

IPC-2614 2010

Sectional Requirements for Board Fabrication Documentation

March 2010

A standard developed by IPC

Association Connecting Electronics Industries



Copyright IPC-Association Connecting Electronics Industries
Provided by IHS under license with IPC
No reproduction or networking permitted without license from IHS
Full document details: IPC 2614 2010



Licensee=Sandia National Lab/5932237100, User=sam, albert
Not for Resale, 05/28/2014 10:49:52 MDT

The Principles of Standardization

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

Standards Should:

- Show relationship to Design for Manufacturability (DFM) and Design for the Environment (DFE)
- Minimize time to market
- Contain simple (simplified) language
- Just include spec information
- Focus on end product performance
- Include a feedback 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.

IPC Position Statement on Specification Revision Change

It is the position of IPC's Technical Activities Executive Committee that the use and implementation of IPC publications is voluntary and is part of a relationship entered into by customer and supplier. When an IPC publication is updated and a new revision is published, it is the opinion of the TAEC that the use of the new revision as part of an existing relationship is not automatic unless required by the contract. The TAEC recommends the use of the latest revision. Adopted October 6, 1998

Why is there a charge for this document?

Your purchase of this document contributes to the ongoing development of new and updated industry standards and publications. Standards allow manufacturers, customers, and suppliers to understand one another better. Standards allow manufacturers greater efficiencies when they can set up their processes to meet industry standards, allowing them to offer their customers lower costs.

IPC spends hundreds of thousands of dollars annually to support IPC's volunteers in the standards and publications development process. There are many rounds of drafts sent out for review and the committees spend hundreds of hours in review and development. IPC's staff attends and participates in committee activities, typesets and circulates document drafts, and follows all necessary procedures to qualify for ANSI approval.

IPC's membership dues have been kept low to allow as many companies as possible to participate. Therefore, the standards and publications revenue is necessary to complement dues revenue. The price schedule offers a 50% discount to IPC members. If your company buys IPC standards and publications, why not take advantage of this and the many other benefits of IPC membership as well? For more information on membership in IPC, please visit www.ipc.org or call 847/597-2872.

Thank you for your continued support.



IPC-2614-2010

Sectional Requirements for Board Fabrication Documentation

Developed by the Electronic Documentation Technology Committee
(2-40) of IPC

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

Contact:

IPC
3000 Lakeside Drive, Suite 309S
Bannockburn, Illinois
60015-1249
Tel 847 615.7100
Fax 847 615.7105



Acknowledgment

Any document involving a complex technology draws material from a vast number of sources. While the principal members of the Electronic Documentation Technology Committee (2-40) 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.

Electronic Documentation Technology Committee

Chair

Karen McConnell
Lockheed Martin EPI Center

Technical Liaisons of the IPC Board of Directors

Peter Bigelow
IMI Inc.

Sammy Yi
Aptina Imaging Corporation

Electronic Documentation Technology Committee

Mark Ambrose, Digital Fusion Inc.

Gerald Leslie Bogert, Bechtel Plant Machinery, Inc.

Jerome Breche, Sanmina-SCI

Gary Carter, Fujitsu Network Communications

Bini Elhanan, Valor Computerized Systems Ltd.

Gary Ferrari, FTG Circuits

Michael Green, Lockheed Martin Space Systems Company

Thomas Hausherr, PCB Matrix A Valor Company

John Kustner, Trilogic EDA

Bing Lee, Lockheed Martin Space Systems Co.

Michael Matthes, Wittenstein Electronics GmbH

Karen McConnell, Lockheed Martin EPI Center

Kristopher Moyer, Motion Control Engineering Inc

Jack Olson, Caterpillar Inc.

Bruce Rietdorf, Zuken USA, Inc.

Bryan Truax, CADParts & Consulting, LLC



Documents

Document Sharing Hub

Table of Contents

1	SCOPE	1
1.1	Purpose.....	1
1.2	Classification	1
2	APPLICABLE DOCUMENTS	1
3	REQUIREMENTS	2
3.1	Terms and Definitions	2
3.2	Design Outputs	7
3.2.1	Design Verification and Reports	8
3.2.2	Fabrication Instruction	10
3.2.3	Fabrication Data Set Updates	11
3.2.4	Re-identified Board (Re-ident).....	11
3.2.5	Final Fabrication Data Set	11
3.2.6	Data Pattern Master (Original Photo Tool or Artwork Master)	12
3.2.7	Production Master	13
3.2.8	Master Pattern Drawing.....	13
3.2.9	Manual Layout.....	15
3.2.10	CAD System Outputs	15
3.2.11	Readme Files	18
4	FORMATTING REQUIREMENTS	19
4.1	Titles and Subtitles.....	19
4.2	Multiple Image Formats.....	19
4.3	Fabrication Data Numbering/Bare Board Part Number	19
4.4	Contract Number	19
4.5	Application Information.....	20
4.6	Distribution Key	20
5	REVISION CONTROL.....	20
5.1	Fabrication Data Revision Level/Bare Board Revision Level.....	20
5.2	Revision Letters	20
5.3	Temporary Revision and Effectivity	20
5.4	Configuration Control	20
6	DETAILED REQUIREMENT DESCRIPTIONS	21
6.1	Numbering of Notes	21
6.2	Notes, Placement.....	21
6.3	Design Requirements.....	24
6.4	Physical Characteristics	24
6.4.1	Board Construction.....	24
6.4.2	Viewing.....	25
6.4.3	Accuracy and Scale.....	26
6.4.4	Multilayer Boards with Blind and/or Buried Vias	27
6.4.5	Constraining Core Printed Boards	27
6.4.6	Hole Schedule	27
6.4.7	Drill Pattern	28

6.4.8	Profile (Board Outline) Dimensioning	29
6.5	Grid Systems.....	29
6.5.1	Datums	29
6.5.2	Dimensions and Tolerances	30
6.6	Plating, Coating, and Marking Requirements	31
6.6.1	Marking and Legends	31
6.6.2	Electrostatic Discharge (ESD) Marking	31
6.6.3	Solder Resist Coatings	32
6.7	Imagery (Conductive and Non-conductive)	32
6.7.1	Conductor Patterns	32
6.8	Process and Final Product Requirements	32
6.8.1	Drill Size Recommendations for Printed Boards	33
6.9	Special Requirements	33
6.10	Mechanical Requirements	34
6.11	Quality Conformance Coupons	34
6.11.1	Optical Fiducial Targets.....	34
6.12	Board Details Including Construction	34
7	FABRICATION DATA SET CHECK LIST	34
APPENDIX A	Note Selection Hierarchy	35
APPENDIX B	Documentation Package Checklist	54

Tables:

Table 3-1	Documentation Process Review Recommendations	8
Table 3-2	Typical Fabrication Detail Requirements	9
Table 3-3	Fabrication Data Set Hierarchy Description Mode Completeness Recommendations	10
Table 6-1	Generic Manufacturing Note Requirements (Optional [O]; Mandatory [M])	22
Table 6-2	Hole Schedule Example (mm)	28
Table 6-3	Drill Size Recommendations Related to Maximum Board Thickness	33

Figures:

Figure 3-1	Hole descriptions.....	2
Figure 3-2	Conductor Spacing.....	3
Figure 3-3	Embedded Inserted Component	4
Figure 3-4	Application Specific Module with Embedded Components Example.....	4
Figure 3-5	Embedded Component Descriptive Hierarchy.....	5
Figure 3-6	Single Image Master Pattern Drawing	14
Figure 3-7	Multiple Image Master Pattern Drawing	15
Figure 3-8	Example Pattern Configuration Control Chart Double Sided Board	17
Figure 3-9	Example Pattern Configuration Control Chart – Six Layer Multilayer Board	18
Figure 3-10	Example of Readme file	18
Figure 6-1	Cross Sectional Description Example.....	25
Figure 6-2	Stackup XML Schema Graphic.....	25
Figure 6-3	Printed Board Viewing Principles	26
Figure 6-4	Electrostatic Discharge Symbols.....	31
Figure 6-5	Example of Connector Key Slot Location and Tolerance, mm [in]	32

Sectional Requirements for Board Fabrication Documentation

1 SCOPE

This standard establishes the requirements for the documentation of printed circuit board fabrication, and identifies the physical attributes and performance requirements of the unpopulated product. The descriptions apply to rigid, flexible, inorganic substrates or any combination thereof. The construction may be single, double, multilayered, or HDI technology and may include embedded (integrated) components. The requirements pertain to both hard copy and electronic data descriptions.

1.1 Purpose

The purpose of the IPC-2614 is to establish a consistent methodology for the descriptions that are transferred from the Design function to the Board fabricators. Included are the physical characteristics as well as the accept/reject criteria governing the quality of the delivered product. Some techniques will be identified as mandatory information in order to avoid ambiguity; other methods may be shown as a recommendation in order to establish consistent communication between design and manufacturing functions

1.2 Classification

Classification of printed board documentation **shall** meet the requirements of IPC-2611, and **shall** be defined by Grade and Completeness mode.

There are three grades of documentation defined in the IPC-2610 series. A specific grade **shall** consist of a letter to define the differences between hard copy and electronic data. In addition, a second classification, consisting of a number, is required that defines the completeness mode of the documentation procurement package. There are also three levels of completeness.

2 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 IPC standards¹

- IPC-T-50 *Terms and Definitions for Interconnecting and Packaging Electronic Circuits*
- IPC-D-310 *Guidelines for Phototool Generation and Measurement Techniques*
- IPC-2220 *Printed Board Design Standard Requirement Series*
- IPC-2581 *Generic Requirements for Printed Board Assembly Products Manufacturing Description Data and Transfer Methodology*
- IPC-2611 *Generic Requirements for Electronic Product Documentation*
- IPC-2611-1 *Generic Requirements for Electronic Product Data Base Storage Recommendation*
- IPC-2612 *Sectional Requirements for Electronic Diagramming Documentation (schematic and logic descriptions)*
- IPC-2612-1 *Sectional Requirements for Electronic Diagramming Symbol Generation Methodology*
- IPC-2613 *Electrical, Mechanical and Discrete Wiring Part Descriptions Documentation (specification control, source control, wire harness and cabling)⁺*
- IPC-2615 *Printed Board Dimensions and Tolerance*
- IPC-2616 *Sectional Requirements for Assembly Documentation (electronic printed board and module assembly descriptions)⁺*
- IPC-2617 *Sectional Requirements for Board and Assembly Testing (electrical, environmental HAST, HALT, etc. test)⁺*

¹ www.ipc.org
⁺ under consideration

- IPC-2618 *Sectional Requirements for Bill of Material Documentation* (complete listing of parts, materials, and procurement documents)⁺
- IPC-6010 *Performance Specification for Printed Boards Series*
- IPC-7351 *Generic Requirements for Surface Mount Design and Land Pattern Standard*
- J-STD-609 *Marking and Labeling of Components, PCBs and PCBA's to Identify Lead (Pb), Pb-Free and Other Attributes*

2.2 American Society of Mechanical Engineers (ASME) Standards²

ASME Y14.5 *Dimensioning and Tolerancing*

3 REQUIREMENTS

The requirements of IPC-2611 are a mandatory part of this standard. That document describes the generic parts of the documentation data set.

3.1 Terms and Definitions

The definition of all terms **shall** be in accordance with IPC-T-50, IPC-2611 and the following. An asterisk (*) by the term indicates that it is a reproduction from the other standards to assist the reader in interpretation of this standard.

3.1.1 AABUS (as agreed upon between user and supplier)

Indicates additional or alternate requirements to be decided between the user and the supplier in the procurement documentation. Examples include contractual requirements, modifications to purchase documentation and information on the drawing. Agreements can be used to define test methods, conditions, frequencies, categories or acceptance criteria within a test, if not already established.

3.1.2 blind via*

A via extending only to one surface of a printed board. See Figure 3-1.

3.1.3 buried via*

A via that does not extend to the surface of a printed board. See Figure 3-1.

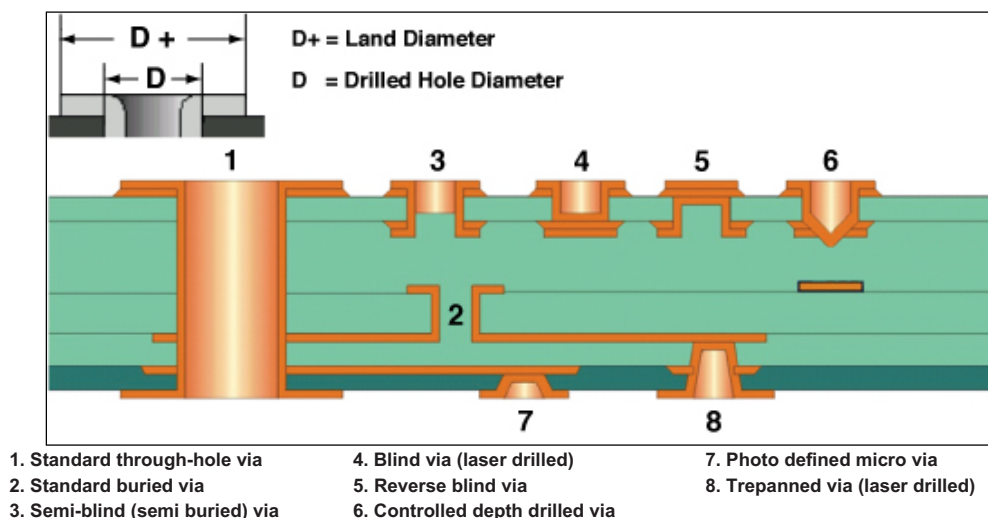


Figure 3-1 Hole descriptions

² www.asme.org
+ under consideration

3.1.4 characteristic impedance*

The resistance of a parallel conductor structure to the flow of alternating current (AC), usually applied to high speed circuits, and normally consisting of a constant value over a wide range of frequencies.

