The Institute for Interconnecting and Packaging Electronic Circuits

IPC-D-325A

Documentation Requirements for Printed Boards, Assemblies and Support Drawings

IPC-D-325A
May 1995

A standard developed by the Institute for Interconnecting and Packaging Electronic Circuits

Original Publication
January 1987

2215 Sanders Road
Northbrook, Illinois
60062-6135

Tel *847 509.9700*
Fax *847 509.9798*
URL: *http://www.ipc.org*

Standardization

In May 1995 the IPC's Technical Activities Executive Committee adopted Principles of Standardization as a guiding principle of IPC's standardization efforts

Standards Should:
• Show relationship to DFM & DFE
• Minimize time to market
• Contain simple (simplified) language
• Just include spec information
• Focus on end product performance
• Include a feed back system on use and problems for future improvement

Standards Should Not:
• Inhibit innovation
• Increase time-to-market
• Keep people out
• Increase cycle time
• Tell you how to make something
• Contain anything that cannot be defended with data

Notice

IPC Standards and Publications are designed to serve the public interest through eliminating misunderstandings between manufacturers and purchasers, facilitating interchangeability and improvement of products, and assisting the purchaser in selecting and obtaining with minimum delay the proper product for his particular need. Existence of such Standards and Publications shall not in any respect preclude any member or nonmember of IPC from manufacturing or selling products not conforming to such Standards and Publication, nor shall the existence of such Standards and Publications preclude their voluntary use by those other than IPC members, whether the standard is to be used either domestically or internationally.

Recommended Standards and Publications are adopted by IPC without regard to whether their adoption may involve patents on articles, materials, or processes. By such action, IPC does not assume any liability to any patent owner, nor do they assume any obligation whatever to parties adopting the Recommended Standard or Publication. Users are also wholly responsible for protecting themselves against all claims of liabilities for patent infringement.

The material in this standard was developed by the IPC-D-325 Task Group (2-22a) of the Documentation and Information Committee (2-20) of the Institute for Interconnecting and Packaging Electronic Circuits.

IPC-D-325A

THE INSTITUTE FOR INTERCONNECTING AND PACKAGING ELECTRONIC CIRCUITS

Documentation Requirements for Printed Boards, Assemblies and Support Drawings

Developed by the IPC-D-325 Task Group of the Documentation and Information Committee of the Institute for Interconnecting and Packaging Electronic Circuits

Users of this standard 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

Acknowledgment

Any Standard involving a complex technology draws material from a vast number of sources. While the principal members of the IPC-D-325 Task Group (2-22a) of the IPC Documentation and Information Committee are shown below, it is not possible to include all of those who assisted in the evolution of this Standard. To each of them, the members of the IPC extend their gratitude.

Documentation and Information Committee

Chairman
Leon Cohen
Formation, Inc.

IPC-D-325 Task Group

Chairman
Leon Cohen
Formation, Inc.

Vice Chairman
Charles Harbin
Teledyne Brown Eng.

Technical Liaison of the IPC Board of Directors

Stanley Gentry
Noble Industries, Ltd.

Cohen, Leon, Formation Inc.
Corbett, David, Defense Electronic Supply Center
DiFranza, Michele, The Mitre Corp.
Ferrari, Gary, Tech Circuits Inc.
Garabedian, Robert, Printed Circuit Corporation
Grande, Paul, U.S. Navy
Harbin, Charles, Teledyne Brown Engineering
Hymes, Les, Les Hymes Associates
Kemp, Cindy, Martin Marietta Corp.
Korchynsky, Stephen, Loral Federal Systems
Kotecki, George, Northrop Grumman Corporation
Kurtz, Thomas, Magnavox Electronic Systems Co.
Morton, John, Loral Federal Systems
Parham, Terry, Tandem Computers Inc.
Rassai, David, 3COM Corporation
Rietdorf, Bruce, Magnavox Electronic Systems Co.
Rosser, Jerald, Hughes Aircraft Co.
Rumps, Don, AT&T Technology Systems
Scherff, Roddy, Texas Instruments Inc.
Tande, Marshall, Magnavox Electronic Systems Co.
Younger, Wally, Nelco Technology Inc.

