

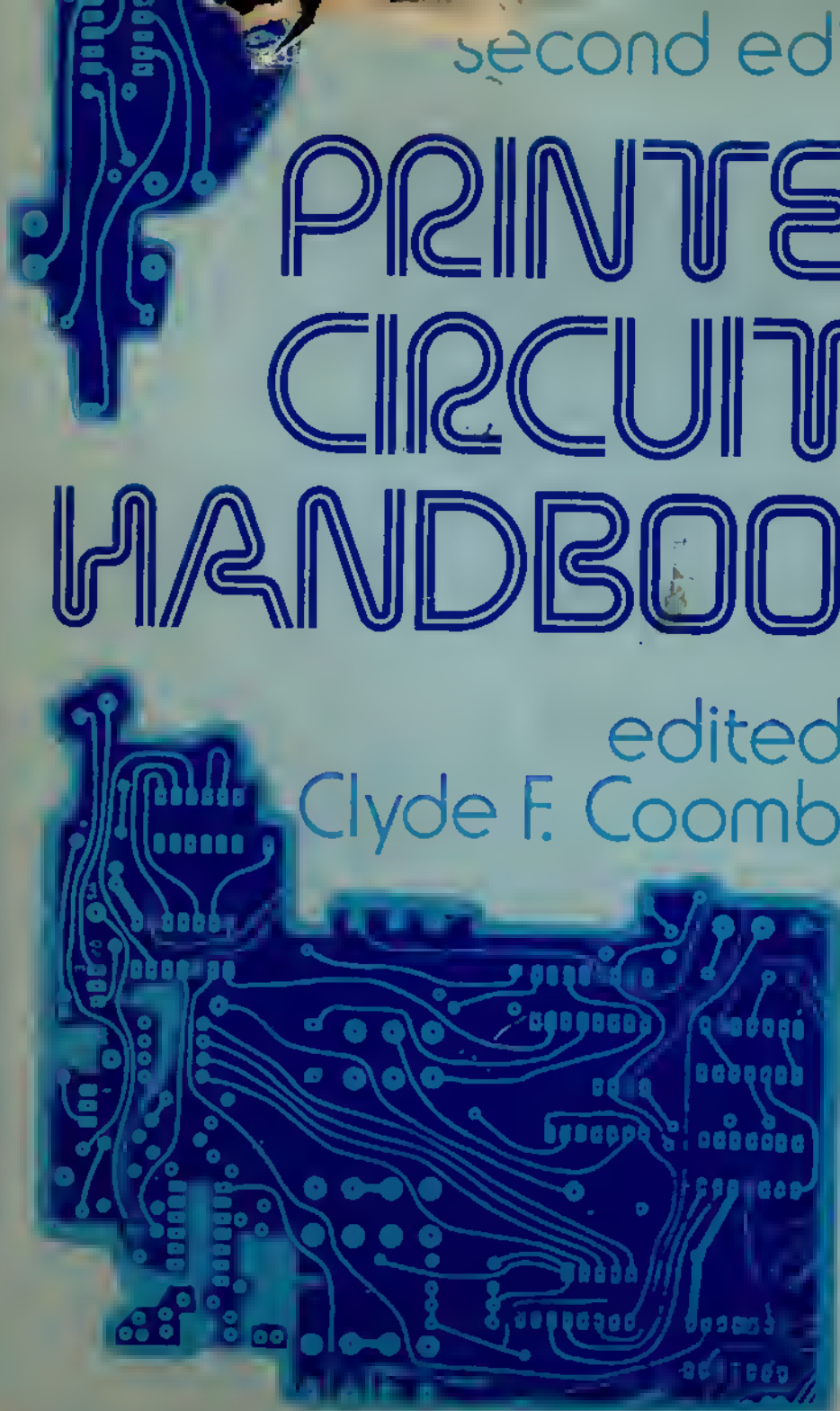
second edition

PRINTED CIRCUITS HANDBOOK

edited by
Clyde F. Coombs, Jr.

PRINTED CIRCUITS
HANDBOOK

second
edition



**Printed
Circuits
Handbook**

Coombs

Second Edition



\$ 32.50

second edition

PRINTED CIRCUITS HANDBOOK

edited by
Clyde F. Coombs, Jr.

634 pages 595 illustrations

In the decade and a half during which this book has been the basic reference on printed circuits, enormous advances have taken place in the design, fabrication, assembly, and testing of these devices. Accordingly, here is the totally revised and updated second edition, featuring 12 chapters that did not appear in its predecessor and important new information throughout its text.

This book is a practical tool for virtually anybody involved with the design or production of electronic equipment. It covers not only every aspect of double-sided plated-through printed boards, but such major variations as multilayer and flexible circuits. It gives specific answers to specific how-to-do-it questions, with enough technical background to enable the user to modify designs and processes for particular applications. And it describes in detail solutions to such crucial problems as waste treatment and pollution control.

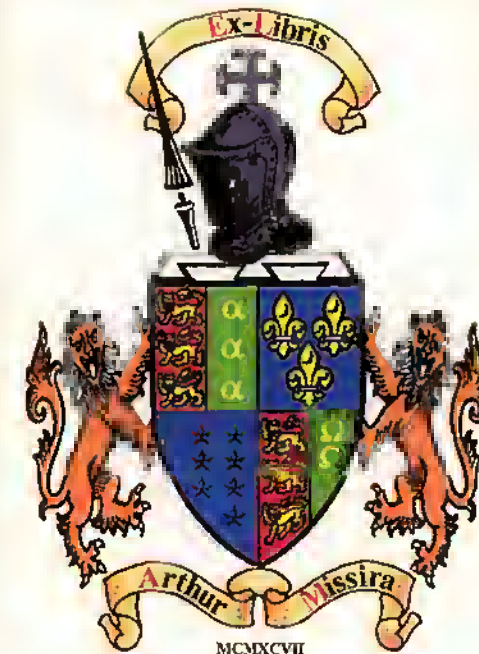
The book can stand alone as a design guide for engineers who must specify designs, materials, and fabrication methods. Production people can use it as a "cook-book" for establishing efficient processes at every stage of manufacturing. It sets standards of quality control for those who monitor the integrity of manufacturing operations. And it provides essential information to managers who need to understand the problems and potential of the entire process.

(continued on back flap)

RML WARRINGTON

QUALITY ASSURANCE

1985.



OTHER MCGRAW-HILL HANDBOOKS OF INTEREST

American Institute of Physics • American Institute of Physics Handbook
Baumeister and Marks • Standard Handbook for Mechanical Engineers
Beevan • Industrial Power Systems Handbook
Blatz • Radiation Hygiene Handbook
Brady and Clauser • Materials Handbook
Burlington • Handbook of Mathematical Tables and Formulas
Burlington and May • Handbook of Probability and Statistics with Tables
Condon and Odishaw • Handbook of Physics
Coombs • Basic Electronic Instrument Handbook
Croft, Carr, and Watt • American Electricians' Handbook
The Diebold Group, Inc. • Automatic Data Processing Handbook
Etherington • Nuclear Engineering Handbook
Fink • Electronics Engineers' Handbook
Fink and Beaty • Standard Handbook for Electrical Engineers
Giacoletto • Electronics Designers' Handbook
Grueenberg • Handbook of Telemetry and Remote Control
Hansher • Communication System Engineering Handbook
Harper • Handbook of Components for Electronics
Harper • Handbook of Electronic Packaging
Harper • Handbook of Materials and Processes for Electronics
Harper • Handbook of Thick Film Hybrid Microelectronics
Harper • Handbook of Wiring, Cabling, and Interconnecting for Electronics
Henney • Radio Engineering Handbook
Hicks • Standard Handbook of Engineering Calculations
Hunter • Handbook of Semiconductor Electronics
Haskey and Korn • Computer Handbook
Ireson • Reliability Handbook
Jasik • Antenna Engineering Handbook
Juran • Quality Control Handbook
Kaufman and Seidman • Handbook for Electronics Engineering Technicians
Klerer and Korn • Digital Computer User's Handbook
Koelle • Handbook of Astronautical Engineering
Korn and Korn • Mathematical Handbook for Scientists and Engineers
Kurtz • The Lineman's and Cableman's Handbook
Machol, Tanner, and Alexander • System Engineering Handbook
Maissel and Glang • Handbook of Thin Film Technology
Markus • Electronics Dictionary
Perry • Engineering Manual
Skolnik • Radar Handbook
Smeaton • Motor Application and Maintenance Handbook
Stout and Kaufman • Handbook of Operational Amplifier Circuit Design
Truxal • Control Engineers' Handbook
Tuma • Engineering Mathematics Handbook
Tuma • Handbook of Physical Calculations
Tuma • Technology Mathematics Handbook

Printed Circuits Handbook

CLYDE F. COOMBS, Jr. *Editor-in-chief*

*Calculator Group Manufacturing Manager
Hewlett-Packard Company
Palo Alto, California*

Second Edition

MCGRAW-HILL BOOK COMPANY

New York St. Louis San Francisco Auckland Bogotá
Düsseldorf Johannesburg London Madrid
Mexico Montreal New Delhi Panama
Paris São Paulo Singapore
Sydney Tokyo Toronto

Library of Congress Cataloging in Publication Data

Coombs, Clyde F ed.
Printed circuits handbook.

Includes bibliographies and index.

1. Printed circuits. I. Title.

TK7868.P7C66 1979 621.381'74 78-16800

ISBN 0-07-012608-9

To Ann

Copyright © 1979, 1967 by McGraw-Hill, Inc. All rights reserved.
Printed in the United States of America. No part of this
publication may be reproduced, stored in a retrieval system,
or transmitted, in any form or by any means, electronic,
mechanical, photocopying, recording, or otherwise, without
the prior written permission of the publisher.

1234567890 KPKP 7865432109

*The editors for this book were Harold B. Crawford and Ruth Weine,
the designer was Naomi Auerbach, and the production supervisor
was Teresa F. Leaden. It was set in Caledonia
by University Graphics, Inc.*

Printed and bound by The Kingsport Press

Contents

Contributors ix

Preface xi

Section 1 ENGINEERING

- 1. Design and Layout 1-3
- 2. Laminate Manufacture and Quality Control 2-1
- 3. Circuit Components 3-1
- 4. Specifications and Purchasing 4-1

Section 2 FABRICATION

- 5. Laminate Machining 5-3
- 6. Image Transfer 6-1
- 7. Plating 7-1
- 8. Etching 8-1
- 9. Pollution Control 9-1

Section 3 ASSEMBLY

- 10. Manual Assembly 10-3
- 11. Automatic Assembly 11-1
- 12. Conformal Coatings 12-1
- 13. Assembly Repair 13-1

Section 4 SOLDERING

- 14. Designing and Presoldering Considerations for Soldering 14-3
- 15. Solder Material and Process 15-1
- 16. Cleaning of Soldered Boards 16-1
- 17. Quality Control in Soldering 17-1

Section 5 QUALITY ASSURANCE

- 18. Quality Assurance Procedures and Methods 18-3
- 19. Acceptability of Fabricated Circuits 19-1

Section 6 MULTILAYER CIRCUITRY

20. Multilayer Design and Layout	20-3
21. Multilayer Materials Selection and Specifications	21-1
22. Fabricating Multilayer Circuits	22-1
23. Laminating	23-1

Section 7 FLEXIBLE PRINTED WIRING

24. Flexible Printed Wiring Design and Layout	24-3
25. Flexible Printed Wiring Materials	25-1

Glossary follows Chapter 25

Index follows Glossary

Contributors

- Ackerman, D. W.** *Universal Instruments Corp.*: CHAPTER 11, AUTOMATIC ASSEMBLY
- Affleck, R. G.** *Hewlett-Packard Company*: CHAPTER 9, POLLUTION CONTROL
- Andrade, A. D.** *Saudia Corporation*: CHAPTER 19, ACCEPTABILITY OF FABRICATED CIRCUITS
- Armstrong, E.** *Nor-Cal Tec, Inc.*: CHAPTER 6, IMAGE TRANSFER; CHAPTER 8, ETCHING
- Bahn, R. J.** *Hewlett-Packard Company*: CHAPTER 7, PLATING
- Bergman, D. W.** *Aerountronic Ford Corporation*: CHAPTER 1, DESIGN AND LAYOUT; CHAPTER 4, SPECIFICATIONS AND PURCHASING; CHAPTER 20, MULTILAYER DESIGN AND LAYOUT
- Cannizzaro, J. R.** *Digital Equipment Corp.*: CHAPTER 24, FLEXIBLE PRINTED WIRING DESIGN AND LAYOUT; CHAPTER 25, FLEXIBLE PRINTED WIRING MATERIALS
- Cole, H.** *Cobar Resources*: CHAPTER 14, DESIGNING AND PRESOLDERING CONSIDERATIONS FOR SOLDERING (CONTRIBUTING AUTHOR: J. Langan *Consultant*); CHAPTER 15, SOLDER MATERIAL AND PROCESS; CHAPTER 16, CLEANING OF SOLDERED BOARDS; CHAPTER 17, QUALITY CONTROL IN SOLDERING
- Duffek, E. F.** *Adion Engineering Co.*: CHAPTER 6, IMAGE TRANSFER; CHAPTER 8, ETCHING
- Ginsberg, G. L.** *General Atomics Corporation*: CHAPTER 1, DESIGN AND LAYOUT; CHAPTER 3, CIRCUIT COMPONENTS; CHAPTER 20, MULTILAYER DESIGN AND LAYOUT
- Grondahl, V. J.** *Rockwell International*: CHAPTER 22, FABRICATING MULTILAYER CIRCUITS; CHAPTER 23, LAMINATING
- Henningesen, C. G.** *Insulectro*: CHAPTER 2, LAMINATE MANUFACTURE AND QUALITY CONTROL; CHAPTER 5, LAMINATE MACHINING; CHAPTER 21, MULTILAYER MATERIALS SELECTION AND SPECIFICATIONS
- Joslin, W.** *Consultant*: CHAPTER 10, MANUAL ASSEMBLY
- Schnorr, D. P.** *RCA Corporation*: CHAPTER 13, ASSEMBLY REPAIR
- Smith, G. A.** *Department of Defense*: CHAPTER 18, QUALITY ASSURANCE PROCEDURES AND METHODS
- Tull, L. E.** *Contouring Technology, Inc.*: CHAPTER 5, LAMINATE MACHINING
- Waryold, J.** *Hunnicut Division, Columbia Chase Corporation*: CHAPTER 12, CONFORMAL COATINGS
- Zimmerman, B. W.** *Department of Defense*: CHAPTER 18, QUALITY ASSURANCE PROCEDURES AND METHODS

Preface

The "printed circuit" process continues to be the basic interconnecting technique for electronic components. There has been no serious competition in the past and none looms in the future. The fundamental building blocks for this process also remain intact, specifically: careful design practices, plating and etching chemistry, metal-clad plastic machining, and photopolymers and plastics control. However, the industry has spent a great deal of time and effort to achieve more precision and lower costs and to find new applications, all with greater quality. The result has been an extension and evolution of the original process rather than the rise of substitute processes. The changes, as they evolved over the past years, have been profound; and while the process described in the first edition will still produce a basic board, it obviously does not address areas that were developed after its publication. This evolutionary process also applies to the material in the second edition of the "Printed Circuits Handbook."

We have revised almost all of the material to update and emphasize those areas in the original process that have changed. We have also added chapters to provide detailed development and control information in the dominant variations of the process, namely multilayer and flexible printed wiring. This book, therefore, is not just a more current set of data on the basics but also contains an almost new book on a large area of material not addressed in the first edition.

Many of these new developments, such as dry film resist, NC drilling, additive circuits, and industry standards, are extremely important to our ability to fabricate, control, assemble, and test circuit boards on an efficient, competitive basis. Some developments are extensions of the basic processes into more complex applications. Other developments—such as pollution control and process-waste treatment—are critical to the survival of a shop in general. All of these developments are described in this edition with the same detailed "how-to" approach used in the first edition.

The term "printed circuit" has been controversial since its original usage. Many have found it less definitive than "etched wiring" or "printed wiring" and, over the years, these more descriptive terms have generally been included in more and more official documentation.

However, "printed circuits" has meant what is in this book to most people from the beginning and seems to have passed into the world's language as the most recognized and used, if not most technically correct, term. Therefore, this book continues to be titled "The Printed Circuits Handbook." In the text of the book itself, however, we have chosen "printed wiring" as the preferred term.

Special appreciation is due to the IPC (The Institute for Interconnecting and Packaging Electronic Circuits) and its Executive Director, Ray Pritchard, for the full and sincere cooperation I have received in the preparation of this book. I'm sure it was a lot more work than was expected. I would also like to acknowledge the support and encouragement of Ray Demere and Bob Watson at Hewlett-Packard. And a special thanks to Sallie Wells Carlson for her help on spelling and other technical production problems.

Clyde F. Coombs, Jr.

Printed Circuits Handbook

Section **1**

Engineering

Chapter 1

Design and Layout

DIETER W. BERGMAN

Aeroneutronics-Ford, Willow Grove, Pennsylvania

G. L. GINSBERG

General Atomics Corporation, Philadelphia, Pennsylvania

Introduction	1-4
General Considerations	1-5
1. Printed Wiring Selection	1-5
2. Decision to Use Printed Wiring	1-5
3. Printed Wiring Types	1-6
4. Packaging Density	1-7
5. Relative Costs	1-8
Technology Description	1-9
6. Printed Boards	1-10
7. Double-sided Printed Boards	1-10
Specifications and Standards	1-12
8. Institute of Printed Circuits	1-12
9. Department of Defense	1-13
10. American National Standards Institute	1-13
11. International Electrotechnical Commission	1-13
12. Abstracts	1-13
General Design Considerations	1-15
13. Initial Design Development	1-15
14. Overall Cycle	1-15
15. Design Checklist	1-16
Mechanical Design Factors	1-17
16. Board Mounting	1-17
17. Board Guiding	1-17
18. Board Retaining	1-18
19. Board Extracting	1-18
Board Size and Shape	1-18
20. Maximum Size	1-18
21. Thickness	1-18
22. Cost of Manufacturing	1-18
23. Board Strength	1-18

24. Partitioning	1-19
25. Testling Considerations	1-19
26. Fault Location	1-18
27. Size Definition	1-20
Material Selection	1-20
28. Mechanical Considerations	1-20
29. Electrical Considerations	1-20
Conductor Patterns	1-21
30. Routing and Location	1-22
31. Conductor Shapes	1-22
32. Width and Thickness	1-22
33. Spacing	1-23
34. Conductor Pattern Checklist	1-24
Holes	1-25
35. Hole Fabrication	1-25
36. Through-hole Connections	1-26
Electrical Design Factors	1-26
37. Resistance	1-26
38. Capacitance	1-26
39. Characteristic Impedance	1-27
Environmental Factors	1-26
40. Shock and Vibration	1-28
41. Thermal	1-31
42. Protective Coating	1-33
Layout	1-35
43. Layout Procedure	1-35
44. Automated Techniques	1-36
45. Layout Checklist	1-39
46. Artwork Factors	1-40
Artwork Generation	1-40
47. Methods	1-40
48. Materials	1-41
49. Considerations	1-41
50. Camera Techniques	1-42
Documentation	1-42
51. How Much Documentation?	1-42
52. Minimal Drawing Package	1-42
53. Formal Documentation	1-43

INTRODUCTION

Initially, printed wiring was developed to adapt an Interconnection method to mass production and assembly schemes and to economize on weight and space in military equipment. It was soon apparent, however, that other advantages would accrue from its use. The close control over and reproducibility of electrical parameters was soon recognized as an important and valuable attribute. Wiring bulk was greatly reduced. Wiring errors, except those in the initial design, were all but eliminated.

Printed wiring was soon found to have the design flexibility to provide characteristics (electrical and mechanical) compatible with all types of electronic equipment. The adaptability of printed wiring to automation, in both fabrication and use, not only resulted in cost savings but also, and more importantly, facilitated controls for enhanced reliability of the total electronic system.

No finished product is ever better than its original design or the materials from which it is made. The manufacturing process can, at best, merely reproduce the design. That is as true of printed wiring assemblies as of any other product the design cycle of which must be repeated with each new board, and the design layout is usually done by a draftsman or other nonengineering person. The need for formalizing layout and design methods and

procedures can, therefore, assume critical proportions. The purpose of this chapter is to provide information that leads to thoughtful decisions and helps to ensure that all pertinent design and layout parameters have been considered.

GENERAL CONSIDERATIONS

"Design and layout" includes the perspective of total system hardware, which includes not only the printed wiring but each and every component in its final orientation. Design and layout considerations must also encompass the relations between and interactions of the components and assemblies throughout the system. Therefore, some major considerations will be the following:

1. User system objectives
2. Product specifications
3. Life expectancy
4. Electronic circuit gain, impedance, voltage, etc.
5. Maintenance concepts at the component level, in service (dynamic), out of service (bench repair), and one-shot with no adjustment or maintenance (throwaway)
6. Environment for storage, assembly, transportation, use, and repair
7. Compatibility of manufacturing method with organization plan, size of production lot, and degree and type of mechanization
8. Material and component sources, performance data, availability, cost, and verification screening for specific relevant design data

These functional and performance requirements, including any particular special considerations, must be properly balanced. As in all kinds of design, many trade-offs must be made in the course of developing the optimum final solution. It is not possible to fulfill all aims completely and adequately and also give top priority to every trade-off involved. Therefore, the least possible number of constraints should be imposed upon the design and the associated artwork.

The trade-off curves of Figs. 1.1 and 1.2 demonstrate a few of the contradictions that often must be resolved. Accurate and meaningful analysis can be achieved best by utilizing the combined judgments of people with diverse design and manufacturing experience.

1. Printed Wiring Selection The basic function of printed wiring is to provide support for circuit components and to interconnect the components electrically. To achieve these results, numerous printed wiring types have been developed; they vary in base dielectric material, conductor type, number of conductor planes, rigidity, etc. As previously stated, the printed wiring designer should be familiar with the variations and their effect on cost, component placement, wiring density, delivery cycles, and functional performance. Otherwise, he will be unable to select the printed wiring structure with the optimum combination of features for the particular electronic apparatus or system requirements.

2. Decision to Use Printed Wiring Although there are frequently good and definite reasons for using printed wiring, there are also potential misapplications; and awareness of the latter will preclude ultimate failure. As with any good thing, there are limitations and associated problems which can cause trouble if not recognized. By careful review of the technology, most such limitations and associated problems can be avoided.

The basic advantages of using printed wiring instead of other interconnection wiring and component-mounting techniques are found in the following areas:

1. The physical characteristics of printed wiring lend themselves to greater versatility in packaging design than does conventional wiring.
2. Wiring is permanently attached to the dielectric base, which also provides a mounting surface for the circuit components.
3. When printed wiring is properly applied, miswiring or short-circuited wiring is not normally possible.
4. High level of repeatability affords uniformity of electrical characteristics from assembly to assembly.
5. The printed wiring technique significantly reduces the bulk and weight of interconnection wiring. Planar construction provides a very neat means of routing conductors.

6. The location of parts is fixed; identification is simplified; and color coding of connectors is eliminated.
7. The printed wiring process lends itself to the use of visual aids to speed accurate component mounting. Assembly errors are thereby minimized, and the complexity, time, and cost of the testing and checking process are reduced.
8. Printed wiring personnel require minimal technical skill and training.
9. Mass-manufacturing processes and automated techniques can be utilized.
10. Maintenance of electronic equipment and systems is simplified.

Some of the more apparent disadvantages of using printed wiring instead of alternate techniques are as follows:

1. The design of printed wiring requires special skills in layout of components and interconnections because of the regimentation imposed by the essentially planar structure.
2. Utilization of equipment space is limited to planar partitioning.
3. Lead time from initial design stage to delivery of the end product may be lengthy.
4. Design is difficult to change after it has been documented and tooled.
5. Low-quantity tooling is relatively expensive.
6. Repair of printed wiring is sometimes difficult and in some applications is not permitted.

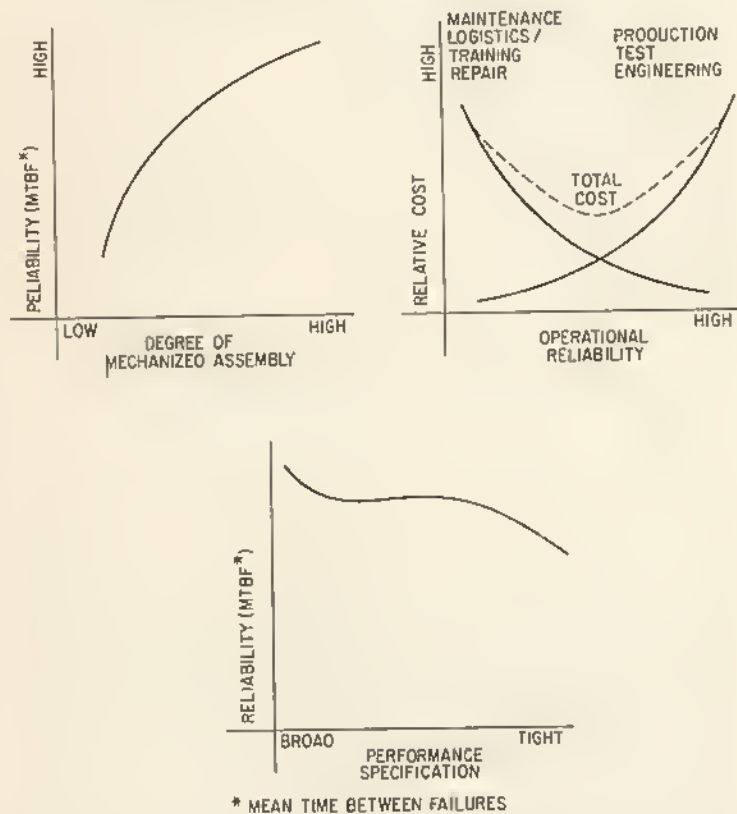


Fig. 1.1 Typical reliability trade-off curves.

3. Printed Wiring Types There are three basic types of printed wiring structures. They are listed below in ascending order of interconnection wiring and component density (either flexible or rigid):

1. *Single-sided* with conductors on only one surface of a dielectric base.
2. *Double-sided* with conductors on both sides of a dielectric base that are usually interconnected by plated-through or otherwise reinforced holes.
3. *Multilayer* structures with three or more conductor layers separated by dielectric material and usually interconnected by plated-through interlayer holes (see Chap. 23).

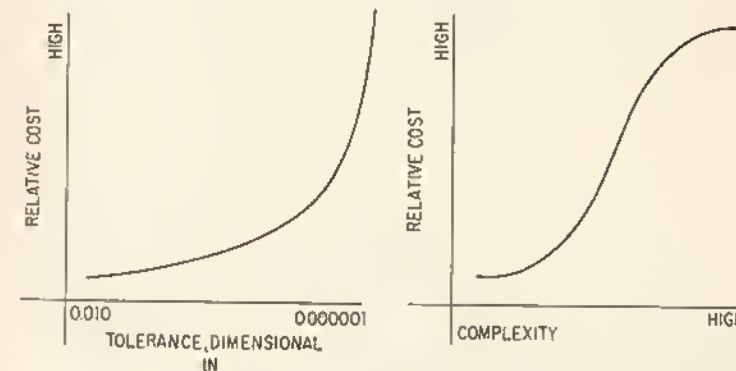
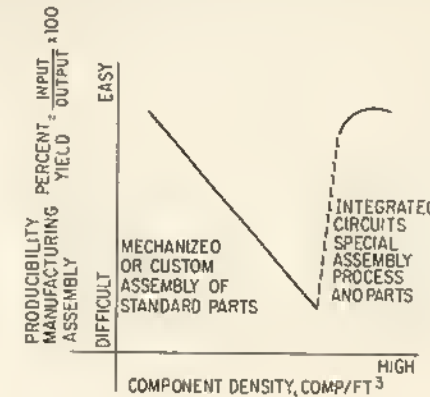


Fig. 1.2 Typical cost trade-off curves.

4. Packaging Density Packaging density is one of the most difficult items about which to generalize. That is due mainly to the differences in printed wiring component sizes and shapes, the number of component leads, and the variable complexity of interconnection. Also, the component lead pattern places a limit on the amount of circuitry that can be effectively packaged on a printed wiring structure. As a measure of the packaging density of a printed wiring structure, the number of component mounting holes per square inch of usable surface can be used. The ratio, although not perfect in application, can be used to estimate the amount of circuitry that can be efficiently interconnected on different printed wiring board types. Based on this design parameter, the following generalized relationships can be stated:

Printed Wiring	Holes/in ²
Single-sided	3-10
Double-sided	10-20
Multilayer	20+

5. Relative Costs Increased interconnection and component densities relate directly to increased costs, manufacturing process complexity, and longer delivery cycles. Relative costs of various printed wiring structures also are difficult to generalize about, since the quantity of each structure produced has a significant effect on the amortization of tooling and setup charges. In addition, the quantity to be fabricated determines the sophistication of the tooling to be used, and that in turn affects fabrication costs.

The difference in material costs must also be taken into consideration when relative cost comparisons are made. A convenient means of comparing printed wiring structure costs is provided in the following generalized relationships:

1. Single-sided printed wiring, rigid, paper-base, phenolic substrate with punched holes; relative cost equals unity (RC = 1).
2. Double-sided printed wiring, rigid, glass-base, epoxy substrate with drilled and plated-through holes; RC = 7.
3. Multilayer printed wiring composite, rigid, glass-base, epoxy dielectric substrate with drilled holes and plated-through holes that are etched back or chemically cleaned, seven layers; RC = 40.

Table 1.1 puts the various techniques in perspective. It indicates that a multilayer board with terminal areas on 0.100-in centers and a minimum line width and electrical spacing

TABLE 1.1 Comparison of Printed Wiring Terminations and Interconnection Capabilities

Type of board	Adjacent terminal area centerline spacing, in	Minimum line width and electrical spacing, in	Termination density, terminations/in ²		Theoretical conductor density, conductors/in		Interconnection capability*
			Theoretical	Practical	Parallel	Crossover	
Single-sided	0.200 ^b	0.030	25	5	4	0	100
	0.150 ^b	0.20	45	10	7	0	315
Double-sided	0.100 ^b	0.20	100	18	10	10	1,000
	0.075 ^b	0.20	177	40	13	0	2,300
Multilayer (7 layers)	0.150 ^d	0.20	45	25	14	14	1,700
							2,260
	0.100 ^d	0.20	100	50	20	20	4,000
							9,200
	0.075 ^e	0.20	177	88	26	26	32,000
400							200

*Interconnection capability (a figure of merit) = terminations/in² × conductors (parallel + crossover) in

^bProduction capability

^cCrossover capability for 88 terminations/in²

^dCrossover capability for 50 terminations/in²

^ePilot production capability

^fDevelopment-model capability

SOURCE: Charles A. Harper (ed.), "Handbook of Electronic Packaging," p. 1-66. McGraw-Hill Book Company, New York 1969.

of 0.020 in has four times as much interconnection capacity as its double-sided-board counterpart.

In the design of any product, after the technical problems have been resolved, an item of concern is the cost. More often than not, potential cost factors change the course of a particular project, equipment, or product. Since initial costing is usually estimated, it is important that all factors be considered in proper perspective in order to avoid canceling an approach that in total would have been the most cost-effective. Some of the key elements of multilayer board costs are (1) board layout, (2) choice of a specification, (3) selection of a base and B-stage material, (4) selection of copper weight, (5) choice of multilayer board thickness and overall size, (6) choice of terminal area pad size and hole sizes, and (7) provisions for maintenance, testing, and repair.

Figure 1.3 identifies most of the factors that should be considered when multilayer printed wiring (MLPW) costs are estimated and indicates how they influence final

pricing. The base costs used in this illustration are for a three-layer board having an area of 24 in² and with 25 plated-through holes of 0.050-in diameter. Adding layers, increasing the number of holes, increasing board area, and/or changing hole size increase costs by certain approximate percents. Base costs used in the chart are for quantities of 500 multilayer boards.

Percent changes from the base costs are estimated by drawing a horizontal line from each factor to the percent line. The total percent can then be applied to the appropriate base cost. If hole size is other than 0.050 in, an additional percent correction is involved. It

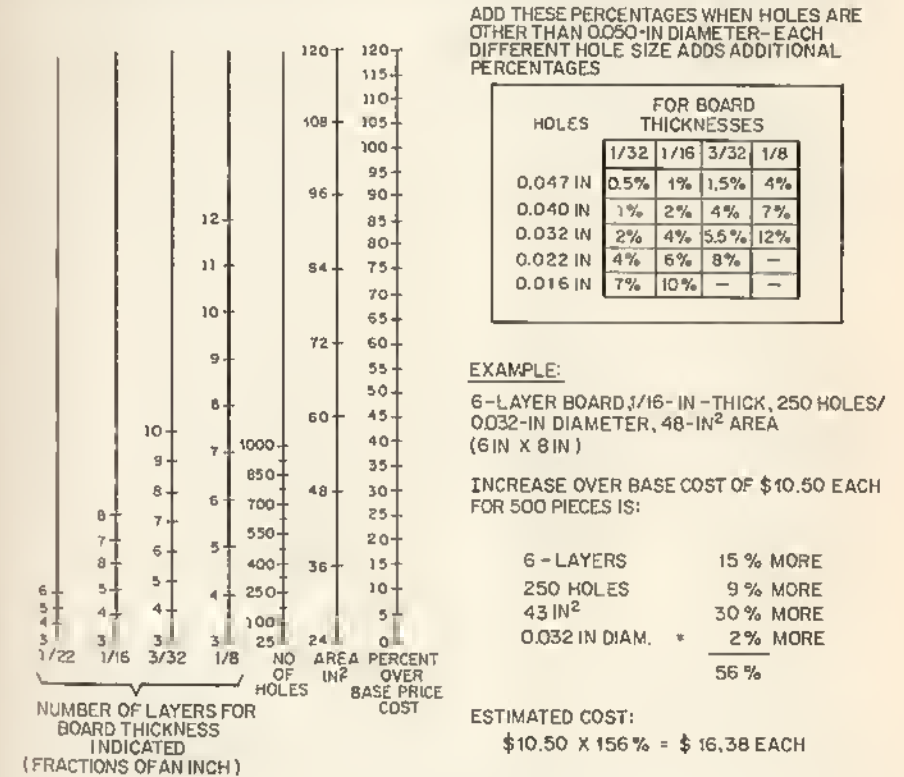


Fig. 1.3 Cost comparison chart.

must be emphasized that these costs are only a guide to the variables in pricing. Other factors can easily raise or lower these estimates, and the base costs themselves are continually subject to change. Each board fabricator should develop its own version of the chart.

TECHNOLOGY DESCRIPTION

Printed wiring, since its inception during World War II, has been predominantly of the rigid type, but a later development of flexible wiring planes necessitated classification as rigid printed circuit board or flexible printed wiring. An important subclassification is by the number of planes, or layers of wiring, which constitute the total wiring assembly or structure.

The printed wiring art has been particularly susceptible to inventions and processes that are of almost infinite variety and novelty; but in spite of the apparent diversity, a relatively small number of processes are still employed to any extent. In this section the most common processes are described. They are classed under the general headings of chemical and mechanical methods, but they will be broken down into single-sided,

double-sided, and multilayer. Boards of the latter type are at present being produced predominantly by chemical processes.

Generally the first—and currently the most widely used—form of printed wiring, the rigid printed board, was early recognized as providing not only a conductive wiring path but also support for and protection of the components it connected and a heat sink to aid in the thermal management of the total package. The chemical methods used in producing rigid printed circuit boards are either (1) subtractive, or etched-foil, or (2) additive, or plated-up. The mechanical methods include stamped wiring, metal-sprayed wiring, embossed wiring, and molded wiring.

6. Single-Sided Printed Boards "Single-sided" has reference to the fact that there is wiring on only one side of the insulating substrate, and that arrangement represents a large

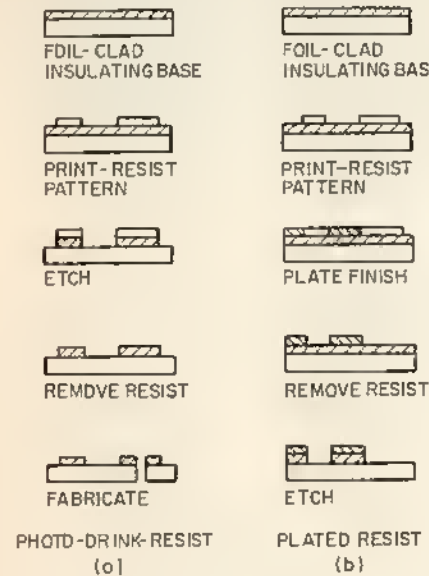


Fig. 1.4 Etched-foil process.

volume of printed boards currently produced. Single-sided boards are used for relatively unsophisticated and simple circuitry, and they are applicable when circuit types and speeds do not demand unusual wiring electrical characteristics.

a. Subtractive, or Etched-foil, Type. As illustrated by Fig. 1.4, the etched-foil process begins with a base laminate composed of a variety of insulators clad with copper or some other metallic foil. Following suitable cleaning and other preparation, a pattern of the desired circuit configuration is printed by using a suitable negative-resist pattern for photoresist or ink resist or a positive-resist pattern if plating is to be used as an etchant. In the case of the latter, gold or solder plating is applied to the nonphotoresist areas. In the next step, etching, all copper not protected by the resist material is removed by the etchant. Following etching, the printed board is stripped, or subjected to removal of resist, and otherwise cleaned to ensure that no etchant remains. The board is then ready to fabricate by drilling, trimming, etc.

b. Stamped, or Die-cut, Type. This process requires a die which carries an image of the wiring pattern desired on the final product. As illustrated in Fig. 1.5, the die may be either photoengraved by using conventional photoresist techniques or machine-engraved. In either case, the die is heated; and when it forces the adhesive-coated foil into the base material, it effects a bond as well as cuts the circuit pattern and embeds it in the base at the conductor edges. Blanking and/or piercing of the board may be done at the same time.

7. Double-Sided Printed Boards When more than one layer of wiring is needed, circuit patterns are placed on both sides of a printed board, and the necessity that arises is for a means of interconnecting the two wiring layers. The usual connection goes through the board rather than around the edge; hence the name "through connection." In many cases the hole through the board serves a dual purpose: it accommodates a component lead and provides a location for some method of interconnecting the two sides. The many processes for effecting interconnection between circuit layers may be generalized as including a plating or a mechanical technique.

a. Plated-through-hole (PTH) Type. There are two variations of the plated-through process; each utilizes a conductor plated through the hole to make the connection. For purposes of comparison, the subtractive process shown in Fig. 1.6a is an extension of the board fabrication process for single-sided boards described above. The additive process shown in *b* is described below.

The subtractive plated-through-hole (PTH) process starts with a double-clad laminate with a series of holes corresponding to the locations where a through connection is

needed. The holes are drilled and deburred, and an electroless coating of copper is applied over the entire board surface including the holes. Next, copper is electrodeposited on the exposed copper foil and sensitized walls of the hole, usually to a thickness of 0.001 in. A negative-, or plating-resist, pattern is then applied and registered to both sides of the material. Resist covers all areas of foil where base copper conductor is not required, and the surplus conductor will subsequently be etched off. The next plating step is to electrodeposit a thin layer of a suitable etch-resist plating, usually solder or gold. The original plating resist, screen or photoresist, is removed, and the circuit pattern is defined by etching away the exposed copper in a suitable etchant. The choice of etchant depends on the type of plating resist used.

The additive process differs from the subtractive process in that no etching is required and the circuit pattern is defined at the same time the through connection is made. Holes are again drilled at desired connection points, and a thin layer of a suitable adhesive is applied as illustrated in Fig. 1.6b. Electroless deposition of copper sensitizes the entire surface and holes and is followed by a "flash" electroplate.

After the registered printing of a plating-resist pattern to both sides of the board, copper is electrodeposited to the desired thickness on exposed areas. Resist is removed; unwanted electroless copper is "flash-etched" off; and excess adhesive is taken off with appropriate solvents. The last step is the curing of the adhesive by subjection to a heat-and-pressure cycle.

b. Mechanical Through-Hole Types. The nonplating techniques, illustrated in Fig. 1.7, preceded through-hole-plating processes for double-sided boards, and they continued in exclusive use until the reliability of the latter was proved and the economic import of

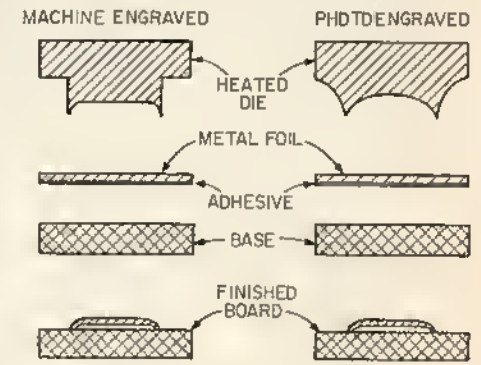


Fig. 1.5 Stamped-wiring process.

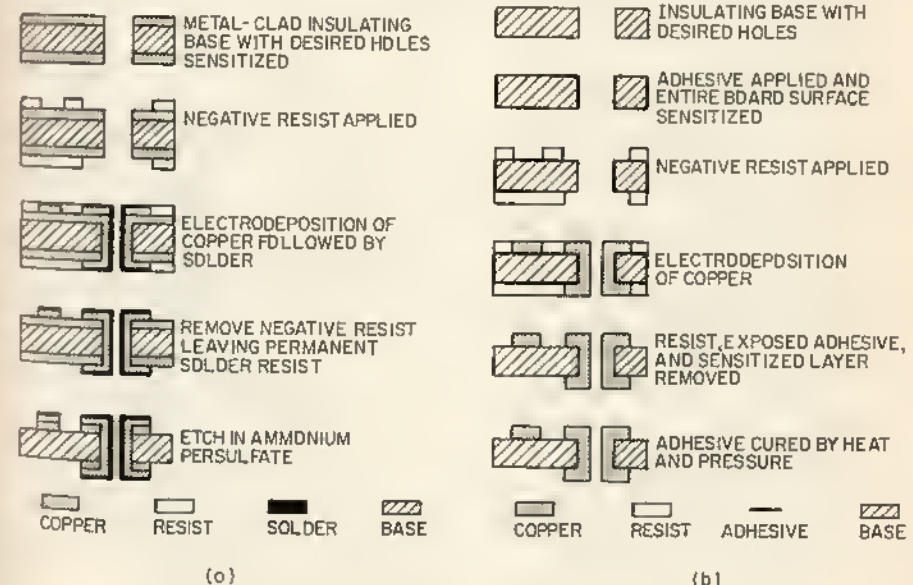


Fig. 1.6 Plated-through-hole process: (a) subtractive, (b) additive.

mass-interconnection techniques was realized. Although not to be construed as an integral part of the board fabrication procedure (as represented by the plated-through hole), mechanical interconnections—actually board assembly techniques—serve the purpose of connecting the two sides of a double-sided board.

A simple and easily made interconnection is represented by the clinched jumper wire illustrated in Fig. 1.7*a*. A formed, uninsulated, solid lead wire is placed through the hole and clinched and soldered to the conductor pad on each side of the board. The lead wires of parts are not normally considered to be interfacial connections.

Three types of eyelets also are commonly used for double-sided board interconnection; they are shown in Fig. 1.7*b* to *d*. Funnel-flanged eyelets are soldered to the terminal areas on the component side of the board prior to insertion of component leads. The other connection is made at assembly when the boards are dip-soldered. The funnel flange, by definition, has an included angle between 55 and 120°. Split-funnel-flanged eyelets differ

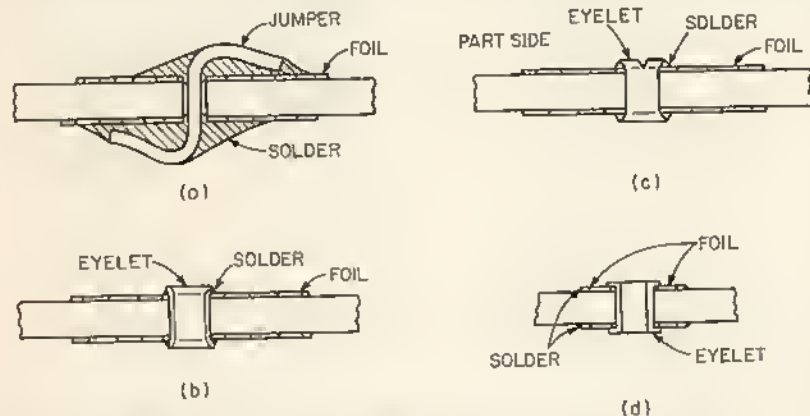


Fig. 1.7 Mechanical interconnections: (a) clinched jumper wire, (b) funnel-flanged eyelet, (c) split funnel-flanged eyelet, (d) fused-in-place eyelet.

only in the split and in the fact that they need not be soldered to the terminal area on the part side of the board before insertion of part leads.

SPECIFICATIONS AND STANDARDS

Specifications are often written to set forth exact needs or help qualify a product or service. They help the contracting agency or buyer monitor or judge the acceptability of the material and/or services supplied, and they put the buyer and supplier on common ground. For products which are obtained on a continual basis or are grouped in a general class, such as printed wiring, specifications and standards which may be supplemented by other documentation or written orders have been developed.

Specifications and guides pertaining to printed wiring may be divided into three general categories: those specifying design requirements, those concerned mostly with materials and acceptability criteria, and those setting forth the performance requirements for printed wiring and associated items.

Although it is not possible to include all of the specifications and standards applicable to printed wiring boards, the most common ones are covered briefly in the following paragraphs.

8. Institute of Printed Circuits Since 1957 the IPC has prepared and released many specifications and standards which are now widely used throughout the printed wiring industry. The initial standard of the IPC, published in 1960, covered the recommended standard tolerances for printed circuit boards. That standard (identified as IPC-D-300) has been revised on several occasions to reflect the increasing sophistication of the printed wiring state of the art.

9. Department of Defense Many aspects of printed wiring design, material selection, and process control are covered by military specifications and standards. It is important that the printed wiring designer have a complete understanding of the military documents that directly affect the design of printed wiring structures. He should also be familiar with the documents which affect design support activities.

10. American National Standards Institute Nearly 4000 standards have been approved to date by the American National Standards Institute (ANSI), formerly the USA Standards Institute. An American National Standard implies a consensus of those substantially concerned with its scope and provisions; it is intended as a guide to the manufacturer, consumer, and general public.

11. International Electrotechnical Commission The "international" specifications covered by this chapter have been prepared by the International Electro-technical Commission (IEC), which is affiliated with the International Organization for Standardization (ISO). A foreword explains the intent of these documents:

The informal decisions or agreements of the I.E.C. on technical matters, prepared by Technical Committees on which all the National Committees having a special interest therein are represented, express, as nearly as possible, an international consensus of opinion on the subject dealt with.

12. Abstracts This section contains abstracts of the major documents that are applicable to the task of printed wiring design. It is intended that an abstract will help the printed wiring designer decide whether to obtain a copy of the latest revision of the document before proceeding with his design.

a. IPC-D-300 (Printed Wiring Board Dimensions and Tolerances, Single and Two Sided Rigid Boards). This specification covers dimensioning and tolerancing limits based on industry capabilities and fabrication costs. It provides five classes for dimensional features to reflect progressive increases in sophistication and cost of tooling, material, and processing.

b. IPC-D-310 (Suggested Guidelines for Artwork Generation and Measurement Techniques). This document has been written to provide a general reference to aid in the making of master patterns required to produce printed circuits. The guide is composed of information submitted by members of IPC committees who represent various companies and government agencies, and it includes techniques suggested in military and commercial practice.

c. IPC-D-350 (End Product Description in Numeric Form). This standard describes record formats for the transmission of end-product description data, in digital form on punched cards and magnetic tape, that are necessary and sufficient for the preparation of tooling and/or the manufacture and testing of printed boards. The record formats may be used in the transmittal of information and data between the printed wiring design phase and the manufacturing facility when the design has been formed by computer-aided processes and/or when the tooling for manufacture of the printed boards is to be performed by numerically controlled machines.

The data contained in each record are general in nature, are not in any particular machine language, and are in a form usable for manual as well as digital interpretation and readout.

d. IPC-D-390 (Guidelines for Design Layout and Artwork Generation on Computer Automated Equipment for Printed Wiring). This guide illustrates accepted techniques used to make printed-wiring artwork by semi- and fully-automated methods. The purpose of this document is to help improve design quality and accuracy of the master pattern, and to help reduce documentation costs.

e. IPC-A-600 (Acceptability of Printed Wiring Boards). This publication is a compilation of visual quality acceptability guidelines for printed boards prepared by the Repairability and Acceptability Committees of the Institute of Printed Circuits. The illustrations in the guide were made to portray certain specific points noted in the title of each. The compendium has been assembled to standardize the interpretations of specifications on printed wiring boards.

f. IPC-CM-770 (Guidelines for Printed Circuit Board Component Mounting). This document has been written to provide a general reference to aid in assembling compo-

nents to printed boards. The guide is composed of selections from mounting specifications submitted by various companies and government agencies and includes techniques suggested for spaceflight, military, and commercial applications.

g. **MIL-STD-100** (Engineering Drawing Practices). This military standard prescribes procedures and format authorized for the preparation of Form 1 engineering drawings and associated lists prepared by or for the departments and agencies of the Department of Defense as prescribed by military specification MIL-D-1000.

h. **MIL-STD-275** (Military Standard, Printed Wiring for Electronic Equipment). This standard establishes design principles governing the fabrication of rigid single- or double-sided printed boards and the mounting of parts (including integrated circuits) and assemblies thereon for use in electronic equipment. The requirements do not apply to parts, such as resistors, inductors, capacitors, or transmission lines, fabricated by using the techniques.

i. **MIL-STD-429** (Printed Wiring and Printed Circuits Terms and Definitions). This standard establishes terms and definitions which have a specific meaning when applied to printed wiring and printed circuit nomenclature.

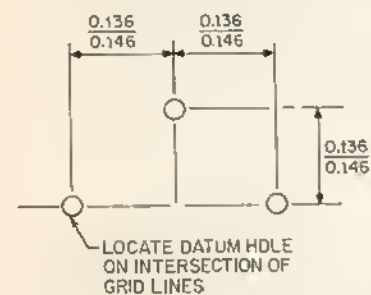


Fig. 1.8 Off-grid hole-mounting pattern.

circuit assemblies by dipping, brushing, spraying, or vacuum deposition.

l. **MIL-P-55110** (Printed Wiring Boards). This specification covers printed boards consisting of a conductor pattern on the surface of one or two sides of an insulating base and associated interfacial connections and standoff terminals.

m. **USASI Y14.5-1966** (Dimensioning and Tolerancing for Engineering Drawings). This standard establishes the rules, principles, and methods of dimensioning and tolerancing used to specify design requirements on engineering drawings. It also establishes uniform practices for stating and interpreting the requirements.

n. **ANSI Y14.15** (Electrical and Electronics Diagrams). This standard contains definitions and general information applicable to most of the commonly used electrical and electronic diagrams. It also includes detailed recommendations on preferred practices for use in the preparation of such diagrams. The recommended practices are intended to eliminate divergent electrical and electronics diagram drafting techniques.

o. **ANSI Y32.16** (Reference Designations for Electrical and Electronics Parts and Equipments). This standard covers the information and application of reference designations for electrical and electronic parts and equipment. The reference designations are intended for uniquely identifying and locating discrete items on diagrams and in a set and for correlating items in a set, graphic symbols on diagrams, and items in parts lists, circuit descriptions, and instructions.

p. **IEC 97** (Grid System for Printed Circuits). The recommended grid system for printed circuits will ensure compatibility between the printed circuits and the parts to be mounted on them.

q. **IEC 321** (Guidance for the Design and Use of Components Intended for Mounting on Boards with Printed Wiring and Printed Circuits). This document gives guidance to the designer, manufacturer, and user of components on matters relating to the specification, design, production, supply, and application of components particularly suited for use with printed circuits. In its present form, the report is intended to be applied to components which are to be soldered to printed circuits.

j. **MIL-D-1000** (Drawings, Engineering, and Associated Lists). This specification prescribes general requirements for the preparation of engineering drawings and associated lists and for the application of intended-use categories to acquisition of the drawings. It reflects Department of Defense policy to buy only the engineering drawings that are needed and to encourage procurement of commercial drawings when they are adequate for the purpose. "Engineering drawings," as used in this specification, include associated lists.

k. **MIL-A-46058** (Insulating Compound, Electrical, for Coating Printed Circuit Assemblies). This specification covers conformal coatings which are suitable for application to printed

r. **IEC 326** (General Requirements and Measuring Methods for Printed Wiring Boards). This recommendation relates to printed wiring boards, however manufactured, when they are ready for mounting of the components. It is intended as a basis for agreement between purchaser and vendor.

GENERAL DESIGN CONSIDERATIONS

Probably the most important considerations in the design of electronic equipment are reliability, satisfactory performance, and maintainability. Those factors are not inherent in printed wiring per se; but by adequate design and proper selection of materials and manufacturing techniques, they can be introduced into the system.

13. Initial Design Development. There is no generally approved method for the development of a design for a printed board, but the following program sequence is useful.

1. Start with a careful design of the electrical circuit and prepare a schematic or logic diagram.

2. Build a breadboard; then analyze the circuit operation by using worst case component values or operating conditions. Use qualified components that have been tested for reliability. Revise the schematic as necessary.

3. Furnish the final schematic or logic diagram and component list to the design draftsman.

4. Select the shape and size for the board to accommodate all components and to fit the available space within the enclosure or the available area.

5. Select appropriate input-output connectors to accommodate all necessary input-output signals and be compatible with space and environmental requirements.

6. Supply specific information on widths of conductors, spacing of conductors and terminal areas, most suitable or critical routing of conductors, hole sizes and locations, type of electrical interconnections, shape and bulk of components, distance between components, method of component lead termination, electroplating requirements, etc., to the design draftsman. Use information from appropriate specifications. Conductors should, whenever possible, be restricted to the widths given in Table 1.2, especially when the

TABLE 1.2 Typical Terminal Area Diameter and Maximum Conductor Width Relations

Terminal area diameter, in	Maximum conductor width, in
0.040	0.015
0.050	0.020
0.062	0.025
0.075	0.025
0.086	0.040
0.100	0.040
0.125	0.050
0.150	0.075
0.175	0.100

dimensions are governed by the numerically controlled drafting machine. (See Width and Thickness, Sec. 32, for more details concerning conductor width selection.)

7. Locate all holes on a modular grid system. When holes for a component having rigid terminals cannot fall on the grid intersection, locate at least one hole on a grid intersection and dimension other hole locations that are not on a grid intersection (see Fig. 1.8 opposite.)

14. Overall Cycle. Printed wiring designers must be aware of the total time required to produce printed wiring. Although printed wiring design usually comes at the end of the equipment design schedule, the printed wiring itself is usually required early in the equipment assembly cycle. An inadequate understanding of printed wiring design, tooling, and fabrication times can lead to serious problems. The prime factors which affect those times are the type of structure involved, the degree of tooling required, the

manufacturing equipment available, and the quantity involved. (Figure 1.9 is an example of a schedule for a small production quantity of a 5 × 6-in printed board containing 45 dual-in-line devices being produced in a dedicated manufacturing facility.)

15. Design Checklist The following checklist covers the universal areas of concern in the design cycle. Other items should be added for specific applications.

a. General

1. Has the circuit been analyzed and divided into basic areas for a smooth signal flow?
2. Does the organization of (1) permit short or isolated critical leads?
3. Has shielding been effectively used where necessary?

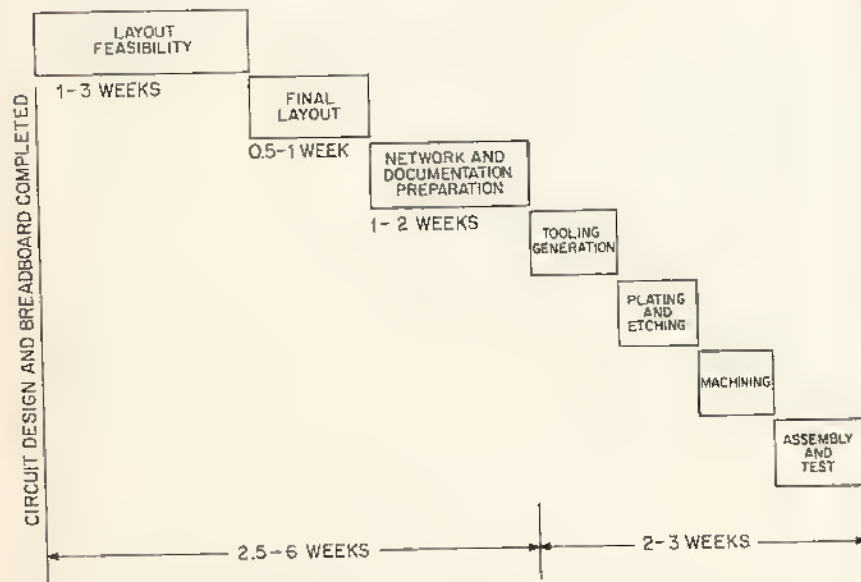


Fig. 1.9 Layout and fabrication cycle. Automated techniques can reduce this cycle by 50 percent or more.

4. Has a basic grid pattern been fully utilized?
5. Is the board size optimum?
6. Are preferred conductor widths and spacings used wherever possible?
7. Are preferred pad and hole sizes used?
8. Are the photomaster and schematic compatible?
9. Is jumper use kept to a minimum? Do the jumpers clear components and hardware?

10. Is lettering visible after assembly, and is it of the proper type and size?
11. Are the larger areas of copper broken up to prevent blistering?
12. Have tool-locating holes been provided?

b. Electrical

1. Have conductor resistance, capacitance, and inductance effects been analyzed— and especially the critical voltage drops and grounds?
2. Are conductor and hardware spacings and shape compatible with insulation requirements?
3. Is insulation resistance controlled and specified where it is critical?
4. Are polarities adequately identified?
5. Do conductor spacings split the geometric mean with respect to leakage resistance, voltage, and noise?
6. Have the dielectric changes related to surface coating been evaluated?

c. Physical

1. Are terminals and control locations compatible with the total assembly?
2. Will the mounted board meet shock and vibration requirements?
3. Are specified standard component lead spacings used?
4. Are unstable or heavy parts adequately retained?
5. Are hot components properly heat-sinked, cooled, or isolated from the board and other heat-sensitive components?
6. Are potentiometers and other multiterminal components properly oriented?
7. Are components organized and oriented for ease of assembly and inspection?
8. Have all possible interferences on board and total assembly been eliminated?
9. Are mating holes properly dimensioned?
10. Are tolerances complete and reasonable?
11. Have the physical characteristics of coatings been controlled and evaluated?
12. Are the ratios of hole to lead diameters within acceptable limits?

MECHANICAL DESIGN FACTORS

Although the printed board lends mechanical support to the components, it should not be used as a structural member of the overall equipment. Support should be provided at peripheral intervals of at least 5 in at the board edge. An analysis of the board for any equipment shock and vibration requirements should reveal whether additional support is required. Board warpage should be considered in design of the mechanical mounting arrangements. When the edge-board type of connector is to be used for the printed board, a careful design is required to assure that the overall board thickness remains within the tolerance capability of the connector. The design of the board-mounting means slides, handles, and other mechanical design features should be such that close tolerances are not required on the board dimensions.

Factors that must be considered in the selection and design of a printed board, include:

1. Configuration of the board—size and form factor(s)
2. Need for mechanical attachment or connector type
3. Circuit compatibility with other circuits and environments
4. Horizontal or vertical board mounting as a consequence of such other factors as heat and dust
5. Environmental factors requiring special attention, such as heat dissipation and ventilation, shock and vibration, humidity, salt spray, dust, and radiation
6. Degree of support
7. Retention and fastening
8. Ease of removal

16. Board Mounting Printed boards should be supported within 1 in of the board edge on at least three sides. As a good practice, boards between 0.031 and 0.062 in thick should be supported at intervals of at least 4 in and boards thicker than 0.093 in at intervals of at least 5 in. The practice increases the rigidity of the board and breaks up possible plate resonances.

The choice of a particular board-mounting technique is normally made after consideration of such design factors as:

1. Board size and shape
2. Input/output terminations
3. Available equipment space
4. Desired accessibility
5. Type of mounting hardware
6. Heat dissipation required
7. Shielding required
8. Type of circuit and its relation to other circuits

17. Board Guiding A major advantage of using plug-in printed wiring assemblies, rather than other circuit-packaging techniques, is the suitability of printed boards for use with board guides for ease of maintainability. The exact board guide technique to be used depends on the shape of the board, the degree of accuracy needed to assure proper mating alignment, and the degree of sophistication desired.

18. Board Retaining Quite often, shock, vibration, and manual handling requirements necessitate that the printed board be retained in the equipment by mechanical devices. The selection of a proper board retainer is important, since the retaining devices will reduce the amount of board area available for component mounting and interconnection and will significantly add to the cost of the electronic equipment.

19. Board Extracting A number of unique principles have been applied to the solution of various problems of printed board extraction, and the consequence has been a constant increase in the number of types of extracting tools. Many of these tools use a minimum of board space (and thereby maximize the amount of board area for circuitry and component mounting) while protecting both the board and the associated input-output connectors from damage during extraction. Selection among the many different types of printed wiring board extraction tools should take into account the following considerations:

1. Board area required to be free of circuit components
2. The extractor's effect on board-to-board mounting pitch
3. The need for special provisions in the printed board design, such as mounting holes and notches
4. The size of the extractor, especially if the extractor is to be stored in the equipment with which it is used
5. The need for an extraction device that is permanently attached to the board assembly, usually by riveting
6. The need for special design considerations in the printed board mounting chassis, such as load-bearing flanges
7. The suitability of the extractor to be used with a variety of size, shape, and thickness of printed board assemblies
8. The cost of using the extractor, both in piece price and added design
9. The degree of access required inside the equipment to engage and use the extraction tool

BOARD SIZE AND SHAPE

The final choice of printed board size and shape will probably be a compromise. Some factors will indicate a large board, others a small one. Herewith is a review of some of the major factors which affect the selection of board size and shape.

20. Maximum Size In order to first determine the maximum size of printed board to be used, artwork generation size and accuracy, throat clearances on drills, platen sizes on presses, and the capacities of etching and plating baths must be checked. Also, any equipment used in board assembly must be considered. What is the largest board that can be wave-soldered? What sizes are the cleaning tanks? One of these points (if not more than one) will place a maximum limit on board size, but none of them will indicate an optimum size.

21. Thickness Printed board thicknesses can vary from 0.020 to 0.250 in, but rigid boards predominantly have a thickness of 0.0625 in (1/16 in). That thickness is common to plug-in printed wiring. Also, the structure thickness must be sufficient to mount components and meet the expected environmental requirements.

22. Cost of Manufacturing Board manufacturing costs vary with board size, tolerances and clearances, finish, quantity, and delivery requirements. Assuming that any size of board can be made (i.e., that the limits mentioned above have been removed), the first point to consider is the material cost. It will vary as the number of boards that can be cut from a sheet of raw laminate varies. The costs of drilling and of the chemicals used vary directly with the area of the board.

Small boards would generally be grouped together on one production blank (see Fig. 1.10), so labor costs can be assumed to follow a step function similar to that for raw material. There will be some constant charges to be added, and an allowance for a fall in yield with larger boards must be made.

23. Board Strength Some of the problems in choosing the size of board depend on the circuits used; others are mechanical in nature. Since the mechanical problems are usually solvable in several ways to fit the best electrical or circuit requirements for the equipment

under consideration, they can be considered first. The matching of the length of the board to the board connector is obvious, and since the length of the connector will be determined by the number of contacts, length can be regarded as an electrical or circuit requirement. The amount of filler needed to insert or withdraw a board must be kept in mind. When connectors which have many contacts are used, the boards can be very difficult to move without the use of some form of tool or cam mechanism. The other purely mechanical considerations are the strength and resistance to vibration of the board, any manufacturing limitations, and the ease or difficulty of dissipating the heat generated in use.

Board strength is unlikely to limit board size; bowing, warping, and the effects of vibration are far more likely to be restrictive. Large boards can be stiffened with supports riveted in hand or in place, or spacers can be mounted on the boards to assure a minimum of clearance between boards.

