6 Underwriters Labs .............................................. 5
  2.7 IEEE ................................................................... 3
  2.8 ANSI .................................................................. 2
3.0 GENERAL REQUIREMENTS ...................................... 4
  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....................... 5
  3.5.2 Boundary Scan Testing ....................................... 5
  3.5.3 Functional Test Concern for Printed Board Assemblies ................................................................. 7
  3.5.4 In-Circuit Test Concerns for Printed Board Assemblies ................................................................. 7
  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
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>9.1.2</td>
<td>Annular Ring Requirements</td>
<td>70</td>
</tr>
<tr>
<td>9.1.3</td>
<td>Thermal Relief in Conductor Planes</td>
<td>71</td>
</tr>
<tr>
<td>9.1.4</td>
<td>Lands for Flattened Round Leads</td>
<td>71</td>
</tr>
<tr>
<td>9.2</td>
<td>Holes</td>
<td>71</td>
</tr>
<tr>
<td>9.2.1</td>
<td>Location</td>
<td>71</td>
</tr>
<tr>
<td>9.2.2</td>
<td>Hole Location Tolerances</td>
<td>71</td>
</tr>
<tr>
<td>9.2.3</td>
<td>Quantity</td>
<td>71</td>
</tr>
<tr>
<td>9.2.4</td>
<td>Spacing of Adjacent Holes</td>
<td>71</td>
</tr>
<tr>
<td>9.2.5</td>
<td>Hole Pattern Variation</td>
<td>72</td>
</tr>
<tr>
<td>9.2.6</td>
<td>Aspect Ratio</td>
<td>72</td>
</tr>
<tr>
<td>9.2.7</td>
<td>Blind and Buried Vias</td>
<td>72</td>
</tr>
</tbody>
</table>

