The IPC-2252, "Design Guide for RF/Microwave Circuit Boards"

Nicholas G. Paulter, Jr.
National Institute of Standards and Technology
Gaithersburg, MD

Abstract
The IPC-2252, "Design Guide for RF/Microwave Circuit Boards,"[1] (superceding the IPC-D-316) was published in July 2002. The IPC-2252 provides information pertaining to the design, fabrication, and test of printed wiring board used in high-frequency (100 MHz to 30 GHz) applications. The design topics include those describing the fabrication procedure and the circuit and board layout. Different types of board interconnects and chip mounting strategies are discussed. This document introduces the IPC-2252 and its contents. The differences between the IPC-2252 and IPC-D-316 are also discussed and a comparison to other IPC documents is provided.

Introduction
The IPC-2252, "Design Guide for RF/Microwave Circuit Boards,"[1] is written as a technical guide for high frequency printed wiring boards. This guide contains information on design processes used in the fabrication of these boards, on methods used to attach other electronic components (such as integrated circuits and discrete devices) to the board, on methods to interconnect other boards, on methods to mount the board into a housing, on the design of the board structure (layering), and on the design of the conductor layout (signal lines and power and ground planes). The IPC-2252 was written to be a guide and not a detailed technical treatise on high-frequency printed wiring board technology. Its primary purpose is to provide information on all the requirements, considerations, and steps necessary to successfully fabricate a radio frequency (RF)/microwave printed wiring board (PWB).

Background
D-21b Task Group
The IPC-2252 was written by the D-21b, the High Frequency Design Task Group. The D-21b is one of three task groups presently within the D-21, the High Speed/High Frequency Design Subcommittee, which in turn is one of the four subcommittees under the D-20, the High-Speed/High-Frequency Committee. Task groups are formed by the subcommittees, with permission from the IPC Technical Activities Executive Committee, to address specific tasks that would otherwise consume too much time for the entire subcommittee membership to address.

The D-21b membership consists primarily of technical people from the automotive and aerospace industries, instrumentation manufacturers, material and laminate suppliers, and printed wiring board manufacturers. Consequently, the technical background of the D-21b membership is quite diverse. This diversity ensures that the IPC-2252 addresses most of the PWB industry concerns for high-frequency PWBs and that it is written to be understandable by people from many industries.

The latest task of the D-21b Task Group was to revise the guide, IPC D-316, "Design Guide for Microwave Circuit Boards Utilizing Soft Substrates."[2] The D-316 was published in 1995 and, due to technical advancement in the printed wiring board industry, was in need of updating. Furthermore, typographical errors existed in some equations that had caused a bit of confusion with users of the D-316 and some information in the document was not presented clearly.

The D-21b was assembled to write the D-316 and, a couple of years after publishing the D-316, the D-21b decided it was necessary to revise the document. Prior to 1998, the activities and business of the D-21b were conducted primarily at two annual meetings, the IPC Annual Meeting and the IPC Expo. In 1998, the D-21b agreed to use e-mail to exchange files for review, to examine suggested changes and comments, and to conduct other discussions. The use of e-mail greatly facilitated discussions and expedited the writing of the IPC-2252, which is the successor to the D-316. (The IPC implemented document renumbering in 2000.)

Intended Audience

As mentioned earlier, the IPC-2252 was written to be a guide and not a detailed technical treatise on high-frequency printed wiring board technology. Although the IPC-2252 may be used by the expert in their respective field within the umbrella of printed wiring board technology, it was primarily written to provide general guidance to the novice, information to managers and administrators, and to be used by the experts needing information outside of their field of expertise (such as a laminate manufacturer wanting to understand high-frequency signal line design issues).