24. Partitioning Minimization of the total number of interconnections can help in determining board size. Partitioning equipment in different ways can result in very different numbers of interconnection points being required. A skeleton block diagram of the whole equipment is needed; it should show all the necessary block-to-block connections. It can be examined to determine the points at which it can be divided so as to break the smallest number of connections; then similar diagrams of the major logic functions within each block can be prepared and split down into smaller areas.

Generally, as board size is increased, the number of interconnections per board increases. The increase is fairly rapid at first; then comes a point at which doubling the board size adds very few more interconnections; and then a further increase in board size can add considerably more.

25. Testing Considerations The time necessary to test each board and locate and repair faults has been cited as a strong argument in favor of the use of very small boards. The reason for making the boards at all is to make a complete equipment or, more probably, an equipment plus a set of spares, and the only cost worthy of consideration is that of testing the boards for the whole equipment. With any form of automated tester, the handling time will be greater than the testing time, so the larger the boards the less the total testing cost.

Another point on board size which must not be overlooked is that, the larger the boards are made, the more money will be tied up while any faulty components found during testing are replaced. Also, the more components put on a board the higher the probability of a board having to be repaired.

How much attention should be paid to this point in selecting a board size depends very largely on what facilities can be provided for rapid repair of faults found in test.

26. Fault Location It is desirable to minimize the time necessary to locate any fault and replace the board concerned. That clearly indicates the use of the largest board possible because, if the whole equipment can be made on one board, the time taken in deciding which board to replace will be zero. However, with most equipment the fault diagnosis time will not be so simple to determine, and attempts to squeeze too much on each board can make fault location more rather than less difficult. The entire equipment should first

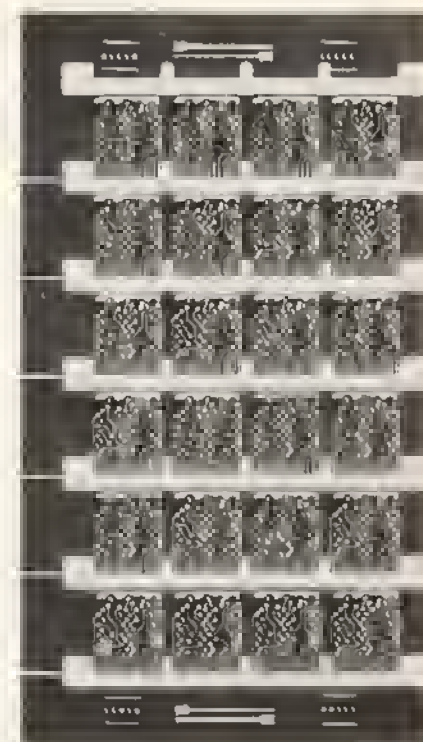


Fig. 1.10 Multiple-image board fabrication panel.

be considered and then partitioned in such a way that a fairly block can be found in the shortest possible time.

27. Size Definition With so many factors to be taken into account, board size must inevitably be a compromise unless the board must be fitted into a frame already designed. In that case, however, the size is not really in question, and only the packing density and type of connector to be used need be settled.

MATERIAL SELECTION

Printed wiring substrate materials are chosen for their mechanical and electrical characteristics and, of course, their relative costs to buy and fabricate. A comparison of the various substrate material properties will help the printed wiring designer select the optimum substrate for the application.

28. Mechanical Considerations Mechanical properties of substrate materials which often have an important bearing on the printed wiring assembly are water absorption, coefficient of thermal expansion, thermal rating, flexural strength, impact strength, tensile strength, shear strength, and hardness. All these properties affect both the functioning and producibility of the structure.

The dielectric base materials for printed wiring boards are, for most applications, one of the following:

1. Phenolic-resin-impregnated paper
2. Acrylic-polyester-impregnated random glass mat
3. Epoxy-impregnated paper
4. Epoxy-impregnated fiberglass cloth

Each of the base materials can be flame-retardant or not. The laminates increase in cost from the least expensive phenolic or polyester bases through the most expensive epoxy-fiberglass bases. There are other differences among these materials as to physical, thermal, and electrical properties and performance; but for general use as a printed board base, all of the materials are adequate.

Materials 1, 2, and 3, above, are punchable and thus are capable of providing low-cost hole formation and permitting other fabrication cost savings when they are used to meet large-quantity requirements. However, the materials have limited use with plated-through holes in printed boards because their dimensional stability is insufficient for high-density designs. The result is excessive expansion in the z axis direction (perpendicular to the board surface), and that leads to frequent cracking of the plated-through-hole barrel when the board is subjected to thermal shocks.

The most popular material for use as printed board with plated-through holes is epoxy-fiberglass. Its dimensional stability is adequate for its use with high-density wiring and minimizes the incidence of cracks in plated-through holes. Its relatively good dimensional stability and its availability in a semi-cured prepreg stage also make it the most desirable type of material for rigid multilayer construction also.

One drawback of epoxy-glass laminate is the difficulty of punching the material in the thickness range usually associated with printed boards. For reason of that difficulty, all holes are usually drilled and routing operations are used to form the board outline.

The temperature levels that various grades of printed board materials can endure in mechanical applications are indicated in Fig. 1.11. They will vary to some extent with thickness of the material.

Although warp and twist in an etched printed board can be attributed to the fabrication process or the conductor pattern configuration, they are more frequently induced during the manufacturing of copper-clad laminate. The designer can help minimize the effects in two ways: First, the direction of conductors should be such that bending stresses released in the etched copper foil on one side of the board are opposed by a similar stress pattern on the other side. That implies there is also a balance between large conductor areas on both sides of the board, when they exist. Second, large conductor areas used as shields and grounds should be broken by etched slots.

29. Electrical Considerations The most important electrical characteristics of the insulating substrate in dc and low-frequency ac applications are insulation resistance, arc and

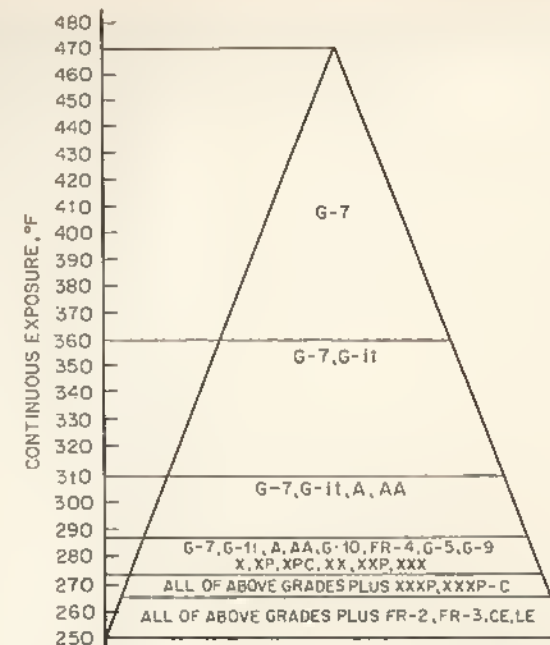


Fig. 1.11 Temperature levels of NEMA-grade materials.

tracking resistance, and flashover strength. In high-frequency and microwave applications, the important considerations are capacitance, dielectric constant, and dissipation factor. In all applications the current-carrying capacity of the printed conductor is important. The electrical characteristics of two commonly used printed board materials are shown in Table 1.3.

TABLE 1.3 Electrical Characteristics of Two Board Materials

Parameter	G-10	G-11
Dielectric strength, perpendicular to, laminate, typical $V/in \times 10^{-3}$	175-525	175-525
Dielectric strength, parallel to laminate, typical, kV/in	45-80	45-80
Insulation resistance, $M\Omega$	$0.2-3 \times 10^6$	$0.1-3 \times 10^6$
Surface resistance, $M\Omega$	$0.1-2 \times 10^6$	$0.01-1 \times 10^6$
Volume resistivity, $M\Omega \cdot cm$	$0.6-3 \times 10^6$	$0.1-1 \times 10^6$
Dissipation factor, as received, at 1 MHz	0.019-0.030	0.013-0.025
Dissipation factor, 24 h in H_2O , at 1 MHz	0.025-0.045	0.015-0.045
Dielectric constant, as received, at 1 MHz	4.6-5.2	4.4-5.2
Dielectric constant, 24 h in H_2O , at 1 MHz	5.0-5.8	4.5-5.8
Dielectric constant at 10 MHz	5.0-5.6	
Dielectric constant at 25 MHz	4.7-5.2	

*Temperature- and humidity-conditioned 96 h at 35°C, 90% RH.

CONDUCTOR PATTERNS

The simplest process for forming conductor patterns is to etch the patterns from the foil laminated to the printed wiring base. The process requires a minimum number of steps and has been widely used in mass production. Foil can be etched on either side or on both sides of the laminate, but etching does not provide for interconnections between conductor planes except by hole plating or by mechanical means. Similar types of printed wiring patterns can be produced by proprietary additive processes or special types of resist.

30. Routing and Location The printed conductor should travel by the shortest path between components within the limits of the wiring rules imposed. The latter may restrict parallel runs of conductors owing to coupling between them. Good design dictates that the minimum number of layers of wiring be used and also that the maximum line width and terminal area size be commensurate with the density of packaging required. Since rounded corners and smooth fillets avoid possible electrical as well as mechanical problems, sharp angles and leads in conductors should be avoided.

Translation of a circuit schematic to its proper physical realization generates many side effects, and crosstalk is one of them. Crosstalk is often a ground plane problem. There are two aspects to the problem: the crosstalk generated in individual boards and crosstalk through the system ground. If careful routing of individual sections of each circuit provides an exclusive conductor to a single ground at essentially a single point, the problem need hardly exist in the point of being harmful. This indicates that a major concern in determining internal circuit layout will be the grounding circuitry. Accordingly, it will partially determine physical subsection size and workable unit grouping.

Conductor routing is the very heart of printed board design, yet many engineers appear to feel it is best left to the layout team to struggle with as best they can. Somewhat helpfully, a realization of the importance of package placement and routing in getting the best possible circuit performance and also the realization that, with more and more complex boards, layout takes an excessive time and causes serious bottlenecks in production have led to the use of computer programs for both routing and layout. So far, however, it seems the best boards can result from manual routing when a well-considered system is used. A computer could do the job if it were fed with sufficient information, but it would take a long time to establish a good layout for a large, complex board and a full routing program. It appears that the best solution is computer-aided design (CAD), in which the computer gets on with the tedious "donkey work," and calls for assistance every time it meets a problem. A moderately experienced layout drafter will spot several possible ways through an apparently blocked board area in the time it would take the computer to try the first couple of dozen logical approaches. The operator might suggest moving several lines or even changing the package allocation to clear the trouble, whereas the computer might give up the problem as insoluble. If the computer displays its interim results on a cathode-ray-tube display, the operator can watch the board layout "grow" and can possibly even prevent the computer from doing something which might block subsequent runs.

One great advantage of CAD for routing is that the final output can be in the form of control tapes for a coordinate-plottting machine and for a numerically controlled drill.

31. Conductor Shapes Sharp corners and acute angle leads in conductors should be avoided when possible because of the additional stresses imposed in both the conductor itself and the adhesive bond and also because of the electrical problems arising from local field intensification. Therefore, although more costly from a drafting standpoint, rounded corners at conductor leads and smooth fillets at the junction of conductors and terminal areas, or lands, are desirable. Such rounded contours will not only minimize conductor cracking, foil lifting, and electrical breakdown but also facilitate solder distribution. The generally preferred and nonpreferred printed wiring conductor shapes are illustrated in Fig. 1.12.

32. Width and Thickness The width of any conductor is a function of the current carried and the maximum allowable heat rise due to resistance. The conductor width should be as generous as possible to tolerate the normal amount of undercutting caused by etching and the manufacturing process, as well as nicks and scratches in the artwork caused by careless handling. The amount of undercutting or line reduction is equal to twice the thickness of the conductor.

The current-carrying capacity of etched-copper conductors for rigid boards is given in Figs. 1.13 to 1.15. For 1- and 2-oz conductors, allow a nominal 10 percent derating (on a current basis) to provide for normal variations in etching methods, copper thickness, and thermal differences. Other common derating factors are 15 percent for continuously coated boards (for base material under 0.032 in and copper over 3 oz) and 30 percent for dip-soldered boards.

33. Spacing Minimum conductor spacing must be determined to preclude voltage breakdown or flashover between adjacent conductors. The spacing is variable and depends on several factors:

1. Peak voltage difference between adjacent conductors
2. Atmospheric pressure (maximum service altitude)
3. Use of a coating
4. Capacitive coupling parameters

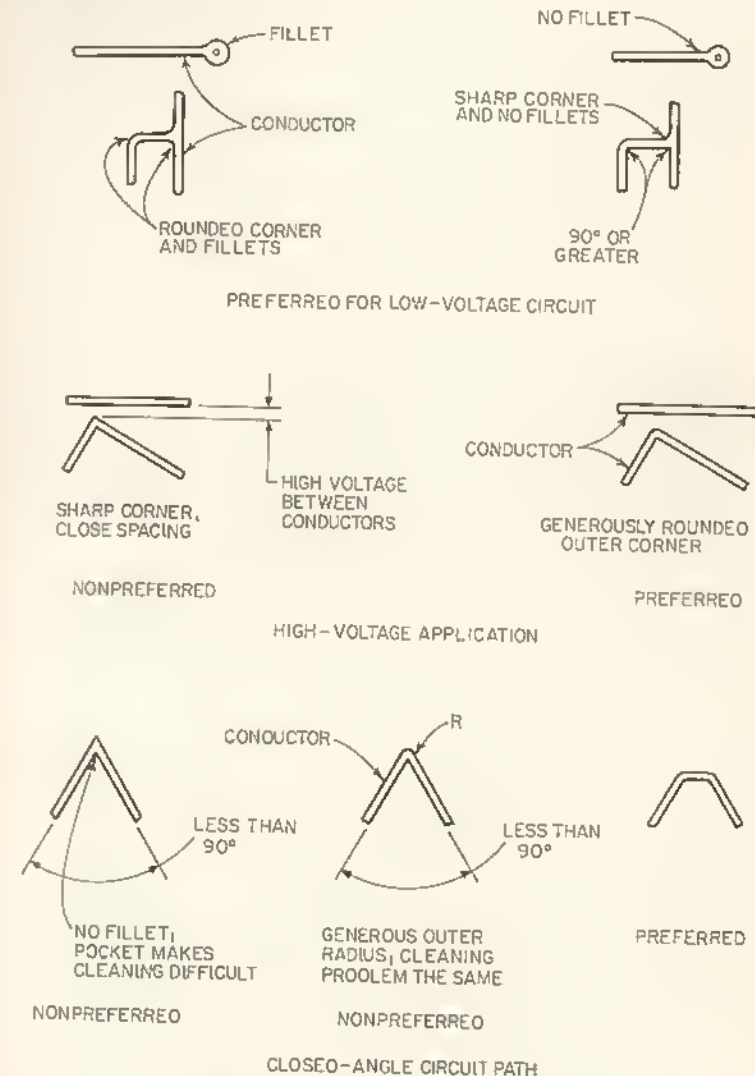


Fig. 1.12 Conductor shapes.

The logician, or circuit designer, may place on board layout certain restrictions which must be heeded. For instance, critical-impedance or high-frequency components are usually placed very close together to reduce critical stage delay. Transformers and inductive elements should be isolated to prevent coupling, and inductive signal paths

should cross at nominally right angles. Components which may produce any electrical noise from movement of magnetic fields should be isolated or rigidly mounted to prevent excessive vibration.

Minimum conductor spacings for various voltages and altitudes for both coated and uncoated boards are shown in Table 1.4.

TABLE 1.4 Spacing of Conductors for Various Voltages

Uncoated boards sea level to 10,000 ft		Uncoated boards over 10,000 ft and coated boards		
Voltage between conductors, dc or ac peak, V	Minimum space, in	Voltage between conductors, dc or ac peak, V	Minimum space, in	
			Above 10,000 ft uncoated	Conformally coated, all altitudes
0-150	0.025	0-50	0.025	0.010
151-300	0.050	51-100	0.060	0.015
301-500	0.100	101-170	0.125	0.020
Above 500	0.0002 (in/V)	171-250	0.250	0.030
		251-500	0.500	0.060
		Above 500	0.0001 (in/V)	0.0012 (in/V)

34. Conductor Pattern Checklist The following checklist pertaining to conductive patterns should be used to help assure that the printed wiring design is of sufficient quality. (An affirmative answer to each question is desirable.)

1. Are conductors as short and direct as possible without compromise of function?
2. Are conductor width restrictions adhered to?
3. Has necessary minimum spacing been maintained between conductors, between conductors and mounting holes, between conductors and terminal areas, etc.?
4. Is close paralleling of all conductors, including component leads, avoided?
5. Are sharp angles, 90° or less, avoided in the conductor pattern?
6. Have all large conductive areas been properly relieved?
7. Is the circuit pattern geometrically arranged to avoid solder bridging?
8. Are component derating practices followed?
9. Has a worst-case analysis been run on the design?
10. Has proper spacing been maintained between conductive patterns and terminal areas and component and board mounting holes?

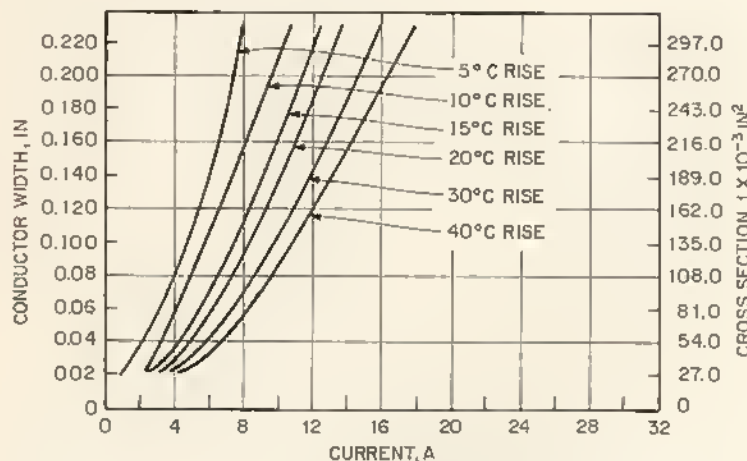


Fig. 1.13 Temperature rise vs. current for 1-oz copper.

HOLES

35. Hole Fabrication Tolerance capability for hole formation is dependent upon the way in which the holes are made. Although holes may be punched in some rigid materials (especially paper-phenolic punching grades) and most flexible materials with a positional accuracy of ±0.003 in, the technique is limited to single-sided wiring only. For double-sided and multilayer boards, holes are drilled in a variety of ways previously described. Tolerances for various methods of drilling holes are as follows:

Method	Tolerance, in
Drilling by eye	±0.010
Fig-horned drill plate	±0.005
Drilling with optical aid	±0.005
Tape-controlled drill, eight spindles	±0.003
Tape-controlled drill, one spindle	±0.001

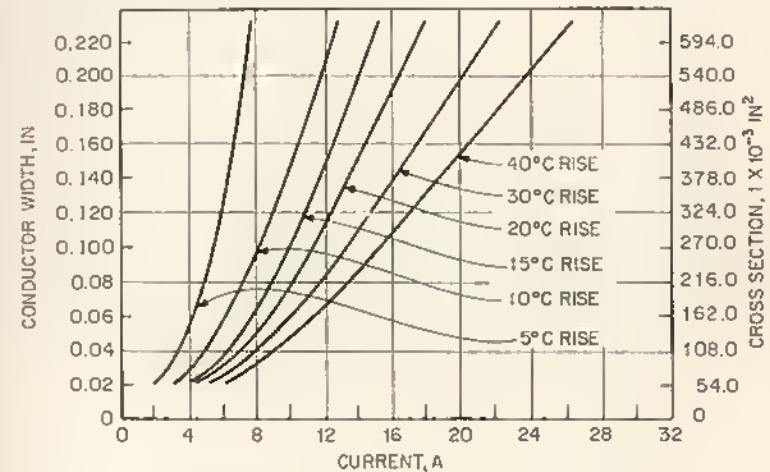


Fig. 1.14 Temperature rise vs. current for 2-oz copper.

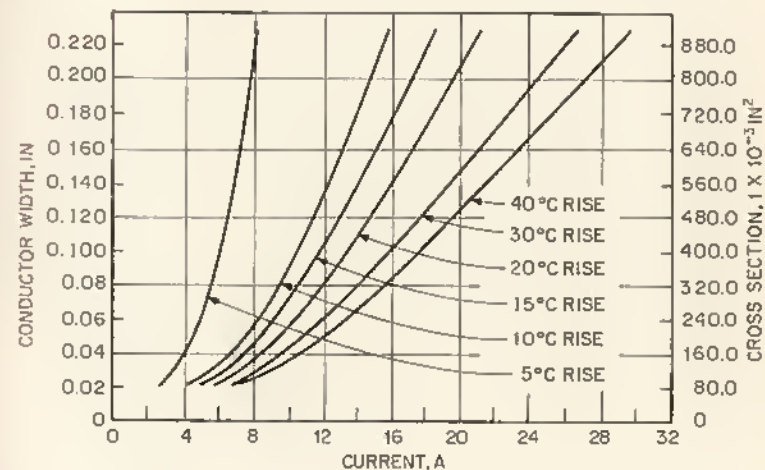


Fig. 1.15 Temperature rise vs. current for 3-oz copper.

Hole size: In single-sided plans should be large enough for accommodation of the stitch, wire, or component lead which is to be inserted. For plated holes, a good soldering relation results when the hole is 0.006 in larger than the lead it is to accept. For plated-through holes, where no lead is used, there is no such restriction, but another limitation exists. At the present time, practical manufacturing techniques can produce reliable plated-through holes with diameters down to one-fourth of the overall board thickness.

36. Through-Hole Connections There are several methods of making through-hole connections, including the fused-in-place flat-flanged eyelet, the funnel-flanged eyelet, the z wire, and the plated-through-hole connection.

Screw terminals are not generally recommended as a means of providing a through connection because of problems encountered with hairline cracks and the cold solder joints that are due to the multiple soldering steps involved.

Of the types of through connections cited, the plated-through hole has much to recommend it from a design standpoint. Terminal areas smaller than those required with eyelets can frequently be used, and all through connections can be made simultaneously—a major advantage when large numbers of such connections are needed. The method is applicable to most types of insulating substrates; hole size control is not critical; and the connection behaves well under shock and vibration. On the distal side, the connection is less strong than an eyelet, is not as repairable, and must be very carefully inspected to be sure that it has been properly made.

The funnel-flanged eyelet also offers a number of design advantages that include easy component installation and ease of inspection, since the solder fillet is open to view. Its disadvantages include the need for greater vertical board clearance, the need for high insertion pressure, and the small amount of direct conductor contact present. (There is a greater dependence on the solder for the electrical contact to the terminal area.)

The fused-in-place flat-flanged eyelet offers the advantages of insertion and fusing in one step under closely controllable conditions. There is less chance of intermittent open circuits through use of the annealed copper eyelet, and there is greater direct contact with the terminal area. As with the funnel-flanged eyelet, the possibility of chemical contamination is absent. Drawbacks include the need for larger terminal areas, properly pretinned eyelets free of oxides, proper tin or solder coating thickness on the board, and maintenance of the electrodes and other fusing variables.

ELECTRICAL DESIGN FACTORS

This section concerns a few of the important electrical circuit factors that must be accounted for in proper printed board design. The information provided here is by no means all-inclusive. Also, the values and equations given should be used only as guides for estimating approximate values of a given characteristic. All electrical parameters of a board are highly interdependent, and empirically measured values will, therefore, rarely coincide exactly with calculated values. That is especially true of high-frequency regions, where it is impossible to make a complete analytic prediction of board characteristics. It should also be pointed out that data given in this section are simplified for purposes of compactness and that many contributing factors have been eliminated.

37. Resistance Current-carrying capacity of conductors is not usually a problem, but the ohmic resistance may be a problem when conductive paths are unusually long or voltage regulation is critical. An approximate calculation of conductor resistance and temperature rise can be made from Fig. 1.16. The resistance and temperature rise may also be calculated from the expression

$$R = 0.000227W$$

where *R* is the resistance per linear inch, in ohms, and *W* is the width of the line, in inches, based on 99.5 percent minimum-purity copper 0.0027 in thick (in 2-oz. copper).

38. Capacitance Capacitance may be of considerable importance, especially at high frequencies. The distributed capacitance between conductors located one above another

must be accounted for when high-frequency circuits are involved; it is of the order of 1 pF/in. The basic capacitance equation serves as a useful, if approximate, guide:

$$\text{Capacitance (pF/in)} = \frac{\text{conductor width (10}^{-3} \text{ in)} \times \text{dielectric constant}}{\text{dielectric separation (10}^{-3} \text{ in)} \times 4.15}$$

When conductor width is at least 10 times greater than the dielectric separation, the equation is generally in close agreement with empirical values obtained but may lie on the low side. The capacitance coupling between conductors can be minimized by limiting the lengths of conductors running in the same vertical plane. Capacitance between

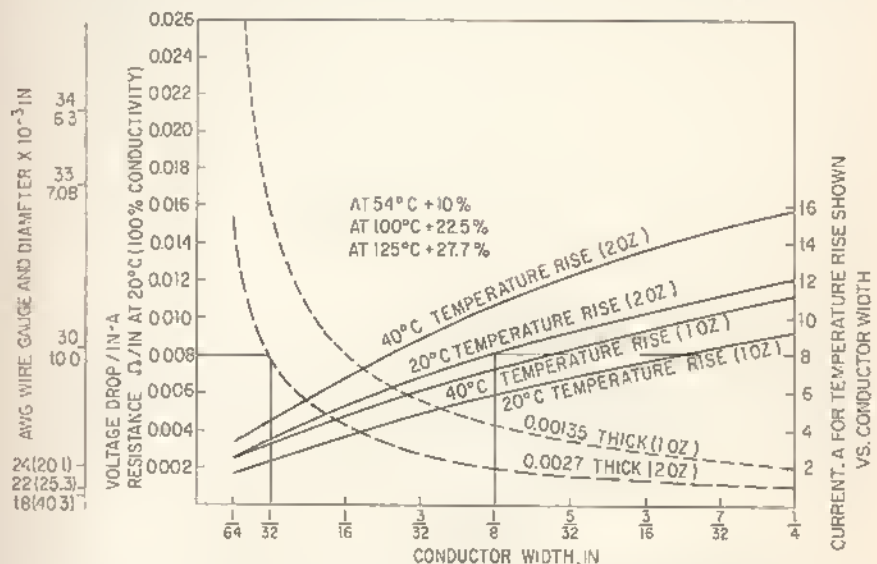


Fig. 1.16 Conductor width, copper conductor resistance, voltage drop and temperature rise conversion chart, rectangular printed wiring vs. round wire. A.C.S. resistance is 0.67879 μΩ/in² and 0.67879/0.00135 = 503 μΩ/in² (1 oz).

adjacent conductors is a function of conductor width, thickness, and spacing, as well as the board material itself. It can be calculated from the expression

$$c = 0.31 \frac{a}{b} + 0.23(1 + k) \log_{10} \left(1 + \frac{2b}{d} + 2h + \frac{b^2}{d^2} \right)$$

- where *k* = material dielectric constant
- a* = conductor thickness, in
- b* = width of conductors, in
- d* = distance between conductors, in

Special attention must be given to circuits located over a shield or ground plane, since the entire length of the conductor is capacitance-coupled to the plane and consequently to similar conductors of similar relations.

It has been shown that, for critical high-frequency circuitry, the electrical characteristics of single-sided circuits made of 1/16-in epoxy-glass or paper are not adequate and that associated ground-plane micro-strip-line construction is a must. The electrical characteristics can also be used as a process control; they give a good indication of process variables.

39. Characteristic Impedance The characteristic impedance of parallel conductors is determined as follows:

$$Z_0 = \frac{R + j\omega L}{G + j\omega C}$$

where $Z =$ characteristic impedance, the apparent impedance of an infinitely long line, ohms

$R =$ resistance per unit length of line, ohms

$L =$ inductance per unit length of line, henrys

$G =$ conductance per unit length of line, mhos

$C =$ capacitance per unit length of line, farads

The method of achieving transmission-line capability with double-sided printed wiring is called "microstrip"; see Fig. 1.17. A specific formula for microstrip is the following:

$$Z_0 = \frac{h}{W} \frac{377}{\epsilon_r}$$

where h and W are given in the diagram and ϵ_r is the effective dielectric constant of the material (considering the effect of air). This analytical method disregards fringing effects and leakage flux and gives validity to the notion that more reliable values may be obtained by measurement. To account for the fringing effects of microstrip transmission line, and if the analytical method must be used, the following formula is recommended:

$$Z_0 = \frac{h}{W} \frac{377}{\sqrt{\epsilon_r \{1 + (2h/\pi W)[1 + \ln(\pi W/h)]\}}}$$

Typical values of the impedance of microstrip in epoxy-glass material when the conductor is exposed to air are given in Fig. 1.18.

ENVIRONMENTAL FACTORS

Printed wiring may normally be expected to exhibit the same properties as the base materials, i.e., laminate and cladding, from which it is constructed. Environmental requirements, therefore, relate back to those elements. An exception is the case of monolithic assemblies of printed boards, which are, by nature of the lamination and interconnection procedure, additionally affected by the environment.

In addition to serving as an electrical connection medium, printed wiring planes provide mechanical support for the active or passive components they are interconnecting. In that way they become an integral part of the package, or assembly, and must therefore be able to withstand the environmental stresses associated with the entire structure.

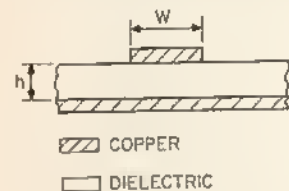


Fig. 1.17 Microstrip transmission line.

40. Shock and Vibration Vibration, flexing, and bowing can become a problem on larger boards. When the board is normally mounted, the board connector will keep one edge of it held firmly, and the two sides will be supported to some extent by the board guides. That will leave only one edge free to bow either because of stresses from manufacturing or under the influence of vibration. A handle along the free edge will help to eliminate those effects and will leave the center of the board as the area most likely to be affected. The problems of minimizing the effects of vibration and warping are exactly the same as those met in any other form of engineering, and similar solutions can be used. A stiffening web, which can be a piece of metal angle, a plastic extrusion, a "top hat" section, or a solid bar, can be fitted across the middle of a large board. Alternatively, smaller pieces of suitable material can be riveted or bonded to the board to change the major resonant frequency.

One of the greatest dangers of bowing or vibration is that components with electrically live cases will be brought into contact with the soldered joints on the back of the next board. Stiffening or the application of damping strips may not be enough to prevent natural warping of a large board from introducing a dangerous bow. If there is any risk of bowing, the board can be fitted with spacers higher than any of the packages on it—as many as may be required being fitted at suitable places. The curves of Figs. 1.19 to 1.21 are a guide to the number of support intervals and level of complexity required for different applications.

Figure 1.19 illustrates the relation between the resonant frequency and length of a beam. The curve is for a beam of modulus 1.8×10^6 psi, thickness 0.62 in, and 0.016 psi.

For example, a cantilever $1\frac{1}{2}$ in long would have a resonant frequency of about 260 Hz, and a cantilever 3 in long would have a resonant frequency of about 65 Hz. The illustration also shows the resonant frequency of a beam fixed at both ends to be about $6\frac{1}{2}$ times the resonant frequency when the beam is used as a cantilever.

Figure 1.20 shows that the resonant frequency of a beam may vary over a range of 3:1 depending on the choice of common materials. The illustration also shows that the same beam fixed at both ends has a resonant frequency about $6\frac{1}{2}$ times higher than when used as a cantilever. The curve is for the instance in which the length is 1.8 in, thickness is $\frac{1}{16}$ in, 0.016 psi. If the board is made from phenolic XXXP, the resonant frequency will be about 100 Hz; if from G-10 epoxy, it will be about 240 Hz; and if from nonwoven continuous glass-filament-base epoxy, it will be on up to about 300 Hz.

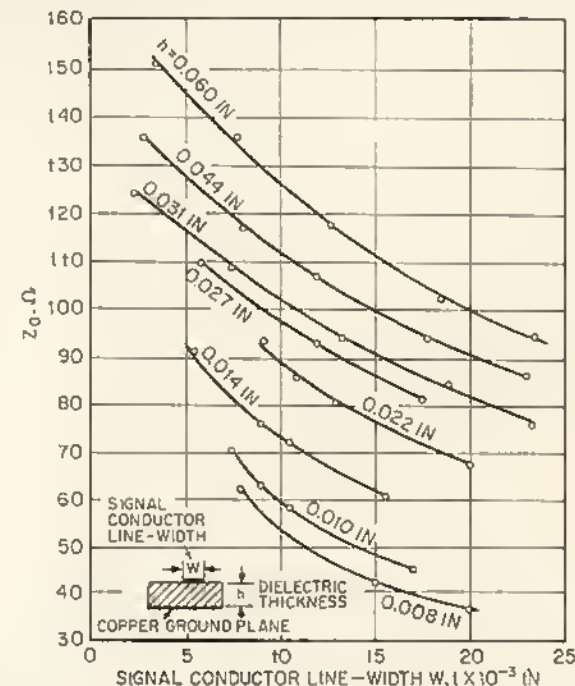


Fig. 1.18 Characteristic impedance vs. line width and dielectric thickness: G-10 epoxy-glass dielectric-microstrip transmission line. Conductors are 2-oz copper. Measurements made at room temperature (26°C) and room humidity. Board material dielectric constant relatively stable with frequency, varying from about 6 at 1 kHz to about 5 at 25 MHz.

Figure 1.21 illustrates the relation between the resonant frequency and the amounts of sag or deflection under the influence of gravity. For example, if the sag is 0.01 in, the resonant frequency will be about 31 Hz. The derivation is simple and is included for those who may be interested. First, the formula for resonant frequency is

$$f_r = \frac{1}{2\pi} \sqrt{\frac{K}{M}}$$

where K is the spring constant and M is the mass. From Newton's laws, $F = Ma$, where a is the acceleration. In this case, $a = g$, the gravitational constant. The relation of the weight to the mass is $W = Mg$. The relation between the force applied to the spring and the deflection of the spring is $F = Kx$, where x is the deflection. In our case the force is equal to the weight.

$$K = \frac{F}{x} = \frac{W}{x}$$

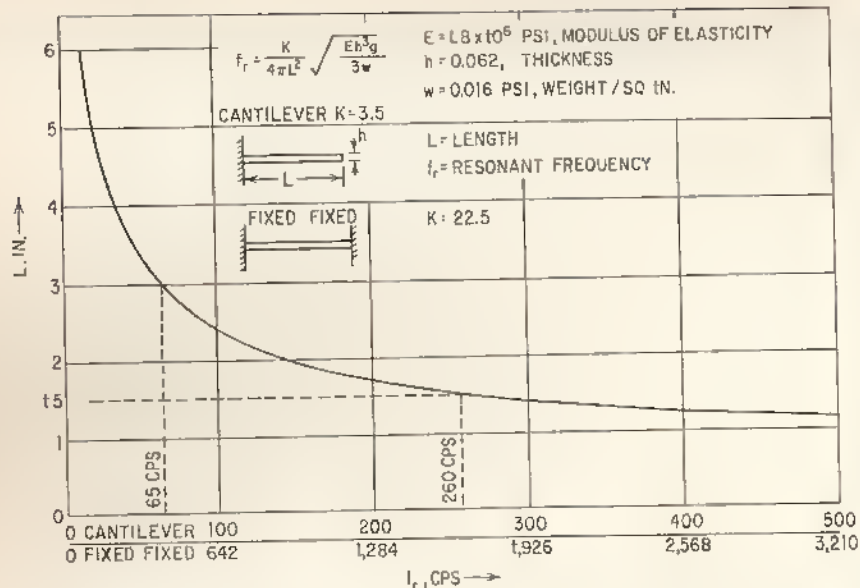


Fig. 1.19 Relation between resonant frequency and length of a beam. $f_r = \frac{K}{4\pi L^2} \sqrt{\frac{Eh^3g}{3w}}$
 $E = 1.8 \times 10^6$ psi, modulus of elasticity; $h = 0.062$, thickness; $w = 0.016$ psi, weight/in²; $L =$ length; $f_r =$ resonant frequency; $K = 22.5$.

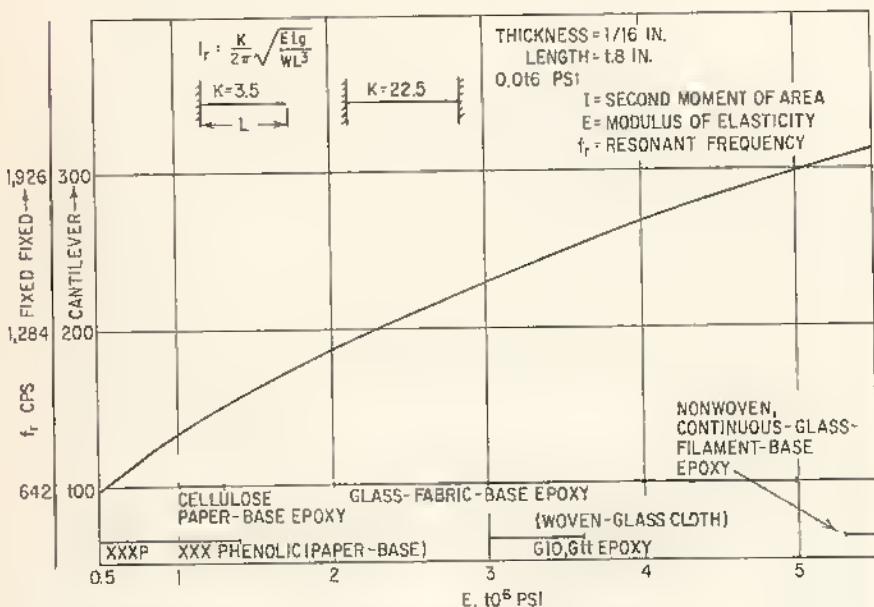


Fig. 1.20 Resonant frequency as a function of E . Thickness = 1/16 in.; length = 18 in.; 0.016 psi. $I =$ second moment of area; $E =$ modulus of elasticity; $f_r =$ resonant frequency.

Substituting for K/M in the formula for resonance gives us the result:

$$\frac{K}{M} = \frac{g}{W} \frac{W}{x} = \frac{g}{x}$$

$$f = \frac{1}{2\pi} \sqrt{\frac{g}{sag}}$$

When a typical flat-card circuit board is mounted, the method most generally employed is either the cantilever or clamp-clamp method.

The level of stress to which the board and its components will be subjected is in principle completely determined by the forces and moments imposed on the surfaces and

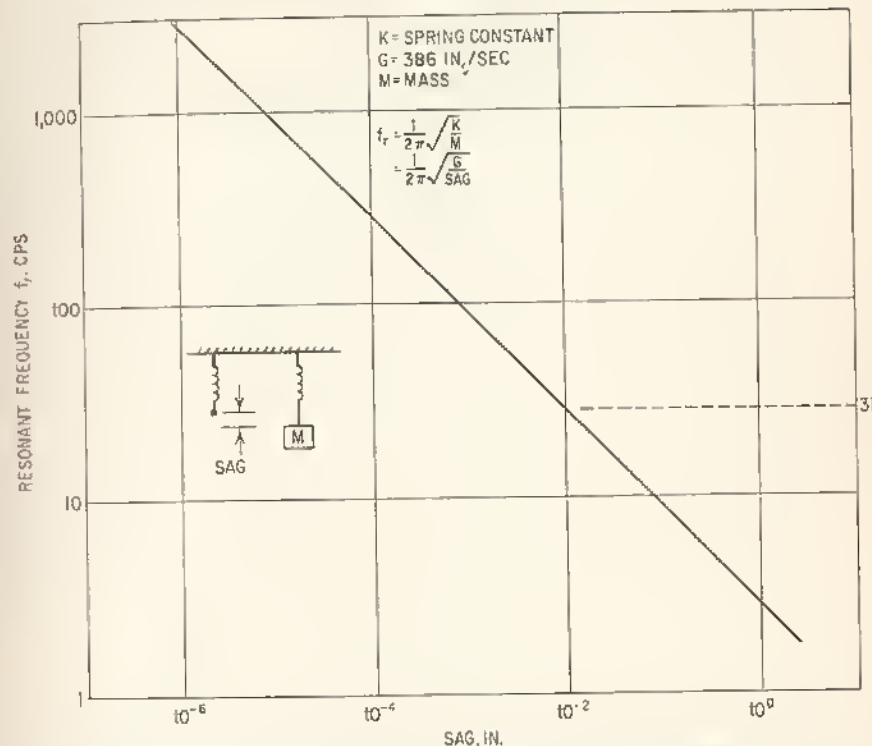


Fig. 1.21 Relation of sag to resonant frequency. $K =$ spring constant; $G = 386$ in/s; $m =$ mass.

by total forces such as those induced by gravity or centrifugal effects. It is vital to know the stress levels for given loads and their effects on the boards and components in terms of strains, deformations, deflections, and displacements. The included graphs will assist in assuring that the stress levels are within allowable limits. Safety factors should be chosen with respect to application and proved by subsequent laboratory and field tests.

In the selection of components to be mounted on a printed board, consideration should be given to the resonant frequency of the board and comparison should then be made with the chart of Fig. 1.22. The acceleration level will then be established for consideration of the amplitude of vibration at the point where the component will be mounted. That energy level should then be referred to the component specifications.

For example, by reference to the dashed line of Fig. 1.22, if a component is vibrating at 0.75 in (total amplitude) at a frequency of 5 Hz, the vector velocity is found to be 11.7 in/s and the acceleration vector 0.05 g 's or 30 R/s².

41. Thermal In facing the problem of heat removal, the printed wiring designer should take advantage of the available conductive cooling, employ high-temperature components

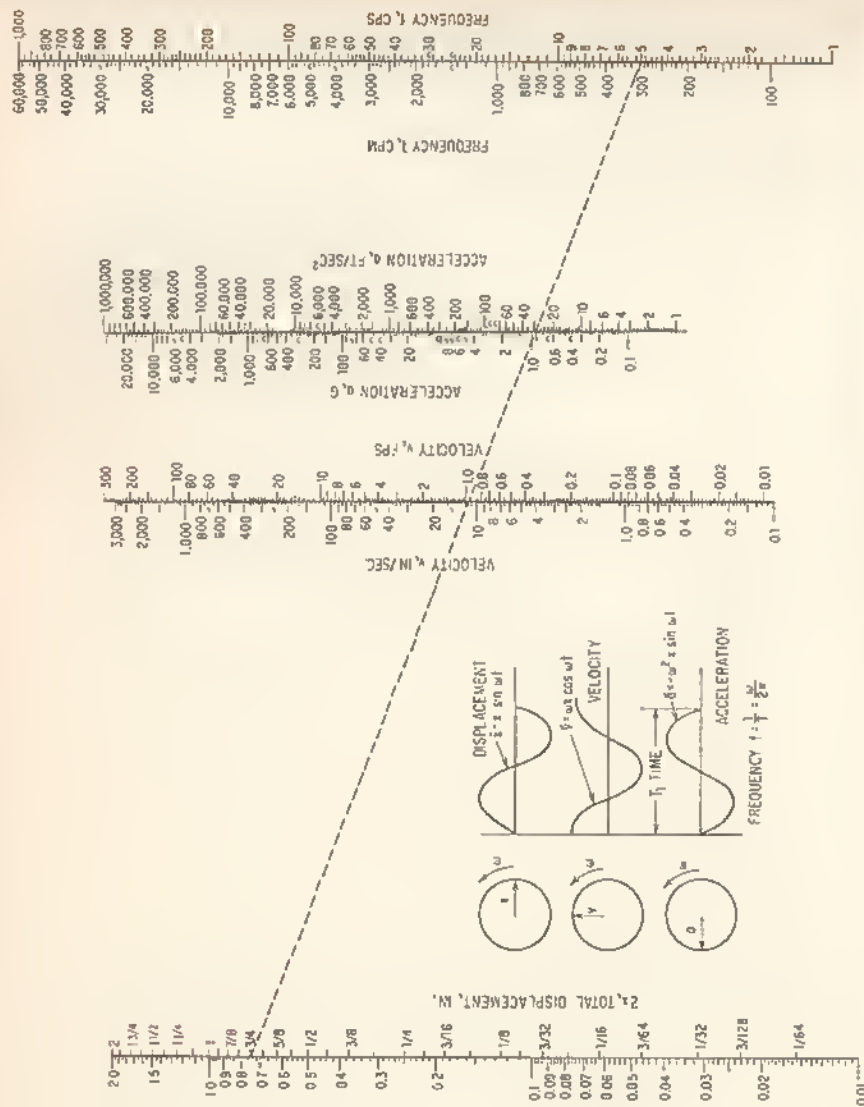


Fig. 1-22 Vibration nomograph.

where possible, and ensure thermal isolation of temperature-sensitive components from high-heat-emitting sources.

All three principal modes of heat transfer—conduction, convection, and radiation—find use in cooling electronic packages.

a. Conduction. Enhancement of heat transfer by conduction may be brought about by the use of (1) materials with high thermal conductivities, (2) direct paths to large heat sinks, and (3) good thermal bonding or coupling between parts involved in the conduction path.

The use of direct large-cross-sectional paths to heat sinks is particularly important in conductive cooling. The shorter the path to the heat sink and the fewer the number of thermal bonds or joints present the better. The printed conductor itself must be considered in the thermal path and should be as large as possible to help effect heat transfer. The contacting surfaces should be large and not subject to oxidation.

b. Convection. Convective cooling can be enhanced by (1) increasing the velocity of the fluid substance over the surfaces to be cooled, (2) increasing the surface area available for heat transfer, and (3) replacing laminar flow with turbulent flow, which increases the heat transfer coefficient and ensures good scrubbing or wiping action around the parts to be cooled.

c. Radiation. Radiative heat transfer can be enhanced by (1) using materials with high emissivity and absorptivity, (2) raising the temperature of the radiating body and/or lowering the temperature of the absorbing body, and (3) arranging the geometry to minimize back reflection to the radiating body itself.

Power transistors and high-wattage resistors are among the components which must be given special consideration to eliminate local hot spots that can damage adjacent components or the board itself. In general, hot components should be mounted close to the frame, which serves as the heat sink, in order to shorten the thermal path. Temperature-sensitive components should be kept as far away as possible from those high-temperature areas. Heat-resistant components should be placed in the hottest part of the package—either in the center or closest to the hot components.

Some typical component-mounting arrangements for controlling heat transfer on printed boards are shown in Fig. 1-23. In estimating the heat dissipation in an electronic package, one must analyze the circuit to obtain the maximum power dissipation for each component and then determine the maximum operating surface temperature to be expected. With that information known, the design must then be arranged, by using the techniques already discussed, to maintain the components below their maximum operating temperature under the worst conditions.

The maximum temperature allowable in the package must be considered in light of the associated insulation present as well as the components themselves.

Some idea of the amount of heat that can be effectively removed from a sealed unit and from an external surface by various types of cooling is given in Table 1.5 for sealed units and external surfaces.

42. Protective Coating It is desirable, and often mandatory, to provide protection for printed wiring assemblies against dust, dirt, contamination, humidity, salt spray, and

TABLE 1.5 Effective Heat Dissipation as a Function of Type of Cooling

Type of cooling	Heat removed, W/in ²
Sealed unit	
Free convection, air	0.15
Metallic conduction	2-5
Forced convection, air	To 7
Direct liquid cooling	To 10
Vaporization cooling	To 20
External surface, based on a 40°C temperature rise	
Free convection, air	0.25-0.50
Free convection plus radiation	To 2
Direct liquid cooling	To 4
Vaporization cooling	To 7

mechanical abuse environments. There are many insulating (conformal coating) compounds that can be applied; predominant among them are epoxies, polyurethanes, silicones, acrylics, polystyrenes, and varnishes. These compounds are similar to encapsulating materials of the same chemical type, but frequently they are used in solution form. The number of variations and combinations of the materials is very large. For very thin conformal coatings (0.0006 in thick and less) vacuum-deposited paraxylene is often used. The most widely used coatings for military applications are in accordance with MIL-I-46058.

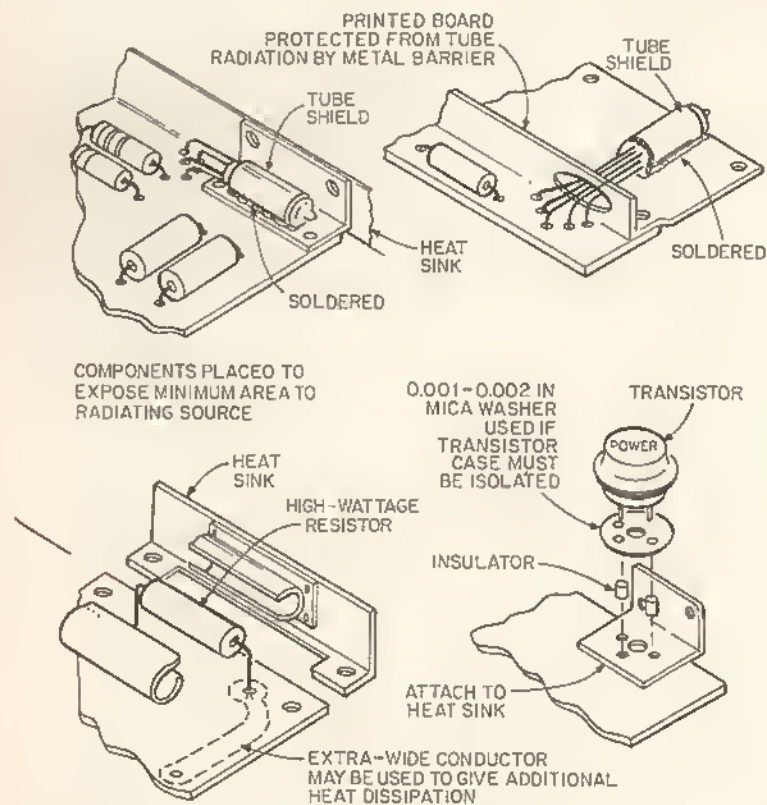


Fig. 1.23 Component-mounting arrangements for facilitating heat transfer in printed board assemblies.

Some of the more important technical considerations involved in selecting a printed wiring assembly coating are:

1. Its ability to prevent the occurrence and migration of circuit, component, and current-carrying corrosion products
2. The protection it affords to the printed wiring insulation
3. The effect of humidity and temperature on insulation resistance
4. The effect of its thickness and the temperature and humidity on important electrical properties such as dissipation factor, dielectric constant, and Q
5. Its flexibility
6. Its resistance to cracking during thermal shock
7. Ease of its application and processing
8. Its ability to be modified in order to control its thickness
9. Its optical clarity to allow for the viewing of board and component markings
10. The ease by which it is removed prior to repairing the printed wiring assembly

LAYOUT

The printed wiring layout is defined as being a sketch that depicts the printed wiring substrate, the physical size and location of all electronic and mechanical components, and the routing of conductors that serve to interconnect the electronic parts. The layout is usually prepared in sufficient detail to permit the generation of documentation and artwork.

The engineer should finish the draft in a well-drawn schematic with as few crossovers and jumps as possible. He should bring in light the areas of coupling incompatibility and those where short circuit paths and isolation of one circuit from another are required.

The position and quantity of the all-board connections to be used to interface with the interconnection system wiring play a very important part in determining where the inputs, outputs, and grounds will be. The board contacts will greatly affect the total order of the final layout. They should be designated alphabetically in the schematic to help guarantee that they will be clearly understood and any necessary fixed contact assignments should be noted. Shields and grounds also must be clearly defined for the layout personnel, especially if they are used for reducing undesired noise or interference from couplings.

43. Layout Procedure Layouts are usually preferred to be as viewed from the component side. Two-sided designs may be represented on a single sheet by a coding system on by another sheet representing the individual sides of the design. The latter method can cause errors and is not the one preferred. Multilayer conductor layouts also are laid out singularly in this manner, as are ground and voltage planes.

If a standard component matrix can be achieved, the preprinted layout format can be made in advance. That is true of the voltage and ground layers as well when those layers are required for the design.

Proper fabrication indexing hole locations are necessary for each layout; two or three indexing holes are desirable. It is preferred that the holes be located in the x and/or y datum plane.

a. Input Data. The normal input data for manual layout design is a logic symbol schematic. That is a document depicting the electronic circuit in symbolic form by utilizing both discrete and logic formats.

Schematic circuit flow is indicated going from left to right on the diagram. Terminals, components, and connector contacts are labeled, as are all input-output terminal designations. Associated with the data input is a list of components and specified critical circuit design holes.

The critical circuit design rules encompass such electrical areas as capacitance coupling, feedback, current, and clock signal grouping. They are the circuit design engineer's responsibility and should be clearly defined in his circuit design rules before the layout designer prepares the design layout.

An alternative input may be a line diagram combined with proposition and logic listings associated with the electrical circuitry design.

b. Development. The sequence of events in the development of a printed board design layout may vary, but the following are the general steps to be taken in the layout development after the input data have been received.

1. An initial evaluation is made of the schematic input, parts list, and special circuit rules. It includes comparison of the data with the physical limits of the board. Physical limitations is concerned with the specific usable board area available.
2. Components are reviewed as to their circuit relation, the necessity of implementing short conductor interconnections, heat sinks, ground and voltage connections, and special conductor width requirements.
3. Signal input and output connector interconnections are then reviewed to assimilate the logic flow organization as it is related to the individual design interface.
4. Specific company standards concerning automatic component insertion requirements, component matrix location parameters, mechanical hardware, and connector usage are then considered as they have impact on the design.
5. Since most of the above process is mental, the actual "hands on" procedure for the manual design layout continues to be carried out by the designer.
6. An underlay of printed-grid polyester base material, utilized as the background for the design layout, is attached to the designer's working surface.

6. Standard board sizes permit the use of a preprinted polyester base material (Mylar, Cronaflex) of the master printed board configuration. All specific rules and/or standards become a part of the master outline.

c. Methodology. It is very convenient initially to make a layout by using templates (paper dolls) of the components and/or the components themselves and making repeated freehand point-to-point wiring diagrams until the interrelations of all components and wiring are compatible with good design practice with respect to mutual coupling and inductive and capacitive effects. This process, with quick rearrangements of components and conductors between steps, will evolve more and more refined designs. It is recommended that at least three of the steps be taken quickly, and three or more additional steps of refinement are perhaps worthwhile for the most complicated layout. Each should be considered to be only a stepping-stone to the next one and not in any way a complete entity.

Electrical parameters of the printed wiring must be literally designed into the more critical circuits. In particular, the resistivity and dielectric properties of the insulating base require consideration for the higher frequencies and higher-impedance circuits, especially when stability in environmental extremes is an important design criterion.

Whenever possible, the component layout should permit automatic assembly unless a short production run is anticipated. Similar units should be placed together in rows or groups—bearing in mind the spacing requirements arising from the manufacturer's recommendations, use of modular grid locations, heat dissipation, electrical coupling, and repair considerations. The component placement should, if possible, allow the electrical values and code numbers to be read easily from one direction, and preference should be given to the electrical values. That will not generally be feasible in the case of automatic assembly. Components should be mounted on only one side of the board and be so spaced and located that each component can be removed without removing any other part, unless, of course, the package is to be expendable.

Heavy components should be located over or near supported areas and heat-emitting components near the board sides or heat sink areas. More is said about this in connection with designing for thermal shock and vibration stresses.

The insulator bodies of tube sockets, connectors, and similar parts subjected to insertion and withdrawal forces must be rigidly anchored without relying on the soldered connections for mechanical support. If tube sockets are not used on subminiature tubes and the flexible tube leads are soldered directly to the printed wiring, the tube envelope must be rigidly supported.

d. Orientations. Assembly and inspection of boards are much easier if all packages are mounted in the same way. Cases do arise when it appears that to reverse the orientation on alternate rows of packages will simplify the interconnections or the power distribution, but in most cases a number of minor snags crop up later in the interconnections and the final gain on the finished board is negligible. If package orientation is standard, automatic or semiautomatic placing and reflow soldering of flat packs, or insertion of dual in-line packages (DIPs), is possible without having to reverse the board or reset the machine as would be necessary if alternate rows of packages were inverted. Also, fault finding on large boards is easier if the orientation is standard. The operator knows, without checking, which is pin 1, which pin is ground, and which is power. Finally, mistakes in wiring are far more likely if there are any inversions in orientation.

e. Grid Systems. All component hole locations, mounting holes, and even the outside board configuration should be laid out on a modular grid system. The basic modular units of location are based on a 0.100-, 0.050-, or 0.025-in system applied along both the *x* and *y* axes. Unless dictated by a particular fixed design parameter, there are few acceptable reasons for a distance between mounting holes on the board length or width to have a dimension of, say, 2.093 in when 2.100 in would suffice or 4.120 in instead of 4.125 in. Manipulation of tolerances can normally adjust any dimension to the nearest 0.025 in and satisfy the grid requirements. Should a high production run be anticipated, numerically controlled program drilling of printed boards can now become a reality based on the 0.025-in rectangular coordinate system.

f. Conductor Routing. The same principles that make a good layout for point-to-point conventional wiring apply to printed wiring. For instance, to guard against undesirable feedback requires either adequate separation or shielding between input and output. Though the principles are the same, however, the dimensions are different. There is

virtually only one plane, and it requires more interconnect planning because normal hookup wire is insulated and may be routed in intersecting paths without concern for contact. Printed circuits, on the other hand, establish a connection at each junction. To overcome that disadvantage, the routing of each conductor must be thoughtfully planned. As described above, that sometimes means some leads may be longer than desired, but with extra thought a jumper or crossover can be used to great advantage.

Conductor routes should first utilize the space available. That, as an example, is where the problem illustrated in Fig. 1.24 exists. It should be noted that the route picked takes precisely the geometric mean with respect to such hazards as

1. Insulation resistance (leakage)
2. High voltage
3. High noise
4. Undesirable feedback or coupling due to electrostatic or electromagnetic coupling
5. Signal losses due to capacitance effects

If all things are considered equal, the conductor route should split the difference at every turn and have equal spacing over parallel runs as shown in Fig. 1.25.

g. Marking and Identification. Almost all printed circuits require the use of nomenclature on the boards to identify components, circuits, test points, and the board itself. The nomenclature may be applied after the board is etched, or it may be etched along with the circuit. If it is applied later, it is usually screen process printed or stamped. In either case, the marking material must be carefully chosen for its electrical and chemical properties. For that reason and for reason of the extra processing steps involved, designers usually include nomenclature in their artwork and etch it along with the circuit. Although such nomenclature is conductive, it does not normally present a problem. The nomenclature should be as small as it can be to survive etching and still be legible. Thickness of the copper and the location of the nomenclature relative to conductors will determine how much deterioration the nomenclature will receive during etching. Very small nomenclature should not be located in open areas when 2- or 3-oz copper is being used.

To facilitate testing and servicing a printed board assembly—and to link the assembly visual aid to documentation—component reference designation marking should be

considered. In addition to reference designations, polarities of capacitors and diodes should be indicated as close to their component mounting holes as possible. Reference designations and polarities should be completely visible with the components in place. On a dual in-line integrated circuit, pin 1 should always be indicated, as should be the tab for a 6-, 8-, 10-, or 12-lead TO-5-type integrated circuit.

Printed boards should carry two distinct identifications. Since the fabricator machines and plates the printed board before the electrical components are installed, he needs the nomenclature and a method of identifying the board. The identification is the printed board fabrication drawing number (master drawing). The printed board assembly number should be located on the component side where assembly of electrical components takes place, from which orders for replacement of spare parts are made, and where servicing and testing is normally done. The number should always be preceded by "ASSY NO." In addition, the serial number of the assembly is necessary for tracing purposes. If space permits, display your company logo on the board. All these features are easily applied by the etched-foil method.

h. Summary. When printed wiring assemblies are being designed, the following basic concepts should be considered:

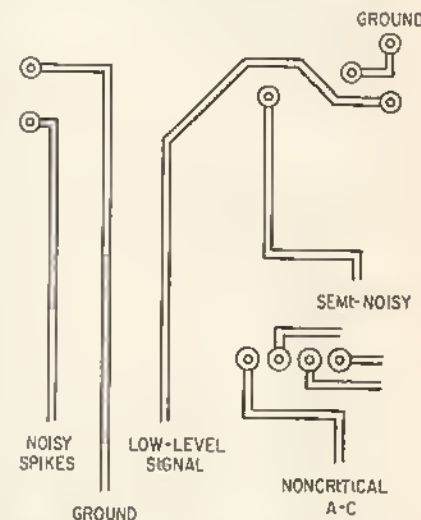


Fig. 1.24 Conductor routes utilize space availability.

1. Specific contractual requirements.
2. Selection of standard parts, materials, and processes according to MIL-STD or commercial standards, in that order of preference.
3. Design that will accommodate the use of standard tools.
4. Safety factors for the equipment.
5. Accessibility of parts.
6. Minimum need for adjustment in operating the equipment.
7. Provision for simple maintenance.
8. Selection of parts resistant to damage by the heat of soldering. Parts to be mounted must not only meet the storage and operating conditions specified for the equipment but also maintain their integrity after having been subjected (during assembly) to heat soldering.

Proper mounting design should fulfill the following basic requirements for mounting parts.

1. Ease of maintenance
2. Fasteners easily removable
3. Minimum loose hardware
4. Adequate clearance between printed conductor and mounting brackets
5. Board supported at proper interval
6. Positive board retention

44. Automated Techniques The application of automated techniques to printed wiring design has become more prevalent. A relatively complete automated system for printed

wiring design includes the following set of functions: logic assignment (component, gate, and pin allocation), logic partitioning (grouping logic functions by physical board), component placement, conductor routing (interconnection layout), design checking, and artwork generation. The desirable final step is the generation of control tapes for numerically controlled production equipment. Building the system requires a hierarchy of optimization steps, with intermediate design data being passed to successive higher levels of assembly.

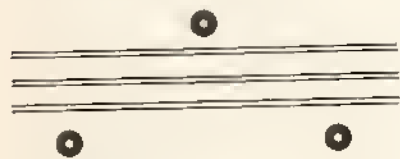


Fig. 1.25 Parallel-run conductor configuration.

When fully implemented, an automated design system will replace nearly all of the manual operations typically required. Key among the manual operations eliminated are component placement, interconnection layout design, and taping and/or digitizing from a rough layout of the board. A number of the widely used board design conventions or "standards" exist simply because they are convenient to a manual design environment and because they impose no restrictions on the human designer. By way of example, the 0.0625-in pad that is in common use provides no particular benefit and seems to derive solely from the 0.250-in "round number" at 4:1 tape-up scale. Another example is the frequent "odd number" 0.156-in spacing of connector pads or fingers. Although these and other conventions may represent acceptable or convenient practice for manual design, they are often awkward or limiting in an automated design environment. A comprehensive CAD system will optimize the pad (terminal area) size to complement the complete system.

a. Grid Pattern. One of the more fundamental considerations in an automated design environment is the use of a grid pattern to define the relative positions of component pins, obstacles, connectors, conductors, etc., on the board. Typical grid sizes are 0.100 in, 0.050 in, and 0.025 in, where the value defines the dimension from grid line to grid line. The physical dimensions of integrated circuit components generally support the latter two sizes, although it is important to note that the components influence rather than control the grid size used.

The minimum practical grid size is typically determined by the desired width of printed circuit conductors and of the air gap clearance between conductors. For example, a 0.020-in conductor and a 0.030-in conductor clearance will result in a 0.050-in grid size. A 0.012-in conductor and a 0.013-in clearance requirement will support a 0.025-in grid

size. Selection of the grid size to be used is influenced by interconnection complexity, pad sizes, and conductor-pad clearances.

It is significant that the use of automated design techniques will often permit the use of a smaller grid size than nonautomated practices will permit. That results from the high precision with which the finished artwork can be produced and the hole drilling process controlled; each provides improved response to manufacturing tolerance requirements. Accordingly, organizations using automated design facilities for the first time should carefully inspect the basis for existing spacing practices. If an approximate 0.100-in grid size is in use for manual work, a 0.050-in size should be strongly considered. If an approximate 0.050-in grid size is in use, a 0.025-in grid size should be considered for the most complex designs.

b. Component Density. Perhaps the single most difficult parameter to generalize upon is the component density (number of components per square inch of board area) that can be supported by automated design techniques. The reason is simple enough: component density is a dependent parameter affected in some degree by all the other factors under review.