3.1.5 coefficient of thermal expansion (CTE)*

The linear dimensional change of a material per unit change in temperature.

3.1.6 conductor pitch*

The distance between the centers of adjacent conductors.

3.1.7 conductor spacing*

The observable distance between adjacent edges (not center-to-center spacing) of isolated conductive patterns in a conductor layer. See Figure 3-2 (See also "Center-to-Center Spacing.").

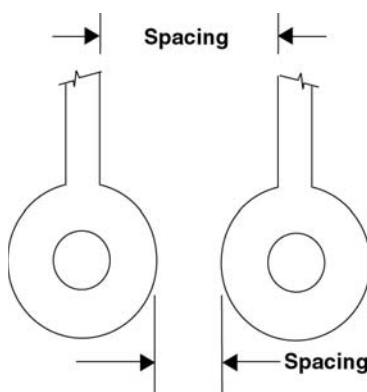


Figure 3-2 Conductor Spacing

3.1.8 center-to-center spacing*

The nominal distance between the centers of adjacent features on any single layer of a printed board.

3.1.9 conductor width*

The observable width of a conductor at any point chosen at random on a printed board as viewed from directly above unless otherwise specified. (See also "Design Width of Conductor" and "Conductor Base Width").

3.1.10 current-carrying capacity*

The maximum electrical current that can be carried continuously by a conductor, under specified conditions, without causing objectionable degradation of electrical and mechanical properties of the product.

3.1.11 decomposition temperature (Td)*

The temperature at which a base laminate material experiences an established percentage of weight loss using Thermogravimetric Analysis (TGA).

3.1.12 dielectric constant*

The ratio of the capacitance of a configuration of electrodes with a specific material as the dielectric between them to the capacitance of the same electrode configuration with a vacuum or air as the dielectric.

3.1.13 dimensional stability*

A measure of the dimensional change of material that is caused by factors such as temperature changes, humidity changes, chemical treatment (aging), and stress exposure.

3.1.14 discrete passive component

A single passive element in its own leaded or surface mount technology (SMT) package. An example would be a single resistor, capacitor, or inductor.

3.1.15 embedded component*

A discrete component that is fabricated as an integral part of a printed board. (See Figure 3-3).

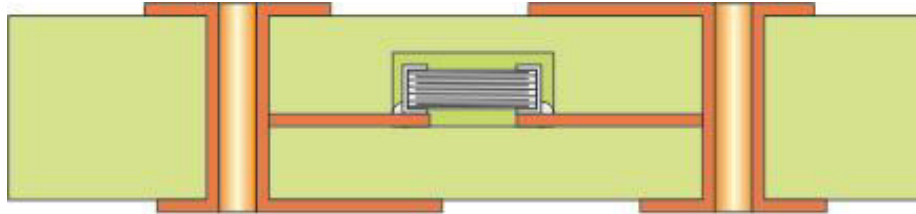


Figure 3-3 Embedded Inserted Component

3.1.16 embedded component (inserted)*

A functional component that is inserted between the layers of the primary interconnect substrate, as opposed to being on the surface.

3.1.17 embedded component (formed)*

A component that is created, from raw materials, inside the primary interconnect substrate, as opposed to being on the surface.

3.1.18 embedded active component (device)

An active device that is inserted between the layers of the primary interconnect substrate. It should have terminals which can connect with the layers of the substrate. It does not include a bare die in the current plastic IC package that is housed inside the encapsulant. (See Figure 3-4).

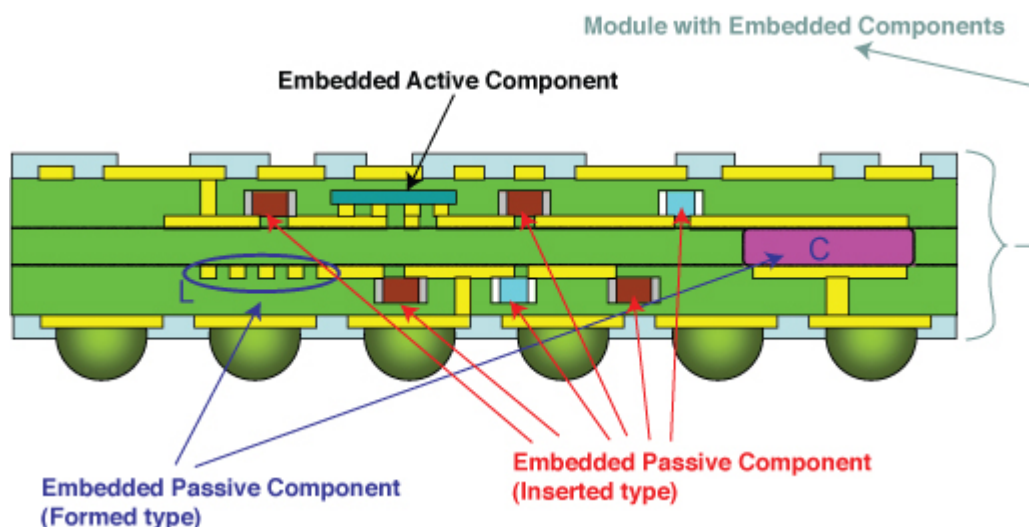


Figure 3-4 Application Specific Module with Embedded Components Example

3.1.19 embedded passive component (device)

A passive component that is formed or otherwise inserted inside the primary interconnect substrate as opposed to being on the surface. (See Figure 3-5).