Table of Contents

Tables

Figures

Documentation Requirements for Printed Boards, Assemblies and Support Drawings

1.0 SCOPE

This standard establishes requirements and other considerations for the documentation of printed boards and printed board assemblies.

1.1 Purpose The purpose of this standard is to establish the general requirements for the preparation of drawings necessary to fully describe end product printed boards, printed board assemblies and related support drawings. Special emphasis is given to the technical requirements necessary to fully describe the fabrication and assembly of various types of printed boards. Regardless of material, construction, layer count, special fabrication requirements, or end product usage, the documentation package may include, but not be limited to the following:

- Master Drawing Requirements
- Specifications
- Board Definition
- Artwork/Phototooling
- Soldermask Requirements
- Master Pattern Drawing
- Production Master
- Assembly Drawing and Parts List
- Electrical Test Requirements
- Final Schematic/Logic Diagram
- Related Support Drawings
- Artwork Plot Data
- Excellon Drill Data

Refer to IPC-D-275, "Design Standard for Rigid Printed Boards and Rigid Printed Board Assemblies," regarding all subjects pertaining directly to design.

This standard may be used for both commercial and military applications. Printed boards and printed board assemblies intended for military usage **shall** be fabricated and/or assembled by a manufacturer that has been qualified to the appropriate military specification, unless otherwise agreed to contractually.

Documentation intended for military electronic equipment **shall** be so noted.

1.1.1 Organization of Information This standard is organized into various sections in order to provide information for the documentation of rigid printed boards and printed board assemblies.

The major sections and their specific emphasis are:

Section 1 – Scope, Purpose and Classification

Section 2 – Applicable Documents

Section 3 – Documentation Requirements

Section 4 – Documentation Package

Section 5 – Sample Figures and Examples

Section 6 – Master Drawing Notes and Check List

Section 7 – Design Outputs

Section 8 – Printed Board Assembly Drawings (Including Figures & Examples)

Section 9 – Printed Board Support Drawings

Section 10– Schematic / Logic Diagrams

1.2 Classification 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 1.2.2), could require level "A" design complexity (preferred producibility) for many of the attributes of the printed board or printed board assembly (see 1.2.3).

1.2.1 Board Types This standard provides design information for different board types. Board types are classified:

Type 1 – Single-Sided Printed Board

Type 2 – Double-Sided Printed Board

Type 3 – Multilayer Board without Blind or Buried Vias

Type 4 – Multilayer Board with Blind and/or Buried Vias

Type 5 – Multilayer Metal-Core Board without Blind or Buried Vias

Type 6 – Multilayer Metal-Core Board with Blind and/or Buried Vias

1.2.2 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 is responsible for determining the class in which his board product belongs.

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.

1.2.3 Producibility Level When appropriate, this standard will provide three levels of design complexity: Levels A, B, and C. Included are special features, tolerances, measurements, assembly, testing of completion, and verification of the manufacturing process. Higher levels of design complexity often result in a reduction of the productibility level and, therefore, increased fabrication costs. These levels are:

Level A — General Design Complexity-Preferred

Level B — Moderate Design Complexity-Standard

Level C — High Design Complexity-Reduced Producibility

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, 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.

1.2.4 Documentation Classification This standard provides three classes for documentation requirements to reflect progressive increases in sophistication of the drawing package. The three classes of documentation are:

Class A — Minimal Documentation

Class B — Moderate Documentation

Class C — Full Documentation

Selection of class should be based on the minimum need, recognizing that less sophisticated classes require more coordination and communication between user and vendor. Requirements for documentation **shall** be specified in the contract order used to procure documentation, equipment or both.