Because there are so many variables which bear on component density, any quantitative statement requires a number of qualifying remarks. For example, nominal maximum values of component density are respectively 1.2 and 1.8 for dual in-line and flat packs in two-sided boards and 3.0 and 4.0 for multilayers. The values are stated for a component count of about twenty-five integrated circuits (two-sided board values) and a grid size of 0.050 in. Assuming that preferred design conventions are present (some already reviewed, others to be reviewed later), the values are further sensitive to board size and proportions, grid size, and pad or tab sizes.

Board size and proportions can be expected to have a direct effect on conductor density—the concentration of conductors in certain portions of the board, which in turn affects component density. The concentration of conductors in the center of a board usually increases as the board size increases. Regarding board proportions, a 5 × 6-in board will support a higher component density than a 3 × 10-in board owing to an improved balance of conductor channels for side-to-side communication.

Grid size has a direct bearing on the gross number of points available for conductor routing. A smaller grid size will therefore support a higher component density with its increased pin count. Although a 0.025-in grid size provides four times the number of points as a 0.050-in grid size, increases in component density in the same or nearly the same proportion should never be assumed. Grid size changes affect only the gross point count and not the effective (available) point count.

c. Power and Ground Bus Patterns. In general, power and ground patterns utilized for manual designs are suitable for automated design techniques also. The prior emphasis on the need for close control of factors which tend to reduce the effective conductor routing area suggests, however, that the width of power and ground buses should be carefully examined and held to the smallest possible dimension. Excessively wide buses alone can reduce the number of conductor channels by 10 percent.

There may be a preferred orientation in which the narrower of the buses, if there is one, would best be placed on the board side which provides the greatest number of unobstructed channels flowing toward the connector strip and be oriented parallel with those channels. Board proportions also can influence the preferred orientation.

Designs which utilize power and ground plane(s) construction provide an advantage in that they do not consume conductor routing area. The manufacturing basis for the board may introduce restrictions, however, if the ground and power plane(s) must be prefabricated with a standard feed-through clearance hole pattern.

d. Summary. Not all printed wiring designs will permit all the recommendations to apply all the time. The important consideration is that the recommendations be reviewed for every new product design in order to maximize the number of the preferred conventions which can be used. The absence of any one, or a few, of the conventions does not void the opportunity to utilize automated design techniques.

45. Layout Checklist The following checklist pertaining to board layout may be used as a guide to help assure that the printed wiring design is of sufficient quality. An affirmative answer to each question is desirable.

1. Are all noncritical component mounting hole locations at the grid intersections?
2. Are all component mounting hole spacings correct?
3. Are indexing holes adequately spaced?
4. Do component mounting hole terminal areas contain the proper size hole?
5. Is the diametral clearance of the component mounting hole within acceptable limits?
6. Are all board mounting holes properly indicated on the drawing list and dimensionally located and specified on the board drawing?
7. Has the number of different hole sizes been kept to a minimum?
8. Have all tolerances been considered to establish the proper relation of terminal area to hole?
9. Are terminal area sizes compatible with the projection system on the numerically controlled drafting machine?
10. Are terminal areas elongated along the direction of lead clinch?
11. Is the space required for the printed board optimal?
12. Are the drawing list and schematic compatible?
13. Are the printed side and the component side properly indicated on the drawing list?
14. Has a suitable tolerance been allowed between plug-in and mating assemblies?
15. Are all tolerances complete and reasonable?
16. Are all board dimensions scaled to the basic grid?
17. Do all corners of the board have a specified radius?
18. Have all required test patterns been provided?

46. Artwork Factors The problem is to obtain a high production yield in a product such that a reject is not detectable until the investment is high. To solve the problem, most vendors have instituted a rigid inspection of the incoming artwork, or film, before the actual fabrication is even begun. That, coupled with rigid inspection of individual layers, is their guarantee of good registration. The drilled hole, after all, must be aimed at the center of the termination pad. Accuracy is absolutely essential.

Several techniques commonly have been used to accomplish accurate layer-to-layer registration. "Accurate" is, of course, a word that means different things to different people. Some might consider a master with a misregistration of 0.010 in to be accurate, whereas others might quarrel with anything over 0.005 in. The several techniques described next are presented in order of ascending difficulty; the more accurate techniques are presented last.

The more basic, but least accurate, technique would be the completely taped or completely inked artwork. Inking is seldom used because of the physical difficulty of producing complicated artwork with it. Tape, on the other hand, is more difficult to photograph and commonly requires the preparation of a contact antipositive as the photographic master. Cut-and-peel films produce high-quality photographic masters, but their preparation is more complex and expensive than tape.

In today's complex printed wiring market many companies are using automated equipment for the preparation of the artwork master from the printed wiring layout information.

ARTWORK GENERATION

Artwork is basically a manufacturing tool, although it is often prepared by the user (the designer). It is the most important manufacturing tool used in fabricating printed wiring because it uniquely defines the pattern to be placed on the board. Since it is one of the first steps in the overall process, the final product can only approach its quality and accuracy. Owing to the nature of the production processes used, the artwork must provide for certain process allowances; and thus close coordination with the fabricator is necessary.

47. Methods The method required to produce artwork to the established sophistication is highly dependent on the size (area) and accuracy required. Those two factors are established by the board design and are affected by the method of manufacture. A guide to the

tolerance capability of the two most common methods for producing artwork (i.e., manual and automatic) is given in Table 1.6. Final scale tolerance is then the tolerance at working scale divided by the reduction factor with a practical limit of ± 0.001 in for location and ± 0.0002 in for size. Except for transmission line applications, accuracy of artwork is primarily directed toward location and size of terminal areas (pads).

Most printed circuit layout draftsmen can position terminal areas and wiring on the masters to within 0.005 or 0.010 in with the proper equipment. Therefore, 10:1 layouts are used on artwork requiring very high accuracy and 4:1 or 2:1 layouts are used on all other

TABLE 1.6 Artwork Tolerance Capabilities at Working Scale

Line and pad	Tolerance, in, for application by	
	Manual taping*	Automatic NC machine†
Size tolerance	± 0.005	± 0.0005
Location tolerance	± 0.012	± 0.001

*Application on film material in normal room environments.

†Application on glass material by NC machines. Controlled environment assumed.

artwork. Drafters doing the layout must remember that the scale is applied to all dimensions (i.e., target diameters, line widths, etc.) and not just those relating to placement. Sometimes it is necessary to choose a scale which is compatible with available artwork material.

The automatic artwork generator utilizes a precision *xy* plotter with an accuracy of better than ± 0.001 in over a usable surface of 4 × 5 ft. The accuracy of the precision plotter is such that artwork is usually prepared at a 1:1 scale, which eliminates photo reduction. The manual technique requires the artwork to be produced at several times working scale and relies on the camera reduction to provide the accuracy required.

Input to the precision plotter can be either paper or magnetic tape. The data may be prepared by either a digitizer or computer program. The digitizer method is presently more prevalent, but preparation of computer programs for conductor routing is making considerable progress. Output of the digitizer is either paper tape or punched cards; the punched cards are then converted to magnetic tape with standard computer programs. The output of the computer program is usually magnetic tape, but paper tape also can be obtained.

48. Materials Materials used in artwork are either glass or polyester film. Glass plates have considerably better dimensional stability than polyester film when there are variations in temperature and humidity. Thus, when accurate artwork (NC machine) is required, glass is preferred.

49. Considerations The use of multiple patterns, as shown in Fig. 1.10, enables the fabricator to optimize the many batch-type processes in printed wiring and thus increase production and reduce costs without reducing the need for artwork quality and accuracy.

In addition to the obvious gain in accuracy by generating artwork automatically, by-products such as drill tapes and test tapes for automatically testing the board are developed concurrently with the information required to generate the artwork. As computer-aided design techniques are perfected and utilized, outputs of the programs can be used directly in plating artwork.

Keeping artwork masters extremely clean is as important to the photographic process as clean metal is to photoresist coating. It is vital if photographers are expected to produce quality photo masks at a reasonable cost. If dirty or stained artwork is submitted for photographing, it should be rejected as unusable until it has been cleaned. Otherwise, photo masks made from it will have to be hand-scrubbed to eliminate unwanted pinholes, lines, and ragged image edges. Opauing takes time, is very expensive, and results in low-quality photo masks which produce low-quality printed circuits.

50. Camera Techniques The general principles outlined in the following paragraphs are not intended to tell all about printed circuit photography. Obviously, the printed circuit photographer must be one of the best. He need not be an expert in studio photography or continuous tone work, but he should understand all photographic principles very well. In particular, he must thoroughly understand the use and maintenance of a process camera. He should have complete facilities for making and checking film reductions of artwork, touch-up tables, and a place where he can make screens for screen printing circuit boards.

1. Proper selection and use of photographic materials is a major consideration in photography for printed circuit work. That is particularly true in the case of film bases and photographic emulsions.

2. Camera scales are unfortunately inaccurate for making photo masks to the accuracies frequently required for printed circuits. Each photo mask should be checked for both image size and distortion before it is approved for use. Such measurements may be made by using a micrometer or an optical comparator if close tolerances are specified.

3. All photo images should be inspected for pinholes, scratches, and other faults which are touched out with photo opaque between stages in the photo reduction procedure. Magnifiers such as those used by photo engravers and lithographers are widely employed for this inspection and touch-up step.

DOCUMENTATION

Whether a board is to be a one-off-kind prototype or a high-volume production article, it should have some documentation that exercises the means to an end. Exactly what should be documented and how the documentation should be prepared must be given serious consideration if a quality product is to be produced within a given budget and schedule. A printed wiring drawing package should include at least the following drawings.

1. A *schematic diagram* indicates with graphic symbols the electrical connections and functions of a circuit and thereby enables the user to test, evaluate, and troubleshoot.

2. The *master drawing* contains all the information necessary for the manufacture of the printed board.

3. An *assembly drawing* shows a pictorial representation of the finished, assembled board and lists all electromechanical items contained on the assembly.

4. *Artwork* is an accurately scaled configuration of the printed circuit from which the master pattern is made photographically.

5. *Miscellaneous drawings* are prepared to support either the master or the assembly drawings.

51. How Much Documentation? Too little printed board documentation results in excessive scrappage owing to misinterpretation, productivity decrease that is due to efforts to fill information gaps, and loss of uniform configuration that is due to word-of-mouth manufacturing. Information becomes too dependent on individuals rather than documentation.

Too much documentation results in increased drafting hours and labor costs and decreased manufacturing productivity owing to a time-consuming interpretation of over-emphasized and confusing drawings.

Adequate documentation conveys to the user the basic electromechanical design concept, the type and quantity of parts and materials required, special manufacturing instructions, and up-to-date revisions. The use of adequate documentation offers increased profits by enabling schedule and budget commitments to be met, and the result is satisfied customers who receive quality products on time and at an equitable price.

52. Minimal Drawing Package In a minimal drawing package, the goal is to prepare the information at the lowest possible drafting cost without compromising the design and integrity of the board. Since that type of documentation is primarily used by model shops for quick-reaction prototype development, the drawings should be neat, legible, and informative.

Master drawings may be prepared by a variety of methods. Miscellaneous drawings, such as mechanical details and modifications to purchased or existing parts, should be made in sketch form at an compromise to design integrity.

53. Formal Documentation Formal documentation should begin only when there is a management decision to manufacture and market a product for which only minimal documentation exists. The drawing package must be more comprehensive and presentable because formal documentation contains information pertinent to the function of several manufacturing processes. The drawing package need not be expensive. In fact, the cost of formal documentation is only moderately greater than that of the minimal drawing package, provided the integrity of the minimal drawing package is maintained.

Chapter 2

Laminate Manufacture and Quality Control

CHARLES G. HENNINGSSEN

Vice-President, Insulectro, Mountain View, California

Introduction	2-2
1. Treating the Base Material	2-2
2. Copper Inspection	2-2
3. Laminate Buildup	2-3
4. Laminate Pressing	2-4
5. Laminating Final Inspection, Testing, and Quality Control	2-5
Laminate Evaluation, Specification, and Quality Control	2-5
6. Surface and Appearance	2-6
7. Copper Surface	2-6
8. Color	2-6
9. Silk-Screen Adherence Test	2-7
10. Punchability and Machinability Testing	2-8
11. Peel Strength	2-8
12. Solder Resistance	2-11
Base Laminate Testing	2-12
13. Delamination	2-12
14. Warp and Twist	2-13
15. Flexibility	2-13
16. Dimensional Stability	2-14
17. Degree of Cure	2-15
Insulation Resistance	2-15
18. Parameters and Test Conditions	2-15
19. Specimen Preparation	2-16
20. Grades and Specifications	2-17
21. Material Designations	2-17
22. Conditioning Designations	2-18
23. Mechanical Properties of Laminates	2-20
24. Effects of Environmental Change	2-21

25. Coefficient of Thermal Expansion	2-23
26. Water Absorption	2-23
27. Thermal Conductivity of the Base Material without Copper	2-23
28. Copper-bond Strength Retention	2-23
29. Fungus Resistance	2-23
30. Flammability	2-25
31. Dielectric Strength	2-25
32. Dielectric Breakdown	2-26
33. Dielectric Constant	2-26
34. Dissipation Factor	2-26
35. Current-carrying Capacity as a Function of Ambient Temperature	2-26
Troubleshooting of Laminate Problems	2-27
36. Need for Traceability	2-27
37. Surface Problems	2-29
38. Appearance Problems	2-29
39. Machining Problems	2-30
40. Warp and Twist Problems	2-31
41. Laminate Measuring or Delamination	2-31
42. Bond Strength Problems	2-32
43. Miscellaneous Soldering Problems	2-32
44. Excessive Dimensional Changes	2-33

INTRODUCTION

It is important to understand the fundamentals of manufacture and quality control of high-pressure thermosetting laminates. Some of the problems caused by laminate inconsistencies can be explained by failure to control a portion of laminate manufacture. Laminates are composed of layers of material pressed together under heat and pressure to form a dense consistent electrical insulator. The basic raw materials of copper-clad laminate are paper and glass cloth or glass mat bonded together with one of various resins, mainly phenolic, epoxy, or polyester.

1. Treating the Base Material The first operation in laminate manufacture, called treating, consists of impregnating or coating the base material, either paper or glass, with resin. The resin is then semicured to a point suitable for storage and the final pressing operation. The treating operation is accomplished by using a machine called a treater, or coater, as shown in Fig. 2.1a.

The paper or glass passes through a dip tank of resin, where it is saturated, and then through a set of squeeze rollers and a drying oven (see Fig. 2.1b). The oven is air-circulating or infrared and can be up to 120 ft long. Most of the volatiles in the resin are driven off in the oven, and the resin is partially advanced to what is called a B stage. That semicured material is also known in the trade as prepreg, and, in general, it is dry and tack-free to handle. The treater illustrated in Fig. 2.1 is a vertical treater and runs principally glass fabrics; the horizontal treater in Fig. 2.2 treats paper.

Rigid process control is applied during treating in order to monitor the ratio of resin to base material, the final thickness of the prepreg, and the degree of resin advancement. Beta-ray gauges may compare the raw material with the final semicured product and automatically adjust the squeeze rollers above the dip tank so that the proper ratio of resin to base material is maintained. The degree of advancement of the resin is controlled by the speed of the material through the oven and by the velocity and temperature of the air.

Now the prepreg material is stored in a temperature- and humidity-controlled area until time for the pressing operation. Each roll of prepreg is tagged with the value of each component, test results, and date of processing.

2. Copper Inspection Besides the base material and resin, the other principal ingredient in copper-clad laminates is the copper foil. Today almost all copper is electrodeposited on large rotating stainless-steel drums. Each roll of copper is inspected by the laminator for

surface quality and pinholes, and a sample is taken for trial pressing. The trial pressing will show peel strength and general surface quality. The side of foil to be pressed against the prepreg is treated with an alloy to improve bondability to the base material. The alloy is a proprietary coating, usually of zinc or brass, in a controlled ratio to enhance the chemical bond between the copper and resin or adhesive.

3. Laminate Buildup The copper and the prepreg go to the laminate buildup room, where press loads of material are put together. Many laminators build up their sheets in clean-room facilities with filtered air-conditioned systems, principally to keep dust particles from the copper and prepreg during the buildup operation. Electrostatic attraction in the

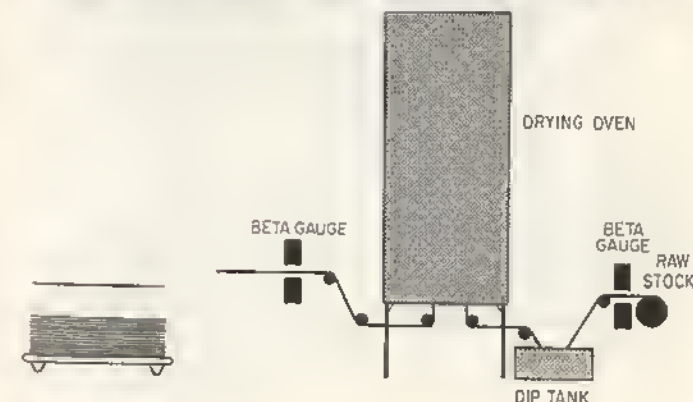
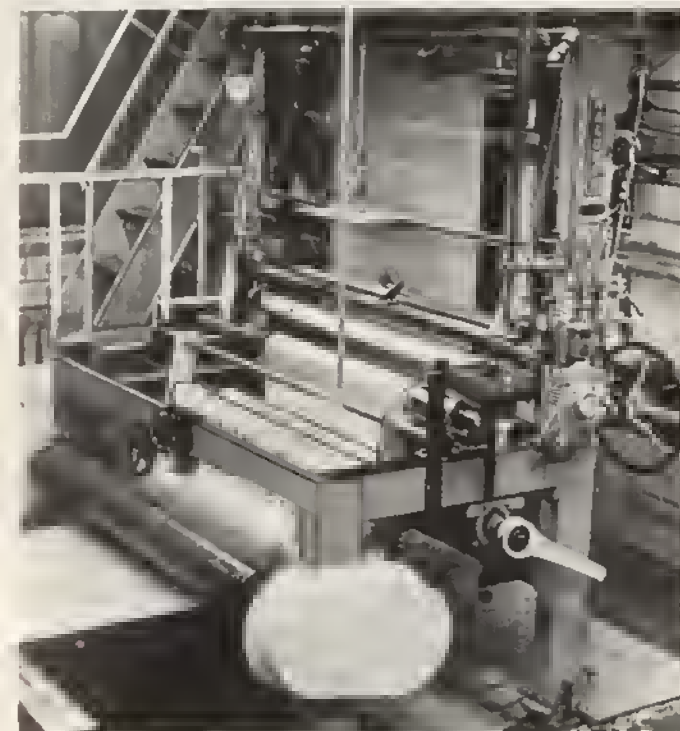


Fig. 2.1 (top) Treater and coater; (bottom) drying oven. (General Electric Company.)

treated material and copper before pressing is a source of pits and dents in the copper surface of the finished laminate. During the buildup operation, the copper foil is first laid against a large polished-steel press pan. Then a number of sheets of prepreg material are laid on top of the copper. The number of layers depends on the desired thickness of the laminate and the characteristics of the prepreg material. Extremely close tolerances in the



Fig. 2.2 Horizontal treater. (General Electric Company.)

final laminate are sometimes difficult to achieve because of the thickness tolerance of the copper and the thickness variance of each press pan. Some of the art of laminating comes in balancing all these variables to produce a final dense material to relatively close tolerances. The final sheet of copper foil is placed on top of the prepreg if the material is to be double-sided copper. If a single-sided laminate is desired, the top press pan can be sprayed with a mold release, or, more commonly, a release film can be laid down on top of the prepreg.

4. Laminate Pressing From the buildup room, the press pans with the material are stacked into a large, multiple-opening press, as illustrated in Fig. 2.3. Several sheets are pressed into each press opening; a typical press is capable of laminating approximately 80 sheets, 36 × 72 in., 1/8 in. thick. The presses, which are hydraulic, develop pressures in excess of 1000 psi. Steam is used as a heat source; it is released into each press platen as pressure is applied in order to cure the material into a final homogeneous sheet. To ensure that the material in the middle of each opening receives enough heat to effect final cure, several thermocouples are buried in the center sheet and used to plot a temperature curve. A

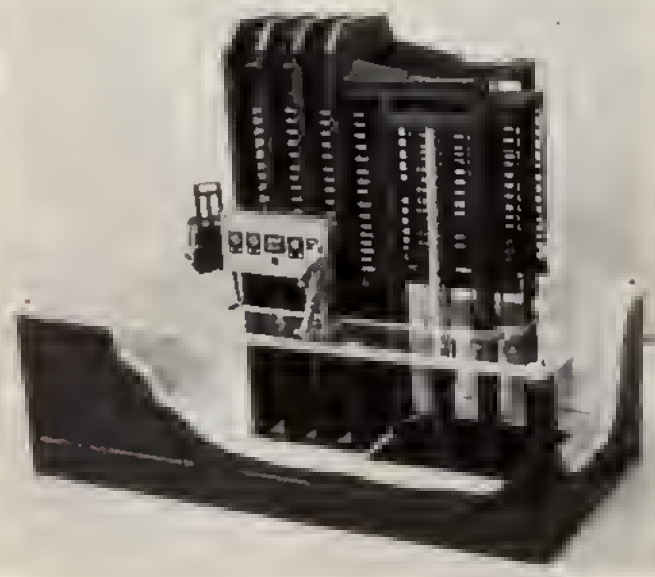


Fig. 2.3 Multiple-opening press. (General Electric Company.)

timer automatically records time against a preset cure cycle; and when the correct time has elapsed, the cure cycle is stopped by running cold water through the press platens. After proper cure, the laminate is trimmed to remove the irregular resin flow areas around the edges. Because of the difference in coefficient of thermal expansion of the materials involved, some manufacturers have found it desirable to place the laminate sheets in an oven at elevated temperatures after pressing. That allows stresses in the material caused by laminating to be relieved.

5. Laminating Final Inspection, Testing, and Quality Control The laminate is visually inspected and the copper surface is checked for scratches, pits, and dents. If the material passes the visual inspection, military, National Electrical Manufacturers Association (NEMA), or customer specification tests are run in a quality or process control laboratory laminate test area. Usually several samples from each press load are tested to assure that

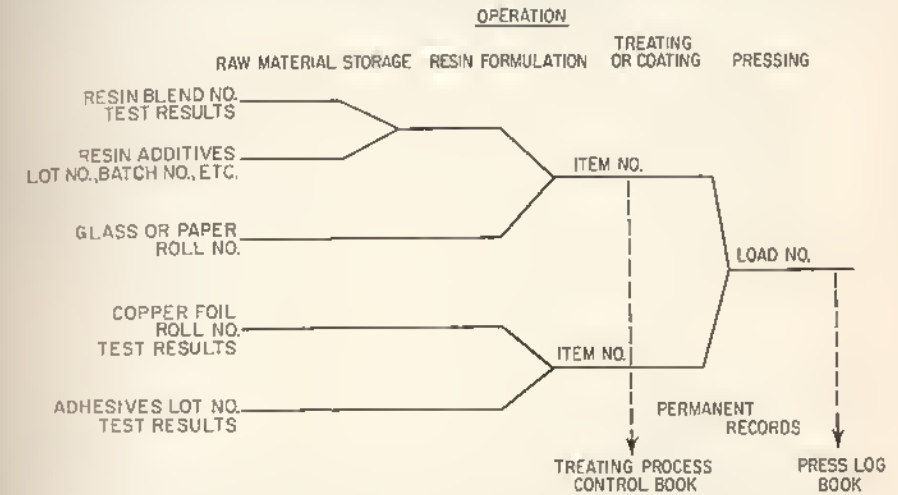


Fig. 2.4 Laminate identification system: permanent product-quality-control records.

the material will meet the appropriate specification. Each sheet is then labeled with the manufacturer's identification, the appropriate specification, and the load or lot number. Most manufacturers keep samples of material from each load for at least a year to enable them to check effectively any processing problems or questions which may result from the use of that lot in the field. Periodically laminators are required to run a complete set of physical and electrical tests as set forth in NEMA or MIL-P-13949E.

Laminators also keep careful records of all constituents of the laminate operation. First, the identity of industrial raw laminate materials is maintained and preserved. As an example, the General Electric Company uses the system of identification shown in Fig. 2.4 to generate its permanent product-quality-control records.

LAMINATE EVALUATION, SPECIFICATION, AND QUALITY CONTROL

In order to develop information for establishing design criteria and fabrication process control, it is essential to evaluate various types and manufacturers of copper-clad laminate. Too often the evaluation is based upon one sheet or sample submitted from each manufacturer. Since the process parameters of laminating are many, it is difficult for a manufacturer to make an entirely consistent material. Generally, electrical and mechanical design and the particular fabrication process will demand certain features and controls of the final material. It is therefore imperative that evaluation tests and specifications be determined by the demands on the finished product. Too often standards are set unrealistically high in many areas which have no relation to the fabrication process requirements of

the final product. In order to determine the pertinent means for test and evaluation, some of the tests most commonly used today are described in the following sections.

6. Surface and Appearance Probably the most difficult test to define adequately has been that of surface and appearance standards. Perhaps more laminate has been rejected for pits and dents in the copper, or for coloration variance of the base material, than for any other reason. That is particularly frustrating to the laminator, who has used all his technical skill to build a consistent insulating material only to find it rejected by the user for cosmetic reasons. Most laminators believe that surface and appearance standards must be applied in light of the finished boards. Some purchasers of laminate require copper surface standards only on critical areas, such as the areas used for tips inserted into edge connectors. In those cases, the user usually provides the laminators with an overlay of the critical copper areas to be inspected on each sheet of material before it is shipped. Thus the surface standard is applied only to the areas which are pertinent to the finished board. With over 90 percent of the copper being ultimately removed, the chance of a pit or dent affecting a critical area is small indeed. Nevertheless, many customers still pay premium prices by requiring the laminator to use special techniques or special selection to provide pit-free copper-clad material.

7. Copper Surface Copper pits and dents are best defined in the surface standards established in MIL-P-13949E. The specification defines the largest permissible dimension of a pit or dent and supplies point values for rating all pits and dents. The maximum total point count for pits and dents, per square foot of panel inspected, is to be less than 30. The point system is as follows:

Point Value	Longest Dimension, in
0.005-0.010	1
0.011-0.020	2
0.021-0.030	4
0.031-0.040	7
Over 0.040	30

A tighter classification (Grade B in MIL-P-13949E) calls for no pit with the longest dimension greater than 0.015 in and no more than three pits per square foot with the longest dimension greater than 0.005 in.

The overlay shown in Fig. 2.5 gives the area and dimension of pits or dents. It is sometimes used to train inspection personnel so they can quickly evaluate pit or dent area. The depth of scratches in the copper foil is measured with a Johansson surface finish indicator (N533). MIL-P-13949E calls for a maximum of five scratches per square foot, 4 in in length, over 140 μ m deep or a maximum of 20 percent of the foil thickness. Foil thickness is specified in MIL-F-55561 and is as shown in Table 2.1.

8. Color Color variance from lot to lot on any particular grade of laminate is usually caused by variation in color of batches of resin, variation of coupling agents on the glass cloth, type of paper used, or variation of alloy coating on the copper. All the raw materials

TABLE 2.1 Foil Thickness

Thickness by weight		Thickness by gages		Tolerance	
oz/R ²	g/m ²	Nom. in	Nom. mm	in	mm
1/2	153	0.0007	0.018	± 0.0002	± 0.0050
1	305	0.0014	0.035	± 0.0002	± 0.0050
2	610	0.0028	0.071	± 0.0003	± 0.0076
3	915	0.0042	0.106	± 0.0004	± 0.0102
4	1221	0.0056	0.143	± 0.0006	± 0.0152
5	1526	0.0070	0.178	± 0.0007	± 0.0178
6	1831	0.0084	0.213	± 0.0008	± 0.0204
7	2136	0.0098	0.246	± 0.0010	± 0.0254
10	3052	0.0140	0.353	± 0.0014	± 0.0355
14	4172	0.0196	0.492	± 0.0020	± 0.0508

are supplied to the laminator, and only by careful inspection of his incoming raw materials can the laminator completely control their final effect on color. The color of most epoxy-glass laminates comes from the coupling agent on the glass cloth. The main function of a coupling agent is to provide a stable bond between two dissimilar surfaces; the agent is applied by the glass manufacturer to help bond the resin to the glass. The two commonly used coupling agents, or binders, are silane (produces white to clear) and laminator chrome (produces green-colored material). If coloration must be specified in a copper-clad laminate, it should be by working carefully with the laminator. A set of samples which will illustrate the acceptable color extremes must be established.

9. Silk Screen Adherence Test In many applications it is important to test for silk-screen ink adherence to both the clad and unclad surfaces of laminates. The laminator has a wide variety of techniques at his disposal to prepare the copper and unclad surfaces to help achieve ink adherence. The method of cleaning the copper will affect copper surface

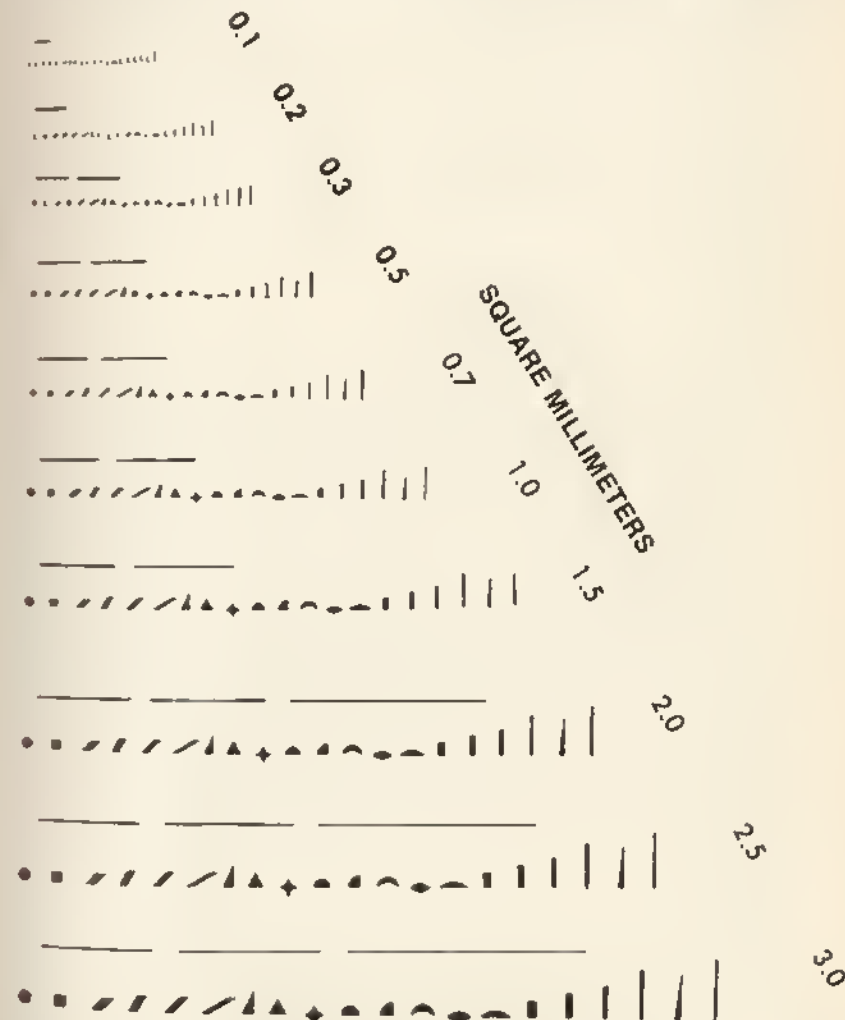


Fig. 2.5 Surface imperfection overlay.

standards, since some copper finishes serve to highlight the copper imperfections much more than others do.

Establishing a standard for ink adherence on the metal side of laminates is especially important when screened component designs are used. Epoxy-glass laminates are sometimes difficult to silk-screen, since mold releases are often used on the press pans. The releases generally must be removed mechanically from the metal side by brushing at the same time the copper is finished. Thin release films also are used, particularly with thin laminates from which other release agents cannot be effectively finished. However, sometimes the release films make the epoxy surface so smooth that, although it is release-free, ink adherence is still difficult to achieve. Occasionally screening also is difficult on the metal side of paper-base laminates. Generally, the laminator should be encouraged to use different release agents, because brushing may lower the insulation resistance of some paper materials as much as ten times in humidity.

In establishing a silk-screen ink-adherence test, it is necessary to specify the ink-mold cycle. The standard tape-adhesive test is common practice in plating (MIL-P-55110), and it can also be used for ink adherence:

The screen process printing shall be used to apply one of the following inks to the metal side of the laminated material.

(a) Use black acid-resisting ink. Dry 30 minutes minimum in an oven.

(b) Use black epoxy ink. Cure eight minutes at 135°C in an oven.

After drying, a strip of pressure-sensitive cellophane tape (conforming to L-T-90, type 1, class A), shall be pressed firmly across the surface of the ink. The tape shall then be removed rapidly by manual force applied approximately perpendicular to the plane of the ink surface. Fresh tape shall be used for every test.

10. Punchability and Machinability The test for punchability and machinability has been subject to much debate among laminators and users. There is as of today no universally accepted test method for either characteristic, and research has found little correlation with values such as Rockwell's hardness or Barcol hardness.

a. Drilling. Each laminator should be consulted on recommended drilling speeds and feeds prior to any evaluation. Sometimes poor drilling from one material to another occurs because each laminator was not given the benefit of recommendations for the material. Sectioning the board will help evaluate the type of hole being obtained and is particularly important on plated-through applications. Many times sectioning will show the material has been heated to such an extent by drilling that the surface is smooth and not platable, is smeared with resin, or is so roughened that glass fibers protrude and will inhibit continuous plated-through lines.

b. Punching. Punchability can be measured in a die simulating the conditions used in the fabrication process. Various hole sizes, spacings, and configurations should be incorporated in the test die. Many paper-base grades will tend to vary in punchability from lot to lot; an care must be taken to measure a wide range of sample panels. Careful physical inspection and sectioning of the holes will tell the type of punching being obtained. The material must be carefully inspected to ensure that no cracking occurs and that there is no lifting of copper around the hole. ASTM Designation D G17-44 describes a standard die and a method of rating the results achieved by the die. The rating system is described in Table 2.2. This standard should be reviewed to determine if either the die or rating system could apply to the user's application. Since a rating system of this type can be subjective, care must be taken that only carefully trained people apply the system. Each laminator should be consulted as to his recommended die tolerance and punching temperatures.

11. Peel Strength

a. Before the Soldering Operation. The basic test pattern for testing peel strength of copper-band strength specified in MIL-P-13949E and by the NEMA standard is illustrated in Fig. 2.6. The pattern should be processed by the same fabrication techniques as in the user's final process with the exception of exposure to various plating solutions or solder which will also be tested. When tested for peel strength, the specimen should be mounted on a flat horizontal surface. The wide copper end of each trace should be peeled back approximately one inch so that the line of peel is perpendicular to the edge of the

TABLE 2.2 ASTM Rating of Punched Specimens

Rating	Point score	Edge	Surface	Holes
Excellent	100	Clean cut	Unaffected	Clean cut, no bulging
Very good	90	Very slight chipping or very slight drag	Microscopic cracks around edges of piece or holes	Slight chipping to walls of holes or very slight bulging around hole
Good	80	Slight chipping or slight drag	Very slight surface cracks around edge of piece or holes or slight raising of surface	Some chipping in walls of holes or slight bulging around hole
Fair	70	Some chipping or drag	Slight surface cracks or some raising of surface around holes	Considerable chipping in walls of holes, some bulging around holes or slight tapering of hole
Poor	50	Objectionable chipping or drag	Surface cracks around edges of piece or holes or distinct raising of surface	Bad chipping in walls of holes, pronounced bulging around hole, or pronounced tapering of hole
Very poor	25	Bad chipping or slight cracking of edges	Pronounced surface cracks or pronounced raising of surface at edges or around holes	Slight cracks extending in body of material from holes, severe bulging around holes, or very pronounced tapering of hole
Worthless	0	Severe chipping or cracking of edges	Very severe surface cracking or raising of surface	Body of material cracked at holes, very severe bulging at holes or closing up of hole with material

specimen. The end of the peeled strip should then be gripped by a clamp which is attached to a force indicator or tensile tester adjusted to compensate for the weight of the clamp and connecting chain. The copper foil is then pulled from the material at a rate of 2 in/min and the minimum load on the force indicator is recorded.

Peel strength of a $\frac{1}{8}$ -in trace is converted to pounds per inch width of peel by dividing the indicated force by the measured width of the strip. The peel strength test pulls the adhesion area directly under the radius formed between the copper being peeled at right

angles and the material. Since 2-oz copper forms a larger radius than 1-oz copper, a greater area of adhesion is being pulled; as a consequence, peel strength for 2-oz copper is increased. For that reason it is always important to maintain a peeling force that is 90° to the copper surface so the radius of the peel is constant. Generally, a 5° deviation from perpendicular is acceptable; and if a large enough distance is allowed between the specimen and force indicator, the 5° variance will not be exceeded during the course of the test. However, many users and laminators have found it desirable to use a testing machine which moves the force indicator along the test specimen as the copper is being pulled, thus keeping the angle of pull constant.

b. During the Soldering. As circuit traces and pads become smaller, the problem of bond strength retention at dip-soldering or

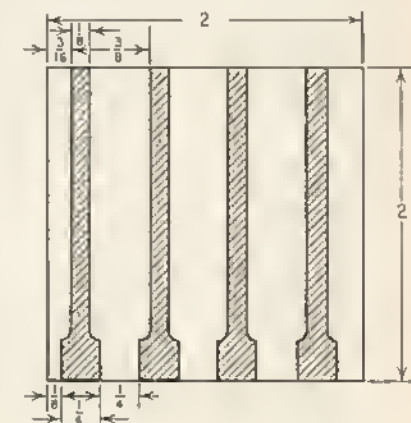


Fig. 2.6 Copper peel strength pattern. Dimensions are in inches.

solder-etch-up temperatures has become an increasingly important factor. The NEMA test, at an elevated temperature, uses the same test pattern; the test sample, if G-10 or FR-4, is immersed in silicone oil for 6 min at 125°C and at 150°C if it is G-11 or FR-5. Peeling is done, as above, immediately after removal from the oil over a 15-s period, and the average peel retention is noted.

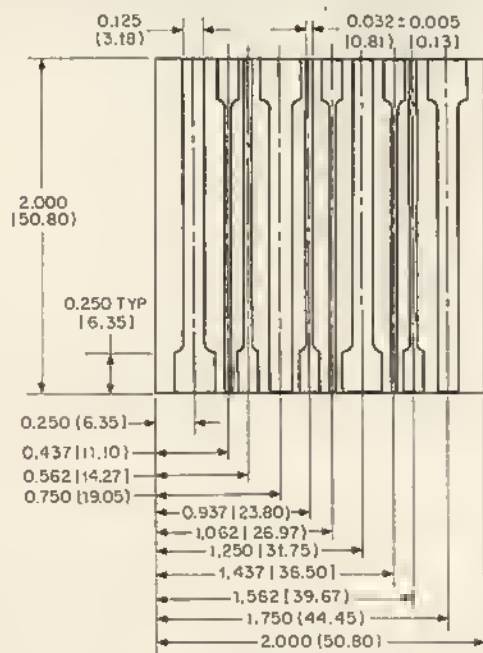


Fig. 27 MIL-P-13949E peel strength test pattern. Dimensions are in inches; numbers in parentheses are metric equivalents in millimeters.

Results on several proposed methods of testing to date have indicated that it is important to immerse the test specimen for a relatively long period of time—at least 6 min—prior to testing. The long immersion is required to bring both the specimen and the test jig to a constant temperature; shorter periods of immersion can result in extremely inconsistent test data. Specimens can also be placed in an air-circulating oven for 60 ± 6 min; the peel test can then be run in the oven at the appropriate elevated temperature. Underwriters Laboratory also has developed a test to determine long-term effects of heat aging on bond strength. The test is the normal peel test after the sample has been aged at 125°C for 1344 h.

c. After the Soldering. It is usually important to test for peel strength after the dip-soldering cycle. The specimen should be floated on the solder pot at 500°F for from 5 to 20 s, depending on the material being used. No flux should be used on the copper, and all excess solder must be removed. Any solder on the specimen will result in extremely irregular peeling. Sometimes petroleum jelly may be applied to the copper before soldering to prevent any solder wetting. After removal of excess solder, the specimens should be visually examined for evidence of blistering and delamination of the metal foil.

d. After Plating. Plating solutions, particularly cyanide gold, can affect adhesion on some copper-clad laminates. Therefore, it is recommended that a peel test be considered after exposure to a normal plating operation. MIL-P-13949E recommends the pattern illustrated in Fig. 27 be used if peel tests after plating are to be run. The procedure recommended is as follows:

1. Degrease in hot trichloroethylene vapor.
2. Immerse for 5 min, at 93.3 to 100°C, in a 30-g/L sodium hydroxide and 30-g/L sodium carbonate solution.

3. Rinse in hot water.
4. Immerse for 20 min at $23 \pm 5^\circ\text{C}$, in a 50-g/L sodium cyanide solution.
5. Rinse in hot water.
6. Immerse for 30 min at $54.5 \pm 2.8^\circ\text{C}$ in a 10-g/L sulfuric acid (specific gravity 1.836) and 30-g/L boric acid solution.
7. Rinse in hot water.
8. Dry for 1 h at 93.3 to 98.9°C.
9. Inspect specimens for such obvious defects as delamination, wrinkles, mending, blisters, and cracks.

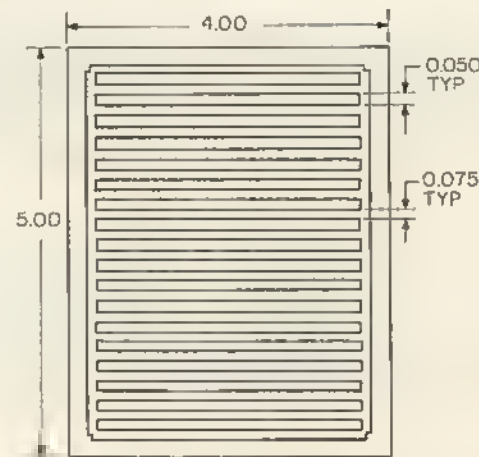


Fig. 28 Dip-soldering test pattern. Dimensions are in inches; 0.050 in = 1.27 mm; 0.075 in = 1.91 mm; 4.00 in = 101.60 mm; 5.00 in = 127.00 mm. Metric equivalents (to the nearest 0.01 mm) are given for general information only and are based on 1 in = 25.4 mm. Tolerances are ± 0.005 in (0.13 mm) for three-place decimals and ± 0.06 in (1.52 mm) for two-place decimals. This pattern is useful with the solderability test, since it clearly defines the contact area of a solder meniscus while limiting the mass of solder adhering to the board. This aids in obtaining consistent test results.

12. Solder Resistance Solder resistance should be measured and evaluated in terms of the soldering demands made on finished boards. A recommended dip-soldering test pattern should include both 1-in² areas of copper until several types of drilled and plated-through holes. The pattern should be processed with normal production techniques (see Fig. 2.8).

Most dip-soldering tests require that the board be floated on the solder surface for a given period of time. The temperature of the bath should be measured 1 in below the solder surface and must be controlled to within $\pm 5^\circ\text{F}$ for consistent results. The time of solder float is heavily dependent on the material used and the temperature of the solder bath. Both MIL-P-13949E and NEMA specifications give minimum solder float times. The test procedure outlined in MIL-P-13949E follows.

Apply the following cleaning procedure to four control specimens:

1. Dip in 10 percent fluoroboric acid for 5 s.
2. Rinse in water.
3. Scrub with fine pumice.
4. Rinse in water.
5. Dip in 10 percent fluoroboric acid.
6. Rinse in water.
7. Dry with clean compressed air or inert gas.

Flux four control specimens immediately after cleaning with a solution of 75 percent isopropyl alcohol and 25 percent water-white rosin by weight. Flux may be applied by brush or dip. The test apparatus, shown in Fig. 2.9, is calibrated as follows:

1. Level the test fixture so that it enters the solder bath in a horizontal position and forms menisci of equal size. The minimum size solder pot which can be used is $8 \times 12 \times 3$ in deep with solder conforming 60 to 63 percent.
2. Preheat the test fixture for 1½ min by dipping it in the solder bath at $232 \pm 12.2^\circ\text{C}$.

3. Raise the test fixture from the solder bath and cool for 1½ min.
4. Cool the test fixture for 1½ min. Just before the cooling period is over, place a control specimen in the test fixture, counterweight with 400 g, and lower the test fixture into the solder bath so that it reaches its maximum depth of dip at the end of the 1½-min cooling period. Skim the inside from the solder bath prior to dipping.
5. Dip the control specimen for the test period. At the end of the dip time, raise the test fixture from the solder bath and remove the control specimen. Place the specimen aside to cool and rest the remaining control specimens.

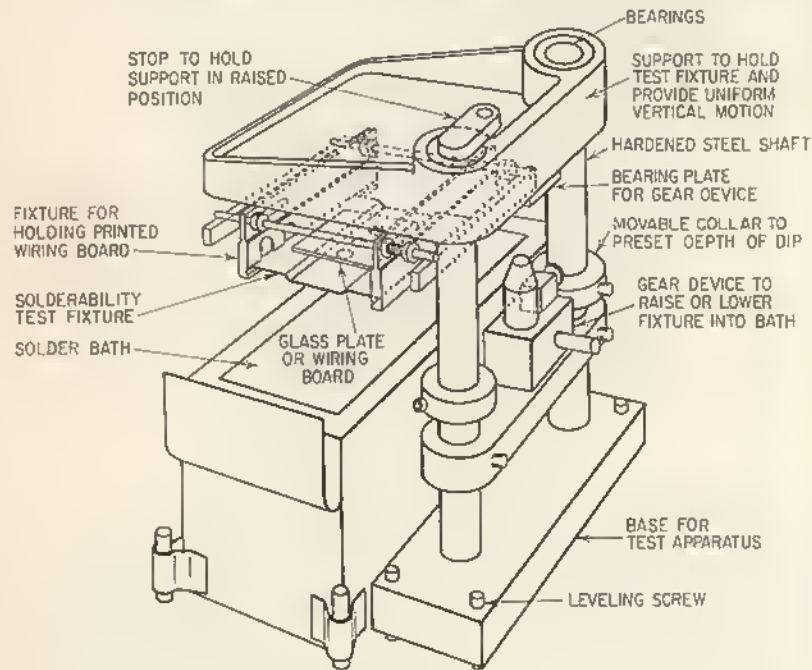


Fig. 2.9 Meniscus test fixture.

BASE LAMINATE TESTING

To perform a base laminate test, cut off a 1 × 6-in strip on two 6 × 6-in boards. Immerse one board in 500°F solder at a 45° angle for 20 s. Remove, allow to cool, and observe for blistering, measling, warp, and twist. Place the other board on edge in a metal rack in a pressure cooker with 15 lb of steam for ½ h. Allow board to dry at room temperature for 1 h and immerse at 500°F and at an angle of 45°. This test is a relative measure of cure, since cure and moisture absorption are proportionally related.

13. Delamination Delamination of materials is defined as a separation of the layers of the board under heat and is usually the result of poor laminate cure or lack of resin in one or more layers. A white checking in the material that is sometimes referred to as "measling" is caused by one fiber of glass pulling away from another at the intersection of the glass fibers in the woven cloth. The effect may be due to stresses pressed into the material and released upon heating, which results in a minute pulling apart of the glass cloth and resin. Also, if eyelets or terminals are too tightly set, measling may occur from external stressing factors. Measling may be present around the periphery of a board if poor shearing or sawing techniques are used. The Institute of Printed Circuits, in its publication "Acceptability of Printed Circuit Boards," has defined measling, crazing, blistering, and weave exposure, as shown in Fig. 2.10. The base laminate should be examined for the conditions illustrated.

The laminate should also be examined for coloration changes after dip-soldering, particularly the lighter paper-base grades. Excessive temperature will quickly discolor paper laminates, and care must be taken not to place excessive demands on the material. The user is encouraged to section the holes in the test pattern after the solder dip, particularly in plated-through-hole applications. Sectioning may show volatiles being driven off through voids or weak points in plated-through holes. The volatiles may be the result of excessive volatile pressed into the material or of moisture absorbed during the plating process because of poor fabrication techniques. Volatiles can cause a huddling of solder in the plated-through holes and give rise to cold solder joints.

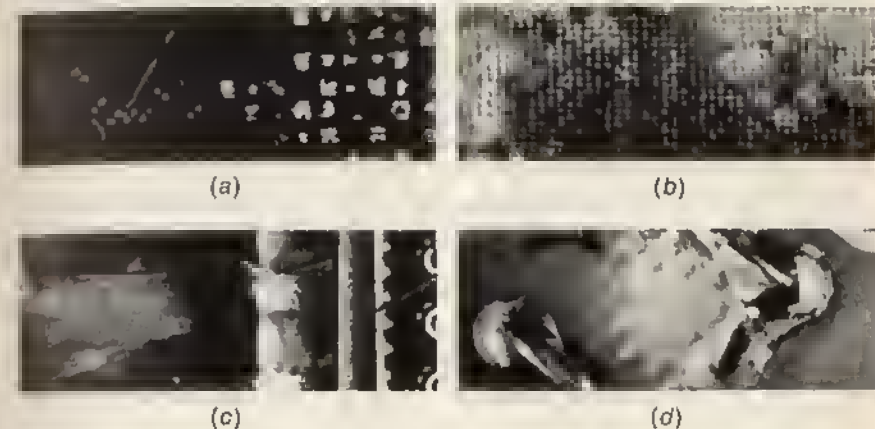


Fig. 2.10 Base laminate defects. (a) Measling: a condition existing in the base laminate in the form of discrete white spots or crosses, below the surface of the base laminate, that reflect a separation of fibers in the glass cloth at the weave intersection. (b) Crazing: a condition existing in the base laminate in the form of connected white spots or crosses, on or below the surface of the base laminate, that reflect the separation of fibers in the glass cloth and connecting weave intersections. (c) Blistering: a localized swelling and separation between any of the layers of the base laminate and/or between the laminate and the metal cladding. (d) Weave texture: a surface condition in which the unbroken fibers are completely covered with resin yet exhibit the definite weave pattern of the glass cloth. Weave exposure: a surface condition in which the unbroken woven glass cloth is not uniformly covered by resin.

The board should also be examined for excessive warp and twist and any hole tilting or deformation. MIL-P-13949E may be consulted to determine warp and twist specifications.

14. Warp and Twist Warp or twist in manufactured sheets with both dimensions 18 in or over is determined by suspending the sheet so that the horizontal level plane touches both corners. No pressure should be applied, and the warp or twist is calculated as follows:

$$C = \frac{36D}{L_x^2} \times 100$$

where C = warp or twist for 36-in dimension

D = maximum deviation from horizontal surface

L_x = length along horizontal straight edge, in

Cut-to-size panels should be measured by placing the convex surface of the panel upward and measuring the maximum vertical height from horizontal surface to edge. In the rare or cut panels, the usual deviation is expressed in inches per inch rather than in terms of a 36-in dimension.

15. Flammability In recent years more emphasis has been placed on flammability of copper-clad laminate, particularly to meet more stringent Underwriters' Laboratory requirements. The UL test is most commonly used, and it is performed as follows:

1. Condition five samples at 23°C, 50 percent relative humidity for 48 h.
2. Flame is provided by 4-in burner with 0.375-in nominal bore. Natural gas (900 to 1200 Btu) or synthetic gas (530 to 550 Btu) may be used. Burner is ignited away from the sample and adjusted to produce a blue flame of approximately 0.75 in.
3. The test sample is clamped vertically so that the lower end is 0.38 in above the top of the burner.
4. The test flame is placed centrally under the lower end of the test specimen; the burner may be tipped 20° to avoid dripping material.
5. The burner is removed after the specified time and the duration of flaming or glowing of the specimen is noted and recorded. When specifications call for two applications, the second test is made within 5 s after combustion stops.

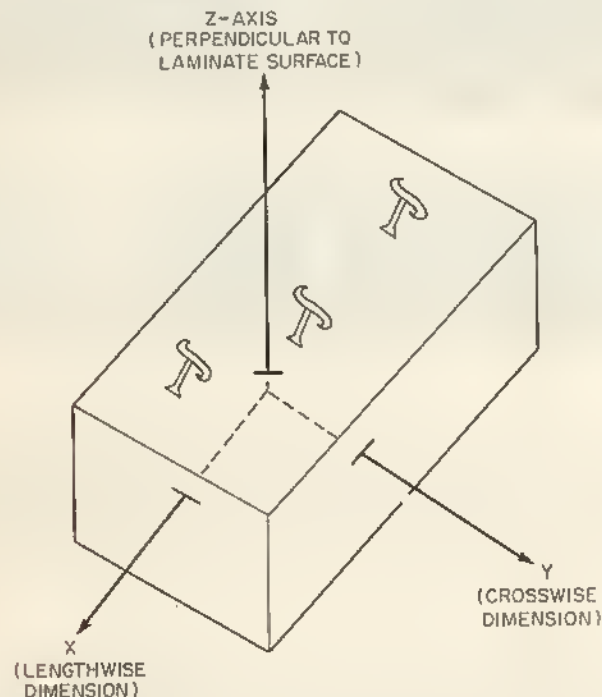


Fig. 2.11 Typical laminate axis definition. Note that x, or lengthwise, direction is defined by a specific part of the manufacturer's watermark.

16. Dimensional Stability As circuit density has increased, so has the need for dimensional stability of finished circuit boards. In general, the dimensional stability in the lengthwise or crosswise dimension (x, y) illustrated in Fig. 2.11 is a function of the laminate reinforcement (glass or paper). The thickness expansion (z) is generally a function of the resin system or resin matrix. It is important to recognize which is the lengthwise direction of the treated material going into laminate construction. The laminator usually identifies the lengthwise dimension by the vertical direction of his water mark, as shown.

To measure dimensional stability, one specimen should be taken from the lengthwise edge and one from the center of each sample sheet. Specimens should conform to Fig. 2.12, which is taken from MIL-P-55617A Military Specification for Thin Laminate Metal Clad.

1. Drill four 0.031-in-diameter holes 0.5 in from each edge and at the midpoint of each side as shown in Fig. 2.12.
2. Using the determined location points, measure from hole to hole the lengthwise and crosswise dimensions and record.
3. Cover the drilled holes with adhesive circular labels and remove all copper.
4. Remove the labels and measure; record the measurement.

5. Subject the specimen, when applicable, to elevated temperature, usually 1 h at the continuous operating temperature for the material being tested.

6. Allow the material to cool and record the dimensions as above.

7. If temperature cycling is required, subject the samples to required cycle. Military specifications call for glass-base material in cycle ½ h at 125°C, ¼ h at 25°C, ½ h at -65°C, and ¼ h at 25°C. Complete five of these cycles before measuring dimensional stability. See the chapter on multilayer materials for a more detailed discussion of dimensional stability in the z axis.

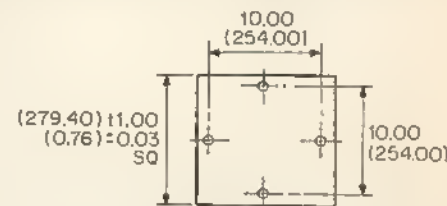


Fig. 2.12 Dimensional stability specimen. Dimensions are in inches. Metric equivalents (to the nearest 0.01 mm) are given for general information only and are based on 1 in = 25.4 mm. Millimeters are in parentheses. Drill four 0.031-in. (0.79-mm) diameter holes 0.500 in (12.70 mm) from each edge and within 0.250 in (6.35 mm) of the midpoint measured from each corner.

17. Degree of Cure The degree of cure of most base resins that go into high-pressure laminates, as well as the measurement of cure, is subject to broad spectrum of interpretation. However, there are several methods involving some instrumentation which might be applicable in determining the degree of cure. Limiting our discussion to epoxy-glass laminates and the methods suitable for those not having extensive laboratory facilities, the following techniques are suggested:

a. Thermal Analysis. Use a differential scanning calorimeter to determine the residual caloric value of the cured sample. This instrument measures heat from the exothermic reaction, caused by the resin curing, as a caloric value. When a laminate is fully cured, there is always some residual caloric value remaining. Measurement of that value in a differential scanning calorimeter usually gives low numbers compared to the very high exothermic reaction measured when uncured samples are tested. Therefore, the measured amount of residual caloric values on acceptable standard laminates will provide a working reference for residual values and for the high exotherm of uncured samples.

b. Spectral Analysis. This is applicable in some instances, but it has some inherent drawbacks. Sample extraction and determination of weight loss during the extraction plus the spectral analysis of the extractables can sometimes provide a measure of the degree of cure.

c. Dip-soldering Techniques. Many users determine relative degree of cure by inserting a 6 × 6-in sample with a 1-in strip of copper etched on it into a 500°F solder pot at 45° inclination. Measurement of deflection as well as warp and twist will often give a measure of the relative degree of cure if enough data have been gathered to determine the basic working range of a particular resin system.

In the final analysis, degree of cure is a relative thing. Its value should be measured under conditions that closely approximate the processing stresses and environmental conditions to which the circuit board is to be subjected.

INSULATION RESISTANCE

The insulation resistance between two conductors or holes is the ratio of the voltage to the total current between the conductors. Insulation resistance is composed of both the volume and surface resistance in a copper-clad laminate. Results of insulation-resistance tests can vary widely if careful control of environmental conditions and process techniques of the test samples is not exercised. While the actual value of insulation resistance may be initially important, it is the change in resistance under a specified environmental condition that is usually more significant. The insulation-resistance test is of greatest value when the test specimen is subjected to the same environment as will be required in the final application.

18. Parameters and Test Conditions The insulation resistance of copper-clad laminates decreases both with increasing temperature and with increasing humidity. Volume resistance is particularly sensitive to temperature changes, and surface resistance changes

widely and very rapidly with humidity changes. Since extended periods of conditioning are required to determine the effects of humidity on surface resistivity, it is recommended, for example, that 96 h at 35°C in a 90 percent relative humidity environment be used. Test data also show that some materials will recover from humidity conditioning much more rapidly than others. Therefore, it may be desirable to cycle humidity and temperature over a specified period of time if this test will best simulate the final environment.

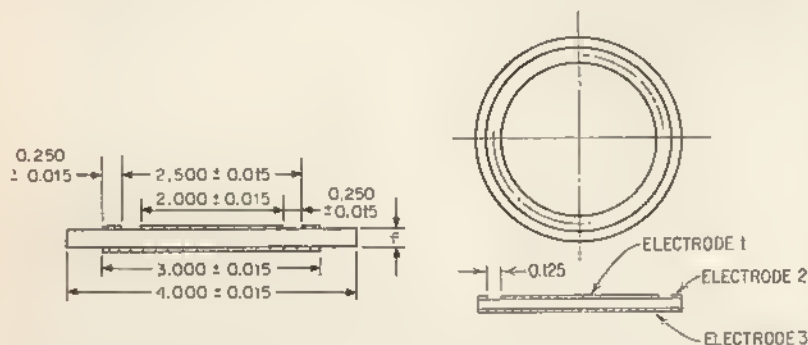


Fig. 2.13 Surface-resistance pattern. Dimensions are in inches.

In order to obtain consistent results, extreme care must be taken in processing and handling the test boards. Etching of the pattern must leave well-defined lines with no cracked or ragged areas. Rubber gloves, preferably surgical, must be used at all times while handling the test boards. Fingerprints can reduce the value of surface resistivity as much as three decades. The following procedure for cleaning test specimens is recommended before any testing proceeds.

1. Place specimens from etch bath immediately in racks in running water at 60°F for 5 min.
2. Place specimens in 10 percent oxalic acid with agitation for 10 min.
3. Scrub with fine pumice.
4. Place specimens in running water at 60 ± 5°F for 30 min.
5. Scrub with demineralized water (one megohm minimum). Rinse with fresh demineralized water. Remove with gloves.
6. Stand specimens on rack in oven at 80°C for one hour (use double clip).
7. Remove with rubber gloves (washed with alcohol), and put into desiccator used as dust-free container.
 - a. Wash gloves in alcohol and store in same dust-free container.
 - b. Touch only sample edges and/or clip leads; wear rubber gloves.

19. Specimen Preparation Either the metal foil of the specimen may be completely removed by etching or metal foil outlines may be left on the specimen to form the edges of the electrodes. The electrodes are generally completed by applying a porous, conductive silver paint to both sides of the specimen. However, for comparative purposes, copper electrodes may be used rather than the conductive paint.

a. **Surface Resistance.** The surface resistance between two points on the surface on any insulation material is the ratio of the dc potential applied between the two points to the total current between them. One of the most commonly used surface-resistance patterns is the ASTM pattern illustrated in Fig. 2.13. This is a circular pattern with a three-electrode arrangement for measuring the surface resistance parallel to the laminate. The third electrode is a guard conductor which intercepts stray currents which might otherwise cause error. For measurement of surface resistance, the resistance of the surface gap between electrodes 1 and 2 is measured. The measured current flows between electrodes 1 and 2, while stray current flows between electrodes 1 and 3, as shown in Fig. 2.14.



Fig. 2.14 Guarding circuit for surface resistance parallel to the laminate.

Measurements should be made after 500 V dc has been applied to the specimens for 60 s + 5 s or - 0 s by means of a megohm bridge having an accuracy of ±6 percent at 10¹¹ Ω. Measurements are to be made at the end of the prescribed conditioning time and condition.

Surface resistivity is calculated as follows:

$$r' = \frac{R'P}{D}$$

where $r' =$ surface resistivity, MΩ

$R' =$ measured surface resistance, MΩ

$P =$ effective perimeter of the guarded electrode, cm

$D =$ Distance between inner circle and outer guard ring, cm

Comb patterns such as the one illustrated in Fig. 2.15 have also been commonly used in surface-resistance tests. The patterns may simulate the final configuration of most copper-clad laminate boards, since the lines on the pattern approximate trace spacing in the board.

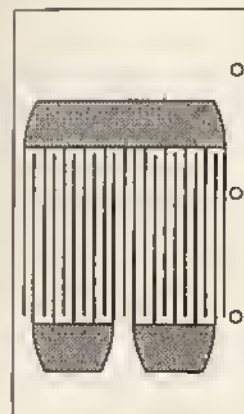


Fig. 2.15 Comb pattern for surface resistance.

Because of the narrow, close pattern, care must be taken to achieve clean, well-defined lines with a ragged meas.

b. **Volume Resistance.** Volume resistance is the ratio of the dc potential applied to electrodes embedded in a material to the current between them. It is usually expressed in ohm-centimeters. The ASTM test pattern shown in Fig. 2.16 can also be used for volume resistance. For the measurement of volume resistance, the measured cur-

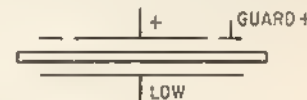


Fig. 2.16 Guarding circuit for volume resistance.

rent flows between electrodes 1 and 3, while stray current flows between electrodes 2 and 3 as shown in Fig. 2.16.

Volume resistivity is calculated as follows:

$$r = \frac{RA}{t}$$

where $r =$ volume resistivity, MΩ·cm

$R =$ measured volume resistance, MΩ

$A =$ effective area of guarded electrode, cm²

$t =$ average measured thickness of specimen, cm

c. **Properties of Copper-clad Laminates.** To establish the design parameters necessary for any copper-clad printed board application it is necessary to know approximately what the principal laminate properties are. Listed in this section are the characteristics which are most readily required in both electrical and mechanical design applications. Some of the tests involving those properties are described in the section on laminate evaluation. The principal materials to be discussed are as follows.

20. Grades and Specifications See Tables 2.3, 2.4, and 2.5.

21. Material Designations The most common method of designating copper-clad materials is described in MIL-P-13949E and illustrated in Fig. 2.17. FL is always used to designate foil-clad laminates. As an example: FL-GF-062-C 1/2-A-2-A means foil-clad, FR-4, 0.062 in thick, copper-foil-clad 1 oz one side and 2 oz the other, grade A pits and dents, class 2 thickness tolerance, and class A wrap and twist.