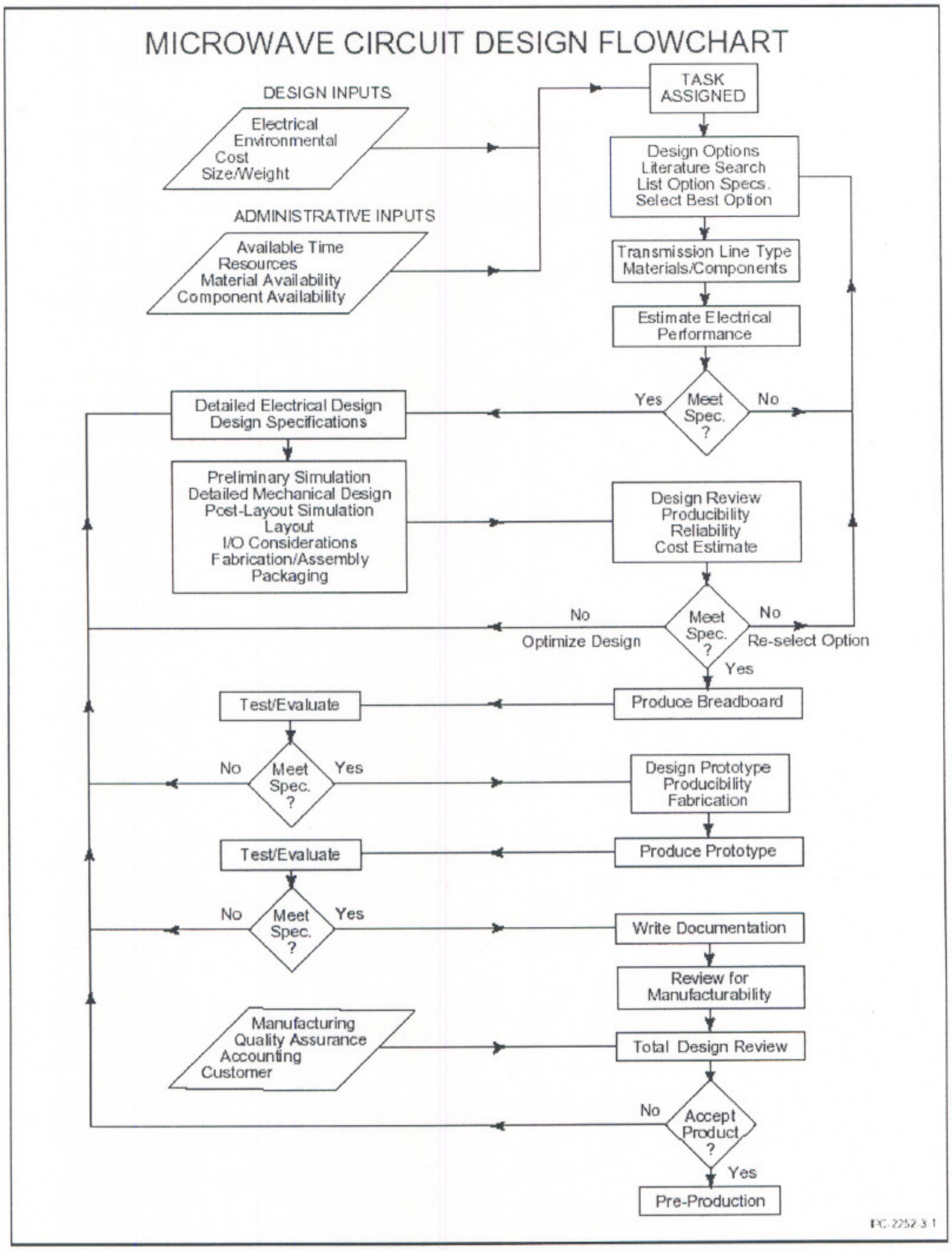


Figure 1 - Microwave Circuit Design Flowchart (Taken from IPC-2252)