Note: Classification of documentation requirements should not be confused with the classification of end item use, as referenced in other IPC standards and specifications which refer to: Class 1) consumer products; Class 2) general industrial; and Class 3) high reliability equipment. The need to apply documentation practices to a particular class of equipment should depend on the complexity of the interface required to produce the printed board; therefore, any documentation class may be applied to any of the end product equipment categories (classes) as required; examples: Class 2B would be industrial equipment supported by moderate documentation.

There are three classes of documentation requirements. These requirements reflect the differences in sophistication and completeness of the documentation packages. The three classes are defined as follows:

Class A — Minimal Documentation

This class of documentation is identified as minimal and consists of layout and artwork only. Class A documentation is usually used for internal use and requires a good deal of coordination between the user and manufacturer of the board. Information may be incomplete in some instances and relies heavily on in-house agreed to manufacturing processes, such as standard material, standard plating processes, standard tolerances, etc.

Documentation is suitable for the application, where the only requirement is that the manufacturer can produce a functional product from information supplied. It may include, as a minimum, the designer's layout or check plot containing manufacturing notes/instructions and single image artwork master.

Class B — Moderate Documentation

Class B documentation package consists of complete board definition, without any description of the manufacturing allowances that have been incorporated into the design. Contractual drawing requirements may apply. Quality conformance coupons may be defined by the design; their position in relationship to the board or the manufactured panel is optional.

The Class B documentation package requires sufficient clarity such that the information may be reviewed by a board manufacturer, in order to establish product producibility using the artwork or other tooling supplied. Since Class B documentation is manufacturer sensitive, responsibility for various aspects of the manufacturing cycle **shall** be agreed to between user and fabricator.

Class B documentation is specifically prepared to convey maximum information to the manufacturer and includes: a master drawing, all manufacturing notes and a single or multiple image artwork master. Performance specifications may be referenced, and contractual drawing requirements may be applied.

Class C — Full Documentation
Class C is a fully documented procurement package. Documentation is to the extent that the information is self-sufficient and may be sent to multiple vendors, with each producing the identical product. This documentation package requires that the full manufacturing allowances are disclosed and documented. Quality conformance coupons are mandatory, as required by the design, with the location illustrated on the master drawing and artwork establishing the relationship between coupons and the board.

Class C documentation includes a formal master drawing and may include a single/multiple image production master, magnetic tape, NC instructions, reference to material requirements, dielectric constant, glass style resin content, etc.; electrical test data, performance testing and sampling plan call outs. In addition, contractual drawing requirements may apply.

1.3 Interpretation "shall" the emphatic 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.

1.4 Documentation Media Two methods for developing documentation are defined in this standard.

The media are:

- Hard Copy Method
- Automated Data Method

1.4.1 Hard Copy Documentation Hard copy documentation may make reference to automated data for defining conductor routing paths, or hole location data. References **shall** be by note indicating the number assigned and type of storage media containing the data.

1.4.2 Automated Data Automated data may make reference to material, performance or testing requirements that exist in hard copy format. References **shall** be contained in the automated data comment records. Automated data may be used to produce hard copy master drawings; however, all automated data must be complete and in user processable format.

The storage media **shall** contain a label describing the exact format, files and revisions contained within the tape or diskette. This will allow the user to immediately process all data without confusion and delay.

When automated data is requested, the media (or electronic transfer) **shall** be agreed to between user and vendor.

The preferred method for generation of CAD data is IPC-D-35X series. Refer to IPC-D-350: This standard specifies record formats used to describe printed board products with detail sufficient for tooling, manufacturing, and testing requirements. The records are also useful when the manufacturing cycle includes computer aided processes and numerically controlled machines.

Other formats however, i.e., Gerber plot data, Excellon N.C. drill, profile data, or test data may be requested. Refer to 4.3.4.