TABLE 2.3 Standard Materials

NEMA grade	Military designation MIL-P-13949E	Resin system	Base	Color	Description
XXXP	None	Phenolic	Paper	Opaque brown	Phenolic paper with no cold-punch stipulation
XXXPC	None	Phenolic	Paper	Opaque brown	Phenolic paper with punchability at or above room temperature
FR-2	None	Phenolic	Paper	Opaque brown	Phenolic paper, punchable, with flame-resistant (self-extinguishing) resin system
FR-3	PX	Epoxy	Paper	Opaque cream	Epoxy resin, paper base with flame-resistant resin system, cold punching, and high insulation resistance
None	None	Epoxy	Paper-glass composite	Opaque cream	Epoxy resin paper core with glass on the laminate surface, self-extinguishing, economic fabrication of paper base, mechanical characteristics of glass fiber, flame-resistant, designed for low-capacitance or high-impact applications
FR-6	None	Polyester	Glass mat	Opaque white	Epoxy-glass, general purpose
G-10	GE	Epoxy	Glass	Translucent green or white	Epoxy-glass with self-extinguishing resin system
FR-4	GF	Epoxy	Glass	Translucent green or white	High-temperature epoxy-glass with strength and electrical retention at elevated temperatures
G-11	GP	Epoxy	Glass	Translucent green or white	High-temperature epoxy-glass with flame-resistant resin system with strength and electrical retention at elevated temperatures
FR-5	GH	Epoxy	Glass	Translucent green or white	Polyimide resin, glass laminate with high continuous operating temperature and high property retention at temperature, low z dimensional expansion
		Polyimide	Glass	Translucent dark brown	

22. Conditioning Designations The conditioning designations used to describe the environments in which tests were run are as follows:

- Condition A. As received; no special conditioning
- Condition C. Humidity conditioning
- Condition D. Immersion conditioning in distilled water
- Condition E. Temperature conditioning
- Condition T. Test condition

Conditioning procedures are designated in accordance with the following:

1. First, a capital letter indicates the general condition of the specimen to be tested, i.e., as received or conditioned to humidity, immersion, or temperature.
2. A number which indicates the duration of the conditioning, in hours, follows.

TABLE 2.4 Materials for High-frequency Application

NEMA grade	Military designation MIL-P-13949E	Resin system	Base	Color	Description
GT	GT	TFE	Glass	Opaque brown	Glass fabric base, PTFE (Teflon) resin, controlled dielectric constant
GX	GX	TFE	Glass	Opaque brown	Glass fabric base, PTFE (Teflon) resin dielectric constant with closer controlled limits than GT
		Polystyrene	Glass	Opaque white	Polystyrene cast-resin base for low-dissipation-factor applications
		Cross-linked polyethylene	Glass	Opaque white	Polyethylene cast base, radiation cross-linked for low dissipation factor

TABLE 2.5 Materials for Additive Circuit Processing

NEMA grade	Military designation MIL-P-13949E	Resin system	Base	Color	Description
Adhesives					
XXXPC	None	Phenolic	Paper	Opaque brown	These laminates are designed for use with additive processes using adhesive bonding techniques.
FR-2	None	Phenolic	Paper	Opaque brown	
FR-3	None	Phenolic	Paper	Opaque brown	
FR-4	None	Epoxy	Glass	Translucent green or white	
Seeded and coated					
XXXPC	None	Phenolic	Paper	Opaque brown	These laminates are used in the patented seeded and coated processes. They are seeded with a small percent of the catalytic seeding agent dispersed throughout the resin system and are coated with a catalyzed adhesive.
FR-2	None	Phenolic	Paper	Opaque brown	
FR-3	None	Phenolic	Paper	Opaque brown	
FR-4	None	Epoxy	Glass	Translucent green or white	
FR-2	None	Phenolic	Paper	Sacrificial aluminum-clad	These laminates are the patented sacrificial additive process. They are clad two sides with a specially anodized aluminum foil. The sacrificial aluminum cladding makes the laminate surface acceptable for the additive process.
FR-3	None	Phenolic	Paper		
FR-4	None	Epoxy	Glass, glass-paper composite		
Swell and etch					
FR-4		Epoxy	Glass	Translucent green or white	This laminate is used in the commercially available swell and etch additive process. The epoxy-glass laminate has a 0.0015-in-thick "resin-rich" surface and a specially designed surface for swell and etch chemicals.

3. After a slant, a second number indicates the conditioning temperature, in degrees centigrade.

4. After another slant, a third number indicates the relative humidity whenever the relative humidity is controlled. For example, C 96/35/90 indicates humidity conditioning for 96 h at 35°C and 90 percent relative humidity.

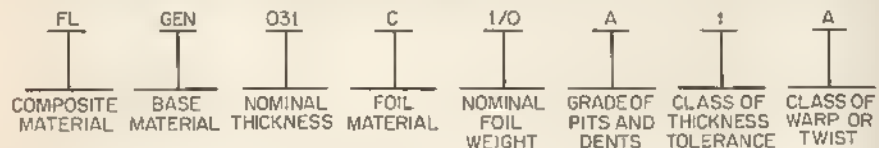


Fig. 2.17 Designations of copper-clad materials.

23. Mechanical Properties of Laminates The principal characteristics usually required in the mechanical design of printed boards are outlined here: If a specific test is required, the test is listed under a particular heading. Whenever they are available, the minimum standards set forth in MIL-P-13949E or NEMA standards for industrial laminates are used.

a. Flexural Strength. Method: ASTM D 790. Unit of value: psi. This test is measure of load which a beam will stand without fracture when supported at the ends and loaded in the center, as shown in Fig. 2.18.

Materials $\frac{1}{8}$ in in thickness are broken over a 1-in span. Materials over $\frac{1}{8}$ in in thickness are broken over a span equal to 16 times the material thickness. "Lengthwise" means the direction of the sheet known to be stronger in flexure, and "crosswise" means at right angles to the lengthwise direction. See Table 2.6.

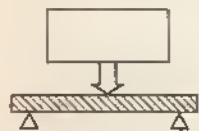


Fig. 2.18 Flexural strength test.

b. Weight of the Base Material. Unit of value: oz/ft² (add 1 or 2 oz/ft² per side for the weight of copper). See Table 2.7.

c. Thickness Tolerance. Nominal thicknesses and tolerances should be as specified below per MIL-P-13949E. At least 90 percent of the area of a sheet should be within the tolerance given, and at no point should the thickness vary from the nominal by a value greater than 125 percent of the specified tolerance. Cut sheets less than 18 × 18 in should meet the applicable thickness tolerance in 100 percent of the area of the sheet. Class of tolerance is as specified in the type designation. For nominal thicknesses not shown in this table, the tolerance for the next greater thickness shown applies. The nominal thicknesses include the metal foil except microwave materials, which are without foil. See Table 2.8.

d. Warp and Twist Percent Variation. Warp and twist values are per MIL-P-13949E. The values apply only to sheet sizes as manufactured and to cut pieces having either dimension not less than 18 in. For intermediate thickness, the next greater thickness value applies. See Table 2.9.

TABLE 2.6 Flexural Strength-Condition A (Minimum average psi)

Material	Lengthwise	Crosswise
XXXP	12,000	10,500
XXXPC	12,000	10,500
FR-2	12,000	10,500
FR-3	20,000	16,000
Epoxy-paper or glass	50,000	35,000
FR-4	60,000	50,000
FR-5	60,000	50,000
FR-6	15,000	15,000
G-10	60,000	50,000
G-11	60,000	50,000
G-1	15,000	10,000
CX	15,000	10,000
Polyimide	50,000	40,000

e. Warp and Twist Variation, Cut Panels. Warp and twist of cut panels with either dimension less than 18 in are per MIL-P-13949E; see Table 2.10. For intermediate thickness, the next greater thickness value applies.

24. Effects of Environmental Change These properties are listed for the mechanical factors which might be affected in a board design by the changing environment.

TABLE 2.7 Base Material Weight

Material	$\frac{1}{8}$ -in	$\frac{1}{16}$ -in	$\frac{3}{32}$ -in	$\frac{1}{4}$ -in
XXXP	3.3	6.7	10	13.4
FR-3	3.6	7.2	11	14.5
G-10	4.7	10.2	14.1	14.5
G-11	4.5	9.8	13.4	19.6
FR-4	5.0	10.0	15.0	20.0
FR-5	5.2	10.2	15.4	20.4

TABLE 2.8 Thickness Tolerance, in Inchas, per MIL-P-13949E

Paper	Class 1 All types		Class 2 All types	Class 3 All types	Class 4 For microwave application
	Class	Class			
0.031 ± 0.0045	0.031 ± 0.0065	0.031 ± 0.0040	0.031 ± 0.0030	0.030 ± 0.0020	
0.062 ± 0.0060	0.062 ± 0.0075	0.062 ± 0.0050	0.062 ± 0.0030	0.060 ± 0.0020	
0.093 ± 0.0075	0.093 ± 0.0090	0.093 ± 0.0070	0.093 ± 0.0040	0.090 ± 0.0030	
0.125 ± 0.0090	0.125 ± 0.0120	0.125 ± 0.0090	0.125 ± 0.0050	0.120 ± 0.0035	
0.250 ± 0.0120	0.250 ± 0.0220	0.250 ± 0.0120	0.250 ± 0.0060	0.240 ± 0.0040	

TABLE 2.9 Warp and Twist per MIL-P-13949E

Thickness, in	Total variation, maximum, percent (on basis of 36-in dimension)					
	Class A			Class B		
	All types, all weights metal (one side)	All types, all weights metal (two sides)	Paper	All types, all weights metal (one side)	All types, all weights metal (two sides)	Paper
0.030 or 0.031	12	5	6	10	2	5
0.060 or 0.062	10	5	6	5	1	2.5
0.090 or 0.093	8	3	3	5	1	2.5
0.120 or 0.125	8	3	3	5	1	2.5
0.240 or 0.250	5	1.5	1.5	5	1	1.5

TABLE 2.10 Warp or Twist of Cut-to-size Panels

Thickness, in	Panel size	Total variation, maximum, in/in	
		All types, all weights metal (one side)	All types, all weights metal (two sides)
0.030 or 0.031	8 in or less	0.015	0.005
	Over 8 in to 12 in (incl)		
0.060 and over	Over 12 in	0.015	0.010
	12 in or less	0.020	0.010
	Over 12 in	0.010	0.005

TABLE 2.11 Operating Temperature

Materials	Temp., °C, for	
	Electrical factors	Mechanical factors
Ordinary applications		
XXXP	125	125
XXXPC	125	125
FR-2	105	105
FR-3	105	105
Epoxy-paper or -glass	105	105
FR-6	105	105
G-10	130	140
FR-1	130	140
G-11	170	180
FR-5	170	180
Polyimide	260	260
High-frequency applications		
GT	220	220
GX	220	220
Polystyrene	110	110
Cross-linked polystyrene	100	100

a. Maximum Continuous Operating Temperature. See Table 2.11.

b. Peel Strength (before and after Solder Dip). Method: See Chap. 2, Sec. 11. Unit of value: lb/in of width. See Table 2.12.

c. Relative Humidity on Surface Resistivity of Glass Epoxy. The effect of humidity on the surface resistance of glass-epoxy was measured by using the ASTM three-electrode

TABLE 2.12 Peel Strength

Materials	Condition A		Condition E-1/150	
	1 oz	2 oz	1 oz	2 oz
Ordinary applications				
XXXP	6	7		
XXXPC	6	7		
FR-2	6	7		
FR-3	8	9	5	6
Epoxy-paper or -glass	8	10	5	6
FR-6	7	8		
G-10	8	10	5	6
FR-4	8	10	5	6
G-11	8	10	5	6
FR-5	8	10	5	6
Polyimide	9	10	8	9
High-frequency applications				
GT	8	10	2	3
GX	8	10	2	3
Polystyrene	6.0	7.0		
Cross-linked polystyrene	6.0	7.0		
Adhesive (typically processal)				
Adhesive-coated	Condition A, 1 oz	Sacrificial aluminum	Condition A, 1 oz	
XXXPC	9	FR-2	9	
FR-2	9	FR-3	9	
FR-3	9	FR-1	9	
FR-4	9	Epoxy-paper or -glass	9	
Sealed and coated	Condition A, 1 oz	Swell and etch	Condition A, 1 oz	
XXXPC	11	FR-4	9	
FR-2	11			
FR-3	11			
FR-1	12			

circular pattern, starting with 97.5 percent relative humidity at 40°C and decreasing the humidity to 64 percent. The results, shown in Fig. 2.19, indicate that the surface resistivity decreases logarithmically with an increase in humidity at approximately the rate of one decade per 20 percent humidity change.

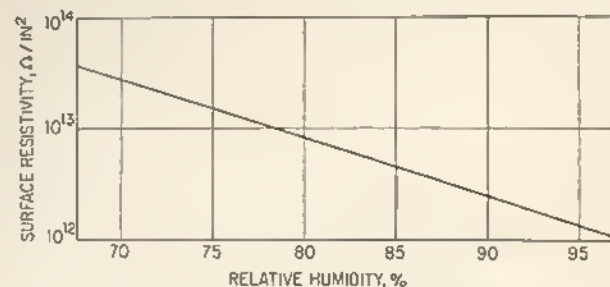


Fig. 2.19 Surface resistivity vs. relative humidity.

25. Coefficient of Thermal Expansion Method: ASTM D696 (at 130°F). Unit of value: in/(in)(°C). The coefficient of thermal expansion, Table 2.13, is the change in length per unit of length per degree change in temperature. The coefficient may vary in different temperature ranges; so the temperature range must be specified.

TABLE 2.13 Coefficient of Thermal Expansion

Materials	Coefficient, × 10 ⁻⁵	
	Lengthwise	Crosswise
Ordinary applications		
XXXP	1.2	1.7
XXXPC	1.2	1.7
FR-2	1.2	2.5
FR-3	1.3	2.5
Epoxy-paper or -glass	1.1	1.7
FR-6	1.0	1.0
G-10	1.0	1.5
FR-4	1.0	1.5
G-11	1.0	1.5
FR-5	1.0	1.5
Polyimide	1.0	1.2
High-frequency applications		
GT	1.0	2.5
GX	1.0	2.5
Polystyrene	7.0	7.0
Cross-linked polystyrene	5.7	5.7

26. Water Absorption Unit of value: percent. Water absorption is the ratio of weight of water absorbed by the material to the weight of the dry material. See Table 2.14.

27. Thermal Conductivity of the Base Material without Copper Unit of value: Btu·h/(ft²)(in thickness)(°F). Thermal conductivity is the rate of heat flow under steady conditions, per unit of area and per unit of temperature gradient, in the direction perpendicular to the measured area. When values are expressed in other units, conversion to the desired units may be calculated by use of Table 2.15:

28. Copper-bond Strength Retention Method: Underwriters Laboratory test, 1344 h at 125°C. Unit of value: percent of retention. Results are summarized in Table 2.16.

29. Fungus Resistance Resistance to various fungi is usually exhibited by lamination with epoxy resins; laminates with phenolic resins are not resistant. See Tables 2.17 and 2.18.

TABLE 2.14 Water Absorption, Condition D 24/23

Materials	1/32-in	1/16-in	3/16-in
Ordinary applications			
XXXXP	1.3	1.0	0.85
XXXXPC	1.3	0.75	0.65
FR-2	1.3	0.75	0.65
FR-3	1.0	0.65	0.60
Epoxy-paper or -glass		0.16	
FR-6		0.40	
G-10	0.50	0.25	0.20
FR-4	0.50	0.25	0.20
G-11	0.50	0.25	0.20
FR-5	0.50	0.25	0.20
Polyimide		1.0	
High-frequency applications			
GT	0.20	0.10	0.09
GX	0.20	0.10	0.09
Polystyrene		0.05	
Cross-linked polystyrene		0.01	

TABLE 2.15 Thermal Conductivity of Base Materials

Material	Conductivity	Material	Conductivity
XXXXP	1.1	G-11	1.8
FR-2	1.8	FR-4	1.8
FR-3	1.6	FR-5	1.8
G-10	1.8		

TABLE 2.16 Copper-bond Strength Retention

Material	Percent	Material	Percent
XXXXP	50	G-11	50
FR-2	50	FR-4	50
FR-3	50	FR-5	50
G-10	50		

TABLE 2.17 Fungus Resistance Tests

Test organisms	Culture no.	Military specification
<i>Aspergillus niger</i>	WADC 215-4247	MIL-F-8261
<i>Aspergillus flavus</i>	WADC 26	MIL-F-8261
<i>Trichoderma T-1</i>	WADC T-1	MIL-F-8261
<i>Chaetomium globosum</i>	USDA 1042.4	MIL-E-4970
<i>Aspergillus niger</i>	USDA 1c 215-4247	MIL-E-4970
<i>Penicillium italicum</i>	USDA 1336.1	MIL-E-4970
<i>Aspergillus fumigatus</i>	WADC 26	MIL-F-4970
<i>Memnoniella echinata</i>	WADC 37	MIL-F-1970
<i>Mycro-Thecium verrucaria</i>	ATCC 9095	MIL-E-5272
<i>Aspergillus terreus</i>	ATCC 10690	MIL-F-5272
<i>Penicillium</i>	ATCC 9849	MIL-E-5272

TABLE 2.18 Fungus Resistance Test Results

Material	Construction	Military specifications		
		MIL-F-8261	MIL-E-4970	MIL-E-5272
XXXXP	Paper-phenolic	Not resistant	Not resistant	Not resistant
XXXXPC	Paper-phenolic	Not resistant	Not resistant	Not resistant
FR-3	Paper-epoxy	Resistant	Resistant	Resistant
FR-4	Glass-epoxy	Resistant	Resistant	Resistant
FR-5	Glass-epoxy	Resistant	Resistant	Resistant

30. Flammability According to Underwriters Laboratories, materials tested for flammability are classified 94V-0, 94V-1, and 94V-2. Definitions of those classifications, as tested by UL flammability procedure, are outlined below.

94V-0 Specimens must extinguish within 10 s after each flame application and a total combustion of less than 50 s after 10 flame applications. No samples are to drip flaming particles or have glowing combustion lasting beyond 30 s after the second flame test.

94V-1 Specimens must extinguish within 30 s after each flame application and a total combustion of less than 250 s after 10 flame applications. No samples are to drip flaming particles or have glowing combustion lasting beyond 60 s after the second flame test.

94V-2 Specimens must extinguish within 30 s after each flame application and a total combustion of less than 250 s after 10 flame applications. Samples may drip flame particles, burning briefly and no specimen will have glowing combustion beyond 60 s after the second flame test.

94H-b This classification is outlined in Table 2.19.

TABLE 2.19 Flammability Classifications

Grade	UL classifications	Grade	UL classifications
XXXXPC	94HB	G-10	94ttB
FR-2	94V-0	FR-4	94V-0 or 94V-1
FR-3	94V-0	G-11	94HB
Epoxy-paper or -glass composite	94V-0	FR-5	94V-0
		FR-6	94V-0

31. Dielectric Strength (Perpendicular to the laminations at 23°C) Method: ASTM D 149. Unit of value: V/mil. Dielectric strength is the ability of an insulation material to resist the passage of a disruptive discharge produced by an electric stress. A disruptive discharge is measured by applying 60-Hz voltage through the thickness of the laminate, as shown in Fig. 2.20. All the tests are run under oil. In the short-time test, the applied voltage is

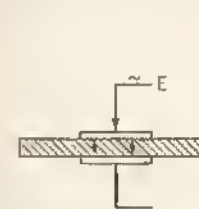


Fig. 2.20 Dielectric strength test.

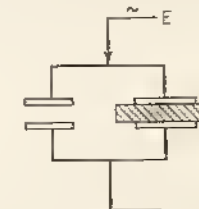


Fig. 2.21 Dielectric constant test.

increased at a uniform rate of 0.5 kV/s. In the step-by-step test, the initial voltage is 50 percent of the short-time breakdown voltage; then the voltage is increased in increments according to a predetermined schedule at 1-min intervals. The test values for dielectric strength vary with the thickness of material, the form and size of electrodes, the time of application of the voltage, the temperature, the frequency and waveshape of the voltage, and the surrounding medium. Step-by-step data for 1/16-in-thick material are as shown in Table 2.20.

TABLE 2.20 Dielectric Strength Data

Material	V/mil	Material	V/mil
XXXXP	740	G-11	600
FR-2	740	FR-4	500
FR-3	550	FR-5	500
G-10	510	Polyimide	750

32. Dielectric Breakdown (Parallel to the laminations at 23°C) Method: ASTM D 149. Unit of value: kV. Condition D 48/50. Dielectric breakdown is the disruptive discharge measured between two electrodes (Pratt and Whitney No. 3 taper pins) inserted in the laminate on 1-in centers perpendicular to the laminations. All tests are run under oil. The short-time and step-by-step tests are performed as in the test for dielectric strength perpendicular to laminations. Step-by-step data on 1/8-in-thick material (type kV) are given in Table 2.21.

TABLE 2.21 Dielectric Breakdown Data

Material	Kilovolts	Material	Kilovolts
XXXP	15	FR-4	45
XXXPC	15	G-11	45
FR-2	15	FR-5	45
FR-3	30	GT	20
FR-6	40	GX	20
G-10	45		

33. Dielectric Constant Method: ASTM D 150. Unit of value: dimensionless. Dielectric constant is the ratio of the capacitance of a capacitor with a given dielectric to the capacitance of the same capacitor with air as a dielectric, as illustrated in Fig. 2.21. The dielectric constant is a measure of the ability of an insulating material to store electrostatic energy. It is calculated from the capacitance as read on a capacitance bridge, the thickness of the specimen, and the area of the electrodes. It varies with temperature, humidity, and frequency. See Table 2.22.

TABLE 2.22 Dielectric Constants

Materials	Dielectric constant, at 1 MHz, under	
	Condition D 24/23	Condition D 48/50
	Ordinary applications	
XXXP	4.1	4.5
XXXPC	4.1	4.5
FR-2	4.5	4.5
FR-3	4.3	4.8
Epoxy-paper or -glass	4.11	5.0
FR-6	4.1	4.1
G-10	4.6	4.3
FR-4	4.6	4.3
G-11	4.5	4.5
FR-5	4.3	4.5
Polyimide	4.8	5.0
	High-frequency applications	
GT	2.8	2.8
GX	2.8	2.8
Polystyrene	2.5	2.5
Cross-linked polystyrene	2.6	2.6

34. Dissipation Factor (Average at 1 MHz) Method: ASTM D 150 (see Fig. 2.22). Unit of value: dimensionless. In an insulating material, the dissipation factor is the ratio of the total power loss, in watts, in the material to the product of the voltage and current in a capacitor in which the material is a dielectric. It varies over a frequency range. See Tables 2.23 and 2.24.

35. Current-Carrying Capacity as a Function of Ambient Temperature Unit of value: A. The design chart, Fig. 2.23, has been prepared as an aid in estimating temperature rises (above ambient) vs. current for various cross-sectional areas of etched copper conductors. It is assumed that normal design conditions prevail when the conductor surface area is relatively small compared with the adjacent free panel area. The curves as presented include a nominal 10 percent derating (on a current basis) to allow for normal variations in etching techniques, copper thickness, conductor width estimates, and cross-sectional area.

Additional derating of 15 percent is suggested under the following conditions:

- For panel thickness of 1/8 in or less
- For conductor thickness of 0.0042 in (3 oz/ft²) or greater.

For general use, the permissible temperature rise is defined as the difference between the maximum safe operating temperature of the laminate and the maximum ambient temperature at which the panel will be used. For single-conductor applications, the chart may be used for determining conductor widths, conductor thickness, cross-sectional area, and current-carrying capacity for various temperature rises. Remember to calculate cross-sectional area with the final plating thickness.

For groups of similar parallel conductors, if closely spaced, the temperature rise may be found by using an equivalent cross section and an equivalent current. The equivalent cross section is equal to the sum of the cross sections of the parallel conductors, and the equivalent current is the sum of the currents in the conductors.

The effect of heating due to attachment of power-dissipating parts is not included. The conductor thicknesses in the design chart do not include conductor overplating with metals other than copper.

TROUBLESHOOTING OF LAMINATE PROBLEMS

36. Need for Traceability It is impossible to build circuit boards in any quantity without having some difficulties that will be blamed principally on the copper-clad laminate

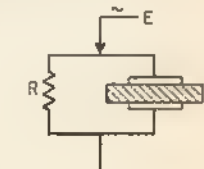


Fig. 2.22 Dissipation factor test.

TABLE 2.23 Dissipation Factors

Materials	Dissipation factor, at 1 MHz, under	
	Condition A	Condition D 24/23
	Ordinary applications	
XXXP	0.028	0.03
XXXPC	0.028	0.03
FR-2	0.024	0.026
FR-3	0.024	0.026
Epoxy-paper or -glass	0.027	0.028
FR-6	0.020	0.028
G-10	0.018	0.019
FR-4	0.018	0.020
G-11	0.019	0.020
FR-5	0.019	0.028
Polyimide	0.020	0.030
	High-frequency application	
GT	0.005	0.006
GX	0.002	0.002
Polystyrene	0.00012	0.00025*
Cross-linked polystyrene	0.0004	0.0005†

*Condition A, 10 MHz

†Condition A, 10 GHz

TABLE 2.24 Dielectric Constant and Dissipation Factor of FR-4, Condition D 24/23

Frequency	Dielectric Constant	Dissipation Factor
100 Hz	4.80	0.009
1,000	4.75	0.012
10,000	4.70	0.015
100,000	4.65	0.018
1 MHz	4.60	0.020
10	4.55	0.022
100	4.50	0.024
1,000	4.45	0.025
10,000	4.40	0.025

material. Too often, the base material appears to be the cause of trouble when actually the fabrication process is out of control. Even a carefully written and executed laminate specification will fail to specify the tests necessary to help identify the laminate as cause of or contribution to a process problem. Listed in this section are some of the most common laminate problems and how to recognize them.

Once a laminate problem is encountered, it should be considered for addition to the material specification. Often the addition, if not made, allows continuing variations and subsequent rejections. Usually any material problem traced to laminate variations will occur in discrete raw material or press load batches manufactured by the laminator. Few users keep records extensive enough to permit identification, in any processing area, of a particular press load or batch of material. So usually the boards continue to be manufac-

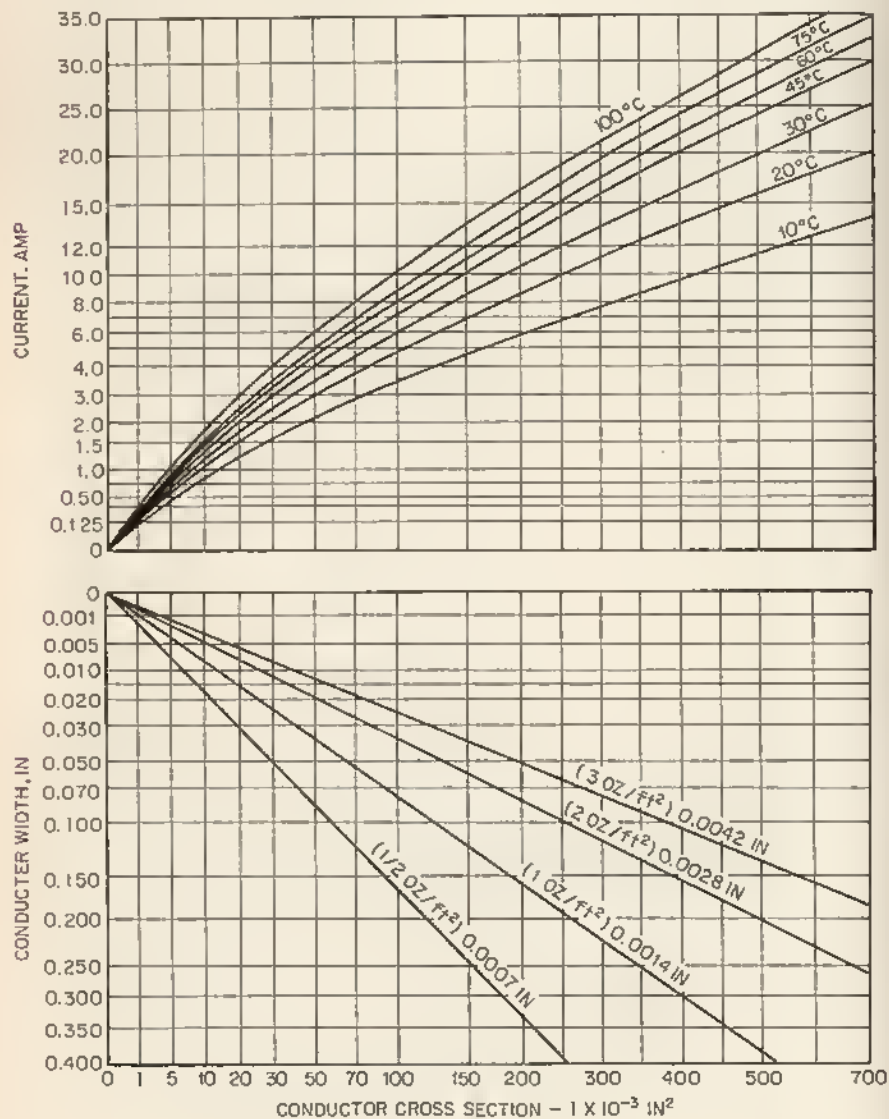


Fig. 2.23 Conductor cross section and current capacity.

tured and loaded and, for example, warpage in the solder pot continues and a large amount of labor and expensive components are lost. If the load number is immediately known, the laminator can check resin batches, copper lots, cure cycles, etc. If, that is, the user does not provide continuity with the laminator's quality control system, he is in the long run penalizing himself. The common problems associated with base material in the fabrication process are discussed in the following sections.

37. Surface Problems

INDICATED BY: Lack of ink adherence, lack of plating adherence, inability to etch certain areas, and inability to solder certain areas.

POSSIBLE DETECTION: Usually made visible by water patterns formed on the board surface.

Possible Causes

1. Unclad surface too shiny because release films have imparted a very dense, smooth surface.
2. Mold release not removed by laminator, usually on unclad side of laminate.
3. Pinholes in copper cause resin to flow and plug up on copper surface; usually seen on foils thinner than $\frac{1}{2}$ oz.
4. Excessive antioxidant compound placed on copper surface by the copper source.
5. Change in resin system, release films, or bushing method by laminator.
6. Excessive fingerprints or oils due to improper handling.
7. Machine oils from pinching, blanking, or drilling operations.

Possible Cures

Suggest laminator use textured films or other release materials.

Contact laminator for mechanical or chemical removal methods.

Contact laminator to check offending copper lots; ask for recommendation of solvent method of resin removal.

Ask laminator for removal procedure. Usually, HCl followed by mechanical removal methods will be recommended. Work with laminator and establish user testing before any change is made in laminate manufacture.

Educate personnel in all phases of the process to handle copper-clad with gloves. Check to make sure laminate is properly interleaved or bagged in transit and that the interleaving paper has low sulphur content or the bags are free from contaminants. Make sure no personnel in contact with copper-clad are using lotions that contain silicones.

Degrease all boards before plating or image-transfer process.

38. Appearance Problems

INDICATED BY: Wide laminate color variances, surface color variances, spotting on surface or internally, films of various colors on the laminate surface.

POSSIBLE DETECTION: Visual.

Possible Causes

1. White checks or spots on surface before processing or after etching (glass-base laminates).

Possible Cures

In the extreme case, this is caused by a resin-starved surface leaving the glass cloth exposed, which today is rare. More commonly seen are microblisters or small white voids on the surface. They are due to a reaction of the glass cloth finish in the resin systems. Boards with a large degree of glass cloth exposure sometimes undergo a degradation in surface resistivity with an increase in humidity. However, boards with microvoids or small blisters usually exhibit no such degradation and the problem is strictly one of cosmetics. Work with the laminator to avoid recurrence and establish an internal standard on acceptability of microblisters.

Possible Causes

2. White mottled surface after processing or a large change of glass cloth exposure.

3. White film on surface after processing, particularly soldering, indicates mild resin attack or foreign deposit.

4. Color variance in base material is outside acceptable cosmetic range.

5. Brown or mottled brown appearance in base material caused by overheating the laminate.

39. Machining Problems

INDICATED BY: Inconsistent punching, shearing, drilling, lack of plating adherence, or irregular plating in the plated-through holes.

DETECTION: Incoming inspection to test any critical machine operation and regular sectioning of incoming laminate after the plated-through-hole process.

Possible Causes

1. Variance in material cure, resin content, or plasticizers can affect drilling, punching, and shearing of the material.

2. Poor drilling, punching, or shearing techniques can give poor or irregular fabrication results.

3. Too much time in the preheating cycle prior to punching or drilling can sometimes affect the laminate cure.

4. Aging of material, principally phenolics, sometimes results in plasticizer leaving the material, which makes the material more brittle than normal.

Possible Cures

Most instances of glass cloth exposure after processing indicate a solvent attack that removes some of the surface resin. Check all solvents and plating solutions with the laminator, particularly the time and temperature in each, to ensure that they are compatible with the laminate used. Follow the laminator's recommendation whenever possible.

Check with laminator to ensure that flux used is compatible with his material. Check for possible process steps in which minerals or inorganics could be deposited. Use demineralized water whenever possible.

Work with laminator to ensure that any major laminate construction or resin change which affects color is approved by the user before the change is made. Sometimes excessive copper alloy transfer will affect color. Work with the laminator in establishing the acceptable cosmetic range.

Check the dip-soldering operations, solder temperatures, and dwell time. Check also for hot components on the board or overall board temperature environment. If the latter is in the upper tolerance range of the laminate being used, brownning can occur.

Possible Cures

Work with laminator to establish tests to simulate the final machinability requirements. Production dies should not be used, or wear and change will affect the results. In any machining variance problem, the laminate is suspect only if the problem is coincident with a change in lot or batch of material.

See Chap. 5 for fabrication recommendations of the various types of laminates. Contact the laminator for specific speeds, feeds, drills, and punching temperatures of each grade. Remember that each manufacturer uses different mixes of resins and base materials and that recommendations will vary.

Preheat laminates cautiously. Take care to find any hot spots, such as under heat lamps, and use the first-in, first-out principle when putting the material under heat.

Check with the laminator to get aging characteristics of his material. Rotate the inventory so stock is generally fresh. Be sure to investigate possible overheating while in warehouse storage.

40. Warp and Twist Problems

INDICATED BY: Warp and twist in the base material either before, after, or during process. Processing also can be indicated by tilting of the holes after soldering.

DETECTION: Possible solder-float tests on incoming inspection; the 45° incline solder test is particularly effective.

Possible Causes

1. Material warped or twisted upon receipt or after sawing or shearing—usually caused by improper laminating, improper shearing, or unbalanced construction in the laminate.

2. Warpage can also be caused by improper storage of material, particularly paper-base laminates, so that the material takes a hove or sets as it sits on edge.

3. Warpage caused by unequal copper cladding, such as 1 oz on one side and 2 oz on the other, unequal electroplating, or a particular board design which will cause copper or thermal stressing.

4. Improper jiggling or holding when soldering. Heavy components also can cause warpage during solder operations.

5. Shifting or tilting of holes in material during processing or soldering caused by improper cure of the laminate or stressed glass construction of the base material.

41. Laminate Measling or Delamination

INDICATED BY: White spots or checks on the surface or in the material, either localized or over a large area.

POSSIBLE DETECTION: Proper solder-float test.

Possible Causes

1. Large blistered areas upon soldering, caused by moisture or volatiles pressed into the material. Cause can also be poor machining practices that delaminate the laminate so that it absorbs moisture during the wet processes.

2. White checks or measling that occur during soldering can be caused by unbalanced laminate construction, improper laminate cure, poor stress relief of the laminate, or lack of ductility of the plated copper.

Possible Cures

Straighten material by hand or stress-relieve in an oven according to the laminator's recommendations on rake angle and material temperature for shearing operation. Work with laminator to make sure materials with unbalanced base material construction are not used.

Store the material flat in the shipping cartons or at an angle such that the material lies flat against the storage rack. Material should usually be placed at an angle of 60° or less with respect to the floor.

Work with the laminator to eliminate unequal cladding. Analyze electroplating and stressing or localized stresses caused by heavy component placement or copper areas. Redesign the boards with the balance of components and copper areas in mind. Sometimes boards laid out with most traces on one side perpendicular to the traces on the other will undergo unequal thermal expansion causing twist; eliminate that type of layout whenever possible.

Boards, especially paper-base materials, must be jigged during solder operation. In some cases heavy components must be compensated for with special jigs or fixtures.

Contact the laminator for any recommended postcure available. In some cases, the laminator may recommend a different laminate for more critical or specialty applications.

Possible Cures

Notify the laminator of the lot involved in the problem; may be isolated. Use recommended machining practices on all materials.

Contact the laminator for instructions on how to stress-relieve boards before dip-soldering. Storage of boards for a period of time in high humidity can cause excessive moisture which will affect solderability of boards. Prebaking of boards and preheating of boards before soldering operations to minimize thermal shock can help either of these problems. (See Chap. 21, on multilayer materials, for data on moisture absorption of stored circuit boards.)

Possible Causes

3. Fiber exposure or heavy measling during soldering operation caused by excessive exposure to solvents which soften the resin, particularly chlorinated solvents.

4. Large components or terminals set tightly can cause excessive stress in the material under heat. The effect is measling around that particular area. Stressing, flexing, or bending the material during or immediately after dip-soldering also causes measling.

42. Bond Strength Problems

INDICATED BY: Pads or traces lifting in the process of soldering operation.

POSSIBLE DETECTION: Adequate testing during incoming inspection and careful monitoring of all wet processes.

Possible Causes

1. Pad or trace lifting in the process may be caused by plating solutions, solvent attack, or copper stressing during the plating operation.

2. Punching, drilling, or piercing can lift portions of the pads. This will become apparent in the plating through operation.

3. Pad or trace lifting in the wave- or hand-soldering operation is usually caused by improper soldering techniques or temperatures. Sometimes poor initial bond in the laminate or poor hot-peel strength will contribute to pad or trace lifting.

4. Occasionally the layout of the board will cause pads or traces to lift in the same areas.

5. Retention by components of heat absorbed in the soldering operation can cause lifting of pads.

43. Miscellaneous Soldering Problems

INDICATED BY: Cold solder joints or solder joints with blowholes.

POSSIBLE DETECTION: Regular sectioning of holes before and after dip-soldering to indicate copper stressed areas and incoming inspection of raw material.

Possible Cures

Work with the laminator on best solvents and application times. Be sure all wet processes, particularly solvent, are checked when materials are changed.

Loosen tight terminals in the swaging operation and remove any heat sinks or heavy components before dip-soldering. Check machining operations, particularly punching, to ensure measling is not minor delamination caused by the operation. Ensure the material is properly jigged and not stressed when hot. Do not quench in a cooler flux remover when hot or under stress.

Possible Cures

Give the laminator a complete list of solvents and solutions used, together with times and temperatures in each step. Analyze plating process to see if copper stressing and excessive thermal shock are occurring.

Careful adherence to machining methods recommended later in this book and regular sectioning of plated-through holes can control this problem.

Most pad or trace lifting is caused by lack of personnel education. Failure to check solder pot temperatures or extended dwell time in the solder pot can be the cause of lifting. Pad lifting in the hand-soldering touch-up operation is a matter of using the proper wattage iron and of education in the proper technique. Some laminators now manufacture grades which have increased peel strength at elevated temperatures for critical soldering applications.

If the board layout causes lifting to occur at the same point in each board, the board must be redesigned. This will be true, generally, in areas of heavy copper or on traces with right angles. Sometimes it may occur on long traces because of the difference in the coefficient of thermal expansion.

Whenever possible, remove heavy components from the board entirely or add them after the dip-soldering operation. Careful technique with a low-wattage iron will generally result in less prolonged heat to the base material than if the component were dip-soldered.

Possible Causes

1. Blowholes or cold solder joints are seen after the soldering operation. In many instances, poor plating techniques followed by expansion during the soldering operation will create voids or blowholes in the wall of the plated-through hole. If this occurs during the wet process, volatiles can be absorbed, plated over, and then driven off under the heat of dip-soldering, which will create centers or blowholes.

44. Excessive Dimensional Changes

INDICATED BY: Material out of tolerance or registration after fabrication or soldering.

POSSIBLE DETECTION: Adequate quality control during processing.

Possible Causes

1. No attention is paid to the grain direction of paper-base materials; expansion with the grain is about half that of across the grain. Also, the material does not return to its original dimension after cooling.

2. Localized stresses not relieved in the laminate will sometimes cause irregular dimensional changes during processing.

Possible Cures

Work to eliminate copper stressing (see the Plating section of this book). Expansion in the z axis or thickness direction of the laminate is usually a function of the material and can contribute to fracturing of plated-through holes. Work with the laminator to get his recommendation on materials which expand less in the z axis.

Possible Cures

Educate fabrication personnel to always cut boards so the grain runs in the same direction. If the dimensional changes cannot be tolerated, consider changing to glass-base materials.

Contact the laminator for his recommendations on how to stress-relieve the material before processing.

Circuit Components

G. L. GINSBERG
 General Atomics Corp. Philadelphia, Pennsylvania

Introduction	3-1
Components	3-2
1. Selection Criteria	3-2
2. Circuit Components	3-3
3. Multiple-Lead Component Circuit Design Considerations	3-3
4. Adjustable Components	3-4
5. Sockets	3-4
6. Bus Bars	3-4
7. Miscellaneous Component Types	3-4
Interconnection Devices	3-5
8. Contact Types	3-6
9. Termination Types	3-6
10. Specifications and Standards for Connectors	3-8
Hardware	3-10
11. Board-Mounting Hardware	3-11
12. Component Mounting	3-12
13. Heat Sinks	3-14
14. Test Points	3-14

INTRODUCTION

The electrical circuit components and mechanical hardware used in printed wiring assemblies vary in both type and shape, Fig. 3.1. Often the selection of such components is limited by specification, availability, and/or cost. However, many commonly available components, such as resistors and capacitors, can be obtained in several configurations, which allows the printed wiring designer some freedom in selecting or specifying the component size, shape, and configuration.

The following text discusses the basic features of printed wiring components and hardware and provides guidelines for their selection.

COMPONENTS

1. Selection Criteria In order to design an efficient and economical assembly, the printed wiring designer should take into account the following component selection criteria:

a. Component Size and Shape. Profile (the height above the printed wiring mounting surface) and printed wiring mounting area are initial considerations to be applied to all circuit components. The parameters govern the placement and density of components on the board and the mounting relation among boards installed side-by-side within an equipment.

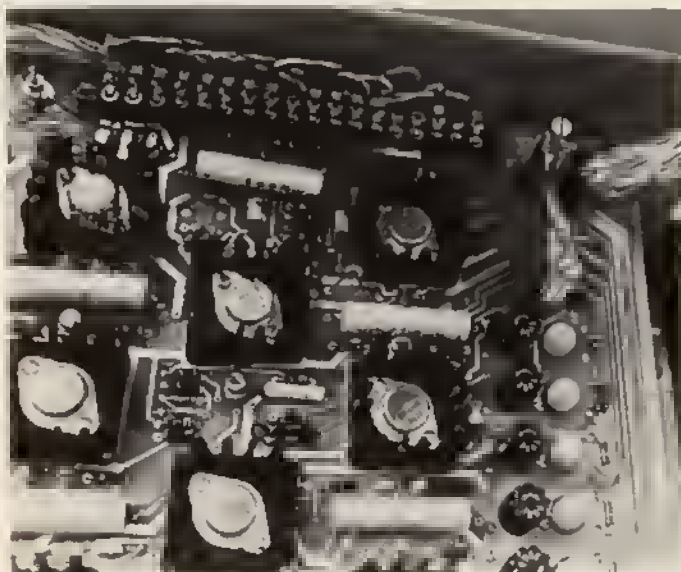


Fig. 3.1 Printed-wiring components and hardware (Winchester Electronics, Oakville, Conn.).

b. Lead Size and Spacing. Lead size and shape are key parameters affecting printed wiring board layout and assembly. Lead size is a prime factor in determining hole size (both drilled and plated) and terminal area size. These in turn relate with lead spacing to determine allowable conductor-routing path locations.

c. Mechanical Tolerances. Proper tolerances on size and shape (for both the component body and its leads) are important to assure that the board and components can be assembled cost-effectively and with a minimum of difficulty. For components with flexible leads, tolerances on lead size and location are not normally troublesome, although assembly time may be more than that associated with rigid leads. The relative magnitude of this consideration also depends on the assembly method used, because compensations that can be made during manual assembly are not possible with high-speed automatic assembly equipment.

d. Component Mounting. Whether the component is self-supporting, has built-in mounting provisions (i.e., threaded inserts or studs), or requires special mounting hardware (such as clamps or clips) is an important consideration. A general rule of thumb is that components weighing $\frac{1}{4}$ oz or more per lead should have a mechanical means of support to ensure that the lead-termination joint is not relied upon for component support.

e. Thermal. The heat-dissipation properties and provisions of a component are sometimes of significant interest. When small quantities of heat are involved, radiation of heat from the component body and/or leads and conduction of heat through leads and mounting brackets, lugs, etc. are usually sufficient to maintain correct operating temperatures. When larger amounts of heat are to be dissipated, heat sinks can be used with most components to remove heat more efficiently.

2. Circuit Components

a. Axial-Lead Components. Perhaps the most common type of printed wiring component is the axial lead component, Fig. 3.2. It usually is cylindrical in shape and has a lead exiting from each of its ends along its neutral axis. The lead is usually round in cross-section; the body is most often formed by molding or dipping; and axial-lead components are the type most suited for automatic insertion. Many resistors, capacitors, and diodes are supplied in this configuration.

b. Radial-Lead Components. Radial-lead components are used in many printed wiring assemblies. This type of component has all of its leads exiting from a common side of the component. The actual body shape is variable; two common types of radial lead components are dipped capacitors and transistor TO cans.

c. Multiple-Lead Components. One of the major advantages of using a printed wiring structure for component mounting and interconnection is the suitability of the structure for use with multiple-lead components, usually integrated circuits (ICs). Multiple-lead components are packaged in several sizes and shapes each of which has distinct advantages and disadvantages when compared with the others.



Fig. 3.2 Axial-lead component.

3. Multiple-Lead Components: Circuit Design Considerations

a. Package Types

1. One of the smallest of the presently existing multiple-lead component types is the flat pack, Fig. 3.3. The body of such a component can be as small as $\frac{1}{8}$ in wide, $\frac{1}{4}$ in long, and $\frac{1}{32}$ in thick. The component leads are normally flat ribbons mounted on 0.050-in centers. Flat packs are presently available with up to 50 leads.

2. An outgrowth of the radial-lead transistor can is the multiple-lead can. This type of component consists of a hermetically sealed can with up to 12 round leads exiting from the bottom of the device (usually in a circular pattern).

3. The dual-in-line (DIP) type of multiple-lead component, Fig. 3.4, closely resembles a large flat pack with its leads formed at a right angle after they exit from the component body. DIP leads exit from the body in a ribbon form but are usually shaped into a V or reduced in size prior to entering the printed wiring mounting hole. The DIP body can be plastic or ceramic. DIP devices are presently available with up to 50 leads on 0.100-in centers.

b. Package Selection. Some major considerations for the selection of a multiple-lead component configuration are the following.

1. Circuit speed has a dominant influence on the size, spacing, and tolerances associated with multiple-lead packages. Circuit conductor lengths and layout relations to other components become critical at speeds of 10 ns and faster.

2. Reliability, serviceability, and environment dictate the need for device hermeticity and ruggedness, and they affect lead configuration and suitability for soldering, welding, or bonding as well. The component mounting method also is dictated.

3. Quantity usage, especially production, determines the degree of assembly tooling and mechanization to be used. Some devices are better suited for high-volume or mechanized applications than others.

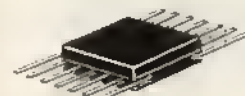


Fig. 3.3 Flat-pack multiple-lead component.

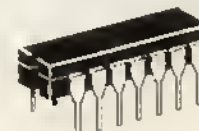


Fig. 3.4 Dual-in-line component.

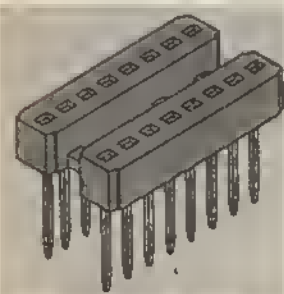
4. Printed board area usage greatly affects the number of ICs per board assembly and quite often the number of assemblies in an equipment. Close multiple-lead-component spacing affects terminal area size, conductor size, conductor spacing, number of conductor layers, and overall fabrication tolerances. Lead spacing also affects those selection parameters.

5. Heat-dissipation characteristics of the package and package suitability with heat sinks must be considered. High component densities readily create hot spots even though heat dissipation per component may appear to be minimal.

c. **Board Layout Considerations.** The following general considerations should be taken into account when printed wiring assemblies with multiple-lead components are designed:

1. The physical dimensions of the multiple-lead component
2. The limitations placed on the board layout area by mechanical or electrical requirements
3. Fabrication restraints and/or requirements affecting hole size and spacing
4. Terminal area size, lead forming, and lead clinching
5. Distribution buses for ground and voltage(s)
6. Artwork size, registration, and tolerances
7. Photography, glass master, and/or screen registration tolerances
8. Registration tolerances for board screening, etching, plating, drilling, and/or punching
9. Automatic, semiautomatic, and manual component insertion tolerances and restraints
10. Component dimensions and tolerances

4. Adjustable Components Adjustable components (usually resistors) are common to many printed wiring assemblies. These devices meet the description provided for radial-lead components but have the additional provision of an adjustment screw. Access to the adjustment feature can be via an exposed edge of the printed board, when a right-angle adjustable component is used. Access to the adjustment feature not on the exposed edge of the printed wiring assembly is achieved with the straight-through type of component.



5. Sockets Some printed wiring applications call for the frequent insertion and removal of some or all circuit components. Printed board component sockets have been developed for that reason. There are appropriate sockets for almost all printed wiring circuit components, including those for DIPs, Fig. 3.5.



Fig. 3.5 Multiple-lead sockets.

6. Bus Bars Board bus bars are constructed with two or three copper conductors (usually tin- or tin-lead-plated) laminated together with a thin dielectric separator. The exterior is generally covered with a plastic barrier material to prevent accidental shorting. The dimensions of the bars depend on their positions on the board. Bus bars that mount under DIPs are slightly less in width than the pin spacing. Bars that stand vertically have widths corresponding to the maximum height of mounted components. Bar length is determined by circuit board dimensions. A typical bus bar configuration for mounting under DIPs is shown in Fig. 3.6.

7. Miscellaneous Component Types There are many other types of printed wiring components which have both electrical and mechanical functions; among them are standoff terminals, relay cans, and switches.

INTERCONNECTION DEVICES

One of the major advantages of using printed board structures, as opposed to other types of component mounting and interconnecting methods, is ease of maintainability through plug-in usage. Board connectors consist of the following types for almost all applications:

1. One-part (card edge)
2. Two-part (plug-and-receptacle assemblies)
3. Discrete contact (plug and/or receptacle)

One-part board connectors, the most common type, use one edge of the printed board as the plug dielectric and printed and plated conductors as the contacts. The other half of the connector is usually an assembly of mating contacts in a chassis-mounted receptacle assembly, Fig. 3.7.

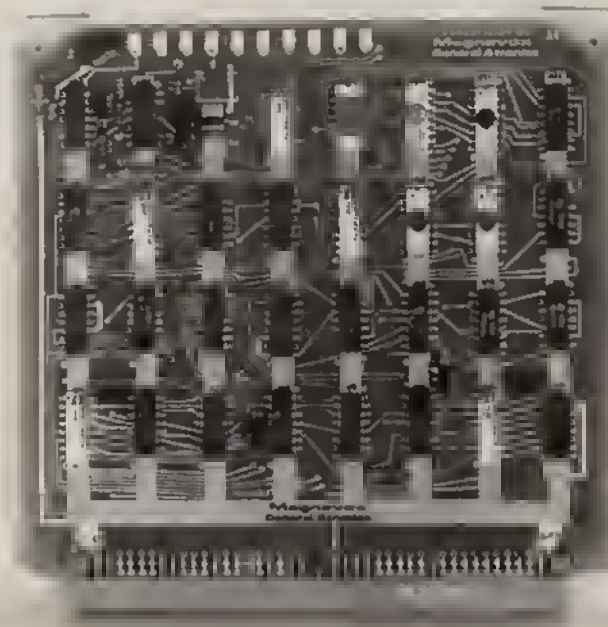


Fig. 3.6 Typical bus bar configuration (General Atomic Company).

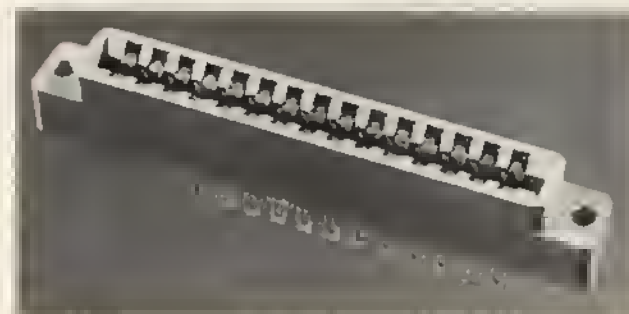


Fig. 3.7 Chassis-mounted receptacle assembly.

Two-part board connectors consist of self-contained multiple-contact plug-and-receptacle assemblies. Usually, although not always, the plug (male) contact assembly half of the connector mounts to the plug-in assembly (Fig. 3.8*a*) and the receptacle (female) half of the connector mounts to an interconnection wiring panel (mother board) or plate (Fig. 3.8*b*).

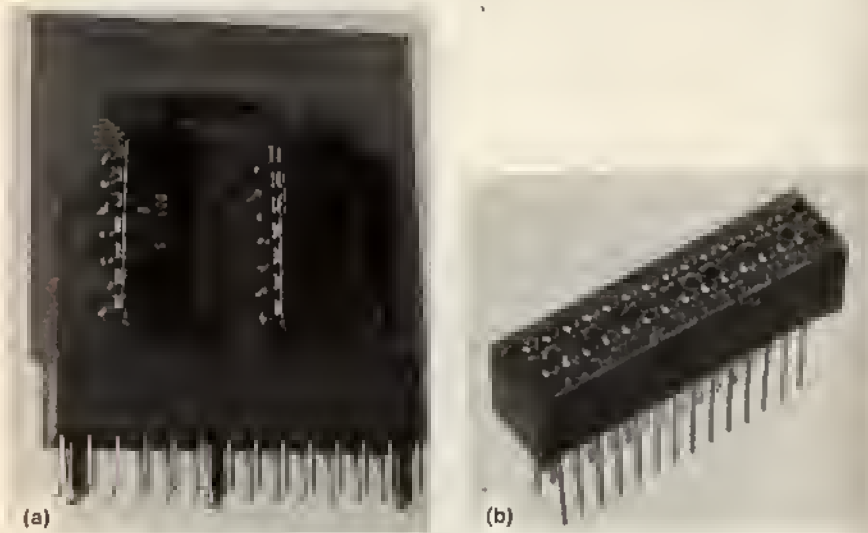


Fig. 3.8 (a) Two-part connector, printed board assembly (with plug half); (b) two-part connector (receptacle half).

The three basic disadvantages of using one-part and two-part board connectors are as follows:

1. Connector location is usually limited to one edge of the board assembly.
2. The direction of insertion (plug-in) of the board assembly is usually limited.
3. The plug and/or receptacle halves are usually limited to a specific number of contacts based on the chosen connector.

When those disadvantages cannot be overcome by acceptable restraints on the design, the use of discrete (plug and/or receptacle) contact connectors is recommended.

8. Contact Types In addition to the three major distinctions among printed wiring connectors just described, a secondary description of a connector relates to the contact types. The most common printed board connector contact types (Figs. 3.9 and 3.10) are:

1. One-part:
 - a. Bellows contact (Fig. 3.9*a*)
 - b. Tuning fork contact (Fig. 3.9*b*)
 - c. Cantilever contact (Fig. 3.9*c*)
2. Two-part discrete contact
 - a. Pin-and-socket contact (Fig. 3.10*a*)
 - b. Blade-and-fork contact
 - c. Hermaphroditic contact (Fig. 3.10*b*)

9. Termination Types The one-part connector and the non-plug-in half of the two-part and discrete-contact connectors are available in several termination types.

a. Solder Terminations. For solder termination interconnection wiring the connector can be soldered directly to a (mother) board by hand-, wave-, or dip-soldering.

When discrete hook-up wiring is to be soldered to the contact terminal, various eyelets, tabs, and tongues are used. An example is shown in Fig. 3.11.

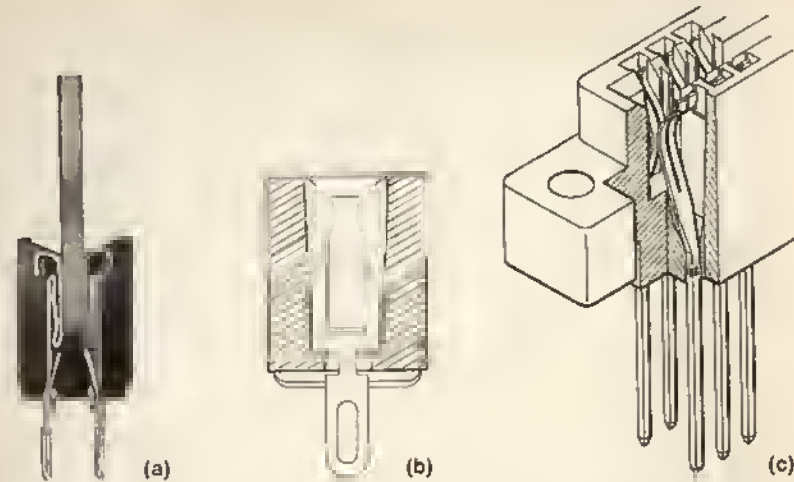


Fig. 3.9 One-part connectors: (a) bellows contact; (b) tuning fork contact; (c) cantilever contact.

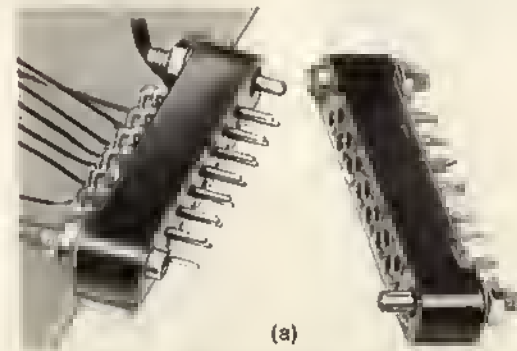


Fig. 3.10 Two-part connectors: (a) pin-and-socket contact; (b) hermaphroditic contact.

b. Solderless (Crimp) Terminations. Many discrete wire hook-up terminations are made to printed board connectors with the crimp method. Crimp terminations are made by either crimping the bare portion of the hook-up wire to the contact portion of the connector or to a tab or pin which mates with the contact terminal. In the former instance the wire becomes a part of a connector assembly; in the latter instance the crimped tabs become a part of the interconnection wiring harness.

c. Solderless (Mechanized) Terminations. Increasing in popularity are solderless printed board connector terminations made by numerically controlled automatic and semiautomatic machines and hand guns. In these applications the connector terminal is usually a solid-metal rectangular-cross-section post. The connection between the post and the hook-up wire can be made by tightly wrapping the bare portion of the wire around the post (solderless wrap), as in Fig. 3.12 or by restraining the wire against the post using a suitable clip.

10. Specifications and Standards for Connectors Although there are many specifications and standards pertaining to most aspects of printed boards, there are few specifications



Fig. 3.11 Solder eyelet contact terminal.

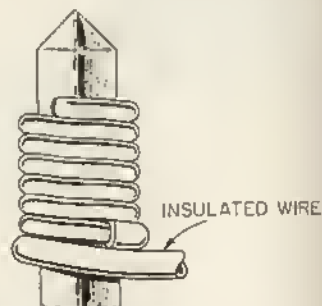


Fig. 3.12 Solderless wrap termination.

and standards covering printed board connectors. The latter are covered briefly in the following paragraphs.

ii. ANSI C 83.88 (IPC-C-405A/EIA-RS-406; General Document for Connectors, Electric, Printed Wiring Board). Document ANSI C 83.88 defines the recommended requirements for multiple-contact electric connectors for the purpose of (1) adapting electric conductors to printed boards and (2) interconnecting one printed board to another. It is intended that only the requirements that apply to a particular application need be tested.

For the benefit of designers and engineers who will use it, document ANSI C 83.88 provides a checklist, Table 3.1, to help in determining the exact requirements of a printed board connector. The four classes of connectors are described in Table 3.2.

b. MIL-C-21097 (Connectors, Electrical, Printed Wiring Board, General Purpose). Specification MIL-C-21097 covers multiple-contact connectors for $\frac{1}{16}$ -, $\frac{1}{32}$ -, and $\frac{1}{8}$ -in printed boards and for interconnections between printed boards. The connectors are intended to provide a reliable connection between printed and conventional wiring. The interconnections are intended to provide connection of printed wiring on one board to printed wiring on another board.

The supplement to this specification contains information concerning specification sheet connectors in accordance with the requirements of this specification. The basic details of MIL-C-21097 specification sheet connectors are listed in Table 3.3.

v. MIL-C-55302 (Connectors, Printed Circuit Subassembly and Accessories). Specification MIL-C-55302 covers connectors (plugs and receptacles) for printed circuit subassemblies and their accessories for use with single-sided printed wiring, double-sided printed wiring, and multilayer printed wiring conforming to MIL-STD-275, MIL-P-55110, and MIL-P-55124, respectively.

The supplement to this specification contains information concerning specification sheet connectors in accordance with the requirements of this specification. The basic details of MIL-C-55302 specification sheet connectors are listed in Table 3.4.

TABLE 3.1 Checklist for Printed Board Connectors

Application: (A brief description of usage, include environmental exposure and temperature ranges.)	
Types:	
Type I:	One-piece connector, dual or single read-out contacts, that make direct electrical and mechanical contact with the printed wiring board.
Type II:	Two-piece connector, dual or single read-out contacts, that mounts directly onto the printed wiring board. Each half of the connector is then mated to the other.
Class: (See Table II):	
Method of Mounting (Where Applicable)	
Through hole	General—
Threaded bushing	Contact spacing
Flanging bushing	
Mounting bracket	No. of contact positions
Force fit	No. of contacts
	Max. current
	Max. mV drop at 1 A
	Type of termination
	Wire size and type
Size: (board thickness)	
Reference Material:	
Required life	
Number of mating cycles	
List any limitations on envelope dimensions	
Polarization	
to position	
Between positions	
References:	
Dielectric material	Why?
Contact material	Why?
Contact plating	Why?
Special requirements:	

TABLE 3.2 Four Classes of Printed Wiring Connectors

	Class I	Class II	Class III	Class IV
Temperature rating	-65 to +150°C	-65 to +125°C	-55 to +105°C	-55 to +85°C
Operating voltage at altitude	50,000 ft	50,000 ft	Not applicable	Not applicable
Insulation resistance	10,000 MΩ initial, 5000 MΩ after testing	2000 MΩ initial, 1000 MΩ after testing	1000 MΩ initial, 250 MΩ after testing	500 MΩ initial, 100 MΩ after testing
Durability	500 cycles	250 cycles	100 cycles	10 cycles
Vibration	5-2000 Hz at 20 g	5-2000 Hz at 15 g	5-500 Hz at 11g	5-55 Hz (0.06-in double-amplitude total excursion)
Humidity	90-95% @10°C 240 h	90-95% @40°C 96 h	90-95% @40°C 96 h	90-95% @40°C 24 h
Mechanical shock	50 g (11 ms)	30 g (11 ms)	30 g (11 ms)	15 g (11 ms)
Contact wear evaluation	5% salt solution for 48 h	5% salt solution for 48 h	Not applicable	Not applicable
Contact resistance at rated current	10 MΩ	10 MΩ	25 MΩ	50 MΩ
Thermal shock, nonoperating	-65 to +150°C	-65 to +125°C	-55 to +105°C	-55 to +85°C
Low-level circuitry	10 MΩ	10 MΩ	Not applicable	Not applicable
Contact identification	Shall be identified	Shall be identified	Not applicable	Not applicable

HARDWARE

For obvious reasons, circuit components and interconnection devices perform a vital function. Often the function of support hardware, i.e., board mounting, component mounting, etc., is not as obvious but is almost as vital in determining the effectiveness of the application of the printed board assembly.