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

| 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 | INDEX | 94 | 95 |

<table>
<thead>
<tr>
<th>Figures</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Figure 3-1</td>
<td>Test Land Free Area for Parts and Other Intrusions</td>
</tr>
<tr>
<td>Figure 3-2</td>
<td>Test Land Free Area for Tall Parts</td>
</tr>
<tr>
<td>Figure 3-3</td>
<td>Probing Test Lands</td>
</tr>
<tr>
<td>Figure 3-4</td>
<td>Example of usable area calculation, mm</td>
</tr>
<tr>
<td>Figure 3-5</td>
<td>Printed board density evaluation</td>
</tr>
<tr>
<td>Figure 5-1</td>
<td>Example of printed board size standardization, mm</td>
</tr>
<tr>
<td>Figure 5-2</td>
<td>Typical asymmetrical constraining-core configuration</td>
</tr>
<tr>
<td>Figure 5-3A</td>
<td>Multilayer Metal Core Board with Two Symmetrical Copper-Invar-Copper Constraining Cores</td>
</tr>
<tr>
<td>Figure 5-3B</td>
<td>Symmetrical Constraining Core Board with a Copper-Invar-Copper Center Core</td>
</tr>
<tr>
<td>Figure 5-4</td>
<td>Advantages of positional tolerance over bilateral tolerance, mm</td>
</tr>
<tr>
<td>Figure 5-5A</td>
<td>Example of location of a pattern of plated-through holes, mm</td>
</tr>
<tr>
<td>Figure 5-5B</td>
<td>Example of a pattern of tooling/mounting holes, mm</td>
</tr>
<tr>
<td>Figure 5-5C</td>
<td>Example of location of a conductor pattern using fiducials, mm</td>
</tr>
<tr>
<td>Figure 5-5D</td>
<td>Example of printed board profile location and tolerance, mm</td>
</tr>
<tr>
<td>Figure 5-5E</td>
<td>Example of a printed board drawing utilizing geometric dimensioning and tolerancing, mm</td>
</tr>
<tr>
<td>Figure 5-6</td>
<td>Fiducial clearance requirements</td>
</tr>
<tr>
<td>Figure 5-7</td>
<td>Fiducials, mm</td>
</tr>
<tr>
<td>Figure 5-8</td>
<td>Example of connector key slot location and tolerance, mm</td>
</tr>
<tr>
<td>Figure 6-1</td>
<td>Voltage/ground distribution concepts</td>
</tr>
<tr>
<td>Figure 6-2</td>
<td>Single reference edge routing</td>
</tr>
<tr>
<td>Figure 6-3</td>
<td>Circuit distribution</td>
</tr>
<tr>
<td>Figure 6-4</td>
<td>Conductor thickness and width for internal and external layers</td>
</tr>
<tr>
<td>Figure 6-5</td>
<td>Transmission line printed board construction</td>
</tr>
<tr>
<td>Table</td>
<td>Description</td>
</tr>
<tr>
<td>-------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>4-3</td>
<td>Final Finish, Surface Plating Coating Requirements</td>
</tr>
<tr>
<td>4-4</td>
<td>Gold Plating Uses</td>
</tr>
<tr>
<td>4-5</td>
<td>Copper Foil/Film Requirements</td>
</tr>
<tr>
<td>4-6</td>
<td>Metal Core Substrates</td>
</tr>
<tr>
<td>4-7</td>
<td>Conformal Coating Functionality</td>
</tr>
<tr>
<td>5-1</td>
<td>Fabrication Considerations</td>
</tr>
<tr>
<td>5-2</td>
<td>Normal Assembly Equipment Limits</td>
</tr>
<tr>
<td>6-1</td>
<td>Electrical Conductor Spacing</td>
</tr>
<tr>
<td>6-2</td>
<td>Typical Relative Bulk Dielectric Constant of Board Materials</td>
</tr>
<tr>
<td>7-1</td>
<td>Effects of Material Type on Conduction</td>
</tr>
<tr>
<td>7-2</td>
<td>Emissivity Ratings for Certain Materials</td>
</tr>
<tr>
<td>7-3</td>
<td>Board Heatsink Assembly Preferences</td>
</tr>
<tr>
<td>7-4</td>
<td>Comparative Reliability Matrix Component Lead/Termination Attachment</td>
</tr>
<tr>
<td>9-1</td>
<td>Minimum Standard Fabrication Allowance for Interconnection Lands</td>
</tr>
<tr>
<td>9-2</td>
<td>Annular Rings (Minimum)</td>
</tr>
<tr>
<td>9-3</td>
<td>Minimum Hole Location Tolerance, dtp</td>
</tr>
<tr>
<td>9-4</td>
<td>Minimum Drilled Hole Size for Buried Vias</td>
</tr>
<tr>
<td>9-5</td>
<td>Minimum Drilled Hole Size for Blind Vias</td>
</tr>
<tr>
<td>10-1</td>
<td>Internal Layer Foil Thickness After Processing</td>
</tr>
<tr>
<td>10-2</td>
<td>External Conductor Thickness After Plating</td>
</tr>
<tr>
<td>10-3</td>
<td>Conductor Width Tolerances for 46 µm Copper</td>
</tr>
<tr>
<td>12-1</td>
<td>Specimen Frequency Requirements</td>
</tr>
</tbody>
</table>
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 criti-
cal. 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 essen-
tial.

### 1.6.3 Producibility Level

When appropriate this standard will provide three design complexity levels of features, tol-
erances, measurements, assembly, testing of completion or
verification of the manufacturing process that reflect pro-
gressive increases in sophistication of tooling, materials or
processing and, therefore progressive increases in fabrica-
tion 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, per-
formance, 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 produc-
ibility 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 Elec-
tronic Circuits (IPC)

- **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 Mea-
  surement 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-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
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
MIL-G-45204 Gold Plating (Electrodeposited)

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

2.7 IEEE
IEEE 1149.1 Standard Test Access Port and Boundary-Scan Architecture

2.8 ANSI
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

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