1.5 Artwork – Generation Most printed boards are now designed using a CAD (Computer Aided Design) System, with the ultimate objective being to generate precision artwork. This is accomplished by producing plot files which are used to drive a precision photo-plotter. The precision photo plotter is used to create the precision plotted artwork (dimensionally stable films). The plotted films may be used for the following purposes:

- Production master (used to fabricate boards). The material may be either film or glass plates, as required.
- Master pattern drawing, part of the drawing package and usually a reproduction of the original artwork (see options #1 and #2, described in 4.2.3).

1.6 Presentation All dimensions and tolerances in this standard are expressed in metric units with millimeters being the main form of dimensional expression. Inches may be shown in brackets as appropriate and are not always a direct conversion depending on the round-off concept or the required precision. Users are cautioned to employ a single dimensioning system, and not intermix millimeters and inches. Reference information is shown in parentheses ( ). Dimensioning should be in accordance with IPC-D-300.

1.7 Conflict – Military Application In the event of any conflict, the following order or precedence **shall** prevail:

1. The procurement contract.
2. The approved master drawing or assembly drawing (supplemented by an approved deviation list, if applicable).
3. This standard.
4. Other applicable documents.

1.8 Order of Precedence In the event of any conflict in the development of new designs, the following order or precedence **shall** prevail:

1. The procurement contract.
2. An approved master drawing or assembly drawing (supplemented by an approved deviation list, if applicable).
3. This standard.
4. Other applicable documents.

2.0 APPLICABLE DOCUMENTS

The following documents form a part of this standard to the extent specified herein. The revision of the document in effect at the time of solicitation **shall** take precedence.

2.1 Institute for Interconnecting and Packaging Electronic Circuits[1]

IPC-A-22 UL Recognition Test Coupon

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

IPC-D-275 Design Standard for Rigid Printed Boards and Rigid Printed Board Assemblies

IPC-RB-276 Qualifications and Performance Specifications for Rigid Printed Boards.

IPC-D-300 Printed Board Dimensions and Tolerances

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

IPC-D-322 Guidelines for Selecting Printed Wiring Board Sizes.

IPC-MC-324 Performance Specification for Metal Core Boards

IPC-D-350 Printed Board Description in Digital Form

IPC-D-351 Printed Board Drawings in Digital Form

IPC-D-352 Electronic Design Data Description for Printed Boards in Digital Form

IPC-D-354 Library Format Description for Printed Boards in Digital Form

IPC-D-355 Printed Board Automated Assembly Description in Digital Form

IPC-D-356 Bare Board Electrical Test Information in Digital Form

IPC-A-600 Acceptability of Printed Boards

IPC-A-610 Acceptability of Printed Board Assemblies

IPC-TM-650 Test Methods Manual

2.1.1 Microsectioning

2.4.22 Bow and Twist

2.6.8 Temperature Cycling, Printed Wiring Board

IPC-ET-652 Guidelines for Electrical Testing of Printed Wiring Boards

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

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

2.2 Department of Defense[2]

2.2.1 Military[2]

MIL-STD-12 Abbreviations for Use on Drawings

MIL-STD-100 Engineering Drawing Practices

MIL-STD-130 Ident Marking of U.S. Military Property

MIL-STD-1686 Electrostatic Discharge Control Program for Protection of Electric Equipment

MIL-S-13949 Plastic Sheet, Laminated, Copper-Clad (for Printed Wiring)

MIL-I-43553 Epoxy Base Ink, Type II

MIL-G-45204 Gold Plating (Electrodeposited)

MIL-I-46058 Insulating Compound, Electrical (for Coating Printed Circuit Assemblies)

MIL-HDBK-263 Electrostatic Discharge Control Handbook

2.2.2 Federal[2]

QQ-A-250 Aluminum and Aluminum Alloy Plate Sheet

QQ-N-290 Nickel Plating (Electrodeposited)

1. Application for copies should be addressed to IPC, 2215 Sanders Road, Northbrook, IL 60062-6135
2. Standardization Documents Order Desk, Building 4D, 700 Robbins Avenue, Philadelphia, PA 19111-5094; (215) 697-2667.