The IPC-2252

Contents

The IPC-2252 contains 9 sections. The first section provides a list of terms and their definitions. Section 2 contains referenced documents, which are mostly other IPC documents and standards for materials and design. Sections 3 and 4 describe design considerations and documentation requirements. Design considerations included are choice of transmission line structure, material, electrical and mechanical designs, and a description of the overall design flow process. Section 5 contains a discussion on materials used in high-frequency printed wiring boards for both dielectrics and conductors. Section 6 contains accurate frequency-domain-based mathematical formula describing the electrical characteristics of stripline and microstrip transmission lines, which are commonly-used transmission line structures in high-frequency circuits. In Section 7, board requirements are discussed and, in Section 8, device attachment and packaging. Section 9 provides a brief description of quality assurance and references IPC documents that have more extensive information on this subject.

Section 1, "General"

Section 1 of the IPC-2252 contains the purpose and scope of the IPC-2252 and defined terms. As stated in the IPC-2252, the purpose of the document is to "aid in the design of manufacturable microwave circuit boards." The key word in the scope is "manufacturable," which implies not only realizable, in the sense of being manufactured with available equipment and procedures, but also within cost margins appropriate for typical and common microwave applications. Therefore, the IPC-2252 may not include some considerations that are unique to space and military applications.

There are 46 definitions in the IPC-2252. These definitions are used to describe the terms used in the document that may be unique to and/or used uniquely by the IPC-2252 and/or commonly-used terms for which no acceptable definition was found. Terms defined include terms for electrical characteristics such as characteristic impedance, effective permittivity, dissipation factor, return loss, scattering parameters; terms for design parameters and features such as coefficient of thermal expansion, copper weight/thickness, ground plane, peel strength, plated-through holes; etc.

Section 3, "Design Considerations"

This section is short and basically describes the design process and considerations. The discussion is based on a circuit design flowchart (see Figure 1). As can be seen from the flow chart, the first task is to determine how the board will be used (Design Inputs) and the resources available to manufacture the board (Administrative Inputs). Once these inputs have been determined and are not in conflict, which in essence defines a manufacturable board, the process of producing a manufacturable board can be started. The first steps are to provide an initial design and estimate the electrical performance of this board. If these steps are successful, then the design details and specifications are produced and a path for board fabrication, assembly, and packaging defined. A prototype board is then made and tested and, if the prototype board functions successfully, production boards are made.

Section 4, "Documentation Requirements"

Documentation in the fabrication of a printed wiring board (PWB) is necessary for several reasons. First, it allows the manufacturer to track the manufacturing process and effects of this process on board performance. Documentation is also a necessary requirement for most quality management systems. The design feature listing is a document that contains all the manufactured features and processes, such as holes, planes, plating, etching, etc. The master drawing contains all the information to layout the board. Therefore, each layer of the board must be detailed, showing all conductors, holes, and other features with dimensions (on-plane dimensions as well as thicknesses) and tolerances.

Section 5, "Materials"

The materials section is the longest section in the IPC-2252. It contains information on material properties, environmental effects, bonding, plating, etc. The material property that is the most important for RF and microwave PWBs is the relative permittivity, ε_r, of the substrate. ε_r is a complex value (ε'_r, the real part, and ε''_r, the imaginary part) and usually expressed in the PWB industry in terms of the dielectric constant (ε'_r) and the loss tangent or dissipation factor ($\varepsilon''_r/\varepsilon'_r$). Typical values of ε'_r for RF/microwave PWBs range from 2.1 to 11 (see ref. 1, Table 5-1). The selected value must meet the requirements for the circuit functions and space limitations. For examples, higher values of ε'_r increase capacitance. In transmission line design, with all other parameters remaining the same, an increase in ε'_r will decrease the characteristic impedance (Z_0) of the transmission line. An increase in ε'_r will also cause an increase in the resonant frequency of resonators. The loss tangent is important in both RF/microwave PWBs and digital PWBs. For the latter, the concern is primarily signal integrity. In RF/microwave PWBs, an increase in loss tangent increases the power dissipated into the board. Increased power dissipation can cause thermal management problems, especially for densely packaged boards.