TABLE 3.3 MTL-C-21097 Specification Sheet Connectors

No.	Type	Contact centers, in	Number of contacts	Rows of contacts
1	A and AD—receptacle, card insertion	0.156	6, 10, 12, 15, 18, 20, 22, 28, 30, 36, 43, 44, 56, 72, 86	1, 2
2	CR—receptacle	0.200	12, 15, 18, 22, 26	1
3	CS—plug			
4	CR—receptacle	0.200	17, 23, 29, 35, 41, 47	2
5	CS—male adapter			
6	D—receptacle	0.156	6, 10, 12, 15, 18, 22	1
7	C—male adapter			
8	D—receptacle	0.156	15, 22, 30, 43	2
9	D—pin, crimp			
10	D—clip			
13	CR—receptacle	0.200	17, 23, 29, 35, 41, 47	2
14	CS—male adapter			
15	CS—male adapter			
16	CR—crimp contact			
17	CR—solder contact			
18	CR—contact removal tool			
19	CR—blade simulator			
20	AD—receptacle, card insertion	0.050	12, 14, 16, 18, 20, 22, 24, 26, 28, 30, 32, 34, 36, 38, 40, 42, 44, 46, 48, 50	2
21	A and AD—receptacle, card insertion	0.156	6, 10, 12, 15, 18, 20, 22, 28, 30, 36, 43, 44, 56, 72, 86	1, 2

11. Board-Mounting Hardware A wide variety of hardware exists for mounting the printed board assembly in the end-product equipment; the range is from complete packaging systems, Fig. 3.13, to individual board hardware, Fig. 3.14. Between the extremes there is hardware for locally mounting a group of assemblies, such as the stacking-spacing hardware shown in Fig. 3.15. The exact hardware to be used for a specific application depends on factors too numerous to mention here, but the broad selection that is available assures success in finding the proper board-mounting hardware for nearly any possible requirement.

12. Component Mounting The shock and vibration to which printed wiring mounted components are subjected during normal handling and environmental testing can damage the lead terminations and lead-to-component body seals. For that reason many printed wiring mounted components, especially those weighing more than 1/4 oz per lead, should be mechanically secured to the mounting base prior to lead termination and during assembly. The more commonly used component-securing methods are: (a) clips, elamps, and brackets; (b) wire and elastic straps; and (c) adhesives.

a. Clips, Clamps, and Brackets. The following are the basic requirements which should be adhered to when components are mechanically secured by clips, clamps, or brackets:

1. All clips, elamps, or brackets should be secured to prevent their rotation, as by

TABLE 3.4 MTL-C-55302 Specification Sheet Connectors

No.	Connector type	Contact centers, in	Number of contacts	Rows of contacts
1	Pin, right-angle	0.150	7, 12, 17, 23, 28	2
2	Socket, straight-through			
3	Pin, straight-through			
4	Pin, right-angle	0.100	9, 17, 25, 33, 41	2
5	Pin, straight-through			
6	Socket, straight-through			
7	Pin, right-angle	0.100	13, 25, 35, 49, 61	3
8	Socket, straight-through			
9	Male, right-angle	0.200	23	2
10	Female, straight-through			
12	Plug, straight-through	0.150	31	2
13	Receptacle, right-angle			
14	Plug, straight-through	0.150	41	2
15	Receptacle, right-angle			
16	Pin, right-angle	0.200	7, 9, 11, 15, 19, 23, 37	2
17	Socket, straight-through			
18	Pin, straight-through			
19	Pin, right-angle	0.150	41	2
20	Socket, straight-through			
21	Pin, right-angle	0.100	92	3
22	Socket, straight-through			
23	Plug	0.075	Decade increments 10 through 120, 150, 160, 180	2
24	Receptacle	0.075	Decade increments 10 through 110	2
25	Receptacle	0.075	120, 150, 160, 180	2
26	Plug	0.100	30 through 110	2
27	Receptacle			
33	Receptacle	0.156	7, 11, 15, 19, 23	1
34	Receptacle			
35	Plug			
36	Receptacle	0.100	11, 17, 23, 29, 35	2
37	Receptacle			
38	Plug			
41	Receptacle	0.156	7, 11, 15, 19, 23	1
42	Receptacle	0.200	11, 17, 23, 29, 35	2
43	Pin	0.100	14, 28	2
44	Socket			
45	Pin	0.100	28, 56	2
46	Socket			
47	Pin	0.100	56, 112	
48	Socket			
52	Receptacle, straight-through	0.100	18, 30, 36, 42, 54, 72	2
53	Plug, right-angle			
54	Plug, straight-through			
55*	Plug	0.100	Decade increments 10 through 70 and intermediate positions 14, 24, 44, 54 and 26, 56, 66	2
56*	Receptacle			
57*	Plug			
58*	Receptacle			
59*	Plug	0.100	90, 100, 120	2
60*	Receptacle			
61*	Plug	0.100	Decade increments 10 through 70 and intermediate positions 14, 24, 44, 54, and 26, 36, 56, 66	2
62*	Receptacle			
63*	Plug			
64*	Receptacle			
65*	Receptacle			
66*	Receptacle			

*USAF



Fig. 3.13 Example of complete packaging system (Techmar Corp., Los Angeles, Calif.).

using two fasteners or one fastener and a nonturn device. Holders designed for single-hole mounting should be capable of withstanding a 14-in-oz torque without rotating.

2. Clamps and brackets which require removal in order to replace a component should be secured with a threaded or other nonpermanent fastener unless the subassembly in which they are used is considered to be disposable or nonrepairable.

3. Spring clips which need not be removed during component replacement may be secured with permanent fasteners such as rivets or eyelets.

4. Spring clips should require their positive displacement in order for the component to be removed.

5. The use of twist-type lugs, tabs, or ears and the clipping of glass envelope components should be avoided.

Figures 3.16 to 3.19 show typical applications for the use of clips, clamps, and brackets for securing components in printed wiring.

b. Straps. When an elastic strap is used for mechanical securing, the strap is wrapped over the component body and passed through holes in the mounting base. When wire is used, it is clinched and soldered in the same manner as component leads to terminal areas. When wire is used

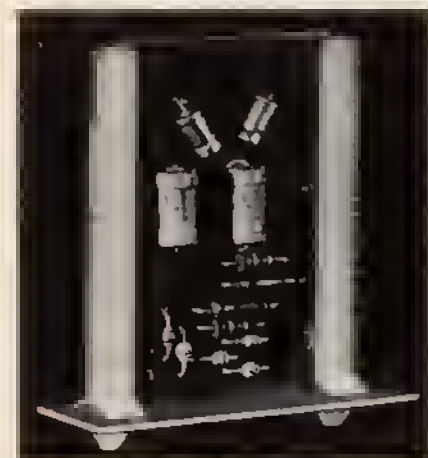


Fig. 3.14 Example of individual board hardware (Richo Plastics, Chicago, Ill.).

with heat-sensitive or fragile components, the part of the wire that touches the component should be covered with a suitable sleeving.

Smaller holes are used with elastic straps than are used with wire. The elastic strap is secured by being stretched to reduce its cross section below that of the hole, and it is returned to its larger than hole size by relieving the tension after it has been passed through the hole. The resiliency of the strap holds the component in place.



Fig. 3.15 Stacking and spacing hardware (Richo Plastics, Chicago, Ill.).

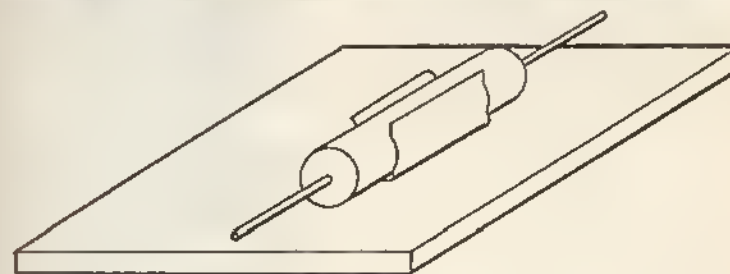


Fig. 3.16 Component holder.



Fig. 3.17 Spring-clip mounting.

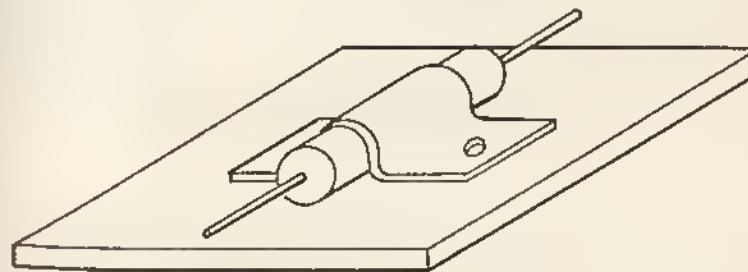


Fig. 3.18 Clip-over mounting.

c. Adhesives. Whenever possible, components should be secured by conventional means; when that is not possible, as in the case of oddly shaped components or when special support is required or when there are special design requirements (limited space, heat transfer, limited access, etc.), a suitable adhesive may be used.

13. Heat Sinks The increasing density of components on printed board assemblies and the use of higher-power components often necessitates the use of board-mounted heat sinks, Fig. 3.20. A heat sink must be considered as a complete thermal system. It consists of four elements: mounting hardware, interfacing materials to bring the mounting surface of the component into intimate contact with the surface of the sink, a surface coating of the metal heat dissipater, and the copper or aluminum thermal dissipater itself. Each element affects overall thermal efficiency, cost, and the frequency of maintenance of the heat sink systems.

Heat sinks range in complexity from a simple, flat metal plate to which a heat-producing component is bolted to a system using thermal feedback sensors to control liquid coolant flowing through tubes surrounding the part. The difference between such thermal dissipaters and the machined metal plate with fingers reaching into and cooled by the ambient atmosphere is that the latter is a passive system that protects a component from thermal destruction at the lowest economically feasible temperature.

For practical purposes, heat sink design must be a compromise. The ideal heat dissipater would encase the device to be

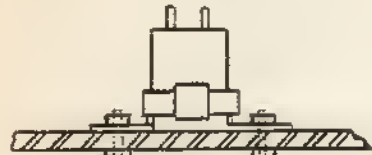


Fig. 3.19 Component cradle riveted to mounting base.

protected in a spherically shaped metal mass consisting entirely of black needlelike protusions radiating directly from the thermal point source. That would provide the most efficient radiating area for a given volume. Although the ideal shape could not be mounted to any flat surface, practical coolers borrow the radial-fin structure from it. When heat sinks of equal volume are compared, the radial-fin coolers are more efficient radiators than are coolers with a series of parallel fins machined or soldered perpendicularly to the surface of the sink. The trade-offs in thermal efficiency, based on shape, are cost, ease of assembly, and installation.

14. Test Points A printed board component which, unlike most others, does not perform an electrical circuit function is the test point. Test points do serve an important maintainability function. This type of printed wiring component can be of the right-angle board-edge type, straight-through type, for local access, or the combination type.

REFERENCES

1. "Printed Wiring Design Guide," Institute of Printed Circuits, Evanston, Illinois.
2. R. L. Peck, New Solutions to PC Packaging Problems Via Bus Bars, *Electronic Packaging and Production*, April, 1973, Kiver Publications, Chicago, Illinois.
3. David Hegarty, Treat the Heat Sink as a System, *Electronic Products*, January 21, 1974, United Technical Publications, Garden City, New York.

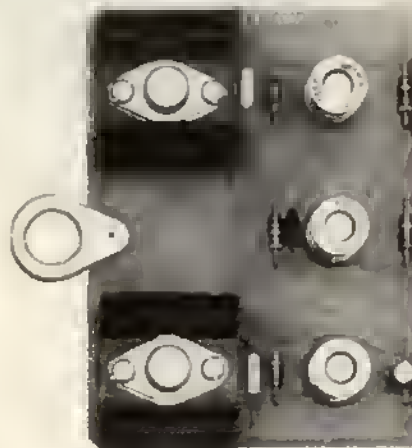


Fig. 3.20 Board-mounted heat sink (Thermalloy Inc., Dallas, Texas).

Chapter 4

Specifications and Purchasing

DIETER W. BERGMAN

Aeroneutronic-Ford, Willow Grove, Pennsylvania

Introduction	4-1
1. Procurement Practices	4-2
2. Prototype Kits	4-2
3. Limited Production Quantities	4-2
Purchasing Guidelines	4-3
4. Evaluation of Supplier	4-3
5. The Make-or-Buy Question	4-3
6. The Purchase Order and Legal Remediations	4-4
7. The Technical Specification	4-5
Specifications	4-5
8. Sample Specification	4-5
Vendor-User Relations	4-6
9. Multilayer Specification Items	4-7
10. Steps in Buying Satisfactory Boards	4-7
Cost Factors	4-9
11. One- and Two-Sided Board Pricing	4-9
12. Multilayer Board Pricing	4-10
13. Delivery Schedules	4-10
Product Definition	4-11
14. Types of Documentation	4-11
References	4-13

INTRODUCTION

The procurement of printed wiring (printed circuit) boards requires that three basic items be understood and accomplished prior to the start of purchasing activity:

1. Knowledge of the environment to which the boards will be subjected

2. Proper evaluation of the capabilities of the board fabricator.

3. Appropriate documentation which clearly defines the requirements of the product. When the above have been properly assimilated, so that the documentation is complete and does not overspecify or underspecify the printed wiring product, the vendors who have been deemed capable of fabricating the particular board may be contacted to obtain competitive pricing.

Printed board products run the gamut from the very simple single-sided variety to sophisticated multilayered versions. Within each category there are many variations. A board whose end-product use is to provide circuitry for a home entertainment product does not require the critical handling, testing, or protection that may be necessary for a board of similar complexity that will see service on a military ocean-going vessel. End-product service considerations make the difference in the cost of the printed board. Too many times in the past, vendor-user relations have been disappointing to both parties either because specifications for a board were too stringent and the cost was higher than expected or because the specifications were too loose and the product did not perform as required. Either failure has a detrimental effect on vendor-user relations.

Just as many types of boards are required by the industry, so there are many types of fabricators. A board-purchasing activity must understand that not every fabricator is capable of producing satisfactory results. That is not meant to imply that the capability scale goes in only one direction and the high-quality house is capable of making all of the various types of printed boards. The opposite is usually true. A fabrication facility that is geared to producing a high-reliability, high-quality product sometimes finds it difficult to compete in making a board that is less complex.

With all those considerations to be taken into account, a company that requires outside services for the fabrication of printed wiring products must have knowledgeable personnel assigned to the procurement of the products.

1. Procurement Practices¹ Historically, the printed circuit is made at home in the high-volume production field. There the economy and reproducibility permitted by the automatic and semiautomatic methods is a dominant factor. That leaves a great many military and industrial projects on the fringes. Although the crossover point varies with board design and specifications, between 500 and 1000 units is typical to justify design and procurement costs. Printed circuit vendors have developed some unique approaches to serve the manufacturer who needs half a dozen prototype boards or perhaps 500 units for a small project.

2. Prototype Kits Starting with a single unit for a prototype or preprototype operation, several firms offer kits that can be used to make circuit boards from specially prepared materials. In addition to the actual material required, the kit normally includes several full-scale layout sheets to aid in designing the board.

As might be expected, the kits reflect several approaches. For example, one company offers its kit in several sizes and a choice of laminate. Conductor runs are applied in the form of tape. Mounting holes are drilled where required. A different approach is one in which the boards contain holes preformed on a 0.100-in grid. The holes, and in fact the whole board, are plated. The desired circuit is either drawn with etch-resist or covered with tape and board-etched.

These do-it-yourself boards can be very helpful, but most prototype and limited-production circuits do not lend themselves to the approach. To meet the limited-volume need, most manufacturers have established separate short-run and prototype departments or ordering procedures. Nearly all vendors acknowledge that some change in procedure must be made for the limited-volume order. Whether a separate department or just expedited handling and simplified paperwork is the best is something that the buyer must determine by contact with individual suppliers.

3. Limited Production Quantities An innovation in the limited-production area is a semi-standard program to reduce the cost. The program is aimed at the 100- to 1000-board market. The quantity range offers some serious challenges in that it is too high for the production techniques often used for short runs but does not justify the cost of fixed tooling. The approach is to limit the selection of mechanical variations while offering full freedom of conductor and hole pattern design. Since in most cases that provides the

designer with all the freedom he needs, the required circuitry can be purchased in limited quantities at reduced price and with quick delivery.

PURCHASING GUIDELINES

Printed wiring poses some special problems because it is unlike electronic components. Although its function is primarily electronic, testing must be largely mechanical.

4. Evaluation of Supplier For long-run, high-volume procurements an evaluation procedure similar to that in MIL-P-55110 or MIL-P-55640 may be practical. Those specifications require that preproduction samples be subjected to a comprehensive series of tests that, according to one supplier, "require two weeks and cost several hundred dollars." After the initial approval, at least one and possibly more samples per day (depending on volume) are given lesser tests. Such a program adds significantly to the cost of any procurement and is prohibitive for short-run commercial projects. Many buyers will find it desirable to trade off some of this "MIL" inspection against a vendor's reputation.

In view of the complications, buying short runs by competitive bid may be overly expensive. More practical may be the procedure of selecting a vendor or vendors (depending on volume) to handle all orders for a fixed period. That will permit both a regular review of performance and good buying without adding excessive purchasing costs to each small order. It should be noted that the boards should be of a similar type.

Since even with a rigorous test program a vendor's quality performance is hard to ensure (the through tests are destructive), vendors should be selected with extra care. Printed circuits would seem to be an area in which the purchasing evaluation trip is particularly important. A plant visit permits the buyer and other members of the buying team to inspect the facilities and meet the people who will be responsible for the quality.

Buying printed circuits does not require a radically different approach but does demand good procurement practices. The best buying tip for users is to purchase only what is required in design tolerances and parameters. Demands for tight tolerances much beyond operating requirements are unnecessary and only add to costs.

Many designers and buyers seem to forget that printed circuits are primarily electrical components rather than mechanical parts and that the reliability of their electrical function is more important than extremely close mechanical tolerances. It is much harder to specify the processing and cleaning procedures necessary to produce printed circuits of high reliability. Since some of the conditions that may lead to unreliability may not show up until long after the circuit is in use, it is advisable to consider only manufacturers with a long record and a reputation for the production of high-quality boards. Buying on price alone can be extremely expensive in this field.

5. The Make-or-Buy Question The make-or-buy question has been a hard one since printed wiring became an industrial reality. It is usual that many equipment manufacturers favor making, since the printed board replaces hand wiring done in their plant and so is unlike the traditionally purchased component. Many major firms currently supply their own requirements of printed boards, at least in part. Quite a number of others, however, either never made their own boards or have stopped making them. The economics of board manufacturing are complex, and there is no one best solution for all users.

To evaluate this problem better, several printed circuit manufacturers were asked to rate the major factors in the make-or-buy decision. Manufacturers are obviously biased in favor of a buy decision, but the comments are pertinent to making a sound evaluation.

The major make-or-buy considerations are price, delivery, and quality. Unless an internal facility is set up to serve the needs of many divisions of a large electronic complex, there will be serious problems in meeting fluctuating requirements. If the internal facility is planned to meet the average yearly requirements anticipated, there will be serious delivery delays during periods of peak requirements and underutilization of equipment and personnel during slack periods.

In a make-or-buy study conducted several years ago, shop-loading figures for six captive facilities were compared with those of an independent fabricator for a similar period. The ratio of minimum requirements to maximum requirements for the captives averaged at 1:5. The independent fabricator's ratio was 1:1.2. That verifies the suspicion that many

captive facilities find it necessary to solicit outside business during slack seasons to remain economically healthy only to find that they can't compete with the independent printed wiring manufacturer in quality or service.

The technical requirements brought about by advanced design and construction place a severe strain on the capital equipment budget of the new entrant into the field. The trend toward more complex boards and more specialized techniques makes it less attractive for companies to consider making their own circuitry. The days when a silk screen and an etching tank were enough to start a printed circuit department have long since passed.

Capital equipment such as drilling machinery and plating controls is often obsolete in two or three years and must be amortized in a relatively short period of time. Those outside the industry are often unaware of that situation and plan on a payback period that either cannot be met or is met only by continuing to use outdated, inefficient equipment at increasingly uncompetitive costs. The printed circuit manufacturer with larger volume and a stable shop had can afford the shorter amortization period and still realize a profit on his capital.

The board user faced with the make-or-buy decision must be able to determine his own costs accurately, because he is often deluded by understated overhead costs. Then unrealistic cost figures are arrived at, and they become the basis for his decisions.

Quality is a major reason why many companies go into competition with their suppliers. In printed circuits, however, most firms are hard-pressed to match the independent shop. The basic reasons are closely related to the causes of the price and delivery problem. The captive is usually smaller and less efficient than the specialist. Because of those factors, neither equipment nor skills are the best available. Research on printed circuitry is the prerogative of the specialists and, of course, the large captive operations.

A more subtle attack on quality can affect even the largest captive shop unless it serves a very diversified company, and that is technical isolation. Within an operation there usually is a tendency to employ as few basic packaging techniques as possible. Although that is often good economy, it can be very dangerous unless there is constant "cross pollenization" with other operations. The independent circuit board producer provides an information exchange link. In the procurement of printed circuits, as of other components, state-of-the-art information is one of the most important services your vendors can provide.

6. The Purchase Order and Legal Ramifications* It is neglect of duty of the worst sort to issue a purchase order for a printed board without (1) carefully investigating and evaluating suppliers or (2) having meetings with the prospective vendor to discuss: artwork requirements, delivery, tooling, tolerances, and performance standards.

When it becomes necessary for a court to construe the meaning of a word employed in the purchase order, the court refers first to a standard dictionary. Words used in connection with a particular trade, however, are given the meaning attached to them by experts in that trade. A trade usage or custom that is known to the parties and prevails in the community where the contract is to be executed and performed is incorporated into the agreement by force of law.

If there is a variance between the common standard English meaning and the legal significance of a word, the legal significance is adopted. When the words or terms used in the purchase order have a definite legal meaning, the parties will be presumed to have intended such words or terms to be given the effect that has been established by the courts, unless a contrary intention appears in the instrument.

The law even permits an implication as to price, if a price is not stated in the contract. If, through inadvertence, no price is established in the contract, the courts infer that a reasonable value was the price which was understood between the parties.

The implications contained in a contract, even though not expressed in the writing, include an obligation on the part of both parties to avoid any act which will make it impossible for the other party to perform.

If a question of interpretation of a purchase order is given to a court, the court will consider the whole instrument, if necessary, in order to reach a proper interpretation. A contract should be interpreted as a whole, and a meaning should be gathered from the entire context and not from scattered or isolated words, phrases, or clauses. It is the spirit rather than the letter of the contract which must control the interpretation. It is also the purpose rather than the name given to it by the parties which gives the contract its real meaning. Calling a purchase order a contract for sale on "trial or approval" does not make

it so if it is, in fact, an absolute contract for sale. Calling the agreement a lease does not make it so if the legal effect of the words employed is to make it an outright sale.

After the general purposes are ascertained, the language employed will be construed to serve those general purposes and not destroy them.

7. The Technical Specification The following is an example of how a general printed board specification might be written. It should be noted that this sample can serve for both military and nonmilitary boards. Table 4.1 of the sample specification has been left open-ended to allow for the continuous tabulation of additional board types.

SPECIFICATIONS

8. Sample Specification This specification defines platings used and pattern tolerances to be met in the fabrication of single- or double-sided printed wiring boards.

1. General Requirements:
 - 1.1. All notes and dimensions specified on the applicable drawing take precedence over this specification.
 - 1.2. Acceptance and rejection of plated-through holes shall be based on the IPC standards.
 - 1.3. Circuitry on the board shall meet the solubility requirements of specification 6S2130.
 - 1.4. Workmanship requirements shall be as specified on the applicable drawing.

2. Pattern Tolerances

- 2.1. Class A (regular):
 - 2.1.1. Conductor patterns shall be positioned relative to the x and y datum lines so that the location of any terminal area shall be within 0.02 in diameter to the true position of the hole it circumscribes.
 - 2.1.2. End-product line widths and terminal areas shall not vary more than ± 0.006 in from the 1:1 dimensions on the applicable printed wiring master drawing.
- 2.2. Class B (premium):
 - 2.2.1. Conductor patterns shall be so positioned relative to the x and y datum lines that the location of any terminal area shall be within 0.010 in diameter to the true position of the hole it circumscribes.
 - 2.2.2. End-product line widths and terminal areas shall not vary more than ± 0.003 from the 1:1 dimensions of the applicable printed wiring master drawing. NOTE: The intent of paragraphs 2.1.1 and 2.2.1 is to satisfy the minimum annular ring requirements specified in paragraph 3.

3. Minimum Annular Ring Requirements:

- 3.1. Boards required to meet MIL-STD-275 will be as follows:

Application	Minimum Annular Ring, in
Unsupported (bare) hole	0.020
Plated-through hole	0.010
Flanged part mounted in hole 55	0.010 beyond flange projection

- 3.2. Boards not required to meet military specification will be as follows:

Application	Minimum Annular Ring, in
Unsupported (bare) hole	0.010
Plated-through hole	0.005
Flanged part mounted in hole	Not less than flange projection

4. Plating Types

- A. All conducting surfaces plus the inside surfaces of holes designated as plated-through shall be plated with 0.0010 to 0.0025 in of copper having 99.5 percent purity.
- B. All conducting surfaces except contact surfaces shall be plated with 0.0005 to 0.002 in of tin-lead plating having a minimum of 50 percent and a maximum of 70 percent tin. (Plated-through holes must have a minimum of 0.0005-in thickness plating.)
- C. All contact surfaces shall be plated with 0.0002 to 0.0004 in of low-stress nickel in accordance with Class 2 of QQ-N-290 plus 0.0005 to 0.00010 in of gold in accordance with Type II of MIL-G-45204. (See Fig. 4.1, p. 4-6.)
- D. All contact surfaces shall be plated with 0.0005 to 0.0010 in of low-stress nickel in accordance with Class 2 of QQ-N-290 plus 0.00005 to 0.00010 in of

gild in accordance with Type II of MIL-C-45204. Roughness of the surface finish of the plated contact shall not exceed 32 μ in.

- E. Bare copper shall be deoxidized and coated with a solder coat of flux, Independence Chemical Company solder coating flux F1 in equivalent.
- F. All surfaces, including contact surfaces, shall be plated with 0.0002 to 0.0004 in of low-stress nickel in accordance with Class 2 of QQ-N-290 plus 0.00005 to 0.00010 in of gold in accordance with Type II of MIL-C-45204.
- G. All surfaces, including contact surfaces, shall be plated in accordance with MIL-STD-275 with 0.0005 to 0.0010 in of low-stress nickel in accordance with Class 2 of QQ-N-290 plus 0.00005 to 0.00010 in of gold in accordance with Type II of MIL-C-45204.
- H. All conducting surfaces shall be coated in accordance with MIL-STD-275 with 0.0003-in minimum thickness of tin-lead or solder.
- I. All conducting surfaces shall be plated with 0.0005 to 0.002 in of tin-lead plating having a minimum of 50 percent and a maximum of 70 percent tin. (Plated-through holes shall have a minimum of 0.0005-in thickness plating.)

5. Board Type Requirements:

NOTE: Boards without contacts shall be as defined as in the following table eliminating contact-plating requirements.

Requirements of Boards without Contacts

Board type	Description	Plating, par. 4	Pattern tolerances, par. 2
I	Type GF double-sided 1-oz copper	A, B, C	A
II	Type GF double-sided 1-oz copper	A, B, C	B
III	Type GF double-sided 1-oz copper	A, B, D	A
IV	Type GF double-sided 1-oz copper	A, B, D	B
V	Type XXXP single-sided 2-oz copper	E, C	A
VI	Type XXXP single-sided 2-oz copper	E, C	B
VII	Type GF single-sided 2-oz copper	E, C	A
VIII	Type GF single-sided 2-oz copper	E, C	B

VENDOR-USER RELATIONS

Buying boards is similar, but not the same, when the boards are single-sided, double-sided, or multilayered. Admittedly the standard advice in any complex buy situation is to evaluate the source. In the case of multilayer boards that advice cannot be ignored. Through vendor-user communications are imperative in order to have a quality buy.

Multilayer boards can be of great value, but one should not evaluate the state of the art by buying the cheapest board available and then deciding that it is the best that industry can do. The small manufacturer of single- and double-sided boards cannot convert to producing multilayered boards without establishing an elaborate inspection procedure.

Micrasectioning and inspection are called for after several of the many manufacturing steps in the multilayer fabrication process. Multilayer boards can cost between 4 and 8 times more than conventional two-sided boards having the same number of holes. Still, multilayer boards pay for themselves when they are the answer to complex interconnection problems.

As has been stated previously and will be reiterated throughout this chapter, a positive vendor-user relation is the most important item in purchasing satisfactory printed wiring products.



Fig. 4.1 Definition of contact surfaces. (1) Contact surfaces are defined as those shown in low datum Z and in the applicable drawing. (2) The amount of plated copper at datum Z should be kept to a minimum.

9. **Multilayer Specification Items** In preparing specifications for the fabrication of printed boards, many items should be considered. They relate to the end-product requirements and the manner in which the end product is derived, i.e., who is responsible for what. The following is a list of items that are usually reviewed before going into a multilayer fabrication cycle. Some of the items are also pertinent to the fabrication of single- and double-sided boards. The conclusions that are reached in reviewing this list have a direct impact on the cost of the individual printed board.

Design trade-offs should be considered early in the cycle to optimize the items that have the most significant cost impact. The list should be reviewed with the particular board fabricator to ascertain the items that are the most cost-sensitive in his manufacturing processes:

1. Conductor thickness per layer^a
2. Conductor width (specify minimum when required)
3. Spacings (specify minimum when required)
4. Tolerances plus or minus (state all that are not standard)
5. Dielectric material to be used
6. Thickness of dielectric between layers
7. Overall laminate thickness with tolerances
8. Number of holes
9. Hole size
10. Hole tolerance
11. Hole location preferable on a grid using an 0.025-in base
12. Actual overall dimensions
13. Number of layers
14. Supplier of artwork
15. Quantity of boards required
16. Thickness of plating in hole (state minimum)
17. Other plating
18. Applicable military or industry specifications
19. Other applicable specifications
20. Type of testing
 - v. Bond (copper to laminate)
 - b. Cleavage (laminate to itself)
 - c. Dip solder
 - d. Hot oil
 - e. Dielectric constant and dissipation factor
 - f. Dielectric strength
 - g. Capacitance
 - h. Impedance
 - i. Vibration
 - j. Temperature cycling
 - k. Altitude
 - l. Humidity cycling
 - m. Insulation resistance
 - n. Flammability

10. **Steps in Buying Satisfactory Boards**¹ The quick guide to what can go wrong and the avoidance of such problems, as listed later in this section is the compilation of inputs from a number of suppliers and users. Careful consideration of this information should save much anxiety and anguish. The points covered take on added meaning when viewed in the light of a total program necessary to use single-sided, double-sided, or multilayer boards successfully. Such a program would include the following:

1. Establish specifications consistent with design standards. Do not assume that normal printed wiring specifications are adequate or suitable. Question each vendor as to what he considers typical, normal, or within reason. Do not overspecify unless you are prepared to relax other requirements.
2. Establish basic board design; use all possible means to reduce the quantity of expensive tooling required.
3. Establish a performance specification to allow complete evaluation of thermal shock, dielectric breakdown, temperature cycling, solder resistance, warpage, etc.

4. Establish or adopt a standard test pattern such as that currently in use by the IPC for multilayer boards.
5. Evaluate vendor's samples. Agree with vendor on realistic reliability-testing procedures.
6. Make microscopic examinations of encapsulated microsections of sample holes from test strips.
7. Produce artwork masters at the precision required for the product. For multilayer boards attempt to achieve as near to perfection as possible.
8. Consult with knowledgeable vendors to avoid built-in trouble areas. Include a test pattern on the master to allow future testing of each board produced.
9. Give vendors an accurate picture of present and future requirements and indicate possible areas of change. Cooperation in this area will decrease the number of tooling changes and allow better tooling to be produced and thereby greatly affect the per card price.
10. Establish on-the-line training in the handling, soldering, and assembly of boards to avoid needless damage.
11. Force the development of good specifications and design guides. "Reproducibility" and "application" are the key words.
12. Ascertain vendor qualifications to produce to applicable specifications.
13. Develop good contacts between vendor and user. In-house contacts must be technically aware of all facets of the board.
14. Make sure that all vendor clarification calls get "quick reaction disposition."
15. Establish firm tooling control.
16. Do not allow engineering or any other activity except that responsible for technical vendor contact (designated by buyer) to communicate with vendor. Also, encourage the vendor to establish his contacts in his house.
17. Allow adequate time in procurement cycle for a good job. Although delivery can be forced, the product cannot be reliably made under pressure.

What can go wrong?

Things that most commonly go wrong are:

1. Layer-to-layer connection discontinuity
2. Opening between conductor and through-hole connection
3. Delamination
4. Measling

Other problems are:

1. Inadequate customer-furnished artwork (registration and tolerances cannot be held)
2. Inadequate plating in hole (high resistance)
3. Poor drilling that creates high resistance in some areas
4. Weak design and inefficient layout (excessive layers or fine line circuitry)
5. Cramping of too much circuitry into too little space
6. Unrealistic tolerances

How to Avoid Problems

1. Evaluate vendor. Pick an experienced vendor.
2. Follow recommendations. To prevent delamination or measling (blistering) during soldering, follow the manufacturer's recommendations on temperature and soldering time. Boards stored before soldering ought to be dried 2 h at 250°F.
3. Use quality control. To avoid getting boards with discontinuities due to poor plated-through holes, make sure manufacturer has stringent process controls. Inspections should be made at many steps. Also, severe heat shocks should be avoided.
4. Test. Do test preproduction samples and test strips. Make microscopic examination of encapsulated microsections. (Use IPC test patterns or coupons.)
5. Establish vendor-user understanding. Make sure there is complete communication between vendor and user and that specifications are thoroughly understood. Do not assume that normal board specifications are adequate or suitable. Performance specifications should allow complete evaluation of thermal shock, dielectrical breakdown, temperature cycling, solder resistance, warpage, etc.
6. Design right. Question several vendors to make sure that design and circuit layout takes full advantage of capabilities without going too far. Ask what is typical, normal, or within reason. Have only experienced people do artwork; perhaps the vendor should do

it. Masters should be the proper size for the tolerance capability (1:1 for precision numerically controlled (NC) artwork generations; 4:1 or more for manually prepared artwork).

COST FACTORS

When ordering printed boards, conventional or multilayer, remember that the quoted price is based heavily on the number of boards ordered, since it is high-quantity processing that gives good economy. That means the manufacturer will process all boards at the same time. One should not expect deliveries spread out over a period of time. For example, don't order 1000 pieces and then expect the vendor to ship them at a rate of 250 every three months just to keep in step with internal equipment production schedules.

Tight tolerances mean high prices, too, because the manufacturer cannot avoid extra problems in meeting the specs. Price depends heavily on yield at a particular tolerance level. With tight tolerances, yields must decrease. Price must therefore go up. Watch the platings. Using gold or other noble platings where solder would do, and using solder where no plating at all would do, can greatly increase the price. Solder plate not only is cheaper than gold but also leads to a greater processing yield.

All these points can be summed up with one statement that applies, actually, to all buying: don't overspecify, either in terms of tolerances, materials, or configurations, and try to standardize board formats.

Don't forget the cost of testing. Try to ascertain the vendor's capability to produce and deliver acceptable boards. The costs of inspection, return procedures, and production delays often can outweigh the money saved by choosing the lowest bidder.

11. One- and Two-Sided Board Pricing: Simple one- and two-sided boards are fairly well understood by users, except for one factor: price. With each job a special, it has often been difficult to compare boards parameter by parameter. With standard price lists, however, one is able to compare price changes caused by changing single parameters. By using standard price indexes, one can make the following comparisons based on a board with these characteristics:

- Board size: 4 × 6 in
- Base material: 1/16-in one-sided epoxy-paper
- Board finish: solder plating
- Number of holes: 80
- Number of different hole sizes: 4
- Board outline: rectangular with rounded corners
- Initial quantity required: 10
- Legend: Printed

The standard price index gives a unit price of x each for 10 boards with the above characteristics. If the initial order is increased to 100, the unit price drops to 25 percent of the 10 price, and if to 500 pieces, the price is 15 percent of the 10 price. This illustrates the tremendous economy inherent in ordering in quantity. The prices are only theoretical and are subject to many changes due to inflation and the labor-machine mixture in the various companies, but the relations between a small and a quantity purchase serve the intended example.

Now, suppose one tries to reduce the board size in an attempt to reduce the price. A 3 × 4.5-in board would reduce the per-board material cost and allow the manufacturer to process more circuits on one panel. The prices would then drop to 92 percent of unit price x each to 10 boards, to 24.5 percent for 100, and to 12.3 percent for 500. With an order of 100 boards, the total price would thus drop by only 0.5 percent of unit price x , which would not be worth the extra effort, but the 500-piece order would price out at savings of 2.7 percent of unit price x . That would be an economy.

If G-10 instead of epoxy-paper is used, the prices increase but the offsetting gains are the superior mechanical and electrical characteristics of G-10. If XXXP is used, the prices go down.

The original specification called for solder plating, apparently because a longer shelf life was needed. If there is a switch to simple etched copper with a protective water-dip lacquer, the prices can be decreased. The savings with 500 boards is about 22 percent.

If nickel-gold is used for longer shelf life, corrosion resistance, and low contact resis-

tance, then the prices go up. The premium here is thus about 42 percent for 500 boards.

Now suppose the number of different hole sizes is reduced from four to two and the number of drilling steps is thereby reduced. The prices drop.

Finally, if the rounded edges are considered not really necessary but included only for aesthetic reasons, the price for the 500 quantity can be reduced to save about 3.6 percent. Putting some of the savings together will revise the characteristics as follows:

Board size: 6 × 8 in (only half as many are needed)

Base material: XXXP

Board finish: etched copper

Number of different hole sizes: 2

Board outline: square corners

Legend: enough space available—etched

With the new characteristics the board prices at the 500 level drop to a point that results in a total saving of 53 percent.

12. Multilayer Board Pricing To get an estimate of the relative costs of multilayer vs. two-sided boards, several manufacturers were asked to give the cost factor multiplier for a multilayer board with twice, four times, and eight times the conductor density of the two-sided boards for a typical quantity of 100 boards. One must remember, however, when making such comparisons, that the costs saved in connectors, assembly, and handling will have a major effect on the overall cost and thus make the multilayer approach attractive. The increased reliability also should be considered.

The most common response was that, if the conductor density was twice as great, the cost of the multilayer board would be twice times as great because of the unavoidable extra work in setting up the multilayer boards with the necessary tolerances to assure acceptable registration. With four times the conductor density, the cost goes up to five times as great for the same reasons as in the first case. With eight times the conductor density, the cost goes up to eight times as much for the multilayer compared with the two-sided board. At that point, the extra multilayer costs are beginning to be absorbed and the cost per unit density is decreasing.

One respondent gave a rough rule of thumb: take the cost of a two-sided board and multiply it by the costs of the layers and add a 20 percent factor for bonding and checkout. Thus for a two-sided board of unit cost y , for four layers we get $4y$ and $0.75y$ extra for bonding, giving $4.75y$ per multilayer board. One factor that can be attached to the economy of multilayer is the cost of connecting each integrated circuit package. A rough way to calculate costs for quantities of about 100 boards of a type is to take between $1z$ and $1.5z$ per simple integrated circuit package, where z is the base price. For large-scale integration (LSI) the price would be about $2z$ to $3z$ per integrated circuit (IC). Again those dollar estimates are general and are based on the standard price index established in Sec. 4.11 above.

Another factor that affects the price of multilayer boards is the number of layers. Many vendors today can fabricate boards in excess of 20 layers, but most vendors and users prefer to stay around 8 layers per board. A limit in the costing is the ratio of the final board thickness to the hole diameter. When the ratio exceeds 4:1, the plating in the hole begins to suffer and special plating techniques which increase cost are required.

One manufacturer concluded that the density of terminal points or holes has a much greater effect on cost than the overall size, number of layers, or complexity of the interconnections has. Costs go up sharply when hole spacings drop below 0.100-in centers. The concept is that the entire tolerance structure for a board with closely spaced holes must be much more sophisticated. The price for the added precision is, of course, added cost.

13. Delivery Schedules How long does it take to get delivery on single-sided, double-sided, and multilayered boards in large quantities? The answer to that basic buying question depends mainly on whether you supply the manufacturer with usable artwork. Minimum delivery time from the date artwork is finally approved is usually one to two weeks for single-sided boards, one and one-half to three weeks for double-sided boards, and four to six weeks for multilayer boards. That is the time needed for the actual production of the boards. Artwork could take from one to three more weeks for production and check out, which would result in a complete delivery cycle of about ten weeks for a sophisticated multilayer board.

Some users are not as skilled in laying out multilayer boards as they are in laying out single- and double-sided boards. In the cases of the latter, the manufacturer will prepare the artwork from sketches and wiring tables. The increased use of NC artwork generators is making that characteristic of the industry disappear.

Changes in wiring can have a profound effect on the circuit layout and artwork and can push delivery dates further back. Be sure that the circuit is complete before starting into the fabrication cycle.

PRODUCT DEFINITION

Basic documentation requirements for printed boards are not unrealistic for even the smallest manufacturer to adapt as a basis for in-house standards. Definition of the printed board product in documented form is still one of the most misunderstood aspects of the industry. The misunderstanding comes about because so many companies are concerned about the high cost of preparing documentation that they attempt to take shortcuts in defining the items that assure them an acceptable product.

Documentation may be very variable as to the quality of the actual recorded information. There are also many techniques for preparing the document that defines the end-product board. One document, however, is the nucleus of all the others, and that is the master drawing. In the IPC-T-50A Terms and Definitions the master drawing is defined as:

A document that shows the dimensional limits or grid locations applicable to any or all parts of a printed wiring (circuit) board (rigid or flexible), including the arrangement of conductive and nonconductive patterns or elements; size, type, and location of holes; and any other information necessary to describe the product to be fabricated.

Figure 4.2 shows a simplified flowchart of how the master drawing fits into the total product development cycle.

14. Types of Documentation* Printed wiring documentation can be divided into three basic groups:

1. Minimum documentation. Used for prototype and small-quantity runs.
2. Formal documentation. Used for a standard product line and boards built in production quantities. Similar to Category E, Form 2 per MIL-D-1000* without source or specification support documentation.
3. Military documentation (Category E, Form 1 per MIL-D-1000). Complies with government contracts specifying procurement drawings for the manufacture of identical items by other than the original manufacturer.

To save time on the master drawing in the minimum documentation phase, a sepia print of the noncomponent (circuit) side artwork can be used. It should indicate board material, finish plating, outline dimensions (if no outline drawing is referenced in the parts list), and drilling data. Notes can be freehand-lettered; capital single-stroke characters $\frac{1}{8}$ in minimum in size are generally acceptable. The contact sepia print should be processed with a light box or similar apparatus. Taped artwork should never be processed through a machine that administers heat, fumes, or a rolling action, because serious damage to the equipment and artwork can result.

Formal documentation should be generated only upon a management decision to manufacture and market a product for which only minimum documentation exists. Formal documentation is similar to Category E, Form 2 per MIL-D-1000 without source or specification documentation. It can make good use of the technique of "form drawings," which, when coupled with the fullest use of the in-house photo facility, make an excellent documentation tool.

Any drafting facility can realize cost savings through the reduction of drafting hours by using form drawings to which is added information that is required by each type of document. For example, a master-drawing form might contain location of the hole chart and an outline of the printed board indicating board dimensions, notes, and title block. An assembly drawing form might contain the board outline, notes, an integrated circuit package outline indicating pin identification, and a title block. In either case, the only information remaining to be added is that which is peculiar to the particular printed board, such as drawing title, number, and drilling information.

*MIL-D-1000 1974 edition, subject to revision.

The photographic facility has primarily been used to transform artworks into master patterns for making printed boards. However, by capitalizing on its extended capabilities, reduced drafting costs can be realized by making screen-tint photopositive transparencies for the drawing package. Here photography is used not only to reduce the photo-positive transparency to a scale convenient for use in the form drawing but also to screen-tint (half-

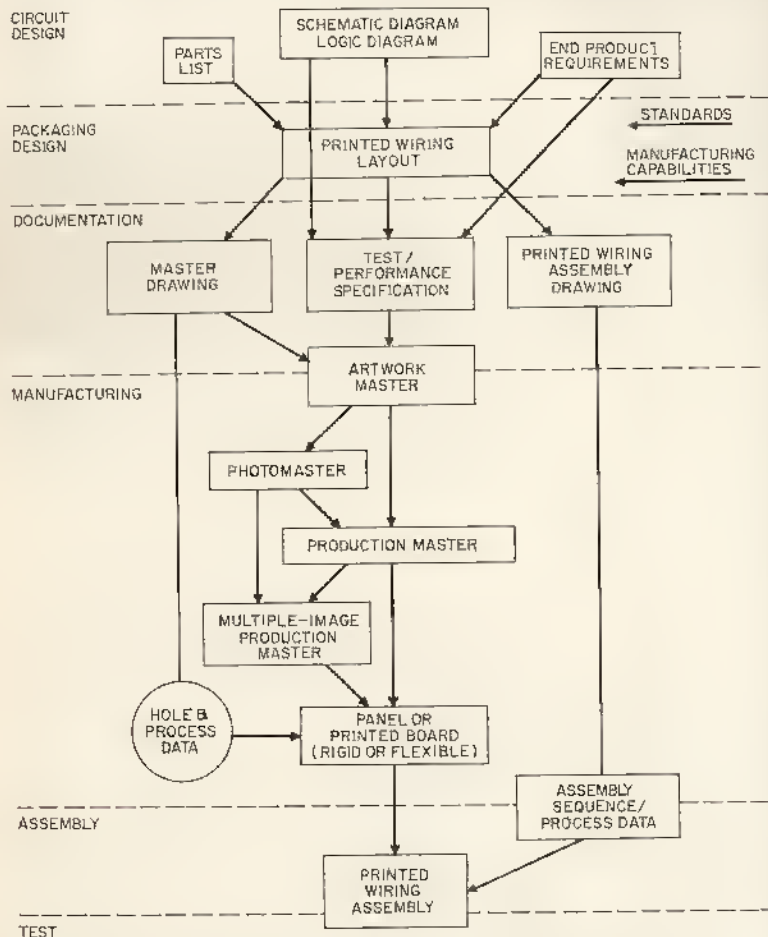


Fig. 4.2 Flowchart of master drawing (Courtesy Institute of Printed Circuits).

tone) the circuit pattern, which is then superimposed over a precision grid. The practice assures positive registration between terminal-area centers and their corresponding grid intersections, which is of primary importance when master drawings are made in compliance with military requirements.

Military documentation (category E, Form 1 per MIL-D-1000) indicates that design establishments are obligated to prepare engineering drawings and associated lists in accordance with MIL-D-1000.

MIL-D-1000 specifies general requirements for the preparation of engineering drawings and associated lists and for application of intended-use categories for their acquisition. The specification reflects the policy of the Department of Defense to buy only the engineering drawings that are needed and to encourage procurement of commercial drawings when they are adequate for the purpose.

Category E, Form 1 specifies the minimum acceptable military requirements for the

preparation of engineering drawings and associated lists by designating the intended-use Category E and Form 1.

Category E engineering drawings are for use as procurement documents to permit competitive procurement and manufacture of items that are substantially identical with original items, including identical repair parts.

Form 1 engineering drawings conform to MIL-STD-100 and are intended for use by the government when controls inherent to that standard are required.

MIL-STD-100A primarily prescribes the procedures and format authorized for the preparation of Form 1 engineering drawings and associated lists prepared for the Defense Department. The standard includes a paragraph stating in part that cost-reduction techniques in engineering drawing preparation may be used when the techniques do not impair the reproducibility quality of Mil-M-9868 or clarity and design disclosure requirements for the kind, category, and form of the engineering document being prepared. Specifically, photographic drafting techniques should be used to the maximum extent practicable.

Although this standard does not stipulate the drawings comprising a printed wiring drawing package, it does specify the requirements for schematics, master and assembly drawings, artwork, and supporting drawings:

1. Diagrammatic drawings, including schematic diagrams, refers to American National Standards (ANSI), ANSI/ANSI Y32.2 and ANSI Y32.16 providing directions for use of symbology relative to diagrammatic drawings.

2. Logic diagrams are required to meet the provisions of USAS Y32.14.

3. Printed wiring master pattern drawings must meet the requirements specified in MIL-STD-100A, MIL-STD-275, and MIL-STD-1495.

4. Printed wiring master drawings must meet the requirements of MIL-STD-100A, MIL-STD-275, and MIL-STD-1495.

5. Assembly drawings, including printed wiring assemblies, must conform to the requirements of MIL-STD-100A.

6. Miscellaneous drawings, such as details, detailed assemblies, tabulated assemblies, inseparable assemblies, specification control drawings, wire lists, which are prepared in support of the printed wiring drawing package, must conform to the requirements of MIL-STD-100A as applicable to the particular document.

REFERENCES

1. Buyers Report on Printed Circuits, *Electronic Procurement*, August, 1963.
2. Orland H. Ellis, Constructing the Purchase Order, *Electronic Procurement*, August, 1963.
3. Evaluation Guide: Multilayer Printed Wiring Boards, *Electronic Procurement*, April, 1965.
4. Bulletin, Methode Electronics, Inc.
5. Specifying Guide: PC Boards, *IEEE*, April, 1965.
6. George R. Jacobs, Jr., Documenting Printed-wiring Packages, *Machine Design*, May, 1969.

Section **2**

Fabrication

Chapter 5

Laminate Machining

CHARLES G. HENNINGSEN

Vice-President, Insulactro, Mountain View, California

LELAND E. TULL

President, Contouring Technology, Mountain View, California

Introduction	5-3
Punching Holes (Piercing)	5-4
1. Design of the Die	5-4
2. Shrinkage of Paper-Base Laminates	5-4
3. Tolerance of Punched Holes	5-4
4. Hole Location and Size	5-5
5. Warming Paper-Base Material	5-5
6. Press Size	5-5
Blanking, Shearing, and Cutting of Copper-Clad Laminates	5-6
7. Blanking Paper-Base Laminates	5-6
8. Blanking Glass-Base Laminates	5-6
9. Shearing	5-7
10. Sawing Paper-Base Laminates	5-7
11. Sawing Glass-Base Laminates	5-7
Drilling Holes	5-8
12. Drill Geometry and Nomenclature	5-8
13. Selection of a Drilling Machine	5-9
14. Backup and Entry Materials	5-10
15. Drilling Feeds and Speeds	5-11
16. Drilling Problems	5-11
17. Routing Laminates	5-12

INTRODUCTION

Laminate machining consists of the mechanical processes by which circuit boards are prepared for the vital chemical processes of image transfer, plating, and etching. Such

processes as cutting to size, drilling holes, and shaping have major effects on the final quality of the printed board. This chapter will discuss the basic mechanical processes which are essential to producing the finished board.

PUNCHING HOLES (PIERCING)

1. Design of the Die It is possible to pierce holes down to one-half the thickness of XXXPC and FR-2 laminates and one-third that of FR-3 (Fig. 5.1). Many die designers lose sight of the fact that the force required to withdraw piercing punches is of the same magnitude as that required to push the punches through the material. For that reason, the question of how much stripper-spring pressure to design into a die is answered by most toolmakers: "as much as possible." When space on the dies can not accommodate enough mechanical springs to do the job, a hydraulic mechanism can be used. Springs should be so located that the part is stripped evenly. If the board is ejected from the die unevenly, cracks around holes are almost certain to occur. Best-quality holes are produced when the stripper compresses the board an instant before the perforators start to penetrate. If the

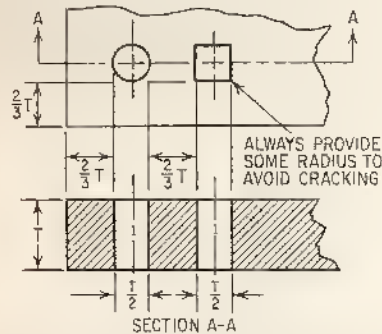


Fig. 5.1 An illustration of the proper sizing and locating of pierced holes with respect to one another and to the edge of paper laminates. Minimum dimensions are given as multiples of *T*, the laminate thickness.

stripper pressure can be made to approach the compressive strength of the material, less force will be required and the holes will be cleaner.

If excessive breakage of small punches occurs, determine whether the punch breaks on the perforating stroke or on withdrawal. If the retainer lock is breaking, the cause is almost certain to be withdrawal strain. The remedy is to grind a small taper on the punch, no more than 1 1/2° and to a distance no greater than the thickness of the material being punched. If the grinding is kept within those limits, it will have no measurable effect on hole quality or size. The other two causes of punch breakage are poor alignment, which is easily detected by close examination of the tool, and poor design, which usually means that the punch is too small to do the job required.

2. Shrinkage of Paper-Base Laminates When paper-base laminates are to be punched, it must be remembered that the materials are resilient and that their tendency to spring back will result in a hole slightly smaller than the punch which produced the hole. The difference in size will depend on the thickness of the material. Table 5.1 shows the amount by which the punch should exceed the print size in order to make the holes fall within tolerance. The values listed should not be used for the design of tools for glass-epoxy laminates, the shrinkage of which is only about one-third that of paper-base materials.

3. Tolerance of Punched Holes If precise hole size tolerance is required, the clearance between punch and die should be very close; the die hole should be only 0.002 to 0.004 in

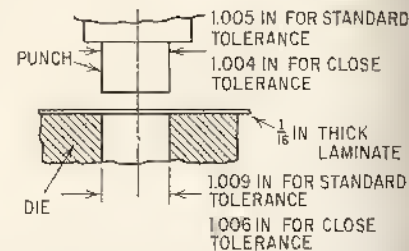


Fig. 5.2 An example of proper tolerance of a punch and die.

larger than the punch for paper-base materials (Fig. 5.2 and Table 5.2). Glass-base laminates generally require about one-half that tolerance. Dies have, however, been constructed with as much as 0.010 all-around clearance between punch and die. They are for use where inspection standards permit rough-quality holes.

A die with sloppy clearances is less expensive than one built for precision work, and wide clearance between punch and die causes correspondingly more break and less shear than a tight die will cause. The result is a hole with a slight funnel shape that makes

TABLE 5.1 Shrinkage in Punched Hole Diameters, Paper-Base Laminates

Material thickness	Material at room temp.	Material at 90°F or above
1/64	0.001	0.002
1/32	0.002	0.003
3/64	0.003	0.005
1/16	0.004	0.007
3/32	0.006	0.010
1/8	0.010	0.013

insertion of components easier. Always pierce with the copper side up. Do not use piercing on designs with circuitry on both sides of the board, because lifting of pads would probably occur.

4. Hole Location and Size Designs having holes whose distance from the edge of the board or from other holes approaches the thickness of the material are apt to be troublesome. Such designs should be avoided; but when distances between holes must be small, build the best die possible. Use tight clearance between punch and die and punch and stripper, and have the stripper apply plenty of pressure to the work before the punch starts to enter. If the distance between holes is too small, cracks between holes may result even with the best of tools. If cracks between holes prove troublesome, plan the process so that the piercing is done before any copper is etched away. The reinforcing effect of the copper foil will help eliminate cracks. Most glass-epoxy laminates may be pierced, but the finish on the inside of the holes is sometimes not suitable for through-hole plating.

5. Warming Paper-Base Material The process of punching paper-base laminates will often be much more trouble-free if the parts are warmed to 90 or 100°F. That is true even of the so-called cold-punch or PC grades. Do not overheat the material to the point at which it crumbles and the residue is not ejected as a discrete slug. Overheated material will often plug the holes in the die and cause rejects. Opening the taper on the take-away holes will reduce plugging, but the most direct approach is to pierce at a lower temperature. Glass-epoxy is never heated for piercing or blanking.

6. Press Size The size of the press is determined by the amount of work the press must do on each stroke. The supplier of copper-clad sheets can specify a value for the shear strength of the material being used. Typically, the value will be about 12,000 psi for paper-base laminate and 20,000 psi for glass-epoxy laminate. The total circumference of

TABLE 5.2 Tolerances for Punching or Blanking Paper-Base Laminate

Material thickness	Base material	Tolerance on hole size, in	Tolerances, in, on distance between holes and slots, 90°F				Tolerances for blanked parts, overall dimension, in
			Up to 2 in	2 to 3 in	3 to 4 in	4 to 5 in	
To and including 1/8 in	Paper	0.0015	0.003	0.004	0.005	0.006	0.003
Over 1/8 in to and including 1/2 in	Paper	0.003	0.005	0.006	0.007	0.008	0.005
Over 1/2 in to and including 1 in	Paper	0.005	0.006	0.007	0.008	0.009	0.008

the parts being punched out multiplied by the thickness of the sheet gives the area being sheared by the die. If all dimensions are in inches, the value will be in square inches. For example, a die piercing 50 round holes, each 0.100 in in diameter, in 0.062-in-thick laminate will be shearing, in square inches:

$$50 \times 0.100 \text{ in} \times 3.1416 \times 0.062 \text{ in} = 0.974 \text{ in}^2$$

If the paper-base laminate has 12,000-psi shear strength, 11,688 lb of pressure, or about 6 tons, is required just to drive the punches through the laminate. Bear in mind that, if a spring-loaded stripper is used, the press will also have to overcome the spring pressure, which ought to be at least as great as the shear strength. Therefore, a 12-ton press would be the minimum which could be considered. A 15- or 20-ton press would be considerably safer.

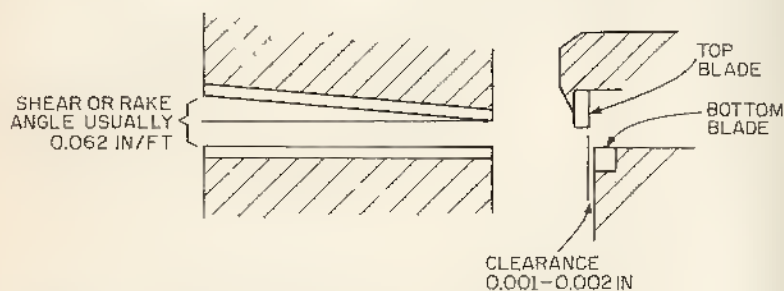


Fig. 5.3 Typical adjustable shear blades for copper-clad laminates.

BLANKING, SHEARING, AND CUTTING OF COPPER-CLAD LAMINATES

7. Blanking Paper-Base Laminates When parts are designed to have other than rectangular shapes and the volume is great enough to justify the expense of building a die, the parts are frequently punched from sheets by using a blanking die. A blanking operation is well adapted to paper-base materials and is sometimes used on glass-base ones.

In the design of a blanking die for paper-base laminates, the resilience, or yield, of the material previously discussed under Piercing applies. The blanked part will be slightly larger than the die which produced it, and dies are therefore made just a little under print size depending upon the material thickness. Sometimes a combination pierce and blank die is used. The die pierces holes and also blanks out the finished part.

When the configuration is very complex, the designer may recommend a multiple-stage die: the strip of material progresses from one stage to the next with each stroke of the die. Usually in the first one or two stages holes are pierced, and in the final stage the completed part is blanked out.

The quality of a part produced from paper-base laminates by shearing, piercing, or blanking can be improved by performing the operation on material which has been warmed. Caution should be exercised in heating over 100°F because the coefficient of thermal expansion may be high enough to cause the part to shrink out of tolerance on cooling. Paper-base laminates are particularly anisotropic with respect to thermal expansion; that is, they expand differently in the *x* and *y* dimensions. The manufacturer's data on coefficient of expansion should be consulted before a die for close-tolerance parts is designed. Keep in mind that the precision of the manufacturer's data is probably no better than ±25 percent.

8. Blanking Glass-Base Laminates Odd shapes that can not be feasibly produced by shearing or sawing are either blanked or routed. Glass blanking is always done at room temperature. Assuming a close fit between punch and die, the part will be about 0.001 in larger than the die which produced it. The tools are always so constructed that a part is removed from the die as it is made. It cannot be pushed out by a following part as is often true when the material has a paper base. If material thicker than 0.062 in is blanked, the parts may have a rough edge.

The life of a punch, pierce, or blank die should be evaluated with reference to the various copper-clad materials that may be used. One way to evaluate die wear caused by

various materials is to weigh the perforators, or punches, very accurately, punch 5000 pieces, and then reweigh the punches. Approximately 5000 hits are necessary for ovalation, because the initial break-in period of the die will show a higher rate of wear. Also, of course, the quality of the holes at the beginning and end of each test must be evaluated. Greatly enlarged microphotos of the perforator can be used for visual evaluation of changes in the die.

9. Shearing When copper-clad laminates are to be sheared, the shear should be set with only 0.001 to 0.002 in clearance between the square-ground blades (Fig. 5.3). The thicker the material to be cut, the greater the rake or scissor angle between the top and bottom shear blade. The converse also is true: the thinner the material the smaller the rake angle and the closer the blades. Hence, as in many metal shears, the rake angle and the blade gap are fixed; the cutoff piece can be twisted or curled. Paper-base material can also exhibit feathered cracks along the edge that are due to too wide a gap or too high a shear angle. That can be minimized by supporting both piece and cutoff piece during the shear operation and decreasing the rake angle. Epoxy-glass laminate, because of its flexural strength, does not usually crack, but the material can be deformed if the blade clearance is too great or the shear angle is too large. As in blanking, the quality of a part produced from paper-base laminates by shearing can be improved by warming the material before performing the operation.

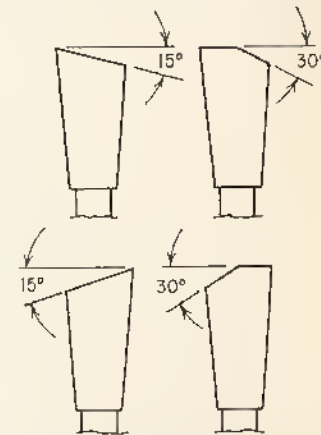


Fig. 5.4 Commonly used sawtooth designs for paper and cloth laminates. At left, two successive teeth on a 15° alternate-bevel saw. At right, two successive teeth on a 30° alternate-corner relieved (AC-30) type of saw.

10. Sawing Paper-Base Laminates Paper-base laminates are much harder on sawing tools than are the hardest woods, and therefore a few special precautions are necessary for good saw life. Sawing paper-base laminates is best accomplished with a circular saw with 10 to 12 teeth per inch of diameter at 7500 or 10,000 ft/min. Hollow-ground saws give a smoother cut; and because of the abrasive nature of laminated materials, carbide teeth are an excellent investment. (See Fig. 5.4 for tooth shape.) When a saw does not last long enough between sharpenings, use the following checklist. (These steps could have a cumulative effect and change saw life by a factor of 4 to 5.)

1. Check the bearings for tightness. There should be no perceptible play in them.
2. Check the blade for runout. As much as 0.005 in can be significant.
3. When carbide teeth are used, inspect them with a magnifying glass to make sure a diamond tool no coarser than 180 grit was used in sharpening them.
4. If the saw has a thin blade, use a stiffening collar to reduce vibration.
5. Use heavy pulleys with more than one V belt. Rotating parts of the system should have sufficient momentum to carry the sawtooth through the work smoothly and without variation of speed.
6. Check the alignment of the arbor and the motor mounting.

All these steps are intended to reduce or eliminate vibration, which is the greatest enemy of the saw blade. If vibration is noticed, find the source and correct it.

11. Sawing Glass-Base Laminates When glass-base laminates are to be sawn, carbide-tooth circular saws can be used; but unless the volume of work is quite low, the added investment required for diamond-steel-bonded saws will be paid for in future savings. The manufacturer's recommendation on saw speed should be followed; usually it will be for a speed in the neighborhood of 15,000 ft/min at the periphery of the saw blade. When economics dictate the use of carbide-tooth circular saws for cutting glass, use the instruction previously given for paper-base laminates (see Fig. 5.4 for tooth shape) and remember that each caution regarding runout, vibration, and alignment becomes more important when glass-reinforced laminates are sawn.

DRILLING HOLES

A good, clean, burr-free hole is essential to good printed circuit quality. As we will see, to obtain that quality of hole consistently, a balance of proper drills, drilling machines, speeds and feeds, hackup board, and copper-clad materials is necessary.

