

Setting the standards for success.(IPC Issues and Answers)(Conference news)

Source:  CircuiTree

Publication Date: 01-JUN-08

One of the best parts of my job as president of IPC is watching it all come together--I recently had that privilege once again at IPC's show in Las Vegas. Staff had worked diligently to make the move to Las Vegas seamless. Our technical team and committees planned a dynamite lineup of conference sessions and courses that broke all previous show attendance records. The exhibitors moved mountains of equipment and manpower to share the latest and greatest of their products, services, and tools. And hundreds of volunteers worked tirelessly throughout the year to deliberate and deliver standards that our industry has come to depend on--and so it was at the show.

[ILLUSTRATION OMITTED]

If you missed out, I hope you will join us next year. For now, I'll just leave you with some parting gifts, thanks to our scores of dedicated standards development committee leaders ... along with hopes that you may join us in our endeavors soon.

Recently updated guidelines and specifications for engineers in the printed circuit board and fabrication industries:

IPC-DR-572A, Drilling Guidelines for Printed Boards. Provides guidelines for generating good quality holes in PCBs made from a wide variety of materials and addresses the topics of drill bits, tooling, drilling stacked materials, drilling machines, drilling operation, drilled hole quality, and trouble shooting.

IPC-SM-840D, Qualification and Performance Specification of Permanent Solder Mask. Establishes the requirements for the evaluation of liquid and dry-film solder mask material and for the determination of the acceptability of use on a standard printed board system. Two classes of requirements, T and H, reflect functional performance requirements and testing severity based on industry/end use requirements. Coverage is given to adhesion, material qualification, resistances to solvents, and electrical requirements. Revision D incorporates resistance requirements to lead-free soldering processes.

IPC-4101B with Amendments 1 & 2, Specification for Base Materials for Rigid and Multilayer Printed Boards. This specification covers the requirements for base materials (laminates and prepreg) for rigid or multilayer printed boards for electrical and electronic circuits. It contains more than 50 specification sheets and now uses search terms to allow the user to find similar groups of materials. It provides additional information and data on PCB materials that are better able to withstand the newer assembly operations employing higher thermal exposures, including those that use lead-free solders.

IPC-4412A with Amendment 1, Specification for Finished Fabric Woven from "E" Glass for Printed Boards. Exhaustively covers the classification and requirements for finished fabrics woven from "E" glass fiber yarns. These yarns are formed from filaments of electrical-grade glass and are intended as reinforcing materials in laminated plastics for electrical and electronic uses. The fabrics covered are all of a plain-weave construction. Document includes two extensive tables of finished fabric glass styles, one in SI units and the other in US units.

IPC-4563, Resin Coated Copper Foil for Printed Boards Guideline. This guideline covers the requirements for resin coated copper foil intended for use in the formation of high-density interconnect surface microvias for printed boards and printed board assemblies.

IPC-5702, Guidelines for OEMs in Determining Acceptable Levels of Cleanliness of Unpopulated Printed Boards. Every electronics manufacturer, whether an OEM or EMS company, must determine if the unpopulated printed boards entering the assembly process have an adequate level of cleanliness. "How clean is clean enough?" is a question that has no definitive answer, as there is no golden number for board cleanliness. The issue is very complex, with many critical considerations, so a single methodology to determine acceptability doesn't exist. Recognizing this, IPC-5702 was written to give printed board or printed board assembly professionals guidance on how to correlate cleanliness related data to electrical function and determine acceptable cleanliness levels.

IPC-9691A, User Guide for the IPC-TM-650, Method 2.6.25. Conductive Anodic Filament (CAF) Resistance Test (Electrochemical Migration Testing). This document provides guidance regarding how this resistance test can best be used for evaluating the effects of mechanical stress, laminate material fracturing, ionic contamination, moisture content prior to press lamination, and other material processing characteristics on CAF test method results. This test method provides a proven standard for determining the risk of temperature, humidity, and bias (THB) failure within rather than on the surface of PCBs, typically filament formation along the boundary between the resin and laminate reinforcement.

Ever wonder how the standards you rely on get developed? Or why a certain criterion was included? Use your technical expertise to provide input on the development or improvement of IPC standards. Meeting participation can occur in person, by teleconference, or by e-mail. Distance shouldn't deter you.

Find out more about IPC committees at www.ipc.org/committees and about the documents currently in development in our Status of Standardization at www.ipc.org/status.

"Coming together is a beginning. Keeping together is progress. Working together is success."

Henry Ford

Dennis P. McGuirk is the President of IPC--Association Connecting Electronics Industries. He can be reached by phone at 847-790-5341 or by email to mcgude@ipc.org. For more information, visit www.ipc.org.