The mechanical properties and thickness of the board affect a variety of board performance parameters. For example, thin substrates are not as dimensionally stable as thick substrates. Consequently, it will be more costly to fabricate on thin substrates because of the difficulty in maintaining design tolerances. Furthermore, mechanical strength is poorer for thin boards than for thick boards; the thin board will have a greater tendency to warp and bow. Soft substrates have more of a tendency to show these effects than rigid substrates. However, if the need is to make a conformal board, then the thin board is required. As mentioned previously, the loss tangent causes power dissipation. This dissipation is into the board. Thicker boards will be able to accommodate greater power dissipation than thinner boards. Also, high peak powers can cause corona discharge. Circuits made on thicker substrates will be less likely to fail because of corona discharge than circuits made on thin substrates.

The effect of the processing and operating environment on the board material is also a very important criterion in the selection of a board material. These effects include not only temperature and humidity, but chemical resistance too. Boards are subject to a large temperature range and different chemical environments during fabrication. For complex board designs, which typically require multilayer boards, the stability of the board material to these environments becomes increasingly important. The board material must be able to accommodate these environments without delaminating, warping, etc.

There are other substrate-related properties that are important to the successful fabrication of the RF/microwave PWB and these include, for example, ε'_r as function of temperature; coefficient of thermal expansion; tensile, compressive, and flexural strengths; and specific gravity.

Bonding films are another component of the dielectrics used in RF/microwave PWBs and their characteristics are important to the operation and function of the circuit. Bonding films are used to glue together layers of dielectric (metal clad or not) to form a multilayer PWB. These bonding films may be thermoplastic or thermosetting in nature. ε'_r of typical thermoplastic bonding films is in the range of 2.1 to 2.4 (see ref. 1, Table 5-2) and that of thermosetting bonding films is 3.5 to 4 (see ref. 1, Table 5-3). These values of ε'_r may not necessarily match that of the substrate materials, which may make the circuit harder to model and design.

The selection of metal conductors to be used in the RF/microwave PWB is also important. The IPC-2252, in Section 5.3, discusses many different possible conductors that can be used in the manufacture of a PWB. The conductors can be plated or clad. For clad conductors, both thin (less than 100 μm thick) conductors and thick conductors (greater than 100 μm thick) are used. Clad conductors are typically roughened to enhance adhesion to the surface of the dielectric. The effect of this roughening can be observed more readily in thin claddings than in thick claddings as an increase in conductor loss, which is frequency dependent.

Plated conductors are usually deposited using an electrodeposition process. However, some plated conductors can be deposited using an electroless method. An electrodeposited metal will require that the dielectric first have a conductive coating, which may be obtained from an electroless metal deposition. Copper, nickel, gold, silver, and tin are some of the electrodeposited metals used for conductors in a PWB. Copper and tin can also be deposited using an electroless process. Gold is used primarily because of its corrosion resistance and high electrical conductivity (4.1×10^7 S/m). Silver is used because it has the highest electrical conductivity (6.1×10^7 S/m) of commonly available conductors, but it tarnishes (oxidizes) easily and may migrate into the substrate material. Because of this, nickel is typically used as a diffusion barrier between the silver and the substrate and rhodium is on top of the silver to prevent oxidation. Copper is the ubiquitous metal used in PWB conductors. Copper has a high electrical conductivity (5.7×10^7 S/m), is readily available, and can be plated with both electrolytic and electroless processes. Tin, because its electrical conductivity (9×10^6 S/m) is relatively low, is primarily used to protect a copper layer.

Conformal coatings are also discussed in the IPC-2252 but are not recommended. Conformal coatings are protective layers that are deposited on top of conductors. Because these coatings typically have an ε'_r value different from that of the substrate material, the electrical performance of the circuit will change. Additionally, the dissipation factor of these coatings is typically greater than that of the substrate thereby increasing dielectric loss.