12. Drill Geometry and Nomenclature The drill bit functions to cut and remove from the hole the copper-clad material being drilled. It is therefore important to understand the function and geometry of each part of the drill. The point angle, as shown in Fig. 5.5,

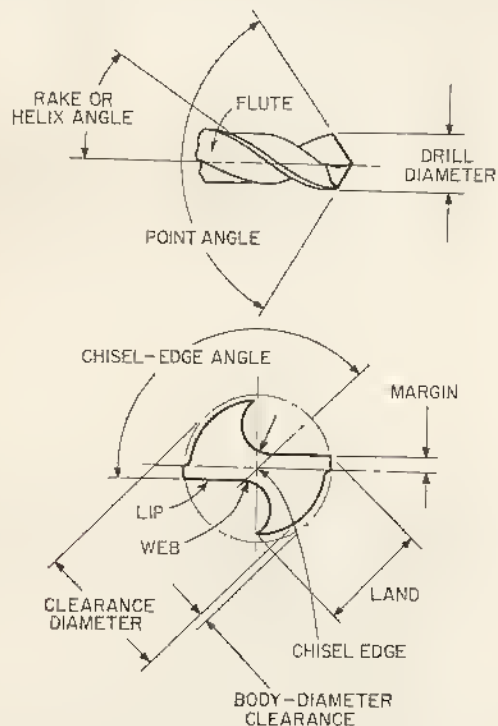


Fig. 5.5 Drawing of typical drill bit geometry.

determines the ability of the tool to cut the material. For laminate drilling, it usually varies between 90 and 130°; for paper-base materials, between 90 and 110°; and for glass-base materials, between 115 and 130°. The drill point has two primary cutting edges which are parallel to each other and are displaced by the web. The cutting edges are connected by the web to a secondary edge called the chisel edge. Both the cutting and chisel edges must be straight for uniform cutting and have no nicks or grinding marks. The surface should be highly polished to reduce friction. The flutes must be equal in length and angle; otherwise, the drill will be off center.

The flute angle, or helix angle, determines the ability of the drill to remove chips from the hole. Helix angles vary from 20 to 50°. A relatively large angle is needed to assure good chip flow and prevent blockage of the drill. The flute of the drill should be polished to reduce friction heat and improve drill life, and an 8- to 10- μ m finish is recommended. The drill is relief-tapered back along the body to allow cooler running. The taper, called relief angle or back taper, ranges from 0.001 to 0.003 in/in. Most printed circuit drilling is done with carbide drills. The usual grade is C2, which has good resistance to heat and high hardness. Printed circuit carbide drills are fragile and will break easily if they are dropped or stressed heavily during drilling.

13. Selection of a Drilling Machine The selection of a drilling machine can be difficult, and it will affect both drilling economy and hole quality. Machines on the market today range from single-spindle manual drills to multiple-spindle machines controlled by mini computers. Within that range are a multitude of drills with differences in speed, capacity, accuracy, and cost.

For the sake of simplification, drilling machines will be classified as manual and numerical control. Manual machines require the operator to position the work and initiate the drilling cycle. In numerical control (NC) machines both positioning and drilling cycle are controlled by some remote input such as punched tape or computer. Drilling machine selection should be based on an analysis of the following factors:

Panel size. What is the maximum panel size to be drilled, and in what quantity will that size be drilled? This factor applies more to multiple-spindle drills than to single-spindle manual machines.

Volume. What will the typical run size be? A high-volume shop will usually be forced to invest in high-output NC machines, whereas low volume, short runs, and prototypes may be effectively handled on manual machines.

Accuracy. What accuracy is needed? Manual drills are necessarily labor-intensive, and the ultimate accuracy is a function of operator skill. True hole location within 0.004 to 0.006 in can be achieved through careful attention to artwork and template fabrication. For true hole locations of 0.003 or less, NC equipment should be specified.

Capital. How much can you afford to spend on a drilling machine? The capital investment for a single-spindle manual machine will range from \$4000 to \$7000; that for a multiple-spindle manual drill will approach \$20,000. NC drills will cost from \$25,000 to over \$150,000.

a. Manual Control. Work is positioned for drilling in a manual machine by one of two basic methods: optical or template. The optical method requires that the hole location be defined either by an artwork overlay or reproduction of the circuit defining the pad. Photoresist is often used when the developed image is sufficient to locate the hole positions, and hoards may be stacked and pinned for registration to the master. In template drilling, the hole pattern must be reproduced in the template material, which usually is aluminum. The hoards are pinned to the template, and the stack is moved under a stylus controlled by the operator. When the stylus drops into a hole in the template, a command is given to drill the board stack, usually from the bottom.

Both locating systems can be used with multiple-spindle manual machines such that either the drill head or the table, rather than the work, may be moved by the operator. In all cases, it is advisable to road-map the artwork and template to guide the operator through the circuit. Without such a guide, he will find it very easy to miss holes.

b. Numerical Control. The complexity and high cost of NC drilling equipment require that a close look be given the components that make up the system.

Two primary systems are currently used to translate and control the coordinate information from a punched tape to machine-positioning movements.

Numerical control is defined as machine-control logic obtained from fixed programs that are designed and hard-wired into the electronics. That is the older of the two systems. When it is properly designed and adjusted, it is very reliable, and failures are usually restricted to components.

In computer numerical control (CNC), the machine-control logic is obtained from software; the computer is programmed to control the machine. The software is usually in the form of a punched tape called an executive program, and it may enter the computer through the system's own tape reader.

The advantages of CNC over NC are flexibility, versatility, and speed. Those advantages are more important to the machine manufacturer than to the user; for they allow the manufacturer to make changes in machine performance at a minimum of cost and effort. The advantages to the user are somewhat less well defined, but they are usually in the form of pattern storage, step-and-repeat capability, and other manipulative processes made possible by the computer memory.

1. Numerical Control Machine Design and Construction. The major elements of all NC machines are frame or base, positioning systems, feedback system, and spindle configuration.

1. Frame. Three basic constructions are used today: (a) a large granite slab as a machine base (b) a stress-engineered steel fabricated frame, and (c) a combination of (a)

and (b). A large cast-iron base is sometimes used in place of a granite slab. In all cases a massive frame is used to resist the extreme inertial forces of rapid table movement as well as support the various machine components.

2. Positioning systems. Machine positioning is through either a dual-axis movement in which the table moves in both the *x* and *y* directions, or through a split-axis movement, in which the table moves along one axis and the drill head assembly moves along the other. There are advantages and disadvantages to neither system, but a well-designed machine will overcome the separate weaknesses.

3. Drive and feedback system. All NC machines utilize precision ball screws to drive the moving members. The lead screws are driven by servomotors that receive commands from the control unit. The control unit must have some way of knowing where the positioning elements are at any given moment. Two feedback methods are used: open loop and closed loop. In an open-loop system, individual incremental movements are added to or subtracted from the preceding positioning data, and a new position is assumed by the control. There is no direct feedback to the control, and any error that develops will be unknown by the control. However, the design of open-loop incremental drives is quite advanced, and the system has proven reliability.

Closed-loop systems are the more common of the two types. Position information is sent back to the control and compared with the position instruction. Until the two agree, no further machine functions will be allowed. Closed-loop systems work in two ways. First, they utilize rotary encoders to count the revolutions of the driving lead screw and feed that information back to the control. Secondly, they utilize linear encoders and an accurate scale to tell the control where the moving member is.

In practice, both systems work well. The linear system is most favored because it is based on the real position of the moving member and not just the motion of the units driving that member.

4. Numerical control spindle configuration. Spindle design and mounting method vary. The spindle may be driven by air or electricity, and direct current or, most commonly, high-frequency alternating current may be used. The spindle bearings may be grease-packed, mist-oil-lubricated, air-cooled, or liquid-cooled depending on whether they are of the ball bearing or the air bearing type. The rpm may be from 3000 to 100,000; most spindles today operate in the 15,000- to 80,000-rpm range.

An important factor in spindle design is the chucking system used. Because drills will be changed often, chucking should be quick and positive to avoid drill slippage either radially or linearly. Air- and hydraulically operated chucks and centrifugal collars are available. Very little choice is offered in any specific machine design, so the fabricator must decide on spindle requirements early in his investigation.

d. Service. Any drilling machine, and particularly an NC machine, is a complex piece of equipment subject to periodic failures. The industry standard for NC equipment is approximately 5 percent downtime. It is extremely important that complete and fast service be available in the event of a failure. Whether it is can best be ascertained by consulting with users in your area rather than relying totally on factory guarantees.

e. Performance. The choice of a drilling machine involves a thorough investigation into many areas of design and performance. Speed of a given machine is often given in hits per minute. It is important, when that speed measure is used as a gauge of machine performance, to know the stroke or stack height as well as the distance between holes. Further, the user should recognize that maximum hits per minute cannot be attained under most drilling conditions, where feed rate and spindle rpm will vary with drill sizes and material.

14. Backup and Entry Materials Proper selection of drill entry and backup material makes a major contribution to getting hole finish of the desired quality; many drilling problems are generated by improper selection of such materials. The primary purpose of entry and backup material is to eliminate the entry burr on the bottom of each stack of material and the exit burr on the top. The material should also help guide the drill during the operation and clean and cool the cutting tool.

The tungsten carbide used in most drill bits is an extremely poor conductor of heat. The frictional heat of cutting continues to increase during drilling but basically remains within the drill point geometry of the tool, and the heat dissipation is poor. The frictional heat

that is generated does not affect the tungsten carbide to any great degree, but a hot-running carbide drill can load up and tear the walls of the drilled glass-epoxy hole.

Entry and backup material should be uniform in thickness so that all drilling stations are of the same height. The material should be thick enough to allow the drill to penetrate at least 0.010 in more than the major diameter of the largest drill being used without breakthrough of the opposite side. It should not contain any ingredients that will contaminate the drilled hole. It should allow the twist drill an easy withdrawal. (Most drills are broken by seizure.) The most commonly employed entry and backup materials are as follows:

1. Oil-tempered or composition hardboards are inexpensive but generally lack a surface hard enough to eliminate all burring. Also, the oils or binders used in them are apt to cause hole contamination.

2. Paper-base phenolic laminates are slightly more expensive than composition hardboards. They should be hard and relatively free from plasticizers; therefore, an XP is preferable to an XPC or XXXPC. It is sometimes difficult to shear them when their thickness is over $\frac{1}{16}$ in, particularly without warming.

3. Epoxy-glass laminates are the most expensive materials, but they will work well in some applications. However, they promote rather than decrease the tendency toward epoxy smear.

4. Aluminum-clad hardboard and annealed aluminum foil are relatively new and moderately priced. The annealed aluminum surface is harder than the other materials and provides good burr prevention. The core material is free from resins that might cause smearing. Sometimes annealed aluminum foil is used for entry with this system; it provides good burr prevention. The aluminum also serves to clean and cool the drills.

Backup materials should be carefully evaluated; the author believes many of them introduce additional problems such as resin smear. The trend has been toward foil-clad hardboards with hard surfaces and minimum amounts of resin or binder to cause additional smear problems.

15. Drilling Feeds and Speeds The feed rate, in inches per minute, and speeds in revolutions per minute, are critical factors in circuit board drilling. If the feed is too fast, rough holes or drill breakage will be prevalent; if it is too slow, the drill will turn excessively in the hole. The results will be heat buildup, a drill loaded with material, and excessive resin smear along the wall of the hole. Ideally, the speed and feed should be so balanced that the drill cuts a predetermined chip load with each revolution.

Both paper- and glass-base materials usually drill best at 0.001 to 0.003 in of penetration per revolution of the drill. With the smaller drills, however, (0.0625 in and smaller) the chip load can approach 0.009 in per revolution. In general, the speed of the drill bit should not exceed 650 surface feet per minute; at higher speeds excessive heat and resin smear may occur. To calculate speeds and feeds, use the following equations:

$$\text{rpm} = \frac{\text{surface feet/minute} \times 12}{\text{drill diameter} \times \pi}$$

$$\text{Feed rate (inches/minute)} = \text{rpm} \times \text{optimum chip load (inches/revolution)}$$

If the rpm determined exceeds sufficient torque to drill large-diameter holes, decrease the chip load until the spindle has sufficient torque to drill without slowing down. Each drilling operation should develop a drill feed and speed guide such as the one illustrated in Fig. 5.6.

16. Drilling Problems

a. Resin Smear. Much has been published about the cause and effect of resin smear in the holes. The usual effect of resin smear is electrical or mechanical discontinuity or roughened walls of the holes. The cause of smear is resin being heated beyond its melting or heat-distortion point and being drawn through the hole by the drill bit. The key to controlling or eliminating resin smear is control of heat buildup in the drill bit. To eliminate resin smear, the following parameters must be reviewed:

1. Drill geometry. Worn drills, drills with improper geometry, or drills with nonpolished or unequal flutes will increase the amount of cutting energy released into the holes and cause resin smearing.

2. Drill speeds and feeds. Improper selection of drill speeds and feeds can cause the drill to turn in the hole without good continuous chip cutting. Increased heat will result, and chip blockage of the flutes will further cause heat buildup.

3. Improperly cured material. Improperly cured laminate, particularly semicured prepreg in multilayers, will lower the heat distortion or resin melting point and cause resin smear.

4. Improper backup and entry material. Entry and backup material can introduce resins or contaminants that are more difficult to remove than the base resin of the material being drilled. Phenolic smear and epoxy smear will usually not be removed by the same chemicals.

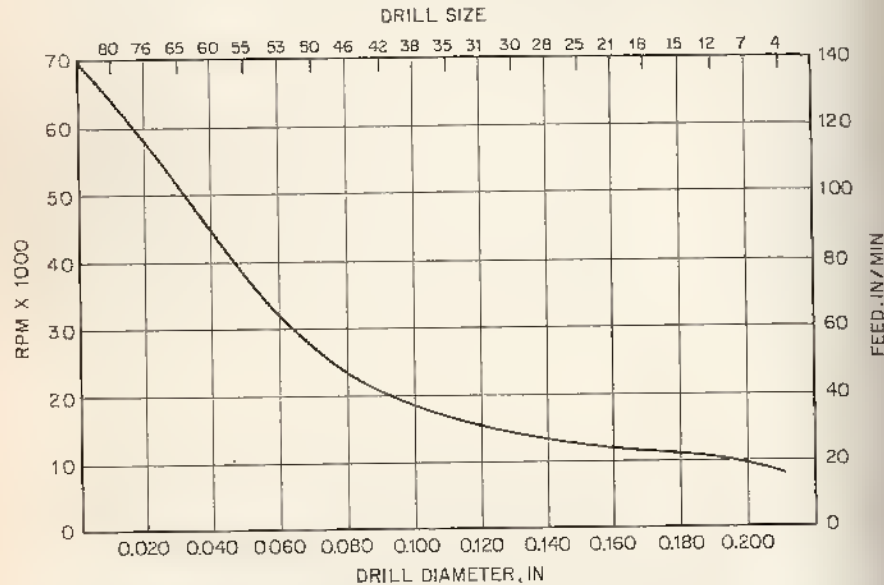


Fig. 5.6 Typical speed and feed chart for epoxy-glass laminate.

5. Stack height and board and backup board design. Too high a stack of material will cause heat buildup. By designing boards with copper lands drilled whenever possible, some heat can be conducted away, which will decrease resin smear. Backup boards with aluminum or copper foil will also help conduct heat generated by drilling.

b. Nail-heading and Burring. Excessive burring (usually burrs of 0.0005 in or greater) and nail-heading occur when drill speeds and feeds are not balanced with drill geometry. Nail-heading and burring are prevalent when the drill is punching and extruding rather than cutting cleanly through copper and base material.

c. Roughened Hole Walls. Roughened hole walls can be due to improper speeds or feeds or poor drill geometry. Poorly engineered or poor-quality drills, with off-center points, unequal flutes, or too low a helix angle, will sometimes cut unevenly and produce rough hole walls. At high speeds small drill bits or bits driven by drills with improper bearings can whip and gouge the hole walls. Worn drill bits exhibit cratering of their cutting edges, which leads to embedding of laminate and thereby results in roughened holes.

d. Hole Location. The location and tolerance of holes are generally determined by drilling machine selection and programming. However, the diameter and geometry of drill bits can contribute to drill wandering. The hardness of entry materials, particularly foils, serves to cut the deviation and wander from proper hole location.

17. Routing Laminates Routing offers the fabricator the advantages of superior edge finishes and closer tolerances than can be obtained from shearing or sawing. It can eliminate the expense of blanking dies and the long lead times associated with hard tooling. Now, when coupled with the use of multiple-spindle machines, the labor costs of

routing can be competitive with those of die blanking. When a design has traces close to the board edge, routing may be the only blanking method that will produce acceptable boards.

Routing is basically a machining operation similar to milling, but it is done at much higher cutter speeds and feed rates. Three routing systems are available to the fabricator: pin routing, stylus or tracer routing, and NC routing.

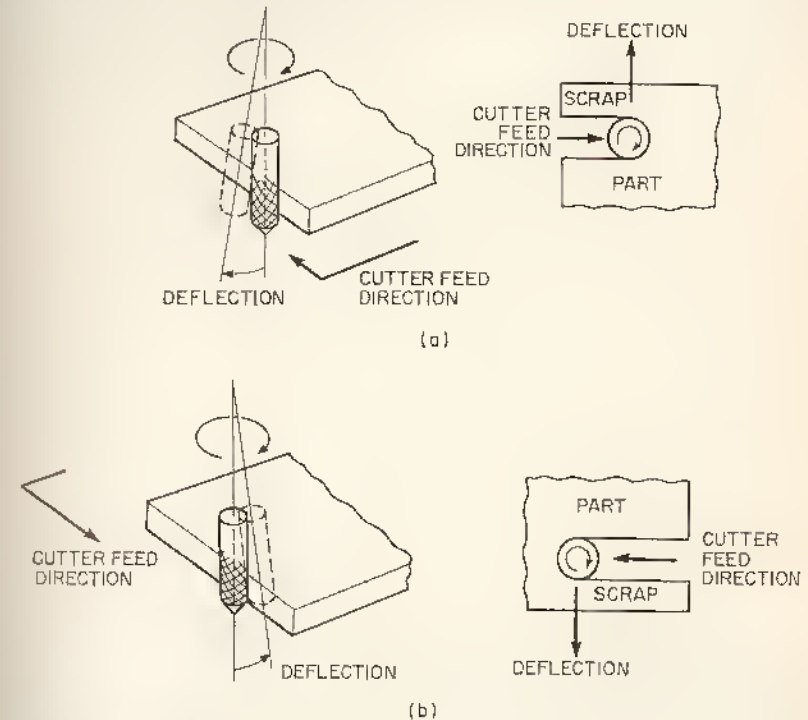


Fig. 5.7 Effect of enter deflection on part size and geometry. (a) Clockwise cutting (recommended for outside cuts) deflects cutter away from part. That leaves outside dimensions large on first pass unless compensated for in programming. (b) Counterclockwise cutting (recommended for inside cuts and pockets) deflects cutter into scrap. Therefore, inside dimensions of holes or cutouts will measure small unless compensated for in programming.

a. Pin Routing. Pin routing requires a machined template, usually of aluminum or steel. The template, which is made to the board outline, includes registration pins for positioning the boards, which are usually stacked three or four high. The package is then routed by tracking the template against a pilot pin that protrudes from the table. The pin height should be a few thousandths below the top of the template; alternatively, a spacer can be used between the template and the adjacent board. The cutter is the same diameter as the pilot pin. Usually two or more passes are required to assure proper tracking of the template, because the cutter tends to force the work away from itself and the pilot pin. Work should be fed against the rotation of the cutter to prevent grabbing. Dwell can cause the cutters to load up with chips and overheat. Pin routing, although labor-intensive not only in cost but also in operator skill, may prove satisfactory for irregular shapes when volumes are low or edge finishes are critical.

b. Stylus or Tracer Routing. In tracer routing a stylus traces the board outline on a template similar to that used for pin routing. The stylus may control the movement of the spindles over a fixed table, or it may control a movable table under fixed spindles. The latter approach is most commonly used with multiple-spindle machines, and the following discussion will concentrate on that machine type.

Templates may be machined to the board outline, with the stylus tracing the external edge, but a closed track 0.005 to 0.010 in wider than the stylus is recommended for greater control and accuracy. In practice, the operator makes his first cut by tracking the outside edge of the template track. (For an outside cut, this assumes tracing clockwise around the template with clockwise cutter rotation.) A second, cleanup pass also is made in a clockwise direction, but with the stylus tracking the inside of the template track. This light cut will relieve most of the cutter load and give the operator better control for final sizing of the board.

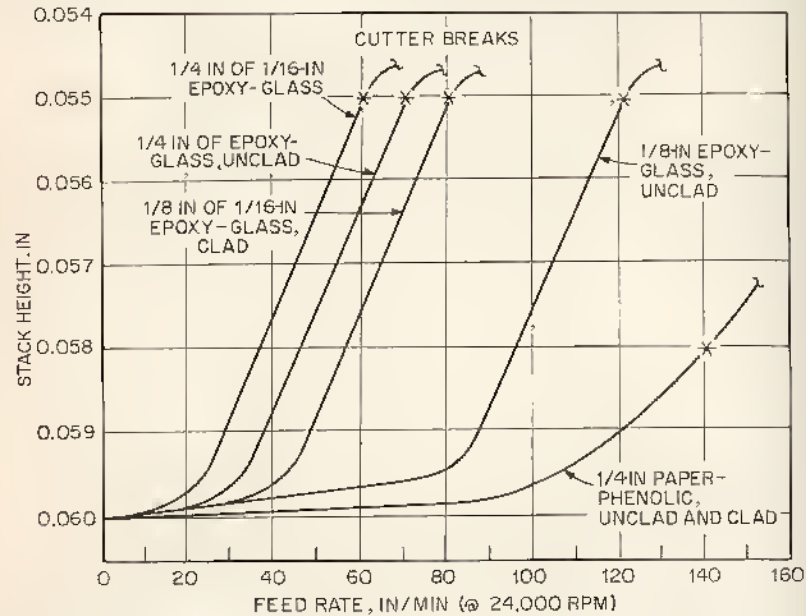


Fig. 5.8 Recommended feed rate, using $\frac{1}{8}$ -in-diameter burr cutter at 24,000 rpm, for varying stack heights of specific thickness of material.

Stylus routing provides better operator control, and accuracy is greater than by pin routing. With multiple-spindle machines, the system can be competitive with die blanking, because up to twenty boards are routed at one time. Sometimes the limiting factor is the ability of the operator to provide the force needed to feed the work past the cutters, but power-assisted table movements can help with some of the load. Tolerances of ± 0.010 in can be achieved at production rates with moderate operator skill and by manipulating stylus sizes; closer tolerances are possible with more skilled operators. Cutter diameters of $\frac{1}{8}$ in can be used successfully on these machines, but cutter breakage is directly related to operator skill and control.

c. Numerical Control Routing. When production volume, flexibility in board design, or short lead time becomes paramount, NC routing may be the best approach to final blanking. Through numerical control of table or spindle movement, the routing operation is reduced to loading and unloading the machine. Complex shapes can be cut to close tolerances in very high volume. Programs can be written quickly, usually under two hours, and design changes can be made in minutes by simple program modifications.

NC router programs consist of a series of simple commands that direct the machine to follow the desired cutter path. Coordinate information is taken from the board blank; added commands must also be entered to direct such functions as feed rate and head up-head down.

Labor costs go down as equipment costs and accuracies are increased by use of the more sophisticated systems. The use of solid carbide cutter bits for routing any of the

commonly used board materials is mandatory. Spindle speeds may vary from 12,000 to above 24,000 rpm, and spindles must have sufficient power to make a cut without an appreciable drop in rpm.

Tooling or registration holes inside the board perimeter are preferred. External tooling holes may be used by leaving tabs at the holes, but the tabs must be removed in a secondary operation.

Although square outside corners can be achieved by routing, all inside corners will have a radius equivalent to the cutter radius. In most designs, that will be acceptable. When square inside corners are required for part clearance, an undercut of 0.015 to 0.030 in along either axis, or a diagonal cut at 45°, will remove material enough that the mating part will have sufficient clearance. The inside corner may also be squared by a secondary operation.

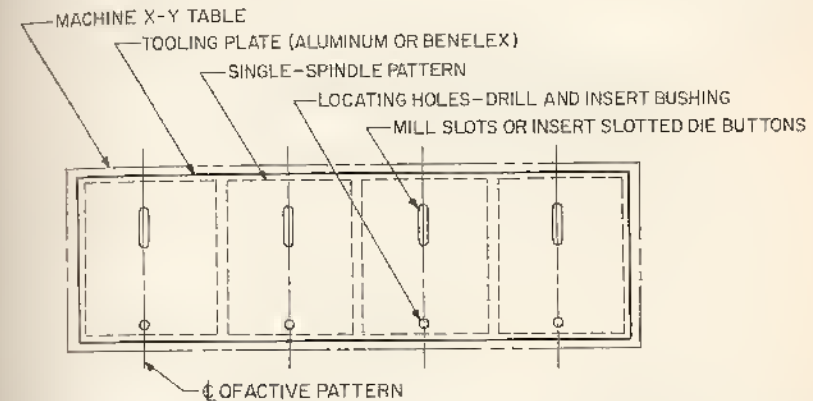


Fig. 5.9 Typical tooling of numerical controlled routing.

The following subsections discuss in detail NC routing procedures.

d. Cutter Offset. Since the cutter must follow a path described by its centerline, it must be offset from the desired board edge by an amount equal to its effective radius. That is the basic cutter radius, and it will vary with the cutter tooth form. Since the cutters deflect during the routing operation, it is necessary to determine the amount of deflection to be added to the basic cutter radius before expending large amounts of time on programming parts (Fig. 5.7).

Variables which affect deflection are thickness, type of material, direction of cut, feed rate, and spindle speed. To reduce those variables, the manufacturer should:

1. Standardize on cutter bit manufacturer, diameter, tooth form, and end cut.
2. Fix spindle speed (24,000 rpm recommended for epoxy-glass laminates, 12,000 rpm for Teflon-glass laminates).
3. Maintain a stack height of 0.250 ± 0.050 in.
4. Rout in clockwise direction on outside cuts, counterclockwise on inside cuts.
5. Standardize on single or double pass.
6. Fix feed rates for given materials. (Note that higher rates will increase part size and slower feed rates will decrease part size.)

e. Direction of Cut. A counterclockwise direction of feed (climb cut) will leave outside corners with slight projections and inside corners with small radii. A clockwise direction of feed (rake cut) will give outside corners a slight radius, and perhaps give inside corners a slight indentation. These irregularities may be eliminated by reducing the feed rate or cutting the part twice.

f. Cutter Speed and Feed Rate. The variables affecting cutter speed are usually limited to the type of laminate being cut and the linear feed rate of the cutter. A cutter rotation of 24,000 rpm and feed rates up to 150 in/min may be used effectively on most laminates. Teflon-glass and similar materials, the laminate binder of which flows at relatively low temperatures, require slower spindle speeds (12,000 rpm) and high feed rates (200 in/min)

to minimize heat generation. The graph of Fig. 5.8 shows recommended feed rates and cutter offsets for most standard laminates at various stack heights. The cutter used is a standard $\frac{1}{8}$ -in-diameter burr type.

g. Cutter Bits. Because of the precise control of table movement in NC routing, cutter bits are not subjected to the shock encountered in pin routing and stylus routing, and therefore small-diameter cutters may be used successfully. However, the fabricator would do well to standardize on $\frac{1}{8}$ -in-diameter cutters because they are suitable for most production work and are readily available from a number of manufacturers in a variety of types. The resulting 0.062-in radius on all inside corners is usually acceptable if the board designer is aware of it.

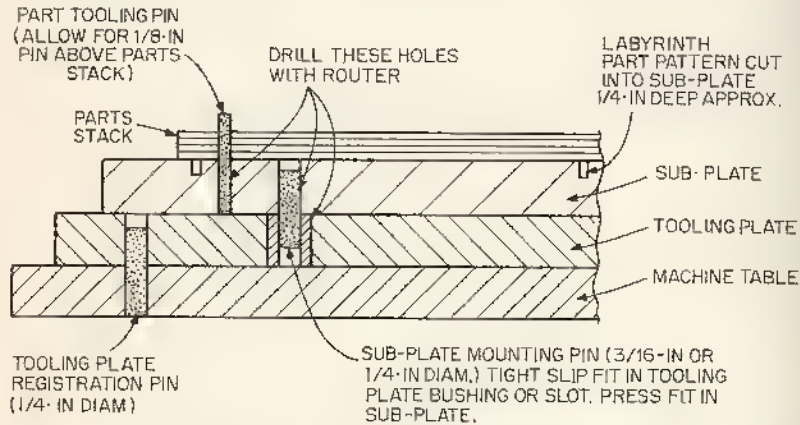


Fig. 5.10 Tooling schematic for numerical controlled routing.

Cutter tooth form is more important in NC than in other routing. Because of the faster feed rates possible, it is important that a cutter have an open tooth form that will release the chip easily and prevent packing. Many standard diamond burrs available on the market will load with chips and fail rapidly. The carbide cutting bit will normally cut in excess of 15,000 linear inches of epoxy-glass laminate before erosion of the teeth renders the cutter ineffective or too small. Cutter life can be prolonged by periodic cleaning to remove epoxy buildup in the reliefs of the teeth. Cleaning can be done by brushing or with chemical cleaners used on drills.

If extremely smooth edges are required, a fluted cutter may be used. Single- or two-fluted cutters with straight flutes should be used when cutting into the foil if minimum burring is desired. It should be noted that such cutters will be more fragile than a standard serrated cutter, and feed speeds should be adjusted accordingly. When a slightly larger

burr can be tolerated, two- and three-flute left-hand spiral cutter hits should be used because of their greater strength. The left-hand spiral will force the workpiece down rather than lift it, assuming a right-hand-turning spindle.

h. Tooling. To simplify tooling and expedite loading and unloading operations, effective hold-down and chip-removal systems should be provided as part of the machine design. Various methods may then be devised to mount the boards to the machine table while properly registering them to facilitate routing the outline. Some machine designs will have shuttle tables available so that loading and unloading may be accomplished while the machine is cutting. Others will utilize quick-change secondary tooling pallets or sub-plates that allow rapid exchange of bench-loaded pallets with only a few seconds between loads.

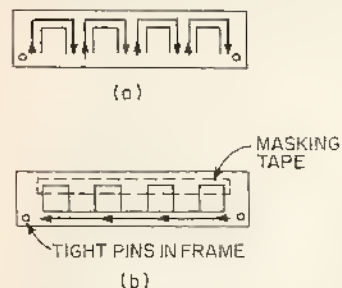


Fig. 5.11 No-internal-pin method. Step 1: Cut three sides (a). Step 2: Apply masking tape (b). Step 3: Cut parts free.

i. Tooling Plates. Tooling plates utilize bushings and a slot on the centerline of active pattern under each spindle. They are doweled to the machine table (Fig. 5.9). The plates may be made by normal machine shop practice, or the router may be used to register and drill its own tooling plate. Mounting pins in the tooling plate should be a light slip fit.

j. Sub-plates. Sub-plates should be made of Benclex, linen phenolic, or other similar material. Sub-plates should have the pattern to be routed cut into their surfaces. The patterns act as labyrinths and aid in chip removal. Part-holding pins should be an

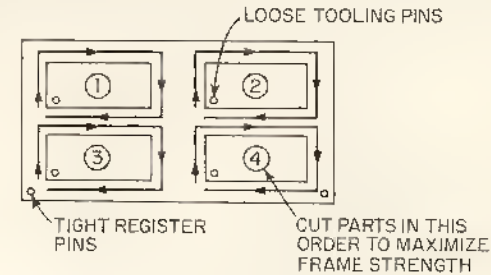


Fig. 5.12 Single-pin method.

interference fit in sub-plates and a tight to loose fit in the part, depending on cutting technique used (Fig. 5.10).

It is recommended that the programmer position the tooling and hold-down pin holes in addition to the routing program. That will provide absolute registration between the sub-plate, full- or part-size, may be used on both sides for a multiplicity of setups and is indexed to the table or master tooling plate with dowel pins to allow multiple tool changes without retooling.

k. Cutting and Holding Techniques. Since the precision required for cutting board outlines, as well as the placement of tooling holes for registering boards, will vary, a number of different cutting and holding methods may be used. Three basic methods are illustrated here. Experimentation will determine which method or combination of methods is most applicable to a particular job. With all methods, the minimum dimension for board separation with a 0.125-in cutter is 0.150 in.

l. No-internal-pin Method. If no internal tooling pins are used, the procedure of Fig. 5.11 may be employed.

Accuracy: ± 0.005 in
Speed: Fast (best used with many small parts on a panel)
Load: One panel high—each station

m. Single-pin Method. The single-pin method is illustrated in Fig. 5.12

Accuracy: ± 0.005 in
Speed: Fast (quick load and unload)
Load: Multiple stacks

n. Double-pin Method. In the two-pin method there is a double pass of cutter offset; see Fig. 5.13. Make two complete passes around each board; the first pass at a recommended feed rate and the second at 200 in/min. Remove scrap after first pass.

Accuracy: ± 0.002 in
Speed: Fast (highest-accuracy system—load and unload slower than single-pin method due to tight pins)
Load: Multiple stacks.

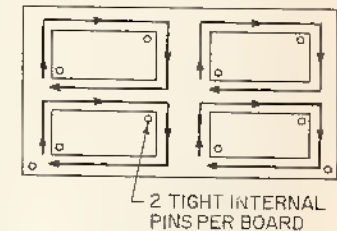


Fig. 5.13 Double-pin method.

Chapter 6

Image Transfer

ERNEST ARMSTRONG

Norcal Tec, Inc., Mountain View, California

EDWARD F. DUFFEK

Adion Engineering Company, Cupertino, California

Introduction	6-2
Photographic Film Preparation	6-3
1. Camera	6-3
2. Film	6-4
3. Camera Room Layout	6-5
4. Film Preparation Work Flow	6-6
5. Photo Film Inspection	6-7
Board Cleaning before Imaging	6-7
6. Vapor Degreasing	6-7
7. Mechanical Cleaning	6-8
8. Rotary Brushing and Hand Scrubbing	6-6
9. Rinsing	6-6
10. Chemical Cleaning; Alkaline Solutions	6-6
11. Acid Dipping	6-9
Photoresists	6-9
12. General	6-9
13. Negative-acting Liquid Photoresists	6-9
14. Positive-acting Liquid Photoresists	6-11
15. Negative-acting Dry Film Resists	6-12
16. Photoresist Application, General	6-14
17. Baking before Exposure	6-17
18. Printing (Exposure)	6-17
19. Developing	6-18
20. Baking after Developing	6-19
21. Removing Resists (Stripping)	6-19
22. Photoresist Facility	6-19

Screen Printing	6-19
23. General	6-19
24. Screen Area Layout	6-20
25. Preparation of Presensitized Stencil	6-20
26. Preparation of Direct Stencil	6-22
27. Removing Screen Stencil	6-23
28. Silk-screen Frames	6-23
29. Squeegees (Hand)	6-24
30. Screen Fabrics	6-24
31. Cleaning the Stretched Screen (Preparing)	6-25
32. Exposing Stencil	6-25
33. Screen Inks	6-26
34. UV-cured Resists	6-27
35. Wet Etchable	6-28
36. Dry-screen Process	6-28
37. Printing Resist Problems	6-28
38. Printing by Hand	6-29
39. Machine Printing	6-29
40. Offset-Press Printing	6-30
41. Hole Pinging	6-31
References	6-31

INTRODUCTION

The basic initial step in the fabrication of a printed board is the generation of a pattern from artwork or photographic film onto a copper- or other metal-clad laminate. In the simplest method, known as the print-and-etch or the etched-foil process, a copper-clad laminate is coated with the exact pattern (positive) in a protective resist material. The unprotected copper areas are subsequently chemically etched to remove the unwanted copper from the laminate. After the etching, the resist is removed, which leaves a replica of the desired pattern in copper.

The second method involves printing the reverse or negative pattern in a resist that leaves the desired areas exposed. The uncovered copper areas are cleaned and electroplated with a protective metal. Plated-through holes are readily incorporated into this approach. The organic resist is removed, and the plated metal (solder, gold, or tin-nickel) now performs as the "resist" in the copper-etching operations. Permanent resists also are applied as solder mask and legend printing.

In either case an organic protective resist which possesses the ability to give the definition and detail required in the circuit must be applied. Chemical resistance and dimensional stability to withstand the action of the processing steps are necessary requirements. Which resist printing method should be chosen depends upon the quality and required line definition of the board, production rates, cost considerations, and compliance with environmental regulations.

Photosensitive resists are capable of the finest line definition. Dry film resists are frequently preferred over liquid resists because they offer good chemical resistance, thick plating capability, and cost savings due to less labor and fewer processing steps. In the print-and-etch process, line definition and resolution depend upon the copper-foil thickness. Typically, 0.005-in line and 0.005-in spacing on 1/4- to 1-oz copper are possible in production. Considerably finer definition is possible in more controlled conditions; for example, 0.001- to 0.003-in lines are possible on thin clad laminate or evaporated metallic films.

Screened resists with medium to long production rates are limited in practice to 0.01- to 0.015-in lines on 1/2- to 1-oz copper. Cost considerations favor screened resists over photosensitive resists, particularly for higher production rates. However, setup costs for screening may exceed photoprinting costs because of the need to make up screens and wash them off. UV- (ultraviolet-) curable and alkali-soluble resist usage has increased for

reasons of possible cost reductions and environmental regulations. Of much interest are screen resists that combine UV curing, alkali etchability, and alkaline stripping properties.

Automatic and semiautomatic screen machine printing offers consistent product and high rates. Offset-printing methods are generally best suited for low-cost high-production rates when definition and resolution are secondary in importance.

The various resist materials and methods of printing the image are considered in this chapter. The printing of the image on the copper-clad laminate includes cleaning, photoresist or screen application, exposure, developing, and baking or ultraviolet curing.

PHOTOGRAPHIC FILM PREPARATION

The positive and negative photographic films taken of the master artwork are used for photographic resist, screen, and lithographic printing processes. What is produced on the film down to the smallest detail will be transmitted throughout all subsequent operations. Equipment and room facilities are prime considerations with regard to the quality of work, fine-line definition, and resolution expected.

1. **Camera** Camera selection should take into consideration the following:

1. Sturdy, vibration-free equipment only.
2. A lens system with no distortion or loss of focus at edges.
3. High-energy lighting which evenly illuminates copy.
4. Back lighting for copyboard. This is essential to avoid fuzzy edges, particularly on taped-up master artwork. It provides contrast of the order of 1000:1, compared with approximately 50:1 provided by front lighting.
5. Maximum size of master artwork to be photographed. This determines copyboard size. In general, the board should be as large as possible.
6. Range of reductions to be made. The range must be established, but as broad a range as possible should be attained.

Most printed circuit board artwork is not laid up greater than 8× final size. When 4× reduction is the maximum available from the camera, the 4× reduction is made first and then the 2× reduction is made from it. The first reduction is to be as large as possible.

An 8:1 artwork for a 4 × 5-in finished card will require a minimum of 32 × 40 in (usable area) copyboard. Since accuracy of final images within standard tolerances of ±0.005 in are more than suitable for most printed boards, it is common practice to use either 2:1 or 4:1 artwork. Tape tolerances can be held to ±0.010 in. That reduces the copyboard usable area requirement to 16 × 20 in for the 4 × 5-in board size. Smaller and less expensive equipment can be used. Commercial equipment with 21 × 25-in and 40 × 48-in copyboards also is available.

Along with the consideration of maximum master artwork expected is the consideration of maximum reduction which may be required. The maximum reduction possible is determined utilizing the equation

$$R = \frac{U - F}{F}$$

where R = maximum reduction attainable

U = maximum attainable distance from lens to copy

F = focal length of lens used

It can be seen that decreasing the focal length increases the maximum allowable reduction. However, lower distortion and other optical considerations favor the use of greater lens-to-copy distances. Focal lengths of 10½ in require a distance of 31.5 in between copy and lens to produce 4× reduction. An extension of the lens-to-copy distance by lengthening the support track coupled with a shorter focal length lens system allows reductions up to 7× on commercially available equipment. Camera units being used for commercial-type printed circuitry have the following attributes:

1. 4× minimum (single-step) reduction capability.
2. Apochromatic (free of spherical aberration) coated, color-corrected, flat-field-type lens.

3. Copyboards no smaller than 16 × 20 in in usable area.
4. Positive copy hold-down, usually by vacuum. Parallelism must be maintained between the copyboard and the film plane to minimize distortion of image.
5. Vibration-free sturdy construction throughout, with solid locking devices which hold the movable camera front and the movable copyboard secure. (Both the front of the camera and the copyboard must be movable in relation to each other in order to obtain focus at the required reduction.)
6. Focus control from back of camera.
7. Aperture adjustment. The setting balances the least residual aberration and fine-line image capability due to diffraction.
8. Automatic timed shutter control.
9. Ground-glass focusing plate at back of camera.
10. Enlarging of at least 2:1 capability as well as reduction capability to allow correction. For example, if the film is to be the same size as the artwork (1:1), it is preferable, rather than try to include allowances for etch undercut in the master artwork, to make the film as a slight enlargement. Uniform shrinkage of copy can also be compensated for in this manner.

11. Front lighting provided by at least two side floodlights. The lights are so attached to the copyboard that the light source intensity and angle of incidence are equal from both sides. The usual minimum angle is 45°. Theoretically, the greater the distance between copy (subject) and light source, the more uniform the illumination. In practice the distance which allows suitable exposure time and uniform illumination is used. Light reflection from the copy into the lens will be at a minimum when the angle between the light path and the lens axis is greater than 45°. Photo-flood lamps are commonly used and are left in operation on a low level controlled by a voltage-varying autotransformer. This reduces heat and glare while adjustments are made and facilitates quick availability of required illumination. Illumination, once established, must remain constant. Exposure time and lens opening can be varied to get the best image for the film being used. Single-color light offers optimum results for most photographic lenses. A green light avoids chromatic aberration, ensures best lens conditions, and simplifies focusing.

2. Film Film used must be fine-grained and have very high contrast, very high acutance,* low photographic speed, and high resolution. Detailed properties and processing techniques are readily available from the trade literature. Orthochromatic film is generally used (except for certain fine-line work) because it is sensitive to green light but not to red light. Therefore, only a red safelight is required, rather than the complete darkness necessary when panchromatic film is used. For finest-line work requiring maximum dimensional stability, glass plates are preferred. They require a camera back with a proper holding frame. A disadvantage is that plates introduce problems in indexing and providing intimate contact during printing. Warp and unevenness in copper-clad laminates, burrs on sheared stock, and plating thickness variations prevent intimate contact during printing of photoresists. Incident light causes loss in definition (poor resolution). Ester-base films (0.007-in preferred) which have been heat-stabilized provide strong, durable, flexible film material which is dimensionally relatively stable and readily conforms to uneven surfaces.

Front and back or different layers are made by master artwork layup on a single transparent or translucent base by using red and blue tapes and exposing through suitable lens filters.

Room light conditions for film handling and processing should be carefully controlled with an adequate number of safelights with proper filters. Simple exposure tests at various points in the room can be made with pieces of the film being processed to determine the effect and number of safelights required. Film developing also must be done in a safelight unless the film being used has been designed for use in normal room (white) light or daylight. Films which can produce either positives or negatives are available. They may be processed in normal room light.

Contact printing is used to make a film positive of the negative produced from the positive artwork copy. Additional negatives can also be made by contact printing onto

*"Acutance" refers to the inherent property of a film that gives it the ability to reproduce very fine detail. The small grain size of the photosensitive crystals form the image precisely.

antopositive film. Equipment is simple and inexpensive. A point source of light provided with intensity range steps and an automatic timer is required. A minimum of 4 ft is recommended from the top of the plate to the light source. Greater distances are even better because intensity throughout the entire area exposed is then equalized. Units are available with a built-in vacuum frame which can be turned over on a swivel. A built-in arc light then exposes the film. Exposure is very fast because of the intensity of light. This, however, may adversely affect film sensitivity.

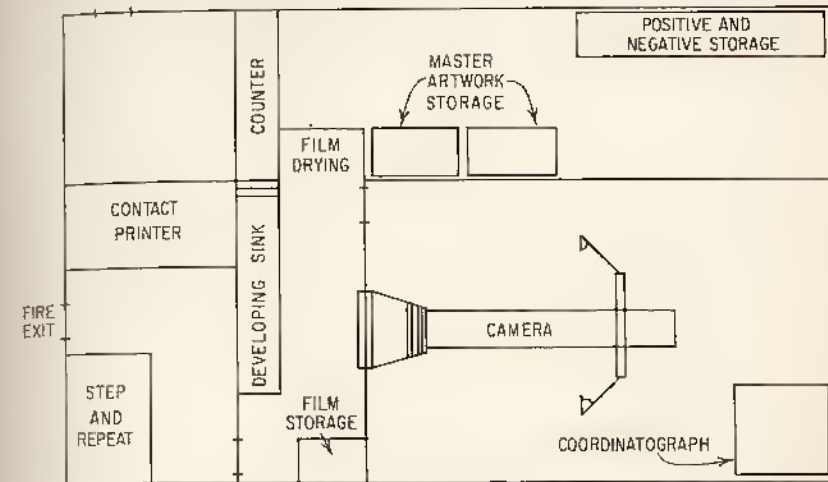


Fig. 6.1 Camera room layout.

A film storage refrigerator in a room with low relative humidity is recommended for bulk storage of unopened packed film. Packages should be removed a day or so in advance of their need to allow them to reach room temperature before opening. This prevents moisture from condensing on the cold surfaces. In addition to bulk storage, a lightproof drawer or cabinet must be provided for daily use. Steel cabinets with light-sealed steel drawers are recommended for storing film. Master artwork and the finished positives and negatives are also stored in clean steel cabinets, but lightproofing is not required. The cabinets should be located in the air-conditioned camera receiving room area.

a. Diao Imaging Film. Molecular photo polymer-based photographic film now available can be used instead of the silver halide type. The following advantages are offered by it:

1. A darkroom is not required.
2. Water processing is not required. Film is exposed and then developed in a machine with ammonia vapor. This eliminates the need for trays of chemicals and water bath temperature controls.
3. Through-put is much faster because drying time is not required.
4. Dimensional stability is equivalent to that of silver halide films.
5. Films are transparent amber or burgundy in color. This allows see-through visual alignment with holes or pads on the printed board.
6. Positive or negative film can be made from available materials. The positive type has long-term stability, however.

3. Camera Room Layout A typical camera-room layout is shown in Fig. 6.1. Air conditioning to a room temperature of $68 \pm 2^\circ\text{F}$ and mean relative humidity of 50 percent are required. The incoming conditioned and filtered air should establish a slight positive pressure with respect to outside rooms. This keeps dust out. Lightproof air outlets must be provided to prevent pressure buildup. The air-conditioning unit, pump, and motors are to be located as far away from the camera room as possible and mounted on solid, independent (isolated from building floor) cement foundation blocks on shock mounts. Compressors and other vibration-generating equipment should also be located as far away as

possible from the camera, as well as being isolated and mounted so as to minimize vibration. These precautions are essential if very fine line definition is to be achieved. The types of films generally used have long exposure times, and any vibration of camera equipment is emphasized and transmitted to the film.

Floors should be tiled or similarly covered to allow complete cleaning in order to reduce dust. Water-trap vacuum cleaners or other good vacuum cleaners should be used to keep floors, shelves, etc., dust-free. All foot traffic that is not necessary should be eliminated. Clean gratings or mats at the entrance to the photographic area are helpful.

The water used in solution makeup should be deionized or distilled and be solids-free. Deionized water is preferred for rinses, but clean, well-filtered (5- μ m or less) tap water is suitable. Straight tap water is commonly used, but it can result in scratched or smeared images because of the presence of particulate matter. A fine-particle filter can be installed as a preventive measure.

The darkroom area in which film is to be handled and processed must be lightproofed, including doors, cracks, ventilators, windows, and light switches. The door to the area must have an internal lock to ensure against accidental entry while film is being processed unless a safelight walk-through is provided.

Electrical switches and controls for automatic camera focusing and shutter operation must be located in the darkroom behind the camera. A similar set of controls is normally available in front of the camera. Room light and copy illumination in the room in front of the camera should be controllable from the darkroom behind the camera. Similar controls are to be provided for copy backlighting.

4. Film Preparation Work Flow A typical work flow with steps taken for film preparation includes the following:

1. Artwork, which has been approved and signed off by quality control, is received and entered in the log. Quality-control inspection prior to sign-off usually utilizes an optical coordinatograph located in the room in front of the camera. Optical drill tape programming machines are even more accurate for inspection.
2. File-card tags for placing on finished film are prepared.
3. Camera lights are turned on low.
4. Camera is set.
 - a. Copyboard and lens are set for reduction required.
 - b. Light integrator is set for reduction being made.
 - c. Shutter on front of camera is set. Suitable allowances for thick-taped artwork should be made.
5. Vacuum pump for copyboard is turned on.
6. Clean white paper sheet is placed on copyboard.
7. Artwork is placed on a white sheet, flat and wrinkle-free.
8. Mylar* cover sheet is rolled over master artwork. This must press artwork down firmly and also be wrinkle-free. With the camera copy set up, the remaining work is to be done in the darkroom behind the camera with the room light on.
9. Developer, stop bath, fixer, and other solutions used to develop the film are prepared (gloves on). Hands are washed and dried when work is completed.
10. Room lights in front of camera are turned off to prevent unwanted copy illumination.
11. Unless walk-through is provided, the darkroom door is locked and the red (safe) light is turned on. The safelight switch should also turn on a red light by the door outside the darkroom. The room light is then dimmed to allow final focusing and positioning of center of copy image onto ground-glass plate at camera back. Once the camera is set, the room light is turned off completely, leaving only the red safelight illuminated.
12. The glass plate door is removed and replaced with the vacuum camera back door.
13. The film is placed in a centered position in the holding board.
14. The vacuum pump is turned on for film hold-down.
15. The film door is closed.
16. The button on the automatic shutter, timer, and exposure control box is pressed; this turns on lights and ensures amount needed for exposure.

*Registered trademark of E. I. du Pont de Nemours & Company.

17. Once the picture is taken, the film is removed and developed (avoid handling). Agitation by tilting the solution trays is of great value in getting suitable film developing.
18. After the film is developed and washed, it is dried in the air at room temperature in a dust-free compartment.
19. Once the film is dry, the identifying tags are attached to the edge of it.
20. The negative is used to contact-print similar films to produce the positive image. Filter of the positives may be used as an intermediary film image to be reduced by being photographed in the same way as the master artwork. When the negative is used, it must be turned over to correct for reversal of the image.
21. When indexed locating pins (standardized tooling) are used, they are pinched in the film while the film is in the camera area.
22. Master artwork is filed.
23. Photographic films are placed in protective envelopes; one set is sent to production; and a master set is filed. The master set is never allowed to leave the camera area. Hanging-type storage is preferred.

When size allows it, a contact print of the master artwork is often made to ensure against changes which may take place because of tape shrinkage or loosening.

Problems that arise are generally attributable to improper exposure or developing, dust, solids in solutions, light leaks, forced drying, finger marks, improper focus, improper reduction, or logged film.

5. Photo Film Inspection The film negative or positive must be inspected and retouched, since every error and blemish will be reproduced faithfully on the copper-clad laminate surface during printing. The film may also be extensively changed to obtain a different circuit pattern or incorporate minor changes.

Faults in the film are generally in two areas; black spots in the transparent area show up as pinholes in the resist or clear spots in the pattern areas. Black spots in the transparency are corrected by removal on the emulsion side. Clear spots in the dark circuit area are corrected by retouch paint.

The actual fabrication of the board is now ready to start. Cleaning is the first step.

BOARD CLEANING BEFORE IMAGING

Improper cleaning prior to resist application is probably the greatest source of printing, etching, and plating problems. Poor photo- or screen-resist adhesion, pinholes, incomplete etching (copper left in patchy areas where not wanted), peeling, and other plating defects are generally related to improper cleaning practices.

Starting with the base material, boards should be handled with soil-free gloves. Finger-print marks "etch" into the surface and often result in peeling of resist or plating.

The basic cleaning steps are given in the following sections.

6. Vapor Degreasing Copper-clad laminates or other laminated materials may require solvent-vapor degreasing to remove oil and soil contamination left on the surface by the preceding manufacturing operations. The contamination includes the mold-release materials used to prevent the laminating press platens from sticking to the copper surfaces. Silicone greases once used as mold releases are most difficult (if not impossible) to remove by degreasing and cleaning.

Two of the most commonly used solvents for vapor degreasing are trichloroethylene (TCE) and perchloroethylene. TCE operates at a lower temperature of $90 \pm 5^\circ\text{C}$. The immersion time varies, but up to 2 min is normally adequate. The TCE must be high-purity-stabilized and be kept clean and dry.

Vapor degreasing may attack and loosen the bond between the copper cladding and the insulating substrate. As processes are established, adhesion and bond-strength tests are required before and after vapor degreasing or other vigorous cleaning procedures.

Care must be taken to protect operators from the hot, toxic degreasing vapors. Since the solvent removes oils so effectively, skin irritation is likely to result from contact with the hot vapors or liquid. Airborne vapor, although limited by the cooling coils, does exist when boards are removed. The best practical protection is to locate the vapor degreaser inside a chemical fume hood and so connect the heater that it turns off when the cooling

water is not flowing. A safety device is a heat sensor, located above the cooling coil and so attached that heating is turned off once a set temperature maximum is reached.

Improvements in laminate manufacturing, chemical cleaners, and machine scrubbers have reduced the need for vapor degreasing.

7. Mechanical Cleaning Mechanical cleaning operations begin with sanding when the boards have been drilled or with abrasive scrubbing on undrilled boards. Hand vibrators have frequently been used, but machines are more effective and practical for production. Wet sanding with 280- to 600-grit water-resistant abrasive paper is preferred. Dry sanding emulsions in the copper materials which may lead to adhesion problems. Also, dry sanding is not effective.

Abrasive fiber rollers on machines are used wet.* The machines also blow the boards dry. Other machines use brushes and a slurry of pumice.† There has been some objection to them on the ground of their leaving embedded or surface residue pumice. Properly working machines present little problem, however, and the boards come out essentially dry. Steam heat, when available, is used to blow-dry wet surfaces. The effect is the formation of a surface with which resists and further plating have good adhesion. In general, printing without sanding on glossy or mirror-bright surfaces will result in resist lifting during etching or plating.

8. Rotary Brushing and Hand Scrubbing Rotary brushing follows sanding in the case of plated-through-hole panels and after vapor degreasing of copper laminates not drilled. Scrubbing is done with a mild alkali cleaner and 4F pumice by using a round fiber brush powered by air. A lightweight motor and shaft are preferred. The speed of rotation is kept low enough to hold the abrasive media in the brush. Scrubbing is done wet by using a rubber-type mat underboard with little or no pressing. It is also possible to use flat metal plates covered with water-resistant abrasive paper. Volume production requires machine scrubbing.

9. Rinsing A high-pressure deionized water spray which ensures complete rinsing is preferred. If one is not available, thorough rinsing must be done with clean tap water. Boards can then be either blown dry with clean dry air or run through drying machines. Ultrasonic units or more wash rinses may be used, depending on the type of pattern to be printed.

10. Chemical Cleaning; Alkaline Solutions Chemical cleaning follows mechanical cleaning of both plated and unplated substrates. Chemical cleaning is generally required regardless of the need for mechanical cleaning. The first step is alkali cleaning. Proprietary metal cleaners are used most frequently. Typical concentrations are between 4 and 12 oz/gal (8 oz nominal) at temperatures between 150 and 180°F. Unless there are larger amounts of surface oils, 160°F nominal is easily maintained and is usually adequate. When a higher temperature is used, evaporation losses must be replenished by frequent water additions. Carbon-steel or stainless-steel tanks are used. Commercial cleaners are composed for the most part of phosphates (trisodium phosphate), sodium carbonate, sodium hydroxide, wetting agents, and chelating agents. Simplified analytical control kits are available. Since most cleaners are low in cost, it is standard practice to discard them and make them up new when there is any sign of poor cleaning. Electrocleaning is done under similar conditions. Commercial solutions have been formulated to be used with direct current, either cathodic or anodic or both. Time in the cleaner will vary with type of soil, temperature, solution composition, current applied, and material being cleaned. One to three minutes for copper-laminate boards at 4 to 8 V has been found effective. Ultrasonic aqueous or solvent cleaning also has found use for special and rigid cleaning schedules.

Rinsing after alkali cleaning is effectively carried out by using filtered (oil- and solids-free) tap water. Water immersion, followed by a strong spray, ensures complete removal of cleaners. Deionized or distilled water may also be used, but since acids and other rinses are to follow, its cost is not warranted at this point. Neutral or acidic cleaners are

*Sommer and Maca Glass Machinery Co., 5501 West Ogden Ave., Chicago, IL 60650

†Bilco, Grandview Blvd., Zetionople, PA 16063

sometimes preferred for reason of the attack of hot alkaline solutions on exposed epoxy or polyimide substrates.

11. Acid Dipping Acid dipping and surface etching follows the rinse to neutralize residual alkali and remove metallic oxides. A 20 to 40 percent (by volume) HCl or 10 to 20 percent (by volume) H₂SO₄ solution is adequate. Following an acid dip the boards may be processed through these solutions: copper etch, water rinse, H₂SO₄ (10 to 20 percent), tap water spray rinse, and deionized or distilled water spray rinse. Finally, the boards are blown dry with clean filtered air.

The adhesion and quality of photoresist films on copper are very dependent on surface preparation. Acid and water rinsing should be the last treatment. A surface finish of 10 to 20 μm rms prepared with complete chemical etching and polishing exhibiting a dull surface is preferred. Mechanical abrasion may leave metallic slivers, high spots, etc., which result in pinholes in the resist, but when it is properly done, excellent adhesion is obtained. Printed boards are held at the edges (using rubber gloves) and on a slight angle vertically so that water drains as it is blown off one corner. When boards are processed in racks, they also should be allowed to drain vertically. Hot air and hot water promote oxidation and should not be used. Oven drying at 90°C (inert atmosphere preferred) for at least 15 min is used to ensure complete water removal prior to printing. Conversion coatings (phosphates, oxides) should not be used unless resist adhesion cannot be obtained by regular processing.

PHOTORESISTS

12. General Liquid photosensitive resists are thin coatings produced from organic solutions which, when exposed to light of the proper wavelength, are chemically changed in their solubility to certain solvents (developers). Two types are available: negative-acting and positive-acting. The negative-acting resist is initially a mixture which is soluble in its developer but after light exposure becomes polymerized and insoluble to the developer. Exposure is made through a film pattern. The unexposed resist is selectively dissolved, softened, or washed away, which leaves the desired resist pattern on the clad laminate. Positive-acting resists work in the opposite fashion; exposure to light makes the polymer mixture soluble in the developer. The resist image is frequently dyed to make it visible for inspection and retouching. The resist pattern that remains after development (and post-baking in some cases) is insoluble and chemically resistant to the cleaning, plating, and etching solutions used in the production of printed boards. A variety of references on photoresist applications may be cited: general procedures,¹⁻¹² fine-line etching,^{13,14} and electroforming.¹⁵ The majority of photoresist materials are available as proprietary materials and are covered by patents. They have for the most part been formulated with well-characterized chemical reaction products; they remain noncrystalline during light exposure; and they have good chemical resistance to and dimensional stability in the agents involved in board processing. Although the composition of proprietary materials has not been released, typical examples may be given to indicate their nature: vinyl cinnamate copolymers,¹⁶ benzal acetophenones,¹⁷ acetophenone plus cinnamate,¹⁸ quaternary salts,¹⁹ and azide polymers.²⁰

13. Negative-acting Liquid Photoresists The primary negative-acting liquid resists are KPR* and DCR.† Selection and typical processing will be considered here.

Formulated originally for making letterpress printing plates and lithographic plates, KPR has become a widely used photoresist in printed circuit manufacturing. It is used primarily when the screen printing and other high-volume methods do not offer fine enough line definition or when a small lot of boards is to be produced. Because of its low solids content and low viscosity (11.7 to 13.1 cP at 25°C) it provides good resolution. Dry KPR dipped coatings are about 0.0001 in thick, which is sufficient for printed board manufacturing. KPR has the advantage of being easily dyed black or blue for inspection and retouching.

The KPR series is generally recommended when copper or copper alloys and anodized

*Product of Eastman Kodak Co., Rochester, N.Y.

†Product of Thiokol/Dynachem Corp., Santa Ana, Calif.

aluminum are to be coated. Other resists cause permanent staining or discoloration on copper. The ultraviolet light sensitivity of KPR is given in the literature.³ In practice, a gold or other ultraviolet-free safelight or orange-tinted glass is used in the resist application and print area. After coating, but prior to exposure, boards are stored in a dust- and light-free cabinet. It has been reported that properly prepared and stored KPR-coated boards have been printed and etched after 2 years' storage. Lower temperatures and humidity controls extend the applied resist life.

a. Problems Associated with KPR

1. Property changes from batch to batch. Remedy: Filter, by using positive pressure, through a 5- μ m (for example, Millipore*) filter or equivalent. This is preferred to vacuum filtration, which tends to remove some of the solvents. Commercially available KPR is filtered.

After filtration, viscosity should be standardized by using either KPR thinner to lower or KPL (filtered) to raise viscosity. Prepared formulations are available. Viscosity measurements are preferably taken at a standard temperature and humidity. A rotary-spindle viscosimeter used at approximately 60 rpm is particularly suitable for the purpose. Permanent-stand explosion-proof models are available.† KPR has a viscosity of about 13 to 14 cP at 25°C and a specific gravity of 0.99 to 1.00. Solids are approximately 7.4 percent by weight. Liquid should be stored in a sealed, dark bottle (small volume preferred) in a light-free, cool area (do not freeze).

2. Poor definition with fine-line etching. Remedy: Reduce viscosity of KPR by adding solvent to get a thinner coat of resist.

3. Poor adhesion and breakdown in etch. Remedy: This is normally due to inadequate or poor cleaning or improper processing.

b. Processing Steps for KPR

1. Apply standardized viscosity, filtered KPR to previously cleaned and dried board by the selected method of application. The spray technique is recommended on drilled plated-through-hole boards.

2. Prebake by baking at 90°C for 15 min after first allowing the boards to air-dry until they are dry to the touch. (Thick coatings will take longer.) When greater chemical resistance is required, temperatures up to 120°C can be used for a longer time.

3. Print (exposure). Photographic film negatives (dark field) are used for print and etch. Photographic film positives (clear field) are used when printing for electroplating. Expose by using a light source such as a carbon-arc or mercury-vapor lamp. Proper exposure time will vary with equipment. Exposure test procedures have been reported.³

Typical time for spray-coated KPR, with mercury-vapor tubes approximately 2½ in from the plate, is about 2 min. Exposure tests must be made to determine the optimum time. A carbon-arc light source can also be used, and white flame-arc exposure time will be less than for the mercury lamp (about ¼ to 1½ min at approximately 72 in with a 75-A arc).

During exposure, the precoated board with the photographic film on it (emulsion side touching the KPR coat) is held in intimate contact in a vacuum frame or a photographer's frame. Writing, letters, numbers, etc., are in proper reading order during this operation.

4. Develop. (a) Use trichloroethylene vapor degreaser. Hang in the vapor only 3 to 4 min (190°F). Spray hot TCE (liquid) for 15 s coming out of tank. Thicker coats will take longer. Boards with holes must be washed out completely by forcing developer through the holes. (b) Or use KPR developer. (c) Or use xylene 2-min immersion, 1:1 xylene-Fetocol spray 15 s. Dip in Fetocol. Air-dry.

5. Dye. Immerse the dry photoprinted and developed board in KPR dye for 30 s. Remove and allow excess to drain off. With cold tap water (65 to 76°F), spray-rinse off excess dye.

6. Retouch. Use plater's resist to touch up one side of the board while making sure all the copper which is to remain is covered. Remove photoresist from areas to be etched away. Oven-dry retouched side at 90°C until dry. Cool and do the other side.

7. Postbake (for plating). Bake at 120°C for 15 min to complete resin cure; bake longer if coat is thick. Postbake is often deleted on print-and-etch copper-clad boards.

