Design Guide IPC-D-330

Foreword

IPC Formed in 1957

Because of a need for technical information, and to develop standards in the field of printed wiring, the IPC was formed in 1957. It stands today as an organization dedicated to the technological advancement of the printed board industry.

The programs of the IPC are made possible only through the active, voluntary support of the membership. The membership of the IPC is made up of representatives of companies from a broad cross-section of industry, and also includes qualified technical experts from Government Agencies, and representatives from colleges and universities. Membership is on a company basis and members include companies that produce printed boards for sale and/or internal use, and companies which are suppliers of material and equipment used by companies that provide assembly services in the electronic interconnect industry.

IPC Mission Statement (Adopted April 1991)

The IPC is a United States based trade association dedicated to furthering the competitive excellence and financial success of its members worldwide, who are participants in the electronic interconnect industry.

In pursuit of these objectives, the IPC will devote resources to management improvement and technology enhancement programs, the creation of relevant standards, protection of the environment, and pertinent government relations.

The IPC encourages the active participation of all of its Regular, Allied and Associate Members in these activities and commits to full cooperation with all related national and international organizations.

Purpose of the IPC Design Guide

The purpose of the IPC Design Guide is to make information available to users and designers of rigid and flexible printed boards as well as hybrid and multichip modules. This documents contains a compilation of valuable and useful data pertinent to the problems encountered in the application of design principles.

The scope of this document is broad. We refer you to the comprehensive Table of Contents which indicates the extent of data to be included in the Design Guide.

Designer's Council

In order to assist printed board designers in their career, IPC has formed the Designer's Council.

The Designer's Council is an organized forum for printed board designers to improve their skills utilizing local technical exchanges, seminars and IPC sponsored workshops. It provides a mechanism for distribution of all design related standards to the local chapters and creates a direct channel for comments on the standards to the responsible IPC committees.

The Designer's Council will provide for a Designer Certification Program. This is an accreditation and training program for establishing recognition of designers that is equivalent to other degreed occupations

Designers are encouraged to form local chapters. Contact IPC for information on getting started. Local chapters are not limited to a specific focus, and may consist of any number of designers.

IPC and the Future

With the continued voluntary cooperative activity of the membership, it is anticipated that through the IPC we can continue to undertake programs and activities which will be of mutual benefit to representatives from the printed wiring industry, and to the entire electronics industry. The IPC will continue to provide a forum through which the objectives of the industry can be attained with the greatest efficiency and economy.

The IPC Design Guide is divided into several major sections to facilitate the organization of information pertinent to design of electronic interconnections. The following test contains a general description of the information contained in each major section of the Design Guide.

Design Guide Contents

Section One—Specifications and Standards

This section provides abstracts of the specifications that relate to the subject of Printed Board Design.

Subjects: Technology Description—Terms and Definitions—IPC Specifications—Military Specifications—Federal Specifications—Other Specifications

Section Two—General Design Information

This section contains information common to the various facets of printed wiring technology. The material includes mechanical, electrical and environmental considerations as well as the fabrication information to be acknowledged in order to design quality printed wiring products.

Subjects: Printed Wiring Selection—Materials—Design Standards—Partitioning—Fabrication

Section Three—Computer Aided Design

This section contains state-of-the-art information concerning the different computerized techniques used to assist in the design of printed wiring. The general descriptions relate to the

degrees of flexibility and limitations of the various systems. Specific areas of this section describe industry attempts toward standardizing printed wiring descriptive languages, art work symbology, and component displacement definition.

Subjects: Techniques—Standardization—Manual Intervention— Numerical Control Equipment—Capabilities and Limitations— Postprocessing

Section Four—Special Performance Design Characteristics

This section deals with the many considerations that have been taken into account with respect to the design of printed boards and assemblies to satisfy special performance characteristics.

Subjects: Basic System Performance—Digital High Speed Interactions—Environmental Stability—Thermal Management

Section Five—Components and Their Mounting Characteristics

This section addresses the effect of component mounting and attachment on printed board design. An overview of component dimensional characteristics is provided. The decision process of connector choice is described. Sockets, switches and bus bars are also reviewed.

Subjects: Electronic Component Packages—Electromechanical Circuit Components—Mechanical Circuit Component Hardware

Section Six—Organic-Base Rigid Printed Board Design

This section contains information applicable to the design of composite printed wiring boards consisting of three or more conductor surfaces on rigid insulating bases separated from each other by an insulating material. This information is supplemented by the information in Section Two in order to completely cover this subject. Particular emphasis is given to materials, types of interconnections between conductor surfaces, lay-up and laminating processes.

Subjects: Design Layout—Design Features—Testing Considerations—Electrical Considerations—Mechanical Considerations—Interconnection—Fabrication

Section Seven—Flexible Printed Circuit Design

This section contains information primarily applicable to the design of flexible printed wiring structures. the various aspects of designing single-sided, double-sided, and multilayer flexible substrate printed wiring are covered herein. This information is supplemented by the information in Section Two in order to completely cover this subject. Particular emphasis is given to materials and the differences between the various techniques.