Section 6, "Electrical Characteristics"

Section 6 contains accurate frequency-domain-based mathematical formulas describing the electrical characteristics of stripline and microstrip transmission lines. These structures are commonly used in RF and microwave circuits. The IPC-2252 defines a microstrip structure as "A transmission line structure that consists of a signal conductor that runs parallel to and is separated from a much wider ground plane" and a stripline structure as a "A transmission line structure that consists of a signal line that runs parallel to and is sandwiched between and separated from two wider ground planes." For stripline, both narrow and wide signal lines are considered. The characteristics discussed for both types of structures are conductor and dielectric loss, characteristic impedance, and propagation function. The formulas presented are synthesis type and include the effects of signal line width relative to its thickness, finite conductivity of the conductors, and conductor surface roughness. The limits of applicability, in terms of transmission line geometry and material properties, are also given. However, as with most formulas for transmission line properties, the ones provided in the IPC-2252 should be used as a tool for understanding the effects of different design parameters on the properties of the transmission line. These formulas may even be useful for the initial design phase (see microwave circuit design flowchart shown in Figure 1). For the final design stages of the board, it is recommended practice that electromagnetic-field simulation software be used to improve success of achieving specified or required circuit operation.

Section 7, "Detailed Board Requirements"

"Board requirements" is a broad category that includes many topics, such as, machined features, imaging, Polytetrafluoroethylene

(PTFE) considerations, metallization details not discussed in Section 6 of the IPC-2252, etching, and bonding. Machined features include plated-through-holes (PTHs), unplated holes, pockets, slots, and cavities. The aspect ratio, R_a, (depth to diameter) for PTHs defines the difficulty in fabricating a PTH. For example, PTHs with $R_a \leq 3$ are easy to fabricate and those with $R_a \geq 10$ are almost impossible to make. High R_a PTHs are more difficult to drill and to maintain plating uniformity along its depth. Hole size after plating may also be difficult to predict because of variations in local plating current densities. Pockets and slots are produced with tools having a radius. Accordingly, the design for these types of features must accommodate the tool used to make them. The subsection on machined features also includes a discussion on tooling hole requirements (size, placement, number), the use of press-fit pins for making ground connections for metal-backed substrates, and dimensioning and tolerancing.

Under tolerancing, bilateral and true-position tolerancing are compared (see Figure 2). Bilateral tolerancing is where both the x and y tolerances (the coordinate system used to describe features on a panel is a two-dimensional cartestian system, using x and y to denote those dimensions) are defined independently of each other. Bilateral tolerancing, therefore, generates a square tolerance zone. True-positioning tolerancing, on the other hand, provides an elliptical tolerance zone, where the x and y tolerances are the major and minor axes of an elliptical tolerance zone. It is recommended that true-position tolerancing be used because it is less likely to accept poor parts or reject acceptable parts than is bilateral tolerancing.