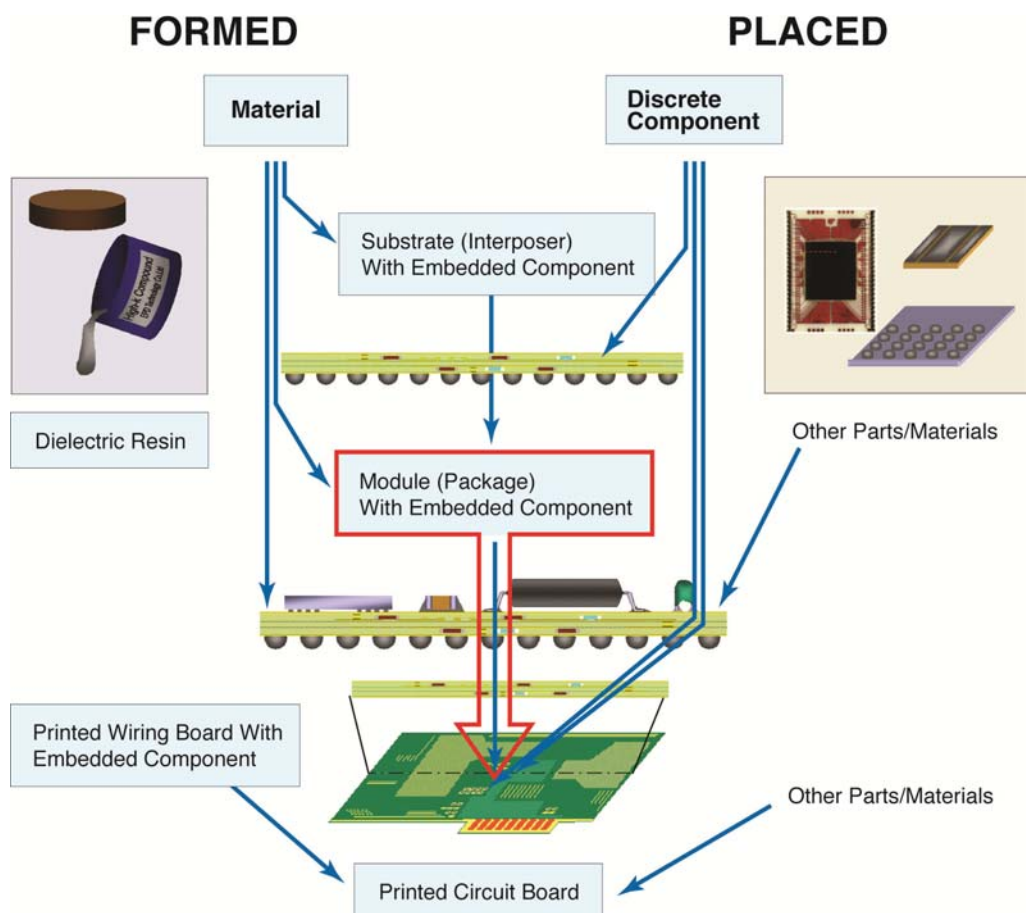


Figure 3-5 Embedded Component Descriptive Hierarchy

3.1.20 fabrication data set

The combination of information provided to the board fabricator that describes the end product requirements of the final product including physical description, fabrication instructions, material references, specification requirements, etc. in any media that facilitates communication between design and fabrication functions.

3.1.21 hole density*

The quantity of holes in a unit area of printed board.

3.1.22 hybrid circuit*

A circuit comprising insulating base material with various combinations of interconnected film conductors, film components, semiconductor dice, passive components and bonding wire.

3.1.23 impedance*

The resistance to the flow of current, represented by an electrical network of combined resistance, capacitance and inductance reaction, in a conductor as seen by an AC source of varying time voltage. The unit of measure is ohms.

3.1.24 inductance*

The property of a conductor that allows it to store energy in a magnetic field induced by a current flowing through it. The unit of measure is henry.

3.1.25 integrated passive component

A general term for multiple passive components that share a substrate and packaging. They may be housed inside the layers of the primary interconnect substrate, which would give them the sub-designation of an “embedded passive component”, or they may be on the surface of a separate substrate that is then placed in an enclosure and surface mounted on the primary interconnect substrate, in which case they would be called “passive arrays” or “passive networks”. (also see hybrid circuit)

3.1.26 interconnection density*

The average number of conductors, based on conductor width and clearance, that may be routed in a prescribed unit area, e.g., cm^2 , considering that there is no restriction within the area to the routing condition and that the conductor length is equal to the unit length of the prescribed area.

3.1.27 known good board (KGB)*

A correctly fabricated printed board that serves as a standard unit by which others can be compared.

3.1.28 land pattern*

A combination of lands that is used for the mounting, interconnection and testing of a particular component.

3.1.29 large-scale integration (LSI)*

An integrated circuit with over 100 gates.

3.1.30 layer*

Stratum of a printed board. Layers are differentiated according to their function (conductor layer, insulating layer) and their location.

3.1.31 multilayer printed board*

The general term for a printed board that consists of rigid or flexible insulation materials and three or more alternate printed wiring and/or printed circuit layers that have been bonded together and electrically interconnected.

3.1.32 passive array

Multiple passive components of like function are formed on the surface of a separate substrate and packaged in a single SMT case. This case is then mounted on the primary interconnect substrate of the system. Examples include an array of capacitors or an array of resistors. (also see hybrid circuit)

3.1.33 passive networks

Multiple passive components of more than one function are formed on the surface of a separate substrate and packaged in a single SMT case. This case is then mounted on the primary interconnect substrate of the system. (also see hybrid circuit)

3.1.34 power plane inductance*

The reactive component inductance, in response to AC noise, seen on a DC backplane system.

3.1.35 printed board*

The general term for completely processed printed circuit and printed wiring configurations. (This includes single-sided, double-sided and multilayer boards with rigid, flexible, and rigid-flex base materials.)

3.1.36 sequential lamination*

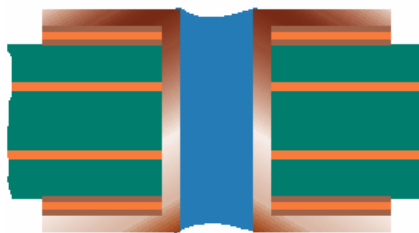
The process of manufacturing multilayer printed boards in which multiple double-sided printed boards with interconnecting holes between conductive patterns on both sides are laminated or combined, after which additional layers (usually single-sided) are attached to the partially completed board stackup.

3.1.37 via*

A plated-through hole that is used as an interlayer connection, but in which there is no intention to insert a component lead or other reinforcing material. (See also "Blind Via" and "Buried Via".)

3.1.38 via, filled (type V via)*

A via with material applied into the via targeting a full penetration and encapsulation of the hole.



3.1.39 voltage plane*

A conductor layer, or portion thereof, that serves as a common voltage source at other than ground potential for an electrical circuit, shielding, or heat sinking. (See also "Ground Plane" and "Signal Plane").

3.2 Design Outputs

The output from the design process, including the CAD system, **shall** be consistent for the products being documented. Consistency refers to the methodology that is used to establish the fabrication data set and the organization of the information, especially the fabrication instructions. Because of technological changes or supplier feedback it is recommended that the process for specifying the end product requirements and the notes that identify them should be reviewed on a periodic basis.

Design cycles usually start with the engineering effort to create a particular functional unit or system. The result of the engineering effort develops the electronic descriptions (diagrams or truth tables), bill of material and product profile information. These descriptions are passed to Circuit Interconnect Design (component arrangement, mounting, and interconnection hierarchy) where the engineering intent is converted to a physical representation. The output of that effort need not be formal but **shall** be sufficient to produce a working prototype of the engineering concepts. Prototypes are tested and modified in order to meet the units' functional requirements, and when satisfied, the documentation is finalized for production. There may be a first article verification that completes the design cycle when all issues have been addressed.