*Millipore Filter Corporation, Box 427T, Bedford, Mass.; also see Refs. 3 & 4.

†Brookfield Viscometer, Brookfield Engineering Laboratories, Inc., Stoughton, Mass.; also see Refs. 3 & 4 for other suppliers.

c. Other Kodak Resists. Later versions of KPR and other Kodak negative series resists have been developed for plating and chemical milling. Consult the first edition of this book and Eastman Kodak Company, Rochester, N.Y., for details on these materials.

d. DCR* (Dynamchem Photo Resist) (Liquid Applied). Somewhat different procedures are followed. This material is narrower in its light-sensitive range than the Kodak similar type of resist (320- to 390-nm specific range). In contrast to KPR, DCR is not designed to be developed in TCE vapor degreasers. Stabilized chlorinated hydrocarbons (for example, Chlorothane-Nuf) can be used. Development time is critical (5 to 10 s maximum) with slight agitation.

Viscosity of available materials is 9.94 cP. Solids are approximately 20 percent by weight, and the specific gravity is 0.90. Process as follows:

1. Clean the copper-clad laminate.
2. Oven-dry in air-circulating oven at 50°C for approximately 20 min.
3. Cool to room temperature.
4. Resist-coat. Dip, spray, and whirl resist as received. Thin with up to 20 percent thinner for roller coating.
5. Dry at 115 to 125°F in a circulating oven. Do not exceed 125°F. This is critical.
6. Expose 30 s to 2 min, typical time for carbon arc. Mercury vapor takes two to three times longer. Black light, mercury-quartz tube, quartz-iodine lamps, or tungsten arc light can also be used. Light-sensitive peak at 360 nm.
7. Develop: Immerse in cold Dynamchem developer 5 to 10 s; 15 to 30 s is recommended at room temperature. Heated solutions are preferred for best results. This is a critical step, and overdeveloping results in poor adhesion of the resist. A drawback lies in the fact that TCE, used frequently to develop resists, attacks DCR.
8. Dye immediately after developing (direct transfer). DCR dye takes 5 to 10 s. Dye-Developer, a Dynamchem one-step dye and developer, can be used in place of steps 7 and 8.

e. Problems with DCR

1. The exposure required by thin films has been found to be up to 18 times as long with the same light sources and intensity as with KPR (mercury arc).

2. Lifting during etch when not postbaked. The 200 to 250°F temperature recommended by the manufacturer must not be exceeded because charring and lifting will occur, starting at the edges.

3. Baked-on DCR is difficult to remove. TCE, as recommended by the manufacturer, does not work unless it is hot and is accompanied by hand scrubbing. Commercial strippers work well, however.

Good chemical resistance to plating solutions has been reported. Pattern plating of copper and plated metal resists has been done on a commercial basis. When thicker coats are used, the exposure problem mentioned above should be considered. Recent DCR formulations are claimed to alleviate these problems.

f. Water-soluble, Colloidal Negative Resist. A water-soluble, liquid negative resist is available for print-and-etch and photochemical metal-etching operations. The material is protein-based and, with the exception of spraying, can be applied by standard methods.

Coatings are processed by prebaking at 70°C, exposure, development in deionized water spray, and postbaking for curing. Stripping is in 15 percent sodium hydroxide. Advantages claimed are low costs and less pollution.

14. **Positive-acting Liquid Photoresists** The positive-acting resists have been marketed as AZ-series materials (modified and refined for printed circuit and microcircuit application by licensing agreements), and AZ-340.† These resists differ from the negative-acting resists in that they become soluble in their developer when exposed to light. The image is printed by using a positive or open-field film. The methods of applying photoresists given in Sec. 16 show some comparisons and advantages of this type of resist. Since the remaining resin, after exposure, developing, etc., is still light-sensitive, multilayer etching possibilities are available. In this case, baking after coating is not required. Typical processing steps for AZ resists are as follows:

*Product of Thiokol/Dynamchem Corp., Santa Ana, Calif.

†Product of Dow Chemical, Midland, Mich.

‡Trade mark Shipley Company Inc., Newton, Mass.

1 **Cleaning.** Degreasing, alkali (electro or soak) rinse, acid dip, and dry are typical steps for copper and its alloys.

2 **Coating.** Whirl or spray coating, dip, roller, and other conventional techniques are possible, but spray at 20 to 40 psi is preferred for plated-through holes. Airless sprays are also suitable.

3 **Drying.** Oven-dry immediately after coating at 120 to 180°F in a circulating-type oven which is well vented; 15 to 20 min at 150°F is sufficient. Infrared may also be used.

4 **Exposing.** Sensitivity range is between 200 and 600 nm, but since most glasses do not transmit lower than 320-nm wavelength, the peak sensitivity through glass is at about 340 nm. Exposure time varies with thickness and light source intensity, and exposure tests must be run to establish time. The image exposed to light is developed off the board, and that not exposed remains intact.

5 **Developing.** AZ developer dissolves the exposed resists very quickly and completely. The developer is an alkali-water solution which is rinsed away in running water. Deionized water spray is preferred. When the properly exposed image does not develop in about 2 min, the developer must be discarded. After thorough rinsing, a 5- to 10-s dip in 10 percent HCl solution is recommended for the plating of copper plated boards. This step is followed by thorough rinsing, preferably with a deionized water spray rinse.

6 **Postbaking.** Generally, postbaking is not recommended, since a high-temperature exposure can be harmful to the resist. For maximum chemical resistance to strong etchants or for plating at high current densities, a bake at 150°F for 20 to 30 min can be used.

7 **Etching.** Etching can be done by using chromic-sulfuric acid solution, ferric chloride, ammonium persulfate, cupric chloride, or other common acid mixtures. White light should be avoided, since the material is made more soluble by exposure.

8 **Plating.** Most printed circuit acid plating solutions can be used with AZ. For optimum results postbaking and white light exclusion are required. Time in alkali solutions should be held to a minimum for all products.

9 **Resist removal.** This is exceedingly rapid with a variety of organic solvents. Removal takes place in ½ to 30 s when resist has not been postbaked and in a longer time when it has been. Acetone and ketones are preferred. Ethyl acetate, cellulose, and other organic solvents also work well. Strong alkalis at 150 to 180°F are effective, particularly when ultraviolet light exposure is made first. Their use makes possible the avoidance of organic solvents which might affect substrates. See Chap. 8.

10 **Rinsing.** Thorough rinsing and in some cases scrubbing and rinsing.

11 **Drying.** Blow dry with filtered air.

15. Negative-acting Dry Film Resists Dry film photopolymer resist films from 0.0005 to 0.003 in thick are commercially available and have replaced liquid resist in many applications. The thicker materials are suitable for use in the most stringent etching and plating operations. Thick pattern plating, for example, can be done on fine lines and fine spacing without problems associated with overplate (mushroom side effect) and chemical attack on the resist.

Thinner coatings of the order of 0.00075 in thick are well suited to print-and-etch applications such as the inner layers of multilayer boards. Care must be taken on thinner resists, which are somewhat brittle and may flake at edges. The laminates are sold in rolls of various lengths and widths up to 24 in.

The photopolymer is sandwiched between a layer of polyester film on one side and a polypropylene sheet on the other side (Fig. 6.2). Both cover sheets protect the photopolymer from damage by abrasion. The polyester sheet further serves as a stable dimensioned base against which photographic film can be placed for exposure.

Light sensitivity is such that all processing before exposure must be done under a gold fluorescent or an equivalent ultraviolet-free safelight. Exposure equipment must be capable of delivering high-intensity ultraviolet light which is collimated and free of excessive heat. High-pressure mercury vapor lamps and mercury quartz lamps work well. Cold-cathode and fluorescent "black lights" are not suitable for fine-line work. The faster the exposure, the finer and more acute the edges will be.

The light is bent as it passes through the polyester cover sheet, but it is transmitted straight to the surface coated. The exposed photopolymer is polymerized with a consequent change in viscosity. Unexposed photopolymer remains unchanged. The higher the energy of the exposure light, the quicker the polymerization will take place. The result is

better resolution. The reverse effect will result from slow exposures from low-energy sources. Heat also causes unwanted polymerization.

Photopolymer resist films, as reported by Butler,²¹ are oxygen-inhibited systems by design. With no ultraviolet light and in the presence of oxygen (air) the resist will not polymerize.

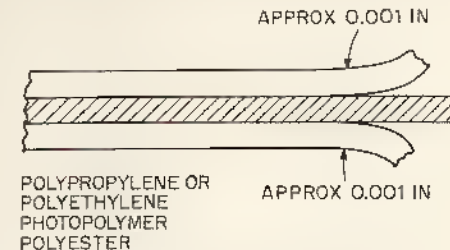


Fig. 6.2 Dry-film photoresist sandwich structure.

a. **Processing.** Typical processing involves the following steps:

1. **Cleaning.**

(a.) Degrease in trichloroethylene vapor.

(b.) Abrasive-cleaner scrub. (Pumice)

(c.) Thorough water rinse. (No residue)

(d.) Blow dry with clean air.

(e.) Oven dry 130 to 150°F, 10 to 15 min.

2 **Surface coating (lamination).** The photopolymer is applied to the board surface by using heat and pressure (Fig. 6.3); it is fed off a roll and onto the board surface as the board passes through the machine.

Boards still hot from oven drying are fed through the machine with heated laminating rolls at 230 to 260°F. Preheating is essential because the laminating rolls cannot supply enough heat at the speeds required. Early machines hurried up heating rollers in trying to overcome the heat deficit by applying more heat. Heated transfer rolls are effective. During this process the polyethylene cover sheet is removed and taken up on a roller. The polyester sheet remains on top of the photopolymer as a protector.

A cooling and normalizing period of 15 min is preferred for dimensional stability. Exposure can come immediately after laminating, but dimensional stability is better after normalizing.

The excess film around the board can be trimmed off with a knife, but care must be taken not to loosen the cover film. Machines for trimming are available.*

3. **Exposure.** Exposure is carried out by registration of the photographic film (emulsion side) onto the polyester sheet; the two are then placed in a vacuum frame for intimate contact between surfaces. The proper time will depend upon light intensity, temperature, photopolymer type and thickness, and type of equipment being used. Photopolymer film manufacture data sheets state that optimum exposure times will give a solid 8 to 10 on a Stouffer[†] 21-step sensitivity guide. Typical exposure times for a variety of exposure machines is given in photopolymer resist films manufacturers' data sheets.‡

4 **Developing.** Developing after stabilization (1 to 15 min) starts with the peeling of the protective polyester cover sheet. The peeling should not be done until just before developing. See Fig. 6.4 for the developing machine.

Developing is done in a 1,1,1-trichloroethane§ solvent. Best results have been obtained by using a hard spray at 60 to 70°F. Vigorous agitation is necessary.

Test panels determine developing time. Optimum time is the time necessary to just dissolve the unexposed resist and thus leave a clear exposed panel image (positive for print-and-etch, negative for pattern plating). Temperature, spray pressure and patterns, and dissolved resist in developer will affect developing time.

*Prime Research & Development Co., Santa Ana, Calif.

†Dynamchem Corporation, Santa Fe Springs, Calif.

‡Stouffer 21 Step Sensitivity Guide, Stouffer Graphic Arts Co., South Bend, Indiana.

§Riston, a product of E. I. du Pont de Nemours & Company

¶Du Pont's methyl chloroform and Dow Chemical's Chlorothane-Nu are suitable.

5 Postbaking. Baking after developing at temperatures between 120 and 150°F for 10 min aids pattern-plating adhesion to a considerable extent. The chemical resistance is enhanced by baking.

These resists may be used in all acid plating baths and etched in all of the common acid etchants. Fluoroborate tin or solder may cause some problems. Pyrophosphate copper, strong alkaline cleaners, and hot ammonium etchants will affect these resists.

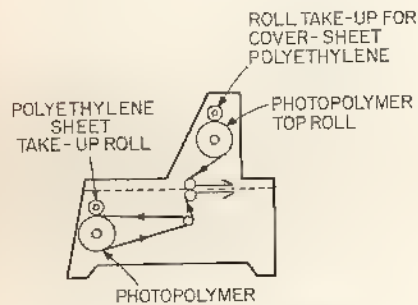


Fig. 6.3 Dry-film photopolymer laminator.

6 Stripping. Stripping can be done by using any of a number of cold commercial strippers.

b. Water-Soluble Dry Film. These systems require alkaline processes. Processing is similar to that described except that developing is done in 2 to 4 percent trisodium phosphate (TSP) or sodium carbonate solutions. Antifoam materials are added to the developer for machine operation. Rinses and residues must be treated in waste disposal systems. See Chap. 9 on disposal.

Acid etchants and plating solutions are inert to this material. Stripping can be done quickly in alkaline solutions or cleaners at 130 to 160°F. A 3 percent solution of sodium hydroxide by weight in water at about 140°F will blister this resist, which can then be lifted or sprayed off with water.

Dry-film photopolymer resists formulated specifically for pyrophosphate plating solutions and ammonia-type alkaline etchants are available. Removal of these films is somewhat difficult. Hot (140 to 180°F) water solution of sodium hydroxide (1.5 percent by weight) is used. Care must be taken with tin and solder boards.

c. Problems Encountered with Dry-Film Resists

1. Thick-film problems are to a great extent traceable to improper surface preparation and to improper application.

2. Lack of adhesion is caused most often by inadequate cleaning or insufficient bonding temperature and pressure. Clean according to the procedure outlined in this section. Make sure that boards are heated prior to entering laminator and that heated rollers are at the right temperature. Finally, check exposure conditions. Incomplete polymerization can also result in poor adhesion.

When plating times in alkaline or near-alkaline solution are long, some lifting may occur at edges of circuit patterns. High current densities exaggerate this problem. Always use the correct resist type for plating or etching applications.

3. Resist in unwanted areas can generally be traced to problems in the film. For scratches and pinholes, touch up films, not boards.

16. Photoresist Application, General The application of "dry film" by using specialized laminators has been discussed above (Sec. 15a, Processing). Resists applied in liquid form lend themselves to several methods of application. Spraying is the simplest and least expensive method, particularly with the negative-acting resists, when plated-through

holes or other cutouts are present. Whatever the method of application, the major requirements for good resolution and definition are that the photoresist film be uniform, pinhole-free, and strongly adherent to the base material. That is why the cleaning and preparation steps prior to resist coating cannot be overemphasized. Moisture absorbed on the substrate or contained in the resist must be kept at a minimum because it also may be the cause of pinholes and poor adhesion. The resist should be applied in a room with about 50 percent humidity. Oven baking or TCE rinsing (hot) is used to eliminate the last traces of water on the substrate. Each of the application techniques will be discussed.

a. Dip Coating and Controlled Withdrawal. Dip coating is a suitable means of applying liquid resist, but a low-viscosity resist will leave a very thin and valueless coating. A thicker coating can be obtained when withdrawal from the liquid is controlled. Slower withdrawal results in the thicker coating found suitable for plating applications.

Dip coating is the easiest and quickest way of coating both sides and is very suitable for many applications. Containers such as glass, stainless steel, and earthenware are suitable. They should be covered when not in use, and they should have a depth greater than the largest workpiece to be processed. That allows contaminating solids to settle and remain undisturbed. Narrow tanks are commonly used. Vertical dipping is preferred to horizontal dipping because of space savings, less disturbance of solids, and smaller surface area exposed with less evaporation and dust collection. By removing printed panels from the tank vertically and slowly, thicker and more even coats can be attained. On cutout and blind holes, the draining resist tends to run in streaks of uneven thickness. The result is incomplete developing. Streaks may also arise from clamps and other holding devices, which should be kept out of the resist.

Heavier thicknesses can be attained by multiple dipping, but each coating must be baked dry prior to the next dip. That ensures complete drying and prevents surface checks or cracks. The adhesion and chemical resistance also are improved. When only one side is to be printed and etched, an even coating thickness can be achieved by dipping and then rapidly removing the board and laying it flat with the desired side up.

It must be remembered that plastic substrates and adhesive materials are affected by the organic solvents in photoresists, and hence the time of immersion of single metal-clad boards should not be greater than the time needed for wetting and good coverage. Filtering is required to remove contaminating solids and oxidized resin products. A 5- μ m filter has been used effectively.

Oven drying can be done under vacuum to remove moisture from resist, but care must be taken to avoid foaming. Lost solvents can be replaced. Solvent loss and moisture absorption, even at room temperatures, are common to dipping operations.

b. Whirl (Spin) Coating. Whirl coating techniques utilize a centrifugal force to distribute the resist over the area to be coated. Coatings are very even in thickness and nearly pore-free. Wheels from 6 to 48 in in diameter have been used. Panels are usually centered on the flat surface and held in place by threaded pin stops, a balanced holding frame, or double-sided tape. The resist material is applied at the center as the wheel turns, or it is poured on and the wheel is then accelerated. Thickness of coating can be controlled by the viscosity of the resist and the revolution and acceleration of the wheel. The speed of the wheel is conveniently controlled by means of a power regulator controlling a driving motor. Variable-voltage regulators and timers wired in series have been used successfully.

A major disadvantage of whirl coating is encountered when double-sided boards are to be coated, particularly boards with through holes. Resist tends to run around the edge or through the holes and cause streaks on the reverse side. Streaks may also form along the holes on the top side. This technique is therefore not suitable for coating plated-through-hole boards. Problems of excessively thin coatings are related to excessive wheel speeds and low resist viscosity.

Hot air, infrared, and other devices can be used to force-dry the resist while spinning takes place. In that event extreme care must be taken to ensure complete venting and fire prevention of evaporating solvents. Spinners or whirlers should be contained in a shielded frame to prevent damage from flying parts. This is especially true of larger and higher-speed units. Splash guards and adequate drains are required. Since the resists are not water soluble, filters should be located at drain outlets and kept clean at all times. Coating should be done in a hood located in a positive-pressure room or in a positive-pressure laminar-flow hood. Clean-room-type conditions are essential for fine-line photoprinting.

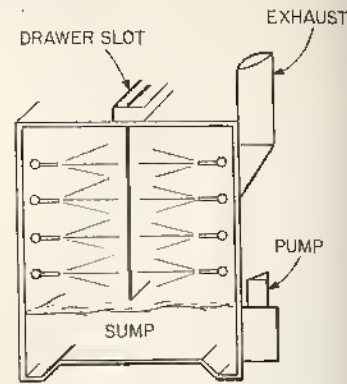


Fig. 6.4 Dry-film-developing machine (vertical spray type).

Shafts must be mounted on sealed bearings, and no oil contamination should be possible.

c. Roller Coating Most photoresists can be successfully roller-coated. Steel rollers with rubber or similar synthetic materials are necessary, since board thicknesses may vary and low and high spots are encountered. Kimbled stainless-steel rollers are suggested, but uneven board thickness and warpage cause problems. The size of roller coaters is a limitation, since the machinery is large and heavy. Perhaps the greatest problem, however, arises from the fact that a large surface area of the resist is exposed on the rollers and in the reservoir. Keeping rollers and working parts clean is therefore time-consuming and difficult.

A possible advantage of roller coating over dipping and whirling is that far less resist gets into holes and/or on the other side of the board. However, application to panel-plated through-hole boards would require extremely even plating or considerable sanding to level surfaces. Particulate matter and air may get into the resist and may result in pinholes and rough spots which will not develop clean.

Unless very fine line work is to be done, offset printing or screening using plating or acid resist inks is recommended. Costs are much lower, and printing and developing steps are eliminated.

d. Spray Coating²⁷ Spray coating can be done with common paint spray equipment (vacuum air gun*). The siphon-type units are commonly used, although airbrush sprays are also used. Recent equipment which does not atomize the lacquer is preferred by some users.

Most of the photoresists can be spray-coated. High-solid mixtures produce thick coatings and tend to run in streaks. It is possible, however, to thin the materials and use two coats rather than try to adjust the gun to produce a fine spray.

Spraying is a good method for keeping excess amounts of resist out of holes when boards are drilled and through-hole-plated. The main objections are resist loss and lack of uniformity. High cost is generally due to overspray (the spraying carried on beyond the edge of boards). Other forms of spraying (airless and electrostatic) are more economical, but their practical utilization with photoresists is limited.

Thickness variations are caused by the operator technique and process control. Viscosity, air pressure, fan pattern, distance of gun from work, degree of atomizing, temperature, humidity, and daily gun cleaning and maintenance must be standardized.

The greatest source of photoresist spray problems is the clogging of the fluid nozzle and/or solid particle buildup on the seat of the needle valve. Cleaning with solvent or resist thinner to remove all remaining solids usually corrects the problem. Intermittent spraying is evidence of this type of problem.

Before spraying the work, check the pattern and adjust the nozzle until the vertical fan forms an ellipse at 6 to 8 in from the test panel. Panel spraying should start at the bottom and proceed across on a line parallel to the surface at a distance of 6 to 8 in. Air used for resist spraying must be free of moisture, solids, and oil. Solid-screen cartridge filters often used with water-trap housings are necessary in compressed-air lines. Dry nitrogen is preferred to compressed air, since it is generally cleaner. Pressures commonly used are between 5 and 20 psi.

e. Electrostatic Spraying. Electrostatic spraying processes utilize the principle of charging the work and the resin particles with opposite electrostatic potentials. Air pressure is still used to atomize the lacquer and help direct it toward the work. Considerable material saving is possible because of reduced overspray. The method is adaptable to automation with increased uniformity. Some photoresist materials can be applied by the method, but equipment costs are high and the process is most applicable to large-volume production.

f. Airless Electrostatic Spraying. Airless electrostatic spraying is an improved version whereby air is not needed to propel the resin toward the work. Fluid is pumped to a rotating head, and centrifugal force carries it to the edge of the head where high electrostatic forces atomize it. The mist formed is highly charged and is drawn to the oppositely charged work by electrostatic attraction. The method virtually eliminates overspray losses. It is also more suitable for automation and large-volume production, but it has the disadvantage of high equipment costs.

*The term "siphon" is commonly used to represent the vacuum system, although the physical principle is the reduction of pressure due to air forced past the opening into the liquid chamber.

g. Ovens. Ovens and other baking techniques used to dry the photoresists properly before exposure and after developing have utilized practically all heat sources: open flames, infrared lamps, gas ovens, electric ovens, plus others. Regardless of which method is used, basic conditions must exist to ensure uniform results. Temperatures must be controllable within $\pm 5^\circ\text{F}$ and sufficient to ensure proper baking. A standard baking time must be established. A common but unreliable practice is to place the boards in an oven and leave them there until the operator is ready to remove them. Once proper baking is done, boards must be removed from the oven and placed in a lightproof storage cabinet where they can cool and wait for exposure without ill effects.

Electric forced-draft ovens which are well insulated and vented to the outside provide temperature gradients of no more than $\pm 1^\circ\text{F}$. Readily available models with temperatures up to 200°C and thermostatic control of $\pm 1^\circ\text{C}$ from room temperature to maximum are required. The judicious use of filters and location of ovens are important considerations in avoiding dust and contamination problems. Cleanliness and consistency, as well as danger, rule out the use of open flames, since photoresists are flammable and toxic.

In selecting an oven, the largest size panel which is expected and the quantities to be processed per day must be considered. A good practice is to have several smaller ovens which will accommodate the largest panels rather than one large oven. This allows greater versatility. It must also be borne in mind that at least two ovens are needed, since prebake and bake temperatures are considerably different for some applications. Boards should stand vertically in the oven and with a sufficient free flow of warm air around them. The vertical position reduces the amount of dust accumulation.

17. Baking before Exposure A bake cycle prior to printing is usually included for photoresist coatings, particularly those on copper-clad laminates. The step starts with drying at room temperature. Prebaking is done at 80 to 120°C (175 to 250°F) depending on type and thickness of resist and the process steps to follow. Infrared lamps may also be used. Prebaking may have the effect of reducing definition and can also result in excessive undercut due to the brittle nature of the resist when print-and-etch boards are processed. For plating applications baking must be more complete to ensure against breakdown in the solutions.

18. Printing (Exposure) The exposing of dry films has been discussed in Sec. 15. The data to be covered in this section apply to the liquid photoresists, but they may also be applicable to the dry films.

a. General. After the copper-clad laminate has been coated with photoresist and prebaked, the next step in the process is to expose or contact-print the desired pattern. In the case of the negative-acting resists, the purpose of the printing operation is to bring about a polymerization that makes the exposed photoresist insoluble in the developer solutions.

The ultimate in resolution is obtained by exposing with a point-type light source or collimated light beam. However, light sources not regarded as point sources, such as the carbon-arc, mercury-vapor, and in some cases tungsten lamps, are available and are well suited for this purpose. The optimum exposure time will vary with the type and thickness of photoresist and the distance from the light source.

b. Double-sided Printing. Matching, tooling, or indexing holes in the film and the board offer the best means of good front-to-back registration when both sides are to be printed. When indexing pins or holes are not used, a common procedure is that of laying the top and bottom films together and lining them up on a light table (emulsion side to emulsion side). Once aligned, the films are adhesive-taped together at one edge on a spacer strip which is the same thickness as the board to be printed. Alignment is checked and adjustment is made until correct. Spacers can be made from strips of copper-clad laminated material.

When indexing is undesirable for short production runs, a greater degree of front-to-back registration can be attained through the use of a pair of tool-steel bars aligned as near as possible to one-half the thickness of the boards to be processed (Fig. 6.5). At the end of each bar there is a drilled line and reamed hole to receive a precision-ground steel indexing pin. The films are aligned on a light table as before, except that this time the films are glossy side up (emulsion sides out). A microscope will be useful during

alignment. Apply double-sided tape to both films along one common edge (do not remove protective backing).

With indexing pins inserted into plates A and B and with plate A on top of B but with $\frac{1}{16}$ in or more between the two plates, insert the film edge covered with double-sided tape (remove the protective backing prior to insertion). This must be done with only one edge free while the main body of film is held firmly in position on the light table with tape. Once the films are inserted between the two steel plates, press them tightly together and thereby cause the exposed tape adhesive to stick to them. So secured, the other tapes holding the film to the table can be removed.

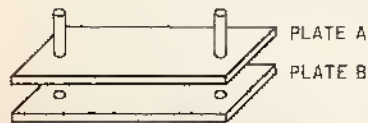


Fig. 6.5 Double-sided printing aligning bars.

Emulsion to emulsion (both inside) is attained by removing indexing pins and placing plate A, with film attached, under plate B and then again replacing the indexing pin. Hinge arrangements also will work, but more loose play is involved (clearance tolerance allows movement). The film must allow a holding area at the edge.

Aligning emulsion to emulsion, punching, and then attaching to the frame with secured indexing pins is also a method used.

Still another technique is to line up the negatives over holes, tape in place at one edge only, and then slip the positive between the negative and the board, align the positive with the negative, tape, and remove the negative (Fig. 6.6).

c. *Coving*. The ultimate in fine-line resolution is affected by the light-scattering process during exposure. Positive- and negative-acting resists can leave a small rounded bottom corner between the developed resist and the surface coated. This is referred to as

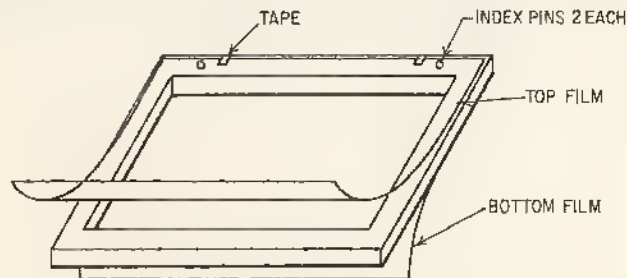


Fig. 6.6 Photoresist exposure jig for double-sided printing.

“coving.” It is due to diffusion of light under the film from a side-angle source and also reflection from the copper surface. Coving may be minimized by using thinner coats of photoresist, unidirectional lighting, and proper spraying or spinning speeds and by avoiding predrying and light exposure.

The reverse action is observed for the positive-acting resists. For that reason positive-acting resists should provide somewhat better registration. Those effects are illustrated in Fig. 6.7.

d. *Work Area Requirements*. A major requirement is that the exposure operation must be carried out in a dust- and contamination-free environment. This is necessary not only to have perfect reproduction of the pattern but also to reduce the damage caused by mechanical contact of the film and substrate. Projection printing would eliminate the problem.

When the film positives or negatives are received for printing, there is little that can be done to correct mistakes or oversights in artwork or photography. Corner rounding (fillet formation) due to distortion in the optical system, density of film, definition, fine sharp lines, curved sides of internal squares, etc., must be allowed for or included in master artwork.⁴

19. Developing The developing operation consists in the removal of the soluble photoresist material. Which developing solution is used depends upon the type of photoresist. Various solvents, such as TCE, mixtures of alcohol and Stoddard solvent, xylene, and

commercial developers, are available for negative-acting resists. The positive-acting resists are readily solubilized by acetone, ketones, esters, and commercially available alkali-base developers.

The developing operation is preferably carried out in a deep narrow tank with an internal spray. Simple immersion may also be used, but spraying will generally yield the ultimate in resolution. Ultrasonic agitation during immersion has been found to aid the development process. Several developing tanks may be used to ensure cleanout.

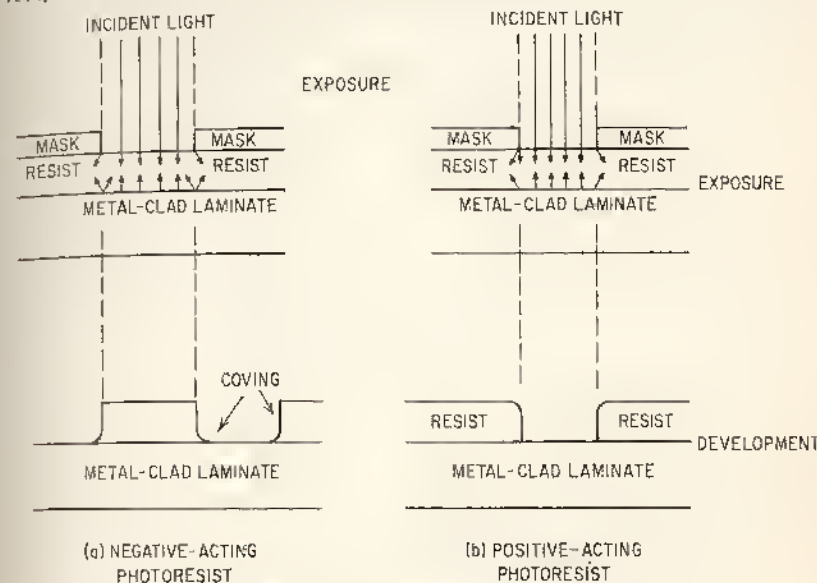


Fig. 6.7 Coving of negative- and positive-acting photoresists.

The time for the developing operation is dependent upon the type and thickness of resist, the developer, and the method used. Typical times are about 2 to 3 min.

a. *Rinsing*. After the developing is complete, the coated substrate is thoroughly rinsed in the developing solution, TCE, xylene, or acetone. A deionized water spray rinse or a dye bath followed by a water rinse completes the developing operation.

b. *Dyeing*. Dyeing is used to reduce retouching time on regular printed boards either plated or print-and-etched. When the ultimate in fine-line images or the best in chemical resistance is required, dyeing is not recommended. Techniques using photographic developers with silver have been used in place of dyeing, but they involve problems of poor color contrast, silver contamination, and discoloration.

20. Baking after Developing For certain applications, a postbake may improve the chemical resistance of the photoresist image during plating or etching operations. An air-circulating oven is used for the purpose. Temperatures and time vary with the type and thickness of resist. The value of the postbake must be determined in each application, since it will affect adhesion of the resist. It generally is not required on print-and-etch copper-clad laminates. Retouching is done prior to postbake operations.

21. Removing Resists (Stripping) Removal of resist image is carried out after the required plating or etching steps. Stripping of resist is detailed in Chap. 8.

22. Photoresist Facility A typical layout of a photoresist area is shown in Fig. 6.8.

SCREEN PRINTING

23. General The techniques of screen printing are particularly adaptable to low-cost print-and-etch and plated printed boards, when the ultimate in resolution and definition

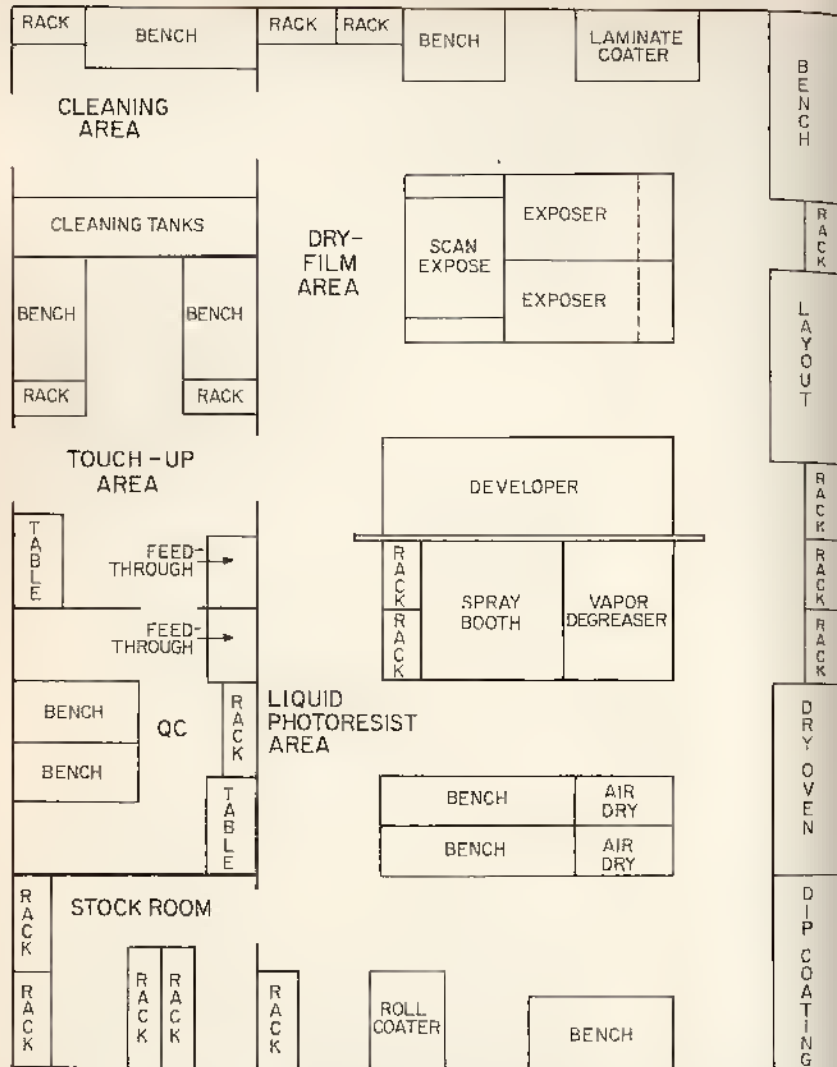


Fig. 6.8 Photoresist area layout.

are not required. Basically it is a stencil operation which depends on the transfer of a resist to the copper-clad laminate surface using a stencil image which is the circuit design. The stencil is firmly attached to the surface of a silk, nylon, or stainless-steel screen stretched drumhead-tight on a frame suspended a short distance above the laminate.

When the direct emulsion process is used, the stencil materials are poured onto the screen and then squeegeed through the mesh, where they dry and are ready to expose and be developed. Resist is forced through the open areas of the screen and onto the laminate by the wiping motion of a rubber squeegee. Additional references on the more common aspects of screen printing may be consulted.^{22,23}

24. Screen Area Layout A typical screen printing area layout is shown in Fig.6.9.

25. Preparation of Presensitized Stencil Screen masking is carried out by transfer of a photographically developed stencil film from its temporary backing to the screen. It is

necessary to consider that photostencil line width will increase slightly over the photographic film size. A requirement is that the stencil material be water-soluble and not be affected by organic solvents, particularly if a vinyl-based resist lacquer is to be used. This type of resist uses lacquer thinners, ketones, or chlorinated hydrocarbons for solvents.

The following is a typical process used to make the screen ready for printing. Standardization is recommended, since it is essential for consistent results.

a. Purpose. This process establishes the procedure to be followed in screen preparation using heat or chemical-hardenable, presensitized photostencil film.

b. Material and Equipment Required.

1. Silk, nylon, or wire cloth mounted in a printing frame. Wash screen by using a fine-bristle brush and fine abrasive cleaner. Cleaning must be followed by thorough water rinsing, making certain that no particles remain in the mesh. Air-dry.
2. Specific photosensitive film being used, including the manufacturer, is to be completely identified and entered here.
3. Vacuum table printer. (Specify type and manufacturer.)
4. Block-out tape.
5. Block-out resin. (Specify type and manufacturer.)
6. Stiff plastic squeegee (2 in long), 60 to 65 Shore durometer.
7. Developer. Specify type and manufacturer. Mix according to manufacturer's directions.
8. Brush with a fine-bristle brush.
9. Build up board, which fits loosely inside the frame area.

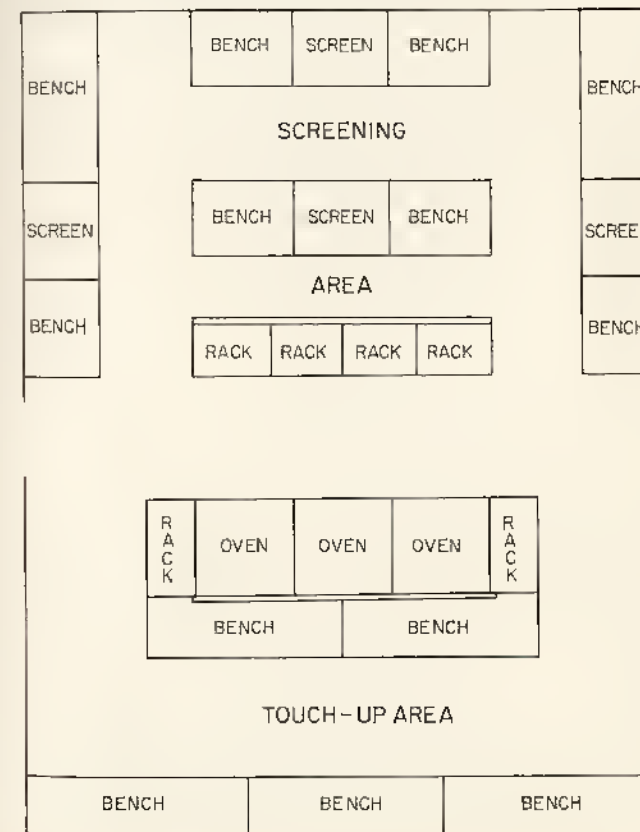


Fig. 6.9 Screen printing area layout.

c. Procedure.

1. Place the stencil film with the plastic side up, into the Arc exposur, or other such vacuum-printing frame. (Make sure both sides are clean.)
2. Place the photographic positive or negative on the stencil film so that the image is reversed.
3. Close exposure frame and start vacuum.
4. Once vacuum is drawn, inspect to make sure that the two films are tightly pressed together. Also make sure that the photographic film is located properly with the emulsion side down.
5. Turn table over and expose for the time determined by test exposures.
6. When exposure is completed, turn table over, turn off vacuum pump, and remove stencil film. Overexposure is indicated by the formation of tiny blisters on the film and poor adhesion to the base.
7. Immerse the stencil film in the prepared tray of developer (see Material and Equipment Required, 7) with the emulsion side up. Developer has a one-day life.
8. Make sure film is immersed; then rock the tray back and forth continuously for 30 s to 1 min maximum. Spent developer causes film to wrinkle and float off plastic sheet backing.
9. Remove film from developer and immerse in warm water (about 110°F).
10. Wash until the design comes completely clean.
11. Water-spray using warm water (110 to 115°F).
12. Place wet film, with the plastic side down, on clean, smooth, flat surface. Use buildup board. Frame edges should not touch surface.
13. Place the screen over the film in position desired and press lightly.
14. Using clean newsprint (not newspaper) or paper towels, blot lightly. Use enough blotting paper to remove excess moisture from the film.
15. Air-dry the film. Forced drying will cause excessive shrinkage and distortion.
16. When the film is dry, peel off the plastic backing sheet. (If the sheet does not peel off easily, allow further drying.) Handle photographic film by edges only.
17. After removal of the backing sheet, wash this side of the screen with naphtha or benzene to remove the adhesive residues.

d. Conversion of the Stencil Film. (Heat-hardened type only.)

1. Oven-bake at 325 to 350°F for 10 min or follow 2, below.
2. With household iron set at "wool," using backup block, iron film side until red color turns purple.
3. Solvent conversion is possible but is not so desirable as ironing, since dimensions may change.

e. **Blocking Out.** Block out area around pattern using a water-soluble block-out. Two coats are applied evenly with a small squeegee.

f. Removal of the Stencil Film from Screen After Using.

1. Spray with hot steam until all the stencil dissolves and washes away from the metal fabric. Removal from silk is not practical.
2. Scrub both sides, using a small amount of abrasive cleaner.
3. Rinse thoroughly and blow dry, using filtered air. NOTE. Presensitized photostencil is light- and heat-sensitive.

26. Preparation of Direct Stencil Direct stencils are generally made by two methods. One involves the use of an emulsion of polyvinyl acetate and polyvinyl alcohol. The unsensitized emulsions, which are sensitized just prior to using, are available commercially. Sensitizing is done by adding ammonium dichromate. Screens coated with the sensitized emulsion are stored three days maximum even in the dark.

Another type of direct emulsion sensitizer is a diazo compound. The diazo compounds are stable several months in solution, and sensitized screens can be stored for up to one year in the dark. Exposure time for the emulsions sensitized with diazo compound must be two or three times longer than when dichromates are used.²⁵ The direct emulsions are applied with a squeegee or squeegeed. The screen is allowed to dry in a vertical position in the dark or safe-light area. Once dry, the sensitizing emulsion is applied to the mesh with a flat scoop coater. Storage in a clean drying area follows.

When dry, the screen is ready for exposure. Photo film is attached to the bottom side of the screen. The entire chase is then put into a vacuum frame which has a thick sponge-

rubber draw-down cover that conforms to the inside of the chase when the vacuum is drawn.

Developing after exposure is simply a matter of washing with a warm-water spray until the image areas are clean. The screen is then blotted with lint-free newsprint. Light-pressure clean air blowing across the surface speeds drying. Hot air used to speed up drying may cause shrinkage problems. The dry screen is then ready to use.

The second type of direct stencil uses a vinyl or polyester film backup sheet 0.002 to 0.005 in thick that is uniformly coated with unsensitized polyvinyl alcohol-polyvinyl acetate. This system is essentially the same as the first one described except the emulsion is in dry form and is easier to apply. A sheet of the material is cut over the size to be printed. It is placed emulsion side up on a plate glass support that is slightly smaller than the inside of the chase. The chase is placed on top of the glass-supported stencil material. The underside of the screen is in contact with the emulsion film.

Next sensitize the liquid emulsion that comes with this type of material or sensitize with ammonium dichromate according to manufacturer's directions. With the sensitized emulsion in contact with the screen, pour additional sensitizer at one end of the screen and squeegee across in one pass using a rounded squeegee. Block out the rest of the open mesh with the same sensitized emulsion.

When ready to expose, follow the same procedure as outlined in the preceding sections. Variations of these techniques can be used for thinner films. Once exposed, the paper, vinyl, or polyester support sheet is removed. The image is exposed, developed, and processed as described.

The polyvinyl acetate system is used because it ensures that the emulsion will not retract to the wires in the mesh and thus leave a sawtooth effect.

Weak points in the polyvinyl alcohol system are as follows:

1. The wire may not be entirely surrounded by the emulsion.
2. Compared with other systems, emulsion washout is slow.
3. Support of underside of screen during washout is awkward.

27. Removing Screen Stencils The stencil materials which hold up best during printing are the most difficult to remove, especially those that are baked. Direct emulsion stencils can be removed from nylon, polyester, and stainless steel (not silk) by using sodium hypochlorite (bleaching solutions) or mixed solution of 20-volume hydrogen peroxide and sodium hypochlorite. Soak, brush, and use high-pressure hot-water spray.

Gelatin base emulsions are used on presensitized sheet films and can be removed from stainless steel, nylon, or polyester materials by using sodium hypochlorite or commercially available enzyme strippers. Avoid burn-out techniques.

28. Silk-screen Frames

a. **Wood.** The lowest initial cost frames are made of kiln-dried white pine. They are relatively light in weight, and warping is minimal. They are used with silk or nylon fabrics. Print-and-etch boards with wide tolerances can be made by using these frames. Problems common to all wood frames include maintaining uniform tension, staple damage with consequent repair time, and dimension instability.

b. **Aluminum.** Lightweight cast-aluminum frames are available commercially. They are made up of two main parts: the outer frame and an inner frame to which stainless steel, nylon, or silk is attached by the supplier. The inner frame, called a filler, is replaceable. The screen is mounted and tightened by means of a flange or channel.

Aluminum frames, although higher in initial cost, are longer-lasting than wood frames. Since stainless-steel meshes can be used and kept taut in them, dimensional stability to closer tolerances can be maintained.

c. **Fabricated Aluminum.** Another available aluminum frame is shown in Fig. 6.10. It utilizes extruded aluminum channeled square tubing with welded aluminum corners. A channel is provided for the inner holding bars, and an outer channel is provided to accommodate the screw used for tension adjustment. Inserts for threads extend the life of the frame.

d. Screen Frame Recommendations

1. Standardize on one frame size large enough for the biggest panel.
2. Use only metal frames such as the fabricated aluminum or similar original design for production work. Initial cost is high, but the frames have a long life. They are

dimensionally stable, can take the tension required for a stainless-steel mesh, and are inert to solvents.

3. Tension should be maintained at a standard value. Excessive tension results in tearing of the screen and requires excessive squeegee pressures which cannot be maintained by operators. Insufficient tension results in jagged-edged lines due to slow retrieval and poor registration. The torque value should be maintained so that the best print and registration are attained with the off-contact distance set an average of 0.052 in or less. That will vary with panel size.

4. Screen holding bias must provide positive holding with no slipping under tension.

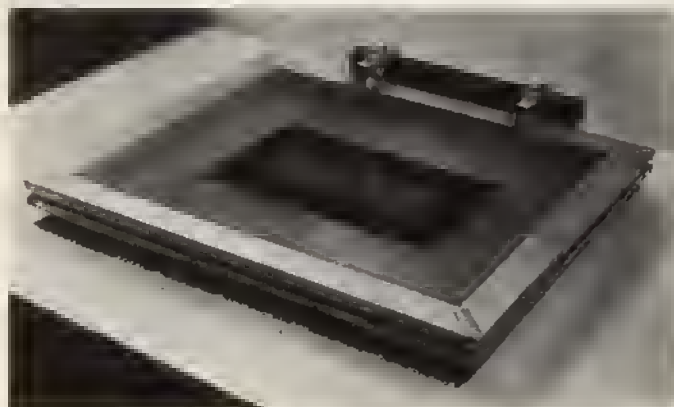


Fig. 6.10 Aluminum screen-printing chase. (Kressilk Products, Inc.)

29. Squeegees (Hand) Clear sharp prints are largely dependent on the squeegee and its use. Rubber and synthetic rubber, $\frac{3}{8}$ in thick and 2 in wide mounted in a $4\frac{1}{2}$ in wooden handle of length that varies with the size of work to be done, are more or less standard. A variety of hardnesses are available. Shore durometer hardnesses are as follows:²²

40-45	Extra soft	60-65	Medium
50-55	Soft	75-80	Hard

Both the length of the squeegee extending out of the handle and the rubber thickness must also be standardized. A short-bladed, thick, soft squeegee may print the same as a harder but longer and thinner one.

Choice will vary with the operator, but for consistent results a standard should be set. The medium (Shore 60 to 65 durometer) is commonly used on fine stainless-steel screens used in printed circuit manufacture. The squeegee must be wiped clean and allowed to dry after each use. Several squeegees should be used during long production runs to prevent excessive softening due to solvent action. They should be allowed to dry while supported by their handles so as not to put weight on the rubber. The edge must be kept straight, with a flat end and square corners. This is a much overlooked area in printed circuit manufacture even though it is clearly recognized that straight, sharp, and square blades are required to produce clear sharp images. Simple machines also are available for this purpose.

The height of the silk screen table is of significant importance, since operator comfort has a direct relation to improvement in rejects and retouching. Generally a 36 to 40-in standard height is adequate. Pushing vs. pulling the squeegee is an operator option. Drop-off support should be provided at finish end.

30. Screen Fabrics Stencil fabric comparison charts are available from fabric suppliers.

a. Silk. Silk screens are in considerable use because of their cost advantage and their ability to take excessive pressures and not deform. For printed circuit work 16XX silk (157 mesh) is typical; it has an opening of about 0.0034 in (diameter).

American and Swiss silks have been found to have the best qualities of mesh uniformity and toughness (tensile strength). For maximum strength and retention of tautness, fabric

strands (weft and warp) must run parallel to frame sides. Silk is unaffected by organic solvents, but it tends to absorb moisture and lose tautness. When restretched, the image is readily distorted. As a consequence, tolerances are very difficult to hold.

As the silk screen is used, the fibers wear and fine filament strands wick. The results are ragged edges, poor definition, and even complete bridged strands between conductor lines.

The distance required between screen and surface being printed is, on the average, about 0.187 in. It results in a parallax when the screen is pressed down by the squeegee. Dimensional allowances may be used to retain tolerances.

Cleaning a silk screen is difficult because abrasives cannot be used without damaging fibers. Baker-on stencils cannot be effectively removed from silk. For that reason silk is not highly recommended for circuit printing. It may be used to print letters, numbers, symbols, etc. when dimensional tolerances and stability are not required. Even then, however, silk should not be used if quantities are high.

b. Polyester. Continuous-filament polyester cloth which has been dyed yellow to prevent unwanted ultraviolet light dispersion can be used instead of stainless steel for cost savings. Once stretched, it holds tension and dimensional stability almost as well as stainless steel. As with other cloths vs. wire, it covers uneven surfaces much better than steel.

The indirect stencil materials do not adhere well to polyester cloth, but the direct emulsions that surround the fiber work well. The cloth is not affected by moisture, is much easier to handle than stainless wire, stretches to proper tension better than nylon does, and does not sag. The yellow dye prevents ragged edges resulting from reflected light during exposure. Exposure time for white cloth is slightly longer.

Nylon fabric is manufactured as a one-filament strand: It is unaffected by organic solvents used in screen printing, but it absorbs water. As is true of silk, water absorption results in a loss of tautness and usefulness.

Nylon of 196 mesh has an opening 0.0033 in in diameter, which is suitable for fine printed circuit printing. Nylon wears well, costs less than silk, and does not form wicking strands. It is, however, stiffer and more difficult to get into the frame than silk. Cleaning may be more rigorous than with silk, but care must be taken not to break the filament.

c. Stainless Steel Type 304. The 200- to 270-mesh stainless steel is a common screen fabric used in printed circuit production. Because of the high tensile strength of the metal, stainless-steel screens have long life, remain taut, and can be made in a finer mesh than silk. In addition, stainless steel has good wear resistance and does not absorb water or screen materials. A minimum parallax is provided with a 0.050-in nominal distance from printing surface to screen. Type 304 stainless steel with a wire diameter of 0.0016 in is typical of the material in use.

31. Cleaning the Stretched Screen (Preparing) The screen must be clean before the stencil material is applied. In general, pumice abrasive materials are not recommended, since they may leave residues. Rinse thoroughly with a high-pressure water spray. Chemical cleaning is preferred. Nylon and polyester fabrics can be cleaned with a soft brush and 20 percent sodium hydroxide solution to sapify animal oils and greases. TSP or other alkaline cleaners also may be used. Soak for about 15 min while avoiding contact with the aluminum frame. Water-rinse thoroughly. Apply a 5 percent solution of acetic acid to the screen with a soft nylon brush. Other organic acids may also be used with nylon. Make sure the stencil fabric is completely dry before applying the stencil.

Stainless-steel wire is surface etched to ensure clean surfaces. A variety of solutions may be used:

1. Ferric chloride 36 to 42° Baumé. Ferric chloride made up with 7 parts ferric chloride etchant 36 to 42° Baumé and 93 parts water. Apply etchant to both sides of wire for one minute and then rinse thoroughly.
2. Chromic acid 10 percent by weight. Apply for 2 to 3 min, water-rinse, and dry.
3. Scrubbing with a 20 percent TSP (trisodium phosphate) solution followed by acetic acid neutralizing and thorough water rinsing.

32. Exposing Stencil The majority of screen stencils are made by using carbon-arc light sources. Stencil films are most sensitive to the blue and ultraviolet portion of the spectrum. They are handled safely in gold fluorescent or red safelight areas.

Besides carbon arcs, high-intensity mercury vapor and pulsed xenon are commonly used. A light-density-time-measurement meter attached to the vacuum frame ensures consistent results.

Uncollimated light, light reflected from the sidewalls, and diffused light from the polyester cover will cause some additional exposure at trace edges. These factors will cause an increase in line width dimensions on negative printed images (pattern plating) and a reduction on screen positives (print and etch). Exposure wedges or other techniques are used to establish the proper conditions.

33. Screen Inks Three general types of screen inks are available: Organic solvent soluble, aqueous alkali soluble, and permanent. Alkali-soluble inks have come into greater use as a result of high solvent costs and pollution standards. New developments are making it possible to use alkali-soluble inks in plating applications. Permanent inks of the epoxy variety are used for solder masking and legend printing. The two soluble-type inks are compared in Table 6.1.

TABLE 6.1 Alkali-soluble vs. Solvent-soluble Screen Inks¹⁰

Alkali-soluble	Solvent-soluble
Primarily used for print and etch.	May be used for print and etch.
Withstands acid plating baths but not alkaline cleaning steps.	Withstands all usual plating baths and cleaning steps.
Can be formulated for fine-mesh screens.	Can be formulated for fine-mesh screens.
Strips in NaOH solutions.	Strips in chlorinated solvents or other organic solvents.
Stripper is cheap, disposable, and presents no health hazard or air pollution problem.	Solvents are expensive; chlorinated solvents are somewhat toxic; other solvents are flammable; all are pollutants. All these problems are controllable.
Used with acidic etchants.	Can be used with acidic or alkaline etchants.
Some inks air-dry; others require forced drying.	Most inks require forced drying and baking.

Although commonly referred to as resist inks, the materials used to print circuit boards are, for the most part, more like resins.

a. Resist Resins. Screen inks generally have a vinyl base, although oil, cellulose, and asphalt resins are available. In some cases baking is required to achieve the best chemical resistance. Resists for use in screen printing, regardless of type, must have the following basic requirements:

1. Chemical resistance to the various cleaning, plating, and etching solutions under the actual processing conditions. Baking of the resist may be required.

2. Viscosity and thixotropic properties must be suitable for screen printing. The viscosity range as measured by a viscosimeter* should be between 4500 and 16,000 cP. The lower-viscosity materials print thinner films and may form blurred images with some materials. They also have a somewhat greater tendency to form pinholes than the preferred higher-viscosity materials. Machine screening high-viscosity resist is controlled by flow rate.

Thixotropic properties of resists are of great importance to line definition and thickness. The property of holding or regelling prevents changes in line width or pad diameter after printing. When lamellar holes are used, the resist may be printed within a few thousandths of an inch of the hole. Without the proper thixotropic properties, resists will run down into the holes even when the viscosity is properly adjusted. Resins which exhibit thixotropic properties tend to decrease in viscosity as they are squeegeed across boards. Once the shearing action stops, viscosity increases to a gel-like consistency.

Another effect on resin rheology is called dilatancy. Materials exhibiting the effect increase in viscosity considerably as they are squeegeed across the board. Once shear action has stopped, they again decrease in viscosity and may be runny.

The resist most suitable for fine-line, small-pad board printing combines thixotropic

*For example, a Brookfield Viscosimeter.

properties in proper blend, which of course must also be adherent and chemically resistant.

3. Resists must dry slowly enough to allow repeated printing without drying and clogging the screen. Once printed, however, they must dry in air or in an oven in a time suitable for production needs. It must be remembered that the metals do not absorb the resists as paper does; therefore, drying takes longer and cannot be accelerated beyond certain limits. If drying is hurried, foaming and bubbling will occur. Well-ventilated and circulating hot-air ovens are required for effective production.

4. The printed and cured resist texture must be such that retouching can be done effectively. Resists which are tarlike when dry cannot be effectively removed (touched up) because they will smear. Those which become brittle flake off when touch-up is attempted, and then a double touch-up is required, i.e., scraping off and brushing on additional resist where the excess was taken off. Baking and drying after these steps would further complicate the process.

5. Adhesion and compatibility of resist to the surface being printed must be good. Good adhesion implies the ability to maintain the original appearance throughout the entire manufacturing process.

6. The resist must be readily and easily removed with solvents which do not damage the substrate or metal.

7. To prevent metal plating over it, the resist must have a high ohmic resistivity and be pure and pinhole-free. Typical thickness is 0.0004 to 0.0006 in.

8. The resist should be colored to contrast with the surface being printed in order to make inspection for printing defects and retouching more effective.

9. Good covering or spreading properties are essential. Screen openings must be filled easily and readily without need for excessive pressure.

10. Resists must not dissolve screen film stencil.

Selection of screen-resist material must be based upon the criteria listed. Oil-base materials suitable for paper are generally not suitable for printed boards because they do not have the required chemical resistance or adhesion. In addition, they dry with difficulty on nonabsorbing surfaces.

Cellulose and acrylic lacquers are thin, very fast drying, brittle when fully oven-dried, and limited in chemical resistance and adhesion. They have been used to spray the back side of metal sheets in processes for printing and etching. During etching the back coating keeps small parts from falling away.

Asphalt-base resists lack a number of the basic qualities required of screen resists. Asphalt-reinforced lacquers are used. Ferric chloride, ammonium persulfate, and ammonium hydroxide etchants have been found suitable at room temperature.

Vinyl-base resists are currently the most suitable for printed-circuit manufacturing. They generally fulfill the basic requirements, provide good screening properties, have good adhesion, and are resistant to hot alkali and cyanide solutions. The overall resistance to acids, etchants, and plating solutions is exceptional. In some cases, vinyl-based resists will hold up in concentrated sulfuric acid and sulfuric-hydrofluoric acid mixtures.

Because of those properties vinyl resists are used extensively for pattern plating circuit boards. Processes include pyrophosphate copper, copper sulfate, copper fluorurate, acid and cyanide golds, nickel sulfamate, Watt's nickel, rhodium, tin-nickel, solder-fluoroborate, and the attendant cleaning and preparation. Baking at temperatures up to 220°F for about 15 min provides maximum chemical resistance and adhesion properties in the plating solutions. However, when the higher temperatures are used, removal is made more difficult. In production, a temperature of 140 to 160°F for about 15 min has been used with consistent good results. Xylene, chlorinated hydrocarbons, other organic solvents, and commercial products are available to strip vinyl resists. Proper venting and safe handling procedures must be used.

Solder mask formulations contain permanently cured resins. Typical screened systems are epoxy-based. The usual screening procedures are followed. In this case, however, they cannot be removed for rework. Dry film solder mask materials are available.

34. UV-Cured Resists The curing of printed circuit resist coatings with ultraviolet light is a new concept which offers air pollution elimination and reduction in energy consumption. UV-curable formulations include three essential components: a base unsaturated polymer

resin, a photoinitiator, and a multifunctional monomer which acts as a cross-linking agent. Typical photoinitiators are mercaptan, ketone, benzoin, amine oxide, and phosphine derivatives. Photopolymerization occurs by free-radical mechanism. Interested readers may consult the literature for additional information.^{26-29,31-32}

UV-cured resists are available as alkali- and solvent-soluble formulations for print-and-etch and certain plating applications, as well as permanent coatings for solder mask application. The evolving technology needs systems that are UV-curable, solvent-free, alkali-etchable, and alkali-strippable for compliance with the Clean Air Act and enforcement of environmental regulations.

35. Wet Etchable* Certain resists that do not require baking or oven drying for their preparation are available. They rely on their dewetting properties to maintain chemical resistance and adhesion to the substrate.

36. Dry-screen Process A dry-screen process involving the use of a thermoplastic resin resist and a heated metal screen also is available. The thermoplastic resist, made fluid by contact with an electrically heated screen, is printed in the conventional manner but solidifies immediately into a nearly dry print. The postdrying and baking of the screening resists is eliminated. Special heat-resistant pattern materials must be used. Higher printing temperatures also bring problems of resist adhesion due to the heat-sink effect of the copper-clad laminate and dimensional changes of the pattern. The chemical resistance of dry-screen resists may also be a problem in certain processes involving vigorous chemical agents.

37. Printing Resist Problems

1. Repetitive printed spots or voids are attributable to solids in the screen. The effect is normally not found on new screens unless the photographic film has pinholes in the emulsion. On reused screens it is attributable to incomplete removal of the old emulsion. Dust and lint also can be the cause. It is therefore important to do printing and store the resist inks in a dust-free area.

2. A "runny" or smeared image is traceable to excessive solvent or extender in the resist. For consistent results, the addition of thinner and extender to resists should be held to a minimum.

3. Failure to print after a number of good prints have been obtained is generally due to resist drying in the screen. When this occurs, the pattern should be cleaned out, the resist removed with solvents, and the screen and squeegee cleaned and dried. Thinning the resist may cure the problem, but the image quality may be damaged. If the problem persists, a better resist should be sought.

4. Poor definition with a tendency to run under the screen is an indication that the resist is too thin. Poor definition, slumping, or spreading out of the printed image when viscosity is high indicates poor thixotropic properties.

5. Jagged line edges which develop during printing, although credited to screening technique, justly belong to the screen-stencil breakdown. The only cure is to select a screen emulsion which has the life expectancy required and make certain it is properly attached to a well-cleaned and prepared screen.

6. Spots which develop after a number of good printings may also be attributable to stencil-film breakdown.

7. Pinholes in resist are usually caused by air or moisture in the resist. Proper procedures involve keeping the containers well closed, mixing on a shaker, and allowing the resist to settle a day or two after shaking before it is used.

8. Poor adhesion of the vinyl-based resists is almost invariably due to improper cleaning of the surface printed. If that is not the case, there is a good chance that the wrong thinner or extender has been added to the resist. The boards may be sanded to improve adhesion.

9. Thin extension of printed image, commonly called shadowing, may be attributed to the following: (a) Board tipping, which can be eliminated by providing a firm mechanical holding. (b) Poor viscosity and thixotropic properties of resist, although these are more evident in an overall running or slumping of the image. (c) Excessive off-contact distance.

*Poll wet etchable.

Contact distance and screen tension must be correlated to produce sharp images which are shadow-free. (d) Insufficient squeegee pressure or improper squeegee angle, although either can also cause incomplete images. (e) Lack of an extension card spacer, called an apron, which prevents the screen from being pushed down too far when the board edge is reached. This is the most likely cause of shadowing. Its thickness should be that of the board being printed. It should be long enough and wide enough to provide positive support of the entire squeegee length, and it should be mounted to allow screen openers to snap clear of all printed images.

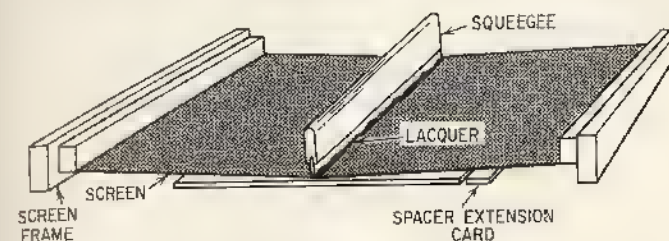


Fig. 6.11 Sketch of screen printing in progress.

38. Printing by Hand A sturdy copy table, the prepared screen, a hinged screen-holding mechanism, squeegee, resist lacquer, spatulas, solvents, cloths, abundant clean newsprint, masking tape, and holding trays are typical essential materials for hand screen printing.

The screen chase is mounted in the hinged mechanism and aligned by eye over the board to be printed. The aligning is done by using the adjustments provided in the hinge mechanism or by means of an adjustable copy board. Movable copy boards are generally preferred, since they provide an adjustable work area. A common system used for locating is to drill, in the board to be printed, small indexing holes which coincide with pins located in the copy board. With the board held in position by the pins, which do not extend through the board, adjustment is made until all holes align with the screen. Adjustments are locked to ensure immovability. Resist lacquer of sufficient quantity to produce a reservoir for many impressions is placed at the upper end of the screen. From this reservoir, an amount sufficient to produce a wave in front of the squeegee along its entire length during the entire pass is scooped away by the squeegee. Figure 6.11 illustrates screen printing in progress.