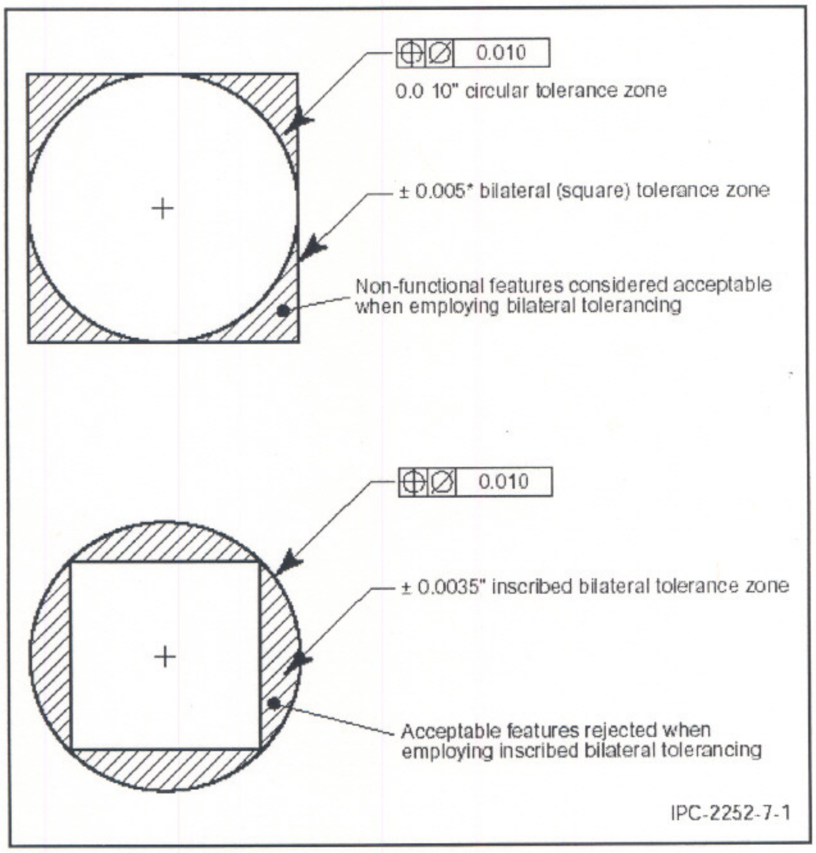


Figure 2 - Comparison of Bilateral and True-Positioning Tolerancing Methods (Taken from IPC-2252)

Guide for generating the artwork and transferring the artwork to the board is discussed in Section 7.2 of the IPC-2252. Negative artwork is usually used to transfer print and etch patterns and positive artwork for pattern plating. Photo-activated organic films (photoresist) are used to transfer the image from the artwork to the board. The photoresist can be either a dry film or liquid. Thinner resists allow for smaller feature sizes than does thicker resists. However, thinner resists are more susceptible to pinhole formation from handling and airborne particulates. The designer should not compensate his/her design geometries for the effect of the fabrication process. This is because the fabricator knows how to compensate for their process to achieve the circuit given by the design information and each fabricator may have different compensation factors.

The board manufacturer needs to be aware that, in the etching process, the copper etch rate will vary across the board, depending on the pattern geometry and conductor thickness. The etch rate of lines will also depend on the proximity of lines. For example, isolated narrow lines separated by a large gap will etch more quickly than a cluster of closely-grouped lines. Undercutting of the etched conductors will also occur. The effect of undercut is greater for thick conductors than for thin conductors. The deposition rate and uniformity of electrodeposited copper will also depend on pattern geometry. That is, plating of copper lines in an area with a lot of conductor coverage should not be expected to plate at the same rate as lines in an area nearly void of conductors.

Section 8, "Device Attachments and Packaging"
Section 8 discusses three levels of attachment, namely, attaching the board to the housing, attaching the connector to the board, and attaching the device to the board. Methods for attaching the board to the housing include mechanical attachment (using various types of fasteners), epoxies (conductive and nonconductive), thermoplastic films, soldering, and direct bonding. Mechanical attachment of soft substrates, such as PTFE, should be done with caution as this may deform the board. One way to minimize deformation of PTFE boards is to use metal plates to distribute the force over large areas.

Conductive epoxies are thermosetting resins that are filled with a metal powder, typically silver. Metal filled epoxies tend to be brittle and do not accommodate differential temperature expansion well. Consequently, these epoxies should be avoided in applications where there will be a large range of temperatures. Since there are many types of epoxies available, the one best suited for the application should be selected. However, bond failure over time may still occur and this may effect the circuit function. Using nonconductive epoxies to secure a board with a ground plane can cause grounding and interconnect problems.