The design cycle may consist of one or many variations and steps and may take place over a long or short period. The time to get a product to market may also impact the data set review cycle and the information being transferred between trading partners. The data set completeness is predicated on many issues however a good review is essential in order to avoid any manufacturing ambiguity. The review of the data set and the documentation methods and details should be based on the recommendations shown in Table 3-1.

Table 3-1 Documentation Process Review Recommendations

Condition	Design Cycle Length	Review Frequency	Comments
Lengthy Design Cycle	One year or more	Yearly basis	Recommend six months
Medium Design Cycle	Six months to a year	Six Month Minimum	Recommend per quarter
Short Design Cycle	Less than six months	Monthly Review	
Change to a referenced standard or specification	Determine applicability specifics	Major review to assess impact	Either during change process or when new data is released
Contemplated change to referenced standard or specification	Determine risk factor related to unfavorable changes	Participate in change process	Determine update of internal procedures
Intending to implement a new technology	Determine time frame	Develop controlled experimentation	Ongoing evaluation until stable

3.2.1 Design Verification and Reports

Each design should be considered in relation to a set of design rules that establish the principles under which the design is accomplished. Some of the rules may be imposed on the engineering principles based on company requirements or by a government authority based on safety or regulatory issues i.e. EMI shielding, hazardous materials, disassembly for recycling.

The physical arrangement of parts and construction of the mounting structure should also be verified according to a set of rules based on the manufacturing capabilities of the supplier(s) who will be involved in the fabrication or assembly process. The Design Rule Checks (DRCs) may be accomplished manually but are most often scripted into the tools used to create the design. The DRCs are based on a specific set of conditions developed by engineering in order to facilitate a smooth conversion from the engineering intent to the fabrication, assembly and testing of the completed product. In most instances the rules do not include manufacturing allowances for variation in the fabrication processes.

The original design rules should be shared with the manufacturing suppliers so that they are aware of the customer expectations. In general, the design rules for printed board fabrication should cover the details defined in Table 3-2.

Table 3-2 Typical Fabrication Detail Requirements

Characteristics	Requirements
Board Details	1. The type, size and shape of the printed board and all tolerance requirements 2. Dielectric separation between layers 3. Bow and twist allowances 4. Overall board thickness requirements including tolerances
Materials	1. Type, Class and Grade of material, including color if applicable 2. Electrical properties of the material i.e. permittivity, loss tangents 3. Plating and coating material(s) type and thickness(es) 4. Marking inks or paint and permanency 5. Embedded component materials
Hole Details	1. The size, location and tolerances for all holes 2. The plating requirements for all holes 3. The hole cleaning requirements including if etchback is required or permitted
Conductor Definition	1. Shape and arrangement of both conductor and non-conductor patterns on each layer of the printed board 2. Conductor width and spacing on the finished printed board 3. Dimensions and tolerances for critical pattern features which may affect circuit performance 4. Conductor layer identification starting with the primary side, the next conductive layer shall be Layer 2 (For assemblies with components on both sides the most complex or densely populated side shall be Layer 1)
Embedded Components	1. Formed passive component requirements 2. Formed active component requirements 3. Inserted passive component requirements (including BOM information and assembly instructions) 4. Inserted active component requirements (including BOM information and assembly instructions)
Marking	1. Printed board identification marking.(may include Bar Code) 2. Printed board revision (may include layer Rev identification) 3. Size, shape and location of reference designation and legend marking, if required 4. Traceability marking and/or date code and serial number (when required) 5. Location and size requirements for fabricator's I.D., UL, ESD marking and cage code 6. Appropriate material marking requirements per J-STD-609
Processing Conditions	1. Processing allowances that were used in the design of the printed board, including but not limited to: a. Conductor width allowance [LMC and MMC] b. Conductor spacing allowance [LMC and MMC] c. Land/hole fabrication allowance [LMC and MMC] d. Solder mask or cover layer registration allowance [DTP] e. Layer to layer registration requirements 2. Applicable processing specifications 3. Location of quality conformance coupons or circuitry
Design Concepts	1. Maximum rated voltage (maximum voltage between two non-connected adjacent conductors with the greatest potential difference) 2. Identification of testing and test point locations (when applicable) 3. Modular grid system(s) used and application (component placement, test points, etc.): a. Metric grid smallest increments. Inch based grid smallest increment
Documentation	1. Terms used on the Fabrication data set shall conform to this standard and IPC-T-50 2. Notes either included on the first sheet(s) or location of notes specified on the first sheet(s)

3.2.2 Fabrication Instruction

The requirements for acceptability in the Fabrication Data Set have traditionally been documented as notes from the user to the supplier. As the technology moves forward these instructions have taken many different forms, however the purpose stays consistent in that details describe the inspectable accept/reject criterion.

Table 3-3 defines the major topics that should be addressed in the instructions and establishes the data completeness mode related to each topic. In a Completeness Mode 1 much of the instruction set is left to the fabricator's best practice methodology. In a Completeness Mode 3 the information can become very prescriptive and in some instance indicates the part number of a supplier whose technology or material **shall** be used to fabricate the printed board. The term "or Equivalent" is used to signify that the fabricator has the option to use an alternative provided that it meets the same characteristics and physical properties as the original. This technique is similar to that of a Specification Control document used for defining unique parts.

The use of any Completeness Mode relates to the relationship between User and Supplier. The least complex Completeness Mode 1 could be used to obtain a quotation in order to see if the fabricators capability matched the Users need. The Mode 1 instruction set could also be used to define the end product requirements, between trading partners who have a history of working together and knowing what each participant in the relationship requires. Mode 2 is used for the occasional order or developing a workable prototype, while Mode 3 is directed toward working with a new supplier who has not built a previous product or "build anywhere" applications. Mode 3 is also recommended for hi-reliability products and long term fabrication cycles. It can also be advantages to use Mode 3 for a prototype intended to be produced in high volume, but not necessarily by the same fabricator.