Subjects: Flexible Printed Circuit Types—Material—Design Practice—Bends and Folds—Electrical—Flexabilty Considerations—Manufacturing Considerations

Section Eight—Hybrid Microcircuit and Ceramic Printed Board Assemblies

This chapter contains an overview of the hybrid microcircuit design procedure. It's purpose is to assist the design activity in early identification of compromises among process flow, component selection and final package configuration, and to permit the selection of design and process alternatives that will emphasize the inherent advantages of this packaging technology.

Subjects: Materials—Components—Thermal Considerations—Microwave Considerations

Section Nine—Multichip Modules

This section contains information concerning features of a printed wiring structure that are used to ascertain its quality and reliability. The text discusses those parameters that are used to ascertain its quality and reliability. The text discusses those parameters that are inspection sensitive and warrant special consideration during the design phase. Samples of patterns and coupons used to aid the inspection processes are included.

Subjects: Materials—Assembly Considerations—Thermal Characteristics—Quality Assurance—Device Protection

Section Ten—Printed Board Backplane Design

This section will help the printed board backplane designer to achieve the optimal technical interconnect design also considering product economy.

Subjects: Connector Systems—Component Fabrication Considerations—Printed Board Backplane Fabrication

Section Eleven—Assembly Considerations

This section contains information concerning the mounting of electrical circuit components and mechanical parts to printed wiring structures. The test relates to specific types of components, their placement, physical retention, and electrical attachment to printed wiring terminal areas. Information is provided describing techniques used with subassembly level interconnection wiring schemes.

Subjects: Components Types—Component Placement—Mechanical Securing—Lead Attachment—Surface Mount Assembly

Section Twelve—Printed Board Documentation

This section contains information on the type and degree of documentation that is necessary in order to communicate complete descriptions of printed wiring end products. This printed wiring design layout process is described, as well as the type of information required to execute a complete layout. Change control procedures are also discussed.

Subjects: General Considerations—Layout—Circuit Diagrams— Phototooling—Assembly Drawings—Data Control and Management

Section One
Specifications and Standards

1.0 Introduction This section of the Design Guide provides abstracts of the specifications, standards and guideline documents that relate to all aspects of the subject of printed board design.

1.1 Institute for Interconnecting and Packaging Electronic Circuits (IPC) The programs of the Institute for Interconnecting and Packaging Electronic Circuits (IPC) are made possible through the voluntary support of its membership. The membership of the IPC is on a company basis and is made up of representatives of companies from a broad and worldwide cross-section of the electronics industry and qualified technical experts from Government Agencies, colleges and universities.

The IPC develops and publishes specifications, standards and guideline documents in order to help establish common terminology and to promote the optimum use of printed boards and related products and assemblies.

IPC documents can be obtained by contacting the association at

IPC
7380 N. Lincoln Ave.
Lincolnwood, IL 60646-1705
Telephone (708) 677-2850 Fax (708) 677-9570
Telex 214814 IPC UR

1.1.1 IPC-T-50

1.1.1.1 Title Terms and Definitions for Interconnecting and Packaging Electronic Circuits

1.1.1.2 Current Issue Revision E, April 1992

1.1.1.3 American National Standards Institute (ANSI) Approval Pending

1.1.1.4 Department of Defense (DoD) Adoption Pending

1.1.1.5 Abstract This document represents the terms and definitions developed by the IPC Blue Ribbon Committee on Terms and Definitions. The Blue Ribbon Committee was appointed in 1973 and assigned the responsibility for the development of concise definitions of terms associated with the electronic interconnection industry as a continuous activity. For this reason, the Blue Ribbon Committee will continue to function, reviewing additional terms and definitions for inclusion into this document.

1.1.2 IPC-SC-60

1.1.2.1 Title Post Solder Solvent Cleaning Handbook

1.1.2.2 Current Issue Original Publication, April 1987

1.1.2.3 American National Standards Institute (ANSI) Approval May 1, 1987

1.1.2.4 Abstract This document describes the following key areas of concern when selecting cleaning solvents/solutions:

- Effectiveness
- Stability
- Safety
- Cost

Other specific topics covered in this handbook are the following:

- Properties of cleaning solvents
- Manufacturing residues
- Cleaning equipment and processes
- Cleaning of printed circuit assemblies after hand solder operations
- Process control
- Environmental considerations.

1.1.3 IPC-AC-62

1.1.3.1 Title Post Solder Aqueous Cleaning Handbook

1.1.3.2 Current Issue Original Publication, December 1986

1.1.3.3 American National Standards Institute (ANSI) Approval February 5, 1987

1.1.3.4 Abstract This handbook discusses the aqueous (water) cleaning of printed board assemblies after they have been soldered, i.e., post solder cleaning. It is organized as follows:

Section I—Introduction to Aqueous Cleaning
Section II—Hardness
Section III—Manufacturing Residues
Section IV—Cleaning Equipment
Section V—Process Control
Section VI—Environmental Considerations
Section VII—Safety Concerns
Section VIII—Cleanliness Determination and Measurement