Soldering is the most common method of attaching the completed board to the housing. In this process, both the board and housing are coated with solder and attachment is achieved when the housing is heated above the solder melting temperature. For large boards, this process may become complicated and require special fixtures to hold the parts in position. Another temperature-based method is called direct bonding. In direct bonding, the board and housing are placed in contact and the temperature is raised above the melting temperature of the board material. Pressure is usually applied to the board during the bonding process.

Several different methods for device attachment are mentioned in the IPC-2252. These methods include welded bonds (resistance welding, parallel gap welding, percussive arc welding, laser welding, and soldering) and diffusion bonding (ultrasonic bonding, thermal compression bonding, and thermosonic bonding). The IPC-2252 provides a brief description of these different methods.

Changes relative to the IPC-D-316
The IPC-2252 contains the same basic information as the document it superseded, the IPC-D-316. Each major section was rearranged to provide better continuity within that section and improve readability throughout the document. Redundant information was deleted and similar concepts, that were presented differently in two or more locations in the IPC D-316, were combined. The definitions in the IPC-2252 were rewritten if they where unclear and deleted if they appeared in and were adequately defined by the IPC-T-50F, "Terms and Definitions for Interconnecting and Packaging Electronic Circuits."[3] Changes were made to some definitions to make them consistent with those from other technical organizations, such as the Institute of Electrical and Electronic Engineers (IEEE). Sections 3 and 4 did not change significantly from the IPC-D-316 other than for improved readability and update of references. The basic content of Sections 5 through 8 did not change significantly either. However, discussions and tables were updated based on current technology and manufacturing capability, and new information added. The errors in the formulas of Section 6 were corrected and the formulas rewritten to more closely parallel those from the references. Section 9 was changed significantly. The IPC-D-316 contained much of the same information that is in one of the following specifications: the IPC-6018A, "Microwave End Product Board Inspection and Test,"[4], the ANSI/IPC-4103, "Specification for Base Materials for High Speed / High Frequency Applications,"[5] and the ANSI/IPC-2221, "Generic Standard on Printed Board Design."[6] This redundant information was deleted in the IPC-2252 and replaced with references to the appropriate IPC specification.

Comparison to Other IPC documents
There are four other IPC documents, two guides and two specifications, that have some degree of overlap with or complement the IPC-2252. The guides are all written with the same intention as that for the IPC-2252, that is, to be informative. The guide IPC D-317A, "Design Guideline for Electronic Packaging Utilizing High-Speed Techniques,"[7] (presently being rewritten by the D-21a Task Group and will become the IPC-2251) provides a discourse on topics pertinent specifically to high-speed circuit design. Because of the similarity in some requirements, the IPC-D-317A and the IPC-2252 have related discussion regarding transmission line electrical characteristics. However, the synthesis formula are different between the two documents because the IPC-D-317A is concerned with broadband signals (pulses) whereas the IPC-2252 is focused on narrowband signals. Similarly, the majority of discussion is directed toward narrowband signals in the IPC-2252 and pulse signals in the D-317A. Consequently, these documents complement each other well by providing guides to high-frequency and high-speed PWBs. The guide, IPC-2141, "Controlled Impedance Circuit Boards and High Speed Logic Design,"[8] (presently being rewritten by the D-21c Task Group) describes topics specific for controlled impedance transmission lines. Controlled impedance typically implies impedance tolerances of ± 10 % or less. There is not significant overlap in the contents of the IPC-2252 and the IPC-2141 other than for board and transmission line design. The IPC-6018A is a specification and, therefore, prescriptive or normative, whereas the IPC-2252 is a guide, an informative document. These two documents contain unique information and together form a more complete instruction on high-frequency PWBs, from design concept to test. The other document that has some relation to the IPC-2252 is

the ANSI/IPC-4103. The ANSI/IPC-4103 is a normative document that provides tables of requirements for different laminates and interlayer adhesives used in high-speed/high-frequency PWBs. Although the IPC-2252 provides certain suggested requirements for laminates used in the fabrication of high-frequency PWBs, it does not provide the detailed requirements contained in the IPC-4013.