Table 3-3 Fabrication Data Set Hierarchy Description Mode Completeness Recommendations

Characteristics	Completeness Mode 1	Completeness Mode 2	Completeness Mode 3
Base material requirement	Mandatory reference	Mandatory specification	Mandatory detailed
Board construction	General description	Stackup defined	Specific construction
Conductive Pattern	Optional description	Pattern defined	Restricted electrical
Solder Mask	Optional description	General description	Specific registration
Surface Finish	Fabricator option	Fabricator specific	Specific requirement
Conductor width	General width	Minimum design width	Specific tolerance
Conductor spacing	General spacing	Minimum design spacing	Specific tolerance
Hole size description	General sizes	Specific sizes	Toleranced sizes
Location Fab allowance	Optional description	Bilateral tolerance	Geometric tolerancing
Conductor Fab allowance	Optional description	Minimum width/spacing	Specific tolerance
Annular ring	General description	Specification reference	Specific detail
Impedance description	General non-tested	Broad tolerance	Tight tolerance dimension
Electrical testing	Fabricator option	Open/short continuity	Specific test spec
Via Fill requirements	None	Optional	Per specification
Through-hole Vias	Estimated number	Specific size and number	Toleranced number
Blind Via	Estimated number	Specific size/no./layer	Toleranced number
Buried Via	Est. number/layer	Specific size/no./layer	Toleranced number
Micro Via	Estimated number	Specific size/no./layer	Toleranced number
Component holes	Estimated number	Specific size and number	Toleranced number
Board periphery	General outline	Dimensioned outline	Dimensioned outline
Cut outs or notches	General location	Dimensioned location	Tight tolerance location
Connector Tab plating	General description	Specific details	Toleranced dimension
Selective plating	General description	Specific details	Toleranced dimension
Marking	General description	Specific details	Multiple requirements

3.2.3 Fabrication Data Set Updates

Once the fabrication data set has been created and released, and as subsequent design changes are incorporated, the designer must remain fully cognizant of the impact and ripple effect to other associated data, drawings and documentation. Especially important is the “mandatory requirement and obligation” of updating and revising all associated information. A Printed Board requiring a change **shall** be updated by raising the revisions of the following:

- a) the fabrication drawing, including revisions of all cognizant pattern description;
- b) the “bare board marking revision”; solder paste stencil pattern;
- c) metal core pattern; laminated heatsinks, etc.
- d) an update to a fabrication data set is permissible, provided the following criteria are met: the change **shall** not affect form (shape), fit (dimensional size) and function (electrical characteristics).
- e) the assembly drawing (if the drawing duplicates information on the PWB drawing – e.g., artwork views)

General Rule: The updated printed board must be a direct replacement for the previous revision board. Quite often, the previous revision must be reworked and made equal to the new revision board.

3.2.4 Re-identified Board (Re-ident)

Whenever a printed board requires a change which does affect the form, fit, or function, the board **shall** be assigned a new part number and treated as a new design. This will require the designer to create new documentation for the new (re-identified) board. The reason is that the new board is physically or functionally different than the old board and is not a direct replacement for the old board. The revision in this situation **shall** be treated the same as an initial release; therefore, the fabrication data set, pattern description and board revisions are assigned a dash (—) revision.

When the revisions do not affect form fit or function, the changed board shall be assigned the next revision identification, consistent with the revision system. The concepts of board replacement **shall** consider all aspects of the configuration control, especially when changing technology such as moving from a tin/lead assembly process to a lead-free application. The conductive pattern may not have physically changed; however, the mounting substrate or surface finish materials may have been altered and thus the maintenance, storage and repair condition are sufficiently different and may warrant a configuration management review. During the review many things are considered including all the documentation requirements and a careful decision is made as to whether a new part number is necessary or whether the original can be updated to the next revision level without a problem in manufacturing, field replacement, or future equipment maintenance.

3.2.5 Final Fabrication Data Set

3.2.5.1 Documentation and Electronic Data

The printed board fabrication data set generally consists of the fabrication drawing, master pattern description or reproductions of the masters (film or paper), Bill of Material, drilling data, profile routing data, bare board test data, and artwork plot data. Due to historical industry usage, the word “drawing” may be used in this document to describe the type of information and not the format. All information may be transferred in any of three formats: Hard copy, Soft copy, and Data.

Note: Electronic data may be supplied, in lieu of reproductions (as noted above), prepared in accordance with Grade C IPC-2581 format (preferred), Grade B IPC-D-356, or other standard industry formats. Information may be supplied via electronic communication channels or if necessary in physical electronic media.

3.2.5.2 Computer-Aided Design (CAD Output)

All CAD system output data **shall** meet the requirements of IPC-2580 series format and/or other industry acceptable standard formats AABUS. The agreed format **shall** be documented in the contractual agreements and assigned the grade and completeness description as defined in this standard

3.2.5.3 Fabrication Drawing

The fabrication drawing as a part of the data set **shall** be prepared in accordance with the requirements of this specification. Typical fabrication drawing documentation requirements are shown in Table 3-2. During the design review, the mode (1, 2, or 3) of documentation **shall** be considered, as shown in Table 3-3.

The fabrication drawing describes all details needed to quote, fabricate, and inspect the printed board and may include the following:

- a) Design requirements
- b) Board details including construction
- c) Materials and process specification
- d) Conductor definition
- e) Minimum spacing requirements
- f) Conductive pattern configuration control chart
- g) Fabrication allowances
- h) Solder mask requirements
- i) Marking requirements (traceability)
- j) Silk screen requirements
- k) Bow and twist requirements
- l) Test coupon requirements
- m) Electrical test requirements
- n) Performance requirements
- o) Conductive pattern drawing
- p) Computer-aided Design (CAD) reference data
- q) Numerical control reference data
- r) A graphical representation illustrating the circuit or masking patterns to the fabrication drawing relationship. The purpose of this diagram is to illustrate the correlation between the original photoplotted pattern, fabrication drawing, fabrication pattern drawing and Production Master

3.2.6 Data Pattern Master (Original Photo Tool or Artwork Master)

Original precision data patterns are usually generated by the CAD tool and are usually defined in terms of photoplotting, using a vector or laser photoplotter. The data pattern may be of a single board, an assembly pallet containing several boards (similar or unique designs), or be a data pattern for the fabrication panel. The data pattern **shall** contain as a minimum, the following information:

1. Board part number and revision.
2. Layer number.
3. For composite printed boards, a complete set of quality conformance coupons is required for the top board, the bottom board and the composite board.
4. Three fiducials (or registration features) **shall** be located on each layer of the data pattern so that their centers form an orthogonal set of vectors that represent the datum measurement system. Each additional layer or data pattern **shall** have fiducials located at the same points which, **shall** register with each other, layer to layer.
5. Marking or silkscreen data patterns (when required) **shall** have a minimum of two alignment marks to coincide with the center-line of features located for that purpose on the board surface to be marked.
6. Component pedestal data patterns (when required for no-lead parts) **shall** have a minimum of two fiducials to coincide with the center line of features located on the board surface.
7. Special tooling data patterns: During the formal design review and prior to layout, special tooling requirements which can be generated during design should be considered.

3.2.7 Production Master

When specified in the contract or purchase order, a production master of each layer may be provided as part of the fabrication data set. When a production master is not provided as part of the data set, the fabricator **shall** have responsibility for creating the production master from the data pattern master, the accuracy of which **shall** meet the requirements of the fabrication data set. Quite often, the fabricator is supplied with a data pattern master from which he must produce the production master. The data is often electronically transmitted.