The image may be printed onto newsprint until it appears clear and repeatable and only then onto the board. The image produced is examined, and adjustments in alignment are made as required.

The wave of resist provides a reservoir suitable for the entire impression plus a slight excess. As a board is printed, the squeegee is pulled or pushed and pressed firmly across the patterns on the screen. The squeegee and excess resist are then returned to the upper end of the screen. The screen chase is raised and held up either by a side foot support or, preferably, by a counterweight balance. The printed board is raised carefully off the indexing pins and placed face up on the drying and holding rack. A new board is inserted and the operation is repeated. Once a number of boards have been printed, they are placed in an oven. Baking is continued until dryness sufficient to allow handling is reached. The other side is printed in a similar manner. When long-production double-sided printing is common practice, at least two printing setups and two operators are required. That minimizes time loss. Equipment costs and space are warranted even for a small facility.

39. Machine Printing With machine printing, various degrees of automation are possible depending on board design, output, and processing requirements. Screen printing is highly labor-intensive and skill-oriented. Automation offers decreased need for the highly skilled operator, high output, low cost, and the means of utilizing new technology. (See refs. 30 and 31). The squeegee, resin feed, and screen raising are automatic. An operator makes adjustments, adds resist to the reservoir, controls the flow of resist, adjusts speed and pressure of the squeegee, and cleans the screen as required throughout the process.

As each board is printed, the operator examines it for acceptability and places it in the rack for oven drying. If the board is not suitably printed, it is placed in a rework tray. Large machines are capable of producing 1000 or more prints per hour. Less expensive bench models are available. The degree of precision construction of the printing machine determines the tolerances and fineness of printing attainable. Repeatable printing ± 0.001 in in two-dimension comparison with screen and piece-to-piece variation of ± 0.0005 in in two dimensions are claimed by some machine manufacturers. Board standardization, tooling, precision drilling operation, room conditions, and a number of other variables must be controlled to maintain these tolerances.

Automatic printing machines are not without problems, which are generally concerned with synchronization, precision tolerance movements, microswitch maintenance, and ink feed. Bench models offer more problems because their adjustment is quite sensitive and varies more than with the heavier-construction floor models. Proper ink flow requires constant and careful cleaning. Squeegees receive considerable wear and must be cleaned often and changed on schedule. Excessive contact with some screen resists softens the rubber. Additional problems include the following:

1. Damaged image or cut screen. When a circuit board is placed incorrectly under the screen, the indexing pins hold it above the proper height or at an angle which exposes the board edges. When the squeegee is drawn across the board, the screen may be cut or the image damaged. In hand printing, the operator can feel the resistance and stop before tearing occurs.

2. Incomplete printing. In some cases the stroke adjustment may drift and travel and not cover the entire area of pattern. Lock screws must be firmly set.

3. Printed hairlines across circuit traces. Machine printing usually involves long production runs; hence the screen surface and resist lacquer are exposed to airborne dust, lint, and contaminations. They are filtered out by the screen and act as block-outs, resulting in printed hair or void lines.

4. Printing of hairlines on unwanted areas may be caused by solid materials scratching through the screen stencil. The materials may come from airborne solids but are more likely to be picked up from board surfaces. Drying racks should be covered as much as possible and stored in a room with clean filtered air. Most airborne dust is generated from the floor. Use only dust-, lint-, and oil-free materials for wiping the boards clean.

5. Misregistration of print due to movement of the board. Warp and rigidity of the printed board make hold-down difficult to maintain. Drilled boards with sung pins are used. Double-sided tape has been effective in attaining the proper hold-down. Warp should be removed prior to screening.

40. Offset-Press Printing The use of printing press equipment for printed circuit manufacturing has a strong appeal because of the potential output capabilities. Tolerances (accuracy) and fine-line-definition potential are good. Large panels can be printed, which allows multiple lay-ups. If three boards are printed on one panel 9×18 in, for example, the labor of handling has been reduced to one-third that of single lay-up. An extremely well skilled pressman is required to operate and maintain the machine.

The greatest application, particularly on large panels, is in the print and etch of single-sided boards. Double-sided printing and mass fill-in of field area required for plated-through-hole boards create a perplexing obstacle.

In operation, a zinc or multmetal lithographic plate picks up the acid-resist lacquer (ink) on the required areas. The ink is transferred from the plate to a synthetic-rubber-blanketed cylinder, which in turn transfers the ink to the board being printed. Detailed operation of this equipment is covered in lithography books.²¹

Professional lithographic personnel are required to make the plates. The preparation of a plate for printing involves adjusting and controlling the flow of ink and water very carefully to provide adequate thickness to achieve the definition required and the chemical resistance. Techniques of printing and then dusting on fusing asphalt powders are cumbersome and inconsistent. Improper heat control can easily result in line slumping and a ruined print.

The panel is placed on the bed and held in position by holding stops. Variations in thickness of board and warp are compensated for in the bed and rubber plate. The machines are essentially constructed of cast iron or steel and so must be protected from

corrosion. Problems caused by dust and dirt contamination are similar to those faced in other printing processes. The machines are large, and considerable area is needed.

The use of UV-cure printing resins offers a tremendous potential for large-scale print-and-etch applications.

41. Hole Plugging An all-copper plated-through-hole board can be made by using a technique of filling or plugging the holes with a resist ink followed by screening a positive circuit pattern. Tin or solder can be applied to the etched board. Both alkali-soluble and solvent-soluble inks are used in the fabrication. As before, the alkali-soluble inks offer advantages for pollution abatement and lower costs on the solvent-type inks. Additional details for this process are given in Chap. 8.

Many manufacturers of equipment and materials for image-transfer processes not mentioned in the text are available. Directories and other listings should be consulted; for example, see refs. 3, 4, and 32-38.

REFERENCES

1. R. J. Ryan, E. B. Davidson, and H. O. Hook, chap. 14 in C. A. Harper (ed.), "Handbook of Materials and Processes for Electronics," McGraw-Hill Book Company, New York, 1969.
2. W. S. DeForest, "Photoresist Materials and Processes," McGraw-Hill Book Company, New York, 1975.
3. "Kodak Photo-sensitive Resists for Industry," supplement to "Kodak Photosensitive Resists for Industry, Physical Properties," "Kodak Photo-resist for Making Photolithograph Plates," Eastman Kodak Company, Rochester, N.Y.
4. "Techniques of Microphotography," Eastman Kodak Company, Rochester, N.Y.
5. Etching, Chemical Milling, and Plating with Kodak Metal-etch Resist, Publ. P-36, Eastman Kodak Company, Rochester, N.Y.
6. AZ Photo Resist Process Instructions, Shipley Company, Inc., Newton, Mass.
7. Dynachem Technical Data Sheet 6-2863.
8. Integrated Silicon Device Technology, vol. III, *Photoengraving*, ASD-TDR-1, 63-316, January 1964.
9. Printed Circuit Image Transfer Symposium, California Circuits Association, Burlingame, Calif., May 20, 1970.
10. Should You Screen or Photoprint Printed Circuits? *Circuits Manufacturing*, 12-16, February 1971. Photo Resist for Plating Replaces Screened Resist, *ibid.*, 24-26, April 1971.
11. K. R. Dunham, *Solid State Technology*, 14:4, November 1971.
12. K. C. Clark, *Solid State Technolingu*, 14:52-55, November 1971.
13. J. R. Sayer, Jr. and J. Smit, Evaluation of Fine Line Etching for Printed Circuit Applications, *Plating*, 48:789-793, 1961.
14. H. S. Holliman, Methods of Producing Fine Lines in Printed Circuits, Proc. 1960 Fall Meeting, Institute of Printed Circuits, Chicago, Ill.
15. T. R. Bates, Photosensitive Resists for Electroforming, *Plating*, 52:673-676, 1965.
16. U.S. Patents 2,610,120, 2,670,285-7, 2,690,966, 2,732,297, 2,739,892, 2,801,233.
17. U.S. Patents 2,706,725, 2,708,665, 2,716,097, 2,716,102-3.
18. U.S. Patent 1,965,710.
19. U.S. Patent 2,908,667.
20. U.S. Patent 2,948,610.
21. Herbert E. Butler, Dry Film Resists: A Product Development History, California Circuits Association Symposium, Burlingame, Calif., May 20, 1970.
22. J. Eisenberg and F. Kafka, "Silk Screen Printing," 2d ed., New York City Schools, McKnight & McKnight Publishing Company, Bloomington, Ill., 1957.
23. A. Kosloff, "Photographic Screen Process Printing," 2d ed., Signs of the Times Publishing Company, Cincinnati, Ohio, 1962.
24. Lithographic Technical Foundation Research Bulletin No. 218, Sensitivity of Dichromate Coatings.
25. John W. Denney, Screen Process Printing Stencils, California Circuits Association Seminar, Burlingame, Calif., May, 1970.
26. C. R. Moigan and A. D. Kettey, Photo-sensitized Thiolen Reactions, ACS Spring meeting, Dallas, Texas, 1973.
27. C. Kehr and W. Uszolek, Thiolen Curable Polymer Technology, ACS Spring meeting, Dallas, Texas, 1973.
28. C. B. Rybny, C. A. DeFazio, J. K. Shahidi, J. C. Trehellos, and J. A. Vona, *J. of Paint Techn.*, 46:596, 60-69, September 1974.
29. J. C. Gardon, and J. W. Freese, "Non-polluting Coatings and Processes," Plenum Press, New York, 1973.

30. H. Oliermaier, An Approach to Automated Screen Printing, *NEPCON*, February, 1974; also published in the Screen Printing Association International Handbook.
31. G. Tumke, Automatic Screen Printing, *Circuits Manufacturing*, 12, Nov. 9, 1972.
32. ASTM Designation D 1706-61, "ASTM Book of Standards," pt. 27, pp. 546-552, American Society for Testing and Materials, Philadelphia, Pa., 1965.
33. A. Marinaccio and B. N. Oshoni, "Exploring the Graphic Arts," D. Van Nostrand Company, Inc., Princeton, N.J., 1942.
34. Printed Circuit Materials and Equipment Directory, Electronic Packaging and Production, yearly publication.
35. Products Finishing Directory, *Products Finishing*, Gardner Publications, Inc., Cincinnati, Ohio, yearly publication.
36. Insulation, Directory Encyclopedia Issue, Lake Publishing Corporation, Libertyville, Ill., yearly publication.
37. Modern Plastics, Encyclopedia Issue, New York. Yearly publication.
38. Institute of Printed Circuits, Evanston, Ill.
39. M. Martin, Institute of Printed Circuits, Spring Meeting, Washington D.C., April 1975.
40. D. Feinberg, Institute of Printed Circuits, Spring Meeting, Washington D.C., April 1975.
41. H. Markstein, *Electronic Packaging and Production*, 15:39-42, June 1975.
42. C. B. Fehrmann, *Electronic Packaging and Production*, 15:29-36, June 1975.

Chapter 7

Plating

ROBERT J. BAHN

Hewlett-Packard Company, Colorado Springs, Colorado

Introduction	7-2
1. Printed Board Plating Flowchart	7-3
Electroless Copper	7-3
2. Function	7-3
3. Electroless Copper Line Steps	7-3
4. Postdeposition Operations	7-4
5. The Mechanism of Electroless Copper	7-5
6. Special Considerations	7-7
7. Troubleshooting the Electroless Copper Line	7-7
Panel and Pattern Plating	7-8
8. Function	7-8
9. Panel Plating	7-9
10. Pattern Plating	7-10
11. Comparison of Panel and Pattern Plating	7-10
Image Transfer and Its Effect on Plating	7-11
12. Discussion	7-11
13. Silk Screen	7-11
14. Plating Problems Related to Silk Screen Resist	7-11
15. Photoresists	7-12
Electroplating	7-13
16. Process Evaluation	7-13
17. Copper Electroplating	7-14
17.1. Copper Pyrophosphate	7-14
17.2. Copper Sulfate	7-16
17.3. Copper Fluoroborate	7-18
17.4. Copper Fluoroborate	7-19
18. Copper (Tin-Lead) Plating	7-19
19. Function and Properties	7-19
20. Nickel Electroplating	7-22
20.1. Function and Properties	7-22
20.2. Nickel Sulfamate	7-23
20.3. Modified Watts Nickel	7-23

Tin-Nickel Electroplating	7-25
24. Function and Properties	7-25
Tin Electroplating	7-26
25. Function and Properties	7-26
Gold Electroplating	7-28
26. Function and Properties	7-28
27. Acid Gold	7-28
28. Alkaline Noncyanide Gold	7-28
29. Gold Strikes	7-29
30. Function of Bath Constituents	7-29
31. Bath Maintenance	7-29
32. Troubleshooting Gold Baths	7-30
33. Equipment	7-30
34. Plating Rate	7-30
35. Gold Solderability	7-30
Rhodium Electroplating	7-31
36. Function and Properties	7-31
Electroless and Immersion Plating	7-31
37. Function and Properties	7-31
38. Electroless Nickel	7-32
39. Immersion and Electroless Gold	7-32
40. Immersion Tin	7-33
Plating Equipment	7-33
41. Selection	7-33
Additive Circuits	7-33
42. Sulfur and Elch	7-34
43. Preconditioned Laminate	7-34
44. Full Additive vs. Semiadditive	7-34
45. Pros and Cons of Additive Circuit Fabrication	7-35
Chemical Safety in the Plating Shop	7-35
46. Function	7-35
47. Identification of Chemical Properties by Label	7-35
48. Solvents	7-35
49. Acids	7-36
50. Oxidizing Agents	7-36
51. Cyanides	7-37
52. Caustic Compounds	7-37
Process Control	7-37
53. Copper Pyrophosphate Solution Analysis	7-37
54. Acid Copper Sulfate Analysis	7-39
55. Acid Copper Fluoroborate Analysis	7-39
56. Solder (60 Sn, 40 Pb) Acid Fluoroborate Solution Analysis	7-40
57. Modified Watts Nickel Analysis	7-41
58. Nickel Sulfamate Analysis	7-41
59. Bright Acid Tin Analysis	7-41
60. Tin-Nickel Solution Analysis	7-42
61. Gold Solution Analysis	7-43
62. Microsectioning	7-43
References	7-44

INTRODUCTION

It is the function of this chapter to present in detail the plating and metallization processes that are currently employed to meet the demands of a quality printed board. Steps both prior and subsequent to plating will be discussed insofar as they have a direct effect on the plating operation. The plating operation itself may affect another operation further downstream.

The principal printed board material discussed will be NEMA grade G-10 or FR-4; each is an epoxy-fiberglass clad with 1/2-, 1-, or 2-oz copper. Other materials, sometimes referred to as exotics, will be briefly mentioned because they can significantly alter plating and related processes.

Through-hole plating has to a great extent become the standard of the industry.^{1-3*} It provides a side-to-side connection with high reliability at a relatively low cost for the subsequent soldering of components. For that reason the emphasis in the chapter will be on the plating of through-hole boards.

In the event the boards being fabricated require adherence to military specifications, some alteration pertaining to plating may be necessary. Several references to pertinent military specifications and their plating mandates will be outlined as the processes are reviewed. In addition to military specifications, the recent legislation regarding environmental and effluent controls are having a significant impact on the printed circuit industry overall. The selection of a particular plating process must be made with ecology in mind.

1. Printed Circuit Plating Flowchart (Fig. 7.1) The flowchart represents the possible routes a conventional printed board may take through the plating operation subsequent to drilling and deburring.

ELECTROLESS COPPER

2. Function It is the function of the electroless copper operation to metallize the entire board with a thin deposit of copper to render the board conductive for subsequent through-hole electroplating. "Electroless" may be a misnomer, since it implies that no current is employed in the deposition. In electroplating, electrons are obtained from a dc power source to reduce metal ions to the metallic state. In electroless plating, the mechanism is similar but the source of electrons is a chemical reducing agent. Hence nonconductive surfaces may be metallized by employing electroless baths, since the baths contain their own source of electrons. Electroplating baths cannot be used for nonconductive surfaces because the electrons cannot flow.

3. Electroless Copper Line Steps The electroless copper line in many large fabrication facilities has been automated. As can be seen from the following steps, the line is very conducive to automation:

1 Rack boards. Boards should be so racked as to permit ready access of the solutions to the holes. Both off-vertical and horizontal racking have been used with success.

2 Clean. Many proprietary alkaline cleaners will do an adequate job; some contain chelating agents that are an aid in oxide removal. Electrolytic cleaning is used on occasion with excellent results.

3 Rinse. Spray rinse or counterflow rinse with soft water.

4 Copper etch. It is the practice of copper foil manufacturers to apply an oxidation inhibitor to the surface of the foil. If the inhibitor is not completely removed, the result will be a "peeler" between the copper-clad surface and electroless copper deposit. To ensure complete removal of the inhibitor, the copper surface is given a light etch. The primary etchant used is ammonium persulfate, although peroxide-sulfuric acid and cupric chloride have been employed with success. Ammonium persulfate is made up at a

*Superscript numbers are those of references listed at end of chapter.

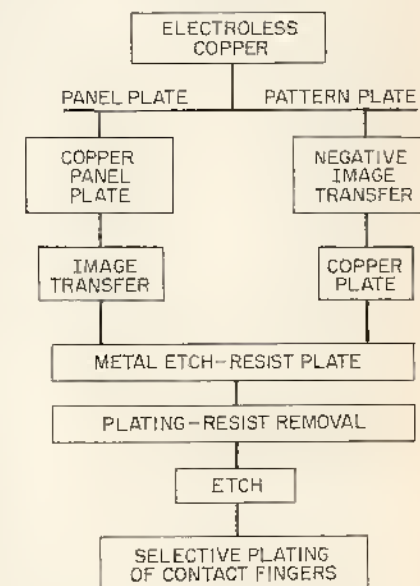


Fig. 7.1 Plating flowchart.

concentration of 2 lb/gal. The temperature is controlled at 70 to 100°F. Temperatures in excess of 100°F can result in chemical decomposition of the solution. Immersion times are of the order of 2 min, or sufficient time to produce a uniform etch. The solution must be made up daily or, in the case of heavy volume, the life of the etchant will be 3 to 4 oz/gal of dissolved copper. It should be borne in mind that all etchants of this nature contain heavy metals in appreciable quantity when they are spent and will require proper disposal. An alternate method that eliminates the need for etching will be discussed later.

5 **Rinse.** Spray rinse or counterflow rinse.

6. **Acid dip—sulfuric acid (10 percent by volume).** This dip is required after an ammonium persulfate etch to solubilize any copper ammonium sulfate which may tend to film on the copper surface in the rinse tank. In the case of a cupric chloride etch hydrochloric acid (33 percent by volume) is employed to remove any cuprous (Cu^+) film.

7 **Rinse.** Spray rinse or counterflow rinse. At this point the copper surface should be completely "water break"-free and have a uniform pink surface.

8 **Hydrochloric acid dip.** Use reagent grade acid, 33 percent by volume, for 2 to 4 min. No rinse.

9 **Activator.** Several proprietary solutions are referred to as activators. Basically they are colloidal solutions of palladium.⁴⁻⁷ Stannic tin forms a protective colloid around metallic palladium. Briefly stated, it is the function of this solution to implant, on a nonconductive surface, a precious metal site that will initiate the desired reaction of plating by chemical reduction. Immersion time is usually 3 to 10 min. These baths are maintained colorimetrically or by titration in accordance with the vendor's instructions. If properly maintained, the life of the activating solution is frequently in excess of a year.

10 **Postactivator.** Also available as proprietaries are postactivator solutions. The function of the solution is to solubilize the protective colloid and expose the precious metal sites for subsequent electroless plating. Immersion time is 3 to 10 min. Control of the bath is accomplished either colorimetrically or by the area processed.

11 **Electroless copper.** At this point the surfaces to be metallized with copper are ready for the electroless bath. Many formulations, both proprietary and nonproprietary, are available. The selection of the electroless copper to be employed requires consideration as to the following:

a. **Work load.** Heavy work loads such as would be encountered with an automatic line should employ regenerable electroless copper for maximum economy. This type of electroless copper can be maintained for several weeks. Light work loads such as encountered in a prototype facility or small shop should be processed in a minimum volume of electroless copper and made up daily.

b. **Deposit thickness.** Most electroless copper formulations have a deposition rate of 1.0 to 1.5 $\mu\text{m}/\text{min}$, but 10 μm is generally considered to be the minimum thickness of electroless copper for subsequent electroplating. There are instances when thicknesses of the order of 100 μm are required, as in additive circuits or as an alternative to a "flash" by electroplating. As can be seen by the plating rates, the time involved with most conventional electroless coppers is prohibitive. To correct that difficulty, several proprietary electroless copper formulations have recently been developed to give plating rates up to 4 $\mu\text{m}/\text{min}$. These baths are regenerable and may be employed for thinner deposits when high speed is desirable.

c. **Image transfer following electroless copper.** Although primary interest is in the coverage obtained in the hole, the deposit obtained on the surface of the copper-clad board must also be considered. When a liquid resist is applied directly over the electroless deposit, the properties of the deposit can have a decided effect on the quality of the image obtained. A spongy, porous deposit will result in resist bleed-in and ragged lines; in the case of a high-density image, it can make the image transfer prohibitive. The ideal deposit will be dense, fine-grained, and oxide-free, and various formulations should be evaluated accordingly.

12 **Rinse.** Spray rinse or counterflow rinse.

13 **Acid rinse—sulfuric acid (2 to 5 percent by volume).** The function of the acid is to neutralize the residual alkaline film from electroless copper.

14 **Rinse.** Spray rinse or counterflow rinse.

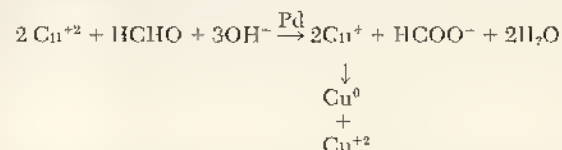
4. **Postdeposition Operations** At this point the electroless copper deposit is essentially complete. There are, however, several operations a board may undergo prior to any other major processing that could be construed as being part of the electroless copper line.

1 **Drying.** Although seemingly insignificant, it is essential that drying be thorough, especially in the case of an electroless deposit that is of the order of 10 μm . Residual moisture will greatly accelerate the oxidation of the thin deposit, and the end result will be a void. An oxidized board also makes any inspection for image transfer quality very difficult. Drying can be accomplished by the employment of one of several commercial conveyorized driers. An alternate method is to immerse the boards while still racked following the final rinse in a water-displacing liquid for several minutes. That is followed by immersion in a vapor degreaser with trichloroethylene or 1,1,1-trichloroethane. Both methods are very effective in drying boards, especially for high-volume production.

2. **Mechanical scrubbing.** The past several years have seen the use of mechanical scrubbing become prevalent throughout the printed circuit industry. The function of scrubbing is to precondition the surface of the board for subsequent image transfer and plating. The scrubbing is accomplished by wet brushing with nylon brushes impregnated with an abrasive. Conveyorized driers can be incorporated into the scrubber units. A properly scrubbed substrate presents a uniform surface on which to apply an image and can substantially reduce the touch-up required. It should also be noted that the electroless copper is removed from the copper-clad surface in the scrubbing operation. Following the image-transfer operation, an acidic cleaning step prior to electroplating will make the succeeding etching operation unnecessary.

3 **Panel-plate flash.** It is a standard practice in many shops to panel-electroplate boards with several ten thousandths of copper immediately following electroless copper. It should be borne in mind that this is an extra operation that will require rerecking. The function of the flash is twofold. When boards must be stored for an extended period of time and there is some probability of copper oxidation in the holes, a flash may be employed to ensure hole integrity. The flash may also be employed when a light etch is used as part of a cleaning cycle prior to electroplating copper. That will ensure that no voids in holes will be caused by the light etch. The flash will be discussed further in the copper plating section.

5. **The Mechanism of Electroless Copper⁸** To enhance the understanding of the electroless copper line, a discussion of the actual reactions involved is in order. The copper reduction may be represented by the equation



This equation indicates that cupric ions are reduced by formaldehyde in a strongly basic medium. The primary product of this reaction is metallic copper, but it should be noted that the reduction proceeds through the cuprous (+1) state. An excess of cuprous oxide formation will cause the reduction reaction to proceed out of control. The result will be loss of the bath and a deposit of copper and copper oxide on the bottom of the solution container that is essentially worthless.

Several methods are employed to inhibit the formation of cuprous oxide. The use of air⁹ is very effective when the air is bubbled slowly through the solution. The cuprous (+1) ions that do not proceed to metallic copper are oxidized back to the cupric (+2) state, which prevents the buildup of cuprous oxide. Complexing agents for cuprous ions are employed in small quantities in many proprietaries to control the formation of cuprous oxide. Care must be taken in formulating this type of electroless copper, since an excess of complexing agent will stop the reaction completely.

An important factor not indicated in the electroless copper reaction is the chelating agent employed for the cupric ions. To a large extent it will govern the plating rate and have a marked influence on the properties of the deposit and the bath stability. Chelating agents that have been successfully employed include amines, gluconates, glycolic-tonates, various EDTAs, and tartrates. In conjunction with chelating agents all electroless formulations are particularly temperature-sensitive. In some instances the plating rate of an electroless copper will double for every 10°F rise in temperature. Too high a temperature will easily result in a runaway reaction and subsequent decomposition of the bath. On the other hand, too low a temperature will result in a bath that will not react at all. As

can be seen, scrupulous attention must be paid to the vendor recommendations regarding temperature. Most baths operate at a temperature of 70 to 80°F.

As indicated in the reaction, hydroxyl ions are essential. Most electroless copper baths operate with a pH in the vicinity of 12. In some instances a bath may be stored by lowering the pH to 8 to 10 with 10 percent by volume sulfuric acid.

Copper, formaldehyde, and sodium hydroxide are consumed in the reaction and must be replenished. Replenishment solutions are based on the usages plus some consideration for drag-out. Most vendors have developed color standards as a simple method of measuring the strength of the bath. A limit must be placed on the area processed through a given bath. As can be seen, formates accumulate as a natural product of the reduction. Copper sulfate is usually employed as a source of copper ions, and that will eventually lead to a precipitation of sulfates as a sodium salt. The by-products will eventually render the bath inoperable.

Two typical electroless copper formulations¹⁰ are as follows:

a	Copper sulfate	29 g/L
	Sodium carbonate	25 g/L
	Rochelle salts	140 g/L
	Versene-T	17 g/L
	Sodium hydroxide	40 g/L
	Formaldehyde (37%)	150 g/L
	pH	11.5
Temperature	70°F	
b	Copper sulfate	25 g/L
	Sodium gluconate	60 g/L
	Sodium hydroxide	20 g/L
	Formaldehyde (37%)	25 g/L
	pH	11.5
	Temperature	75°F

As a catalyst in electroless copper reduction, palladium is essential for initiating the reaction on a nonconductive surface. Once copper has been deposited, the reaction is autocatalytic. It should be mentioned that most precious metals will act as catalysts for the reaction. Palladium has been widely used primarily because of its adaptability to the various catalyzing systems. The colloidal system for depositing precious metal sites on a nonconductive surface has been previously mentioned.

The classic method of activating nonconductors has been employed successfully with printed boards. It consists of two solutions. One is a sensitizer:

Stannous chloride	30 g/L
Hydrochloric acid	30 g/L

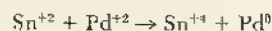
The sensitizer is made by first dissolving the stannous chloride in concentrated hydrochloric acid and then diluting to the proper volume with water.

The second solution is a supersensitizer:

Palladium chloride	0.25 g/L
Hydrochloric acid	1 g/L

The supersensitizer is made by dissolving the palladium chloride in concentrated hydrochloric acid to form chloropalladic acid. Once dissolved, the solution is diluted to the proper volume with water.

The boards are immersed in the sensitizer for 2 to 5 min. Following a thorough rinse, they are immersed for 1 min in the supersensitizer. That is followed by another thorough rinse prior to immersion in electroless copper. An examination of the system will reveal that, in the first step, stannous ions are adsorbed onto the surface to be catalyzed. The stannous ions subsequently reduce the ionic palladium in the supersensitizer to the metallic state and establish the precious metal sites necessary to initiate the electroless copper reaction. The redox reaction is represented by



Although this system is very effective for catalyzing most nonconductive surfaces, there are two reasons why colloidal systems are preferred on printed circuits.

a. The stannous tin solution will form an inorganic polymer as the solution ages. It acts as a parting compound, and the final result will be a "peeler" with bond failure between the electroless copper deposit and the surface.

b. In addition to the nonconductive surface, consideration must be given to the effect of the activating system on the copper-clad surface. Palladium, when in the ionic form, will displace metals less noble than itself and will plate out by immersion. Such is the case with the classic activating system. That, in turn, will rapidly deplete the palladium in the supersensitizer. In contrast, in the colloidal system, since the palladium is already in the metallic state, no displacement can occur.

6. Special Considerations This section concerns materials that require handling other than that normally carried out in the electroless copper line.

1. **Multilayer boards.**¹¹ The function of the electroless copper deposition is to provide side-to-side electrical continuity as well as connection to the innerlayers.

a. **Etchback.** The primary modification in the electroless copper line to be made when multilayer boards are processed is the incorporation of an etchback process. The function of etchback is to ensure an integral copper-to-copper bond between the layer conductors and the electroless copper deposited on the hole wall. A by-product of the drilling operation is an epoxy smear on the inner-layer conductors. An electroless copper deposit over a smeared conductor can result in an open circuit. Although drilling equipment and technique can minimize epoxy smear, a standard practice of many multilayer board fabricators is to employ some form of etchback for insurance. Several etchback variations are available:

Sulfuric acid-hydrofluoric acid. This mixture (1:1) is employed at elevated temperatures (120 to 140°F). It will attack both epoxy resin and glass.

Fluorosulfonic acid. This acid is used to attack epoxy resin.

Sulfuric acid-chromic acid. This is a strong oxidizing mixture available as a proprietary. It is designed to etch epoxy with a minimum amount of residue.

Concentrated sulfuric acid. This is employed at room temperature and will remove epoxy resin by charring. The acid will absorb water from the atmosphere and gradually lose its potency, which necessitates periodic changes.

Etchbacks are generally employed as a first step in the electroless copper line. Optimum immersion times will have to be determined by the user, since all resin systems will not react the same. After the etchback step, the boards should be given a thorough rinse followed by immersion in an alkaline cleaner to ensure complete neutralization of any residual etchback solution.

Although safety should always be rigidly enforced in the plating shop, special consideration should be given to the properties of the etchbacks.

b. **Electroless copper.** Following etchback, conventional processing is employed through the electroless copper line. Several sections should be made of boards processed beyond the electroless copper step, and a panel should be electroplated to ascertain the quality of the interconnection obtained.

2. **Thermoplastic and thermosetting plastics.** Besides epoxy-glass, there are a number of copper-clad insulators that may require through-hole plating. They will vary greatly with regard to the fillers employed, chemical resistance, machining properties, and temperature resistance, all of which can significantly affect processing through the electroless copper line. Materials that are attacked by solvents must not be subjected to degreasing operations. Heat-sensitive substrates and adhesives may require low-temperature cleaning methods. Materials that are difficult to wet may require the use of a proprietary conditioner or some changes in the activator system or possibly both. As can be seen, before any firm cycle for depositing electroless copper on a new substrate can be established, a thorough evaluation must be conducted.

7. Troubleshooting the Electroless Copper Line The following data are intended to be a guide in troubleshooting the electroless copper line.

1. **Voids.** A lack of electroless copper in a hole is called a void. A void may be partial in one hole or in all the holes of a board. The causes of voids can be many and are varied. It is possible that the cause is not in the electroless line itself but instead is incomplete final drying or storage in a corrosive atmosphere that results in oxidation of the electroless

copper. Trapped inclusions can be responsible for partial voids; they are more prevalent in electroless copper plating when hydrogen is generated during the copper reduction. A combination of improved racking and agitation should solve the problem.

A significant reduction in temperature can lower the plating rate of electroless copper to such an extent that the copper will be barely discernible. The resultant deposit will lack continuity and will be unsuitable for subsequent electroplating. The effect is essentially a void. Strict monitoring of temperature controls will correct the situation. The plating rate can also be adversely lowered by overdepletion of the bath constituents. Strict monitoring of the bath in accordance with vendor instructions for replenishment will serve to maintain a uniform plating rate.

The activators employed for establishing the precious metal sites to initiate electroless plating can become so depleted that no deposit will be obtained and a void will be obtained instead. The activators can be checked for strength either by titration or colorimetrically and their depletion thereby avoided.

Voids can occur on multilayer boards if any residual etchback is not neutralized. The retained etchback will leach out in processing and prevent electroless deposition.

2 Electroless copper decomposition. The decomposition of electroless copper is sometimes referred to as triggering. It consists of a rapid plating out of the copper from the electroless solution. Essentially, the copper reduction proceeds out of control; once begun, it is difficult to stop. The end result is a copper-plated tank. This is a costly phenomenon, since, in addition to stopping the line and downtime, an expensive solution must be replaced. On very rare occasions the triggering of an electroless copper bath can be attributed to an error in formulation, but the prime factors are inadequate process control, technique, and maintenance. Excessive activator drag-in is the most likely cause of triggering. Inadequate rinsing and poor racking are the largest causes of activator drag-in. Racks should be so designed as not to trap any solutions.

Foreign metallic materials and copper fragments are conducive to initiating decomposition. Although minute quantities of the metallic materials cannot be completely avoided, daily filtration by an all-plastic filter with a polypropylene cartridge capable of removing particles over 10 μm in size will minimize their effect. Areas of the electroless copper tank that come in contact with either boards or racks may have some electroless plating initiated on them. That can contribute substantially to solution depletion. In addition to filtration, it may be periodically necessary to pump the tank and etch away the plating sites.

Just as low temperature can contribute substantially to a low plating rate with a room-temperature (70 to 80°F) formulation, so too high a temperature can be a principal cause of decomposition. When there are wide fluctuations in temperature, a thermostatically controlled jacketed tank has been found to be very helpful in maintaining a uniform temperature.

3 Copper-to-copper bond failure. A failure of the bond between the copper-clad surface and the electroless copper deposit will in many instances not be detected until final electroplating has been completed. If that is indeed the case, the area of the bond failure must be isolated if the problem is to be corrected. One test method that has been used with success is to apply a drop of 10 percent ammonium persulfate to the underside of the "peeler" and to the copper substrate. The surface having electroless copper will turn black within 15 to 30 s. The change is due to residual palladium from activation. If both surfaces turn black, the surface having the darker deposit will usually be the one having the electroless copper. The situation will occur occasionally because of the relatively poor cohesive strength of electroless copper deposits. If the indication is that the electroless copper deposit is still on the substrate, the electroless copper line may be eliminated as the source of the bond problem.

If the electroless copper is on the underside of the peeler, then the bond failure is between the electroless copper and the copper-clad surface. That indicates the problem is probably in the electroless copper line. A discussion of probable causes follows.

Cleaning can always be a problem in any plating operation. As indicated previously, it is a standard practice for the copper foil manufacturers to employ an oxidation inhibitor on the surface of the copper foil. When the foil is subsequently heated in the laminate-pressing operation, the inhibitor can become extremely difficult to remove by conventional cleaning methods. A light etch as outlined above is employed to undercut the

inhibitor off the surface. The etch employed may in a loose sense be considered part of the cleaning operation. Following the cleaning operation, a water-break-free surface must be obtained before processing proceeds further. The presence of an organic contaminant can easily result in the lifting of subsequent plating.

Residual etchant and oxidation products resulting from the cleaning operation can be a cause of bond failure if those materials are not removed. In the case of an ammonium persulfate, a thin film of the double salt copper ammonium sulfate will form upon water rinsing. The film formation can be minimized by warm rinsing prior to immersion in a 10 percent sulfuric acid solution. If available, the substitution of sodium persulfate for ammonium persulfate will greatly minimize the problem.

The use of cupric chlorides for a light etch can also produce a film that can cause bond failure. In this instance the film is a cuprous oxide. It is removed in a 33 percent hydrochloric acid solution. A quick check for complete removal of the cuprous oxide film consists of immersing a panel in an alkaline cleaner. The appearance of a yellow film indicates the presence of a cuprous oxide film.

As indicated previously, the function of the postactivator is to act as a tin solubilizer and expose precious metal sites. Saturation of the solubilizer with tin will allow tin to remain on the copper-clad surface and cause bonding problems. There is also an indication that tin may be a factor in causing electroless copper decomposition. Adequate rinsing will prevent saturation from occurring prematurely. In addition, the area processed should be monitored so that the solution is periodically discarded.

It is essential that raw chemicals, particularly acids, be free of contaminants that could become a problem source. Plasticizers, miscellaneous organics, and metallics that are occasionally introduced by repackaging will readily cause bonding problems.

PANEL AND PATTERN PLATING

8. Function. At this point a thin deposit of electroless copper has established metal continuity through the holes. It is the function of the next plating step to build up the metal deposit in the holes to a minimum copper thickness of 0.001 in. This may take one of two routes, commonly known as panel and pattern plating. Panel plating consists of copper plating the entire board area including holes immediately following electroless copper. A negative circuit image is then applied and one of several metal-etch resists is plated onto the exposed copper. Pattern plating consists of plating only holes and circuitry with a minimum of 0.001 in of copper followed by the desired metal etch resist. That is accomplished by applying a negative circuit image following electroless copper plating.

9. Panel Plating. The following steps detail a typical plating cycle for panel plating. Panels are received directly from the electroless copper line and are racked for electroplating.

1. Clean. Alkaline or acid cleaners.
2. Spray or counterflow rinse.
3. Acid dip. HCl or H₂SO₄ (30 percent by volume).
4. Spray or counterflow rinse.
5. Copper-plate 0.001 in to 0.003 in in holes. A mat finish is preferred in this instance as a base offering good adhesion in the subsequent image-transfer process.
6. Spray or counterflow rinse.
7. Dry. This may be accomplished by any one of several means. Air drying by hand is probably the least desirable, since it is time-consuming and expensive. A water-displacing liquid followed by a degreasing operation has been employed with success. There are several conveyorized driers that employ air knives. In addition, there are combination scrubber-driers that dry as well as condition the surface for an image-transfer process.
8. Apply negative image. Silk screen, dry photosensitive film, or liquid photoresist.
9. Bake if applicable.
10. Rack for electroplating.
11. Alkali-clean (soak in electrolytic). If electrocleaning is used, anodic cleaning is preferred. Care must be taken not to damage the plating resist by rigidly controlling cleaner concentration, temperature, and current density.

12. Spray or counterflow rinse.
13. Light etch with ammonium persulfate. The employment of an etch is optional; it ensures complete removal of any residual photoresist.
14. Spray or counterflow rinse.
15. Acid dip. Sulfuric acid 10% by volume.
16. Spray or counterflow rinse.
17. Plate metal etch resist.
18. Spray or counterflow rinse. In the event the final metal etch resist is gold, a drag-out rinse should be incorporated in the line.

From this step upward, standard stripping and etching procedures are followed.

10. Pattern Plating The following steps detail a typical plating cycle for pattern plating. Panels from the electroless copper line go directly to image transfer.

1. Apply negative image. Silk-screen dry photosensitive film or liquid photoresist.
2. Rack for electroplating.
3. Alkali-clean (soak or electrolytic). Anodic electrocleaning is preferred if the image-transfer process employed was a photoresist process. That will ensure the removal of any residual not removed in developing. Care must be taken not to damage the plating resist by rigidly controlling cleaner concentration, temperature, and current density.
4. Spray or counterflow rinse.
5. Light etch. This step ensures the removal of any residual photoresist not already removed in developing the image. In the event a conventional electroless copper of the order of 0.000010 in has been applied, an etch is not recommended. A flash panel plate or heavy electroless deposit of at least 0.000100 in of copper must be applied before any form of etching is incorporated into the cycle.
6. Spray or counterflow rinse.
7. Acid dip. Sulfuric acid 20 percent by volume.
8. Spray or counterflow rinse.
9. Copper-plate 0.001- to 0.003-in in holes.
10. Spray or counterflow rinse.
11. Etch resist metal plate.
12. Spray or counterflow rinse. In the event the final metal etch resist is gold, a drag-out rinse should be incorporated in the line.

From this step onward, standard stripping and etching procedures are followed.

11. Comparison of Panel and Pattern Plating **1 Copper plating.** The largest single difference between panel and pattern plating occurs in copperplating. In panel plating the copperplating is completed prior to any image-transfer process. In pattern plating most of the copperplating is accomplished following image transfer.

A significant effect on circuit size is produced by pattern plating. The width of circuit lines and pads normally increases on each side about as much as the surface thickness during plating. That is, a 0.001-in plated surface thickness will increase line width approximately 0.002 in. On dense circuitry, where tolerances are critical, circuit growth must be allowed for when the master layout is made. Since no resist is present in the copperplating phase of panel plating, contamination from organic resists is eliminated. Isolated high-current-density areas that are prone to "burn" in pattern plating are not present in panel plating. Fewer steps are required in pattern plating. Less power is consumed, and material usage in copper is substantially reduced. Current density must be carefully controlled, especially when the plating areas are not uniformly distributed.

2 Metal etch resist and etching. A large portion of the copper plated in panel plating must be etched away. In a typical operation in which 1-oz copper-clad laminate is used and the holes are copperplated to 0.0015 in, the etching requirements are doubled. That in turn doubles etchant consumption, etchant time, and waste disposal and greatly increases the chance of unacceptable undercut. To minimize the effect of undercut in panel plating, a heavier metal etch-resist deposit is sometimes employed. It will to some extent aid in locking in any overhang from etching.

It is apparent from the foregoing that the excessive etching required is the prime disadvantage of panel plating. The availability of ½-oz copper-clad laminate and even thinner clad laminate may serve to alleviate the etching problem.

IMAGE TRANSFER AND ITS EFFECT ON PLATING²

12. Discussion Image transfer consists of the reproduction of a pattern from artwork or photographic film onto a copper- or other metal-clad laminate. In through-hole plating a negative image is applied following either electroless copper or copper electroplating depending on whether panel or pattern is selected. Although image-transfer processes may vary considerably in individual chemical and physical properties, their common purpose is to function as a plating resist for the particular plating process employed.

13. Silk Screen Silk screening is one of the major methods of applying a plating resist to a printed board. The name "silk" does not necessarily mean silk mesh is employed in the process, since nylon and stainless-steel are also used extensively. The negative image is formed by a stencil firmly attached to the screen mesh. The board is then aligned beneath the screen, and the screen is flooded with plating resist. The resist is forced through the open areas of the screen by the pressure of a squeegee drawn across the screen. After one side of the board is dry, the procedure is repeated on the other side. This is followed by a cure per the vendor's instructions. Subsequent to final plating, the resist is stripped, which leaves exposed a copper surface that will be etched away to form conductors, pads, and plated-through holes.

14. Plating Problems Related to Silk-Screen Resist Prior to the incorporation of any resist, a careful evaluation should be made to determine the compatibility of the resist with existing plating processes. There are several problems that become apparent in plating but actually can be traced to silk-screening.

1 Resist lifting. The lifting or breakdown of resist can have several causes. A steady recurrence may indicate that the wrong resist has been selected for a given plating process. Resists that are formulated to strip in an aqueous medium should not be processed through baths such as alkaline cleaners, copper pyrophosphate, or cyanide-based solutions.

There is always the chance that a vendor error has been made. In the event a material has been used successfully in the past and a lifting problem has suddenly developed, one area to be checked is the resist itself. Most vendors will mark their containers with lot numbers. A switch to a different lot-numbered material will quickly indicate whether the problem might lie in the resist itself.

Lifting along the edges of traces and pads may indicate several problems. In the event electrolytic cleaning is used, too high a current density, temperature, or concentration may cause this type of resist lifting in the cleaner itself or so weaken the bond that the resist lifts in a subsequent plating operation. The excessive use of inefficient plating solutions is a prime factor in causing edge lifting of resist. Inefficient baths such as strikes or cyanide golds evolve hydrogen at the cathode (plated part). Gas evolution has a decided scrubbing action that will loosen plating resist, especially when the effort is to build heavy deposits. The use of high-efficiency plating baths will minimize the effect.

Boards coming from electroless plating or panel plating into the image-transfer area should normally be clean, but there are always the routine handling operations that can be a source of organic contamination and subsequent resist lifting. To ensure against this reason for lifting, adequate controls such as gloves, board storage, restricted access, and adequate cleaning procedures should be standard practice.

2 Resist in holes. Resist in a hole can easily be mistaken for a void. Generally the error will occur when it is necessary to strip and rescreen a board. A faulty job of stripping will leave a difficult-to-detect film of resist in the hole. The ultimate result will be a partially plated or unplated hole.

3 Random-plated metal specks and nodules on the board surface. Extraneous metal on a board surface can come from several sources. It can be a form of resist breakdown that indicates there may be a problem with the resist being incompatible with the plating processes employed. Many resists will exhibit nodules to a minor degree, especially when they are used in conjunction with acid copper. Minor noduling will normally not cause a problem since the nodules will usually be removed in the etching process.

Pinholes in the resist will cause nodules in plating. Occasionally they may be traced to inclusions in the resist. The stencil as well as the artwork should be checked to ensure the problem is not in the silk-screening process itself.

In areas where the resist is marginally thin, too high a current density may produce nodules. Some "burning" in high-current-density areas may also be indicative of excessive current density.

4 Broken circuits. Broken circuits or a void in circuitry will usually become apparent following etching. The basic cause can be found in the artwork or the silk-screen process itself. A fault in the artwork will produce a defective stencil that will in turn produce a defective image on the board.

Encounters with fibers, dust, and other foreign materials on the silk screen are frequent. A fine fiber extending across circuits on a screen is difficult to detect and can produce a series of repetitions shorts. The same type of fiber when trapped across a board circuit can produce an ultimate broken circuit by acting as a resist for plating unless it is caught in silk-screen touch-up. An experienced operator will readily detect any irregularities by making frequent inspections of the screen and the image produced on the board.

15. Photoresists Photoresists are thin organic coatings that undergo a chemical change when exposed to light of the proper wavelength. The photosensitive coatings are classified as negative- or positive-acting. Exposure to light will either polymerize (negative type) or depolymerize (positive type) the coating and thereby make the affected area soluble (positive type) or insoluble (negative type) in a given developer. The foregoing mechanism is employed to transfer the desired image to a printed board. A negative or positive film (depending on the type of photoresist selected) is used to permit or prohibit the transmission of light. Subsequent to exposure, the unwanted resist is solubilized in the appropriate developer. The image is now formed and may act as a plating or etch resist.

Photoresist processes as pertains to image transfer for through-hole plating may be subdivided into several categories.

1 Liquid photoresists. The standard liquid photoresist is available as either positive or negative. Developers are either chlorinated-solvent-based or a mildly alkaline aqueous solution, depending on the resist employed. The resists are applied by dipping, spraying, or centrifugal force from a spinning apparatus. Following application, the resist is cured per vendor recommendations and is ready for exposure.

This type of resist is employed primarily on print-and-etch boards. Its use in through-hole plating is somewhat limited by problems encountered in both stripping and developing the resist out of holes.

2 Modified liquid photoresists. This category of photoresist is a modification of the standard photoresist formulation. The viscosity of the resist has been raised to enable the resist to be applied by a roller coater, which is a great aid in keeping photoresist out of the holes. Conveyorized roller coaters that incorporate an infrared curing station are available. They enable the rapid coating and curing of boards that are ready for exposure. To ensure a uniform coating free of inclusions and pinholes, rollers and working parts must be kept clean.

The selection of a photoresist will to a large extent be determined by the existing or proposed plating processes. Many photoresists, especially those that are developed in alkaline developers, are prone to break down in alkaline cleaners, copper pyrophosphate, or cyanide baths of any type. As with silk-screen resist, excessive current and low-efficiency plating baths will cause the photoresist to break down under the scrubbing action of evolved hydrogen at the cathode.

3 Dry film photopolymers. The dry film photopolymer is a marked improvement over its liquid resist counterparts and has been incorporated into the image-transfer processes of many board fabricators. The dry film is laminated to the metal-clad surface with hot rollers. It is available in varying thicknesses depending on the degree of chemical resistance required. The problem of resist in the hole is eliminated because there is no flow; the resist is a true dry film. Both negative- and positive-acting resists can be obtained. Developers are usually of the chlorinated solvent variety, but they may be a mild alkaline aqueous medium. In the chlorinated solvent category, 1,1,1-trichloroethane is currently used. Trisodium phosphate will function as a mild alkaline developer. Developer selection will depend on the photopolymer used.

As with the liquid counterpart, the selection of a dry film photopolymer will depend on compatibility with existing or proposed plating processes. The photopolymers that are developed in alkaline developers are not compatible with alkaline plating processes.

Both silk-screen and liquid photoresists, as applied, are relatively thin; they are of the order of 0.0002 in. With normal electroplating (in excess of 0.001 in), an appreciable growth in circuit and pad width will be noted. Dry film photopolymers may range in thickness from 0.0005 to 0.0025 in depending on what is selected by the fabricator to be optimum. Dry film that is 0.0013 in thick is typical of the thickness selected to function as a plating resist. A good collimated light source will produce an image that will have sharply defined walls on circuits and pads. The plating buildup will not expand circuit or pad width, since expansion will be prevented by the walls of the image produced by the dry film. The use of dry film as a plating resist for pattern plating has a potential problem that does not readily occur with silk-screen and liquid photoresist. After the metal etch resist is plated, the dry film is stripped and the board is ready for final etching. The walls of the circuitry and pads that have been formed by electroplating copper are now exposed to any subsequent etching process. This is very conducive to creating a severe undercut problem, especially where the board is exposed to the etchant for an extended period of time. The time exposed to etchant can be reduced by using thinner copper-clad laminate such as ½ oz. The use of an additive circuit process is ideal for a dry film photopolymer, since etching is virtually eliminated. In contrast to the foregoing, the growth of circuits and pads in electroplating, although cited as a disadvantage when silk-screening and liquid photoresist are used, does have one benefit. The enlarged circuit or pad when plated with a metal etch resist will shield the copper circuit or pad buildup from direct spray etching and thus greatly reduce undercut.

ELECTROPLATING^{16,17,18}

Electroplating is the process of depositing an adherent metallic coating upon a negatively charged electrode by the passage of an electric current in a conducting medium. Normally the medium is water in which metal salts have been dissolved as a source of metal ions. It is the purpose of electroplating to obtain a metallic coating that has different properties or dimensions than those of the basis metal.

As applied to through-hole plating, the negatively charged electrode is the printed board itself (cathode). The function of electroplating a printed board may be sevenfold:^{16,17}

1. Hole wall buildup
2. Conductivity
3. Wearability
4. Low contact resistance
5. Solderability

To accomplish the foregoing, many different plating processes are available. No blanket endorsement of any plating process can be given. This section is designed to present what is available and some of the advantages, disadvantages, properties, and control methods.

16. Process Evaluation To evaluate plating processes, the following areas should be taken into consideration:

1. Simplicity is of prime importance. Baths with only a few constituents will be easier to maintain within a given set of parameters, and problems will be readily pinpointed. The task of training personnel is greatly minimized.
2. Cost is a factor when any process is evaluated. Plating thickness, precious metals, proprietary additives, downtime, etc. are prime areas of consideration.
3. The effect of any given plating process on other processes should be part of the test criteria. Many image-transfer processes are incompatible with a given plating process. There is also the opposite consideration: what effect will existing processes have on the plating process being tested? The advantages in a new process may be sufficient to warrant changing some existing processes.
4. Waste disposal costs can be excessive if an elaborate treatment facility is necessitated by the process employed. Every bath in a testing program must, in addition to its performance, be evaluated as to the details of its treatment to comply with pollution criteria. Any testing program should include safety aspects such as emissions, flammability, and reactivity. These data should be available from vendors.
5. The large majority of plating solutions are made up from raw chemicals to which proprietary agents are added. The additives may be considered to be expensive, but the

experiences accumulated by a vendor may be used to maximum benefit by a plater and can more than offset the cost. A vendor's reputation for service in the event problems arise should influence any decision regarding the implementation of a process.

COPPER ELECTROPLATING

Copper has met with wide acceptance as the base conductor metal in printed circuit fabrication. It has excellent electrical conductivity (second only to silver), is readily plated, is low in cost, and gives highly reliable results. Copper is readily activated, and hence a good metal-to-metal bond is obtained between copper and other plated metals. Printed Wiring for Electronic Equipment, MIL-STD-275C, states that "plated-through holes shall be of copper and that the thickness of the copper plating in the hole shall not be less than 0.001 in." The three most commonly used copper solutions are pyrophosphate, acid sulfate, and acid fluoroborate.¹⁸

17. Copper Pyrophosphate^{19,20} Copper pyrophosphate has long been a standard of the industry for fulfilling the requirements of through-hole plating. The current efficiency in pyrophosphate copper is nearly 100 percent, which minimizes hydrogen gassing and the ensuing resist problems. A 1:1 ratio of hole thickness to surface thickness can be obtained if the bath parameters are maintained per Table 7.1.

TABLE 7.1 Specifications for a Copper Pyrophosphate Bath

Component	g/L	oz/gal
Copper metal, Cu ⁺²	22-38	3-5
Pyrophosphate, P ₂ O ₇ ⁻⁴	150-250	20-33
Oxalate	15-30	2-4
Nitrate, NO ₃	5-10	0.7-1.4
Ammonia, NH ₃	1.5-3	0.2-0.4
Orthophosphate, HPO ₄ ⁻²	No more than 10 to 13 oz/gal	
Operating conditions		
pH	8.2-8.8	
Ratio of pyrophosphate to copper	7.0-8.0	
Temperature, °F	110-140	
Cathode current density, A/ft ²	10-75	
Air agitation, ft ³ /min	t ft ² /ft ² of solution surface	
Anode-to-cathode ratio	t, 2:1-2.0:1	
Anodes	OFHC, rolled electrolytic, electrolytic sheet	

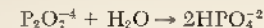
The deposits normally obtained without the use of proprietary additives are semibright, permitting the employment of a prepacked carbon filter for constant use to forestall any downtime because of organic contamination. To have consistent functioning without additives, very rigid controls must be maintained on the bath.

Proprietary brighteners and levelers are available to be used with the basic formulation. In the case of brighteners, the hardness of the deposit is increased along with the reflectivity. The use of a leveler will generally soften the deposit and produce a mat appearance. In general, it may be said that the bath is easier to run with additives. However, stress cracking and organic contamination due to additive degradation can occur, and the bath must be given a purifying treatment.

1. Function of bath constituents. Copper is present in the bath as potassium copper pyrophosphate. The compound is formed by the reaction of potassium pyrophosphate with copper pyrophosphate to form the complex anion Cu(P₂O₇)⁻⁶.

In addition to forming the soluble complex compound with copper, pyrophosphate increases the conductivity of the bath and aids in anode corrosion. Oxalate acts as a buffer to stabilize the pH of the bath. Nitrate reduces the amount of polarization at higher current densities by inhibiting the reduction of hydrogen ions. The operating current density is thereby increased without the danger of burning. The nitrate content is generally added on a proportionate basis with pyrophosphate.

Ammonia is added to assist anode corrosion and add luster to the copper deposit. Ammonia is lost by evaporation and must be replaced by ammonium hydroxide. Orthophosphate is present in the bath as a product of hydrolysis of pyrophosphate.



The prime contributors to orthophosphate are low pH, high operating temperature, and a high ratio. As orthophosphate increases, bath conductivity is lowered and the plating range is decreased. Eventually the orthophosphate buildup will cause the bath to be discarded.

2 Bath maintenance

a. pH. The pH of the bath must be kept within the limits specified in Table 7.1. Depending on the ammonia content, the pH can be raised by the addition of ammonium or potassium hydroxide. To lower the pH, pyrophosphoric acid is employed.

b. Temperature. To insure against excessive orthophosphate formation, local overheating must be prevented. I-type heaters and heat exchangers are effective in circumventing local overheating. Thermocouples for heating control units should be checked to ensure against overall bath overheating.

c. Anode area. It is essential for bath performance and balance that the anode area be maintained in excess of the cathodic area. Anodes must be checked frequently, especially with heavy work loads, to determine the extent of the anodic corrosion and the need for additional anodes. It should be noted that anode bags are not recommended for pyrophosphate copper.

d. Filtration. Continuous or occasional filtration of the bath is recommended to remove dirt and insoluble matter. Periodic carbon treatment should be performed to remove organic contamination.

e. Air. Only air supplied from a blower should be used. Compressor air is notorious for oil contamination. An air filter should be installed on the inlet of the blower.

f. Analysis. Analysis should be made on a regular basis for copper pyrophosphate and ammonia. Heavily used baths should be analyzed very frequently to ensure the proper balance of bath constituents. Orthophosphate content should be occasionally checked. The procedures for the analysis are given in the process-control section.

3 Troubleshooting copper pyrophosphate. To diagnose any plating problem, the first step is to ensure, by analyzing the solution, that the bath constituents are not out of balance. Common problems encountered in pyrophosphate copper may be listed as follows.

a. Rough deposits. Foreign matter can cause rough deposits. It can originate from several sources: inadequate cleaning, lint, dust, fibers, nodules, and carbon. Adequate filtration will normally remove foreign matter from a bath.

Improper anode corrosion can result in rough plating; it is caused by too large an anode area, a pyrophosphate-to-copper ratio of less than 6.5, or a low ammonia content.

b. Stressed or cracked deposit. Stress-cracking normally appears at the orifice of the plated-through hole. It is detected by soldering the board and subsequently microsectioning the hole. Stress-cracking is caused by excessive additive use or by degradation by-products of the additives. It is corrected by a carbon and peroxide purifying treatment.

c. Step-plating. Step-plating, skip-plating, or fish-eye, as it is commonly called, consists of high-current-density areas that are unplated or skipped. The condition is generally found around a plated-through hole and pad. It is overleveling caused by the use of excessive additives. The situation can be corrected by adjusting current densities until the additives come back into the proper range.

d. Pitted deposits. Pitting in a copper bath is usually due to grease or oil in the bath; on occasion it can be attributed to the cleaning cycle. The oil source is frequently the air supply. Oil and grease can be removed by carbon treatment. A small amount of a nonionic wetting agent is beneficial in minimizing the effects of oil and grease contamination.

e. Poor throwing power. Hole sectioning will readily detect any loss of throwing power. Additive breakdown and orthophosphate buildup are the prime causes of poor throwing power.

f. Lifted plating. The loss of plating adhesion is usually found to be linked to something other than the plating bath itself. The first step in correcting the problem is to firmly establish where the bond is failing. This will ensure the failure is not an electroless copper line problem.

Once the problem has been isolated to the copper electroplating line, the cleaning cycle should be closely scrutinized. Contaminated cleaners and acids are always suspect.

Anodic electrocleaning in an alkaline cleaner is very effective in removing stubborn soils, much more so than simple immersion cleaning. Care must be exercised and parameters established when anodic electrocleaning is used in conjunction with pattern plating and a resist.

g. Plating racks. Plating racks can be a source of multiple contamination, both metallic and organic, if they are not kept in good repair. Racks are continually recycled through cleaners, acids, and plating baths. Lifted rack coatings easily trap those solutions and can drag them through entire lines. Periodic inspection and maintenance will minimize the effect.

4 Equipment. Materials of construction for tank linings, filters, and other equipment are given in Table 7.21.

5 Plating rate. Plating rates are given in Table 7.5.

16. Copper Sulfate^{19,20} Copper sulfate is one of the copper-plating baths classified as acid copper. Sulfuric acid is added to the copper sulfate bath for conductivity. These are the two main constituents of the bath, and they are relatively easy to control. Historically the copper sulfate bath has been in use for many years, but until recently its use on printed boards has been somewhat limited owing to poor throwing power.

The deposit obtained without additives is generally dull, soft, and somewhat coarse; it can be significantly altered by the use of additives. The additives can be effective in grain refinement, leveling, and hardening and as a brightener or a means of increasing the current-density range. They include glue, peptone, resorcinol, thionrea, molasses, and gum arabic. Proprietary additives are used frequently because they are more controllable and highly systematized and afford the user the benefit of a vendor's experience and expertise.

Table 7.2 lists the composition and operating conditions for a copper sulfate bath. It should be noted that there are two variations. The first is the standard formulation, and the second is a formulation consisting of low metal and high acid to increase throwing power.

1 Function of bath constituents. Copper ions are added to the bath as copper sulfate. It is essential that a good plating grade of copper sulfate be used; it is readily available from most vendors. Sulfuric acid is used for conductivity and anode corrosion. Care must be taken to select a good grade of sulfuric acid, because many commercial grades are prime sources of organic contamination.

Chlorides are essential in the brightener and leveler mechanisms of some proprietaries. Their addition depends on the particular process employed. It should be noted that the basic formulation contains no chelating or complexing agents. That greatly simplifies any waste disposal treatment.

2 Bath maintenance

a. Temperature. The degree of temperature control necessary will depend to a large extent on the additives employed. The proprietary bright baths require temperatures of the order of 75°F. Excessive temperatures will result in additive breakdown and a

TABLE 7.2 Specifications for Acid-Copper Sulfate Baths

Component	Standard		High Throwing Power	
	g/L	oz/gal	g/L	oz/gal
Copper sulfate, CuSO ₄ · 5H ₂ O	160-300	24-40	60-90	8-t2
Copper, Cu ²⁺	40-75	6-10	15-22.5	2-3
Sulfuric acid, H ₂ SO ₄	52.5-135	7-18	187.5-225	25-30
Addition agents	As required		As required	
Chloride ion	20-80 ppm		20-80 ppm	
Operating conditions				
Temperature, °F	65-90			
Cathode current A/ft ²	20-50			
Agitation	Blower air			
Anodes	Phosphorized copper (0.02-0.08% P)			
Anode bags	Dynel or polypropylene			
Filtration	Continuous or occasional			

loss of throwing power. Too low a temperature will lower the efficiency of the bath. To maintain the proper operating temperature, the appropriate heating or cooling equipment should be installed.

With additives that are not temperature-sensitive an increase in operating temperature will permit the use of higher current densities.

h. Anodes. For optimum results phosphorized anodes should be employed. To contain small copper fines that occur in the corrosion process, the anodes should be bagged with polypropylene or Dynel bags. Cotton or nylon bags should not be used. Bags should be periodically removed, washed out, and checked for holes. New anode bags should be soaked in 10 percent sulfuric acid prior to use.

c. Filtration. For optimum solution clarification continuous filtration is recommended, although periodic filtration may sometimes suffice. Dynel or polypropylene is recommended for filter tubes or bags. Diatomaceous earth or asbestos may be used as a filter aid. A cellulose filter aid should not be used. Gation-type filter cartridges cannot be used in all applications, since they can remove additives. Their use should be a function of the system employed.

d. Agitation. As indicated for pyrophosphate copper, only blower air should be used for acid copper. The degree of agitation will depend largely on the surface area and spacing of the boards in the bath. Close spacing will usually require more vigorous agitation. The agitation should be kept as uniform as possible. In some instances cathode bar agitation has been used beneficially in conjunction with air agitation.

e. Analysis. Analysis should be made on a regular basis for copper, sulfuric acid, and chloride content where applicable. The use of the specific gravities in Table 7.3 will

TABLE 7.3 Total Concentrations of Copper Sulfate plus Sulfuric Acid vs. °Baumé at 25°C

°Baumé	CuSO ₄ + H ₂ SO ₄	°Baumé	CuSO ₄ + H ₂ SO ₄
	oz/gal		oz/gal
12.0	20.0	20.0	35.9
12.5	21.0	20.5	37.0
13.0	21.9	21.0	38.1
13.5	22.9	21.5	39.2
14.0	23.8	22.0	40.4
14.5	24.7	22.5	41.6
15.0	25.7	23.0	42.8
15.5	26.8	23.5	43.9
16.0	27.8	24.0	45.0
16.5	28.8	24.5	46.1
17.0	29.8	25.0	47.3
17.5	30.8	25.5	48.5
18.0	31.8	26.0	49.7
18.5	32.8	26.5	51.0
19.0	33.8	27.0	52.3
19.5	34.9		

give a rough estimate of the copper metal concentration. The acid concentration is determined and subtracted from the total concentration indicated by the table for the particular degrees Baumé. Although the concentrations of the constituents in copper sulfate are not as critical as those in pyrophosphate copper, it is essential that the constituents