Future of the IPC-2252

Because PWB technology changes rapidly, revisions of published documents are often considered immediately after their release. People interested in participating in a subsequent revision of the IPC-2252 should contact John Perry of the IPC (johnperry@ipc.org) and indicate this interest and, to ensure their participation, ask to have their names added to the D-21 subcommittee and D-21b task group rosters.

Acknowledgement

Nicholas G. Paulter, Jr. - Electricity Division, Electronics and Electrical Engineering Laboratory, Technology Administration, Department of Commerce. Official contribution of the National Institute of Standards and Technology, not subject to copyright in the U.S.A.

Conclusion

The IPC-2252 supercedes the IPC-D-136 and was published in July 2002. The IPC-2252 was introduced herein by describing its contents. The differences between the IPC-2252 and IPC-D-316 were also discussed and a comparison to other IPC documents provided.

References

1. IPC-2252, Design Guide for RF / Microwave Circuit Boards, IPC, Northbrook, IL, July 2002.
2. IPC-D-316, Design Guide for Microwave Circuit Boards Utilizing Soft Substrates," IPC, Northbrook, IL, June 1995.
3. IPC-T-50F, Terms and Definitions for Interconnecting and Packaging Electronic Circuits, IPC, Northbrook, IL, June 1996.
4. IPC-6018A, "Microwave End Product Board Inspection and Test," IPC, Northbrook, IL, January 2002.
5. ANSI/IPC-4103, "Specification for Base Materials for High Speed / High Frequency Applications," IPC, Northbrook, IL, January 2002.
6. ANSI/IPC-2221, Generic Standard on Printed Board Design, IPC, Northbrook, IL, February 1998.
7. IPC-D-317A, Design Guidelines for Electronic Packaging Utilizing High-Speed Techniques," IPC, Northbrook, IL, January 1995.
8. IPC-2141, "Controlled Impedance Circuit Boards and High Speed Logic Design," IPC, Northbrook, IL, April 1996.

MINNEAPOLIS CONVENTION CENTER ■ MINNEAPOLIS, MN

IPC's Annual Meeting

SEPTEMBER 28 - OCTOBER 2, 2003

Visit www.ipc.org/annual ■ Call: 847-509-9700 ■ Fax: 847-509-9798

IPC ANNUAL MEETING PROCEEDINGS

TABLE OF CONTENTS

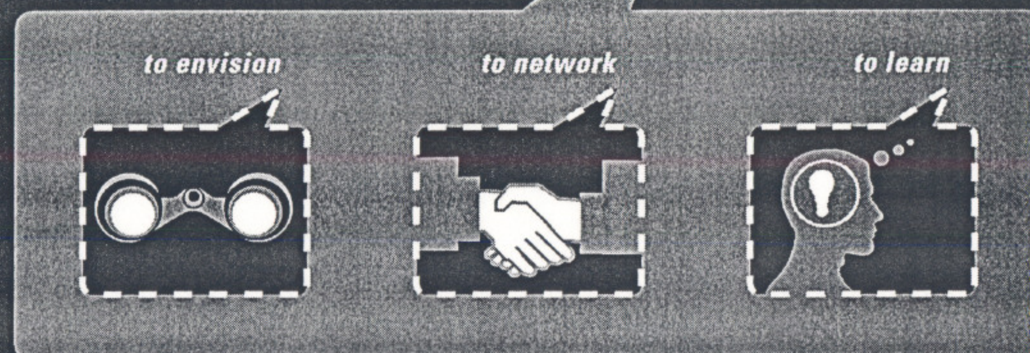
AUTHOR INDEX

SEARCH

DISCLAIMER

ABOUT IPC

it's TIME



IPC

ASSOCIATION CONNECTING ELECTRONICS INDUSTRIES®