When production masters are supplied as a supplement to the engineering fabrication data set, they **shall** be supplied as readily usable 1:1 precision masters including all manufacturing allowances (phototool stretch/shrinkage, etch factor, plating outgrowth etc.). Production masters may be a single image or multiple image patterns.

3.2.7.1 Single Image Production Master

A single image production master consists of the conductive and non conductive images needed to produce a single board. The single image production master is usually contained in a rectangular panel or pallet. When conformance test circuitry is required in order to validate the production quality the test coupon images **shall** be included in the single image production master.

3.2.7.2 Pallet Image Production Master

A pallet image production master consists of the conductive and non conductive images needed to produce an assembly pallet. The board images on the pallet may be of a single board, duplicated several times, or several board types that are part of a board functional set. The pallet image production master is usually contained in a rectangular panel with individual boards arranged in an area array (rows and columns) or in a random orientation. When conformance test circuitry is required in order to validate the pallet production quality the test coupon images **shall** be included in the pallet image production master. The method of excising individual boards should be determined and the appropriate information **shall** be included in the assembly pallet description either on a separate document or as part of the manufacturing instructions included with the board fabrication data set. The details and edge finishes **shall** be indicated in a manner to provide the accept/reject condition of the final board configuration and tolerance.

3.2.7.3 Panel Image Production Master

A panel image production master consists of the conductive and non conductive images needed to produce one or more individual boards or assembly pallets. The board images on the panel may be of a single board, duplicated several times, or several board types that are part of a board functional set. The panel image production master is usually contained in a rectangular panel with individual boards arranged in area arrays (rows and columns) or in a random orientation. When conformance test circuitry is required in order to validate the panel production quality the test coupon images **shall** be included in the panel image production master. The method of excising individual boards or pallets should be determined and the appropriate information **shall** be included in the arrangement of the boards/pallets to panel descriptions either on a separate document or as part of the manufacturing instructions included with the board fabrication data set. The details and edge finishes **shall** be indicated in a manner to provide the accept/reject condition of the final board configuration and tolerance.

3.2.8 Master Pattern Drawing

The master pattern drawing (Figure 4-1), part of the fabrication data set, may be supplied using one of the following options:

Option #1: A reproduction of the original pattern or data base prepared on a drawing format See Figure 3-6. The pattern layer image(s) may be reproduced using photographic techniques or pen plotted, when practical. All layers of the pattern, including solder masks(s), silk screen(s), laminated heat sinks and solder paste stencil(s), when applicable, **shall** be illustrated; more than one pattern layer may be arranged on one drawing sheet. See Figure 3-7.

Option #2: Electronic data may be supplied, in lieu of reproductions (as noted above), prepared in accordance with IPC-2581 (preferred), or IPC-D-350 Format or other standard industry formats.

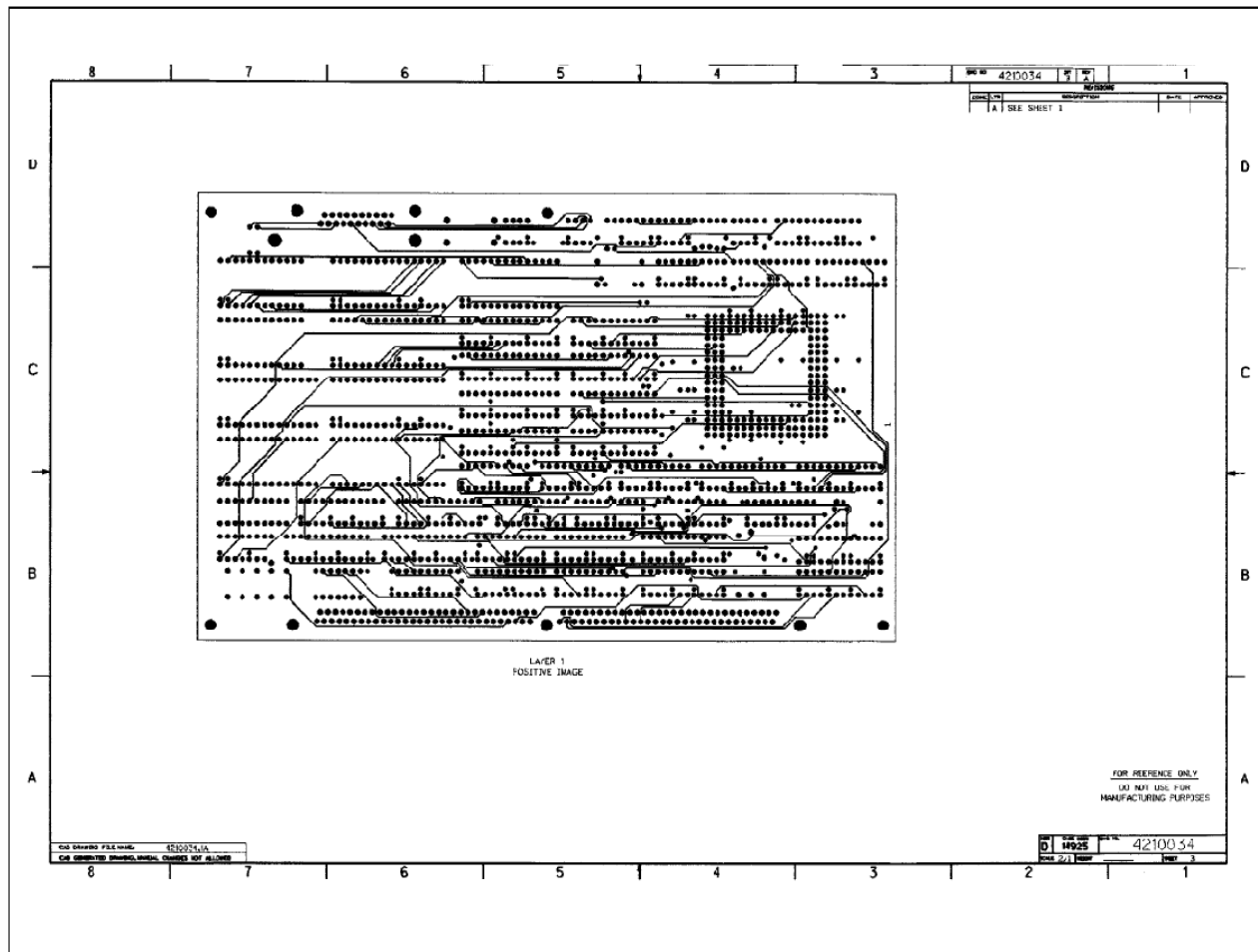


Figure 3-6 Single Image Master Pattern Drawing

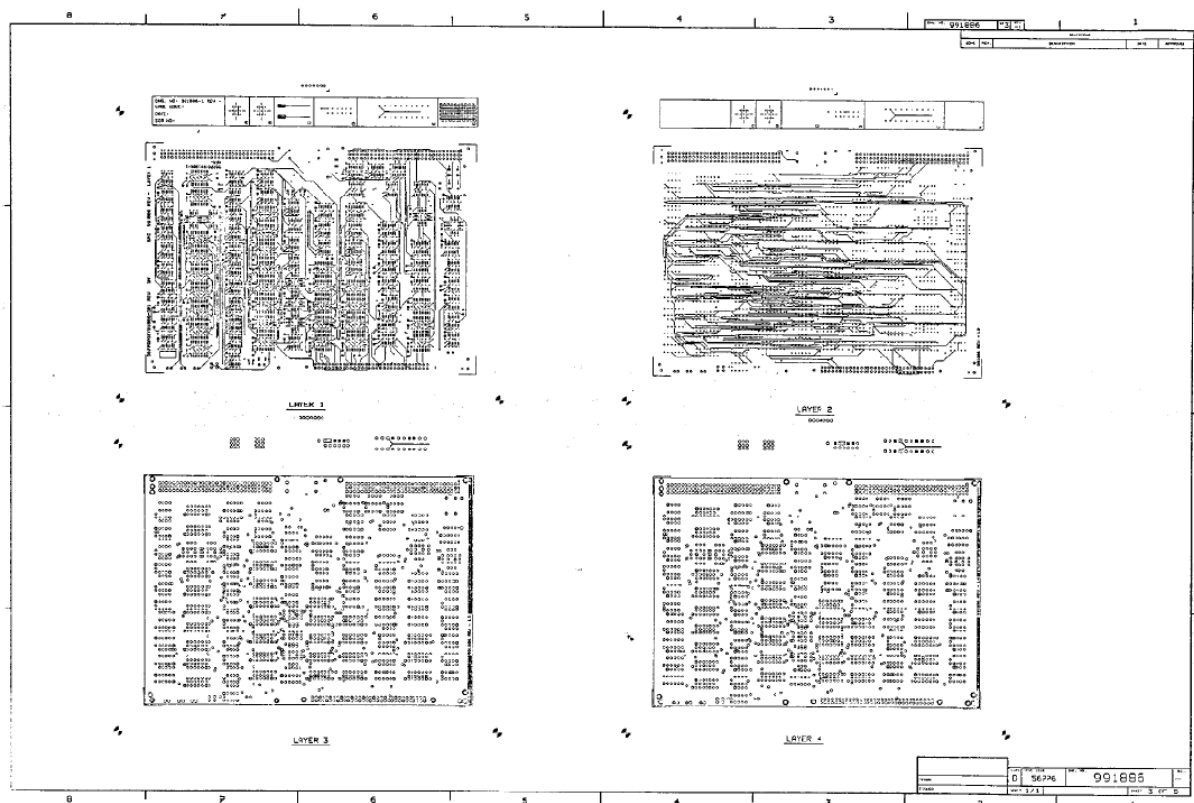


Figure 3-7 Multiple Image Master Pattern Drawing

3.2.9 Manual Layout

Manual layouts should be completed with appropriate notations, marking requirements, and revision level definition. It is necessary to structure this information such that there can be no misunderstanding by anyone viewing the layout. Notes are especially important during the engineering review cycle, the digitizing effort, and whenever the document is to be used by someone other than the originator.

3.2.10 CAD System Outputs

When Computer-Aided Design (CAD) systems are used in the design process, software is used to accomplish some of the tasks which were formerly done manually. These tasks include defining components, component foot prints, associated land patterns, automated placement of components, definition of layers, automated routing of conductors, design rules checking, and generation of check plots, etc.

Automated systems, such as CAD equipment, computer aided engineering (CAE) equipment, computer aided manufacturing (CAM) equipment, test equipment, photoplotters, automated assembly equipment (auto insertion and/or pick and place equipment), automated optical inspection (AOI) equipment, **should** be capable of communicating with each other.

It is recommended that standard industry formats be used when communicating this data between systems. The IPC-2580 series of documents and record formats have been developed to serve as a standard format used to facilitate the interchange or electronic transfer of "Printed Board Description in Digital Form", between CAD systems and other facilities. Electronic transfer and/or delivery of computer generated data (as part of the fabrication data set) should meet the requirements of IPC-2580 series of standards. Archived data should be in accordance with IPC-2581, 2582, 2583, 2584 and 2588.

When using computer aided design (CAD) equipment, the database should include, in complete detail, all information needed to produce the printed board. This information **shall** include all notes, board details including construction and lay-up, materials and process specifications, hole details, conductor definition, minimum spacing requirements, fabrication allowances, solder mask requirements, marking requirements, silk screen requirements, bow and twist requirements, test coupon requirements, electrical test requirements, performance requirements, numerical control reference data, etc.

The use of check plots is important in helping to verify that the data base matches the requirements of design.

IPC-2580 Series format is the preferred output format. The CAD system outputs illustrated below have been in use for many years and are still very popular. The following outputs are shown for illustration purposes only. CAD system outputs may include but are not limited to the following:

- Aperture list (ASCII format)
- Special A/W features (HPGL format)
- Artwork layers (Gerber format)
- Solder mask(s) (Gerber format)
- Silkscreen(s) (Gerber format)
- Solder paste screen(s) (Gerber format)
- N.C. Drill data & drill bit list (Excellon format)
- N.C. Board profile routing data (N.C. data)
- Electrical test data (bare board) (IPC-D-356 format)
- Net list (ASCII format)
- Automatic component insertion data
- Surface mount pick and place data
- Assembly drawing and parts list (HPGL format)
- Functional test data (ATE) (assembled board) (GenRad format)
- Schematic/logic diagram (HPGL format)

3.2.10.1 CAD System Archiving

CAD Systems are usually archived using commercially available configuration data management systems. In lieu of such a system, the user should archive CAD data to a DVD, CDRom or tape drive. Archive should include CAD data files, library data and software version used to create the data.

3.2.10.2 Data Transfer

Generation of master patterns and measurement techniques **shall** be in accordance with the requirements of IPC-D-310. The appropriate pattern tolerances and processing allowances **shall** be incorporated into the production master. Single-image masters, as well as multiple image masters may be provided as part of the fabrication data set. Master patterns are usually provided electronically but may also be produced on film or photographic glass plate media as required. IPC-D-310 specifies the dimensional changes due to changes in temperature and humidity on each of these products.

Note: Modern laser photoplotters have the ability to produce precision patterns within exceptional limits. Typical precision laser plotting accuracy exceeds:

- Image Tolerance: $\pm 12.5 \mu\text{m}$
- Line Width Tolerance: $\pm 12.5 \mu\text{m}$
- Feature Repeatability: $\pm 7.5 \mu\text{m}$
- Accuracy (Overall): $\pm 25 \mu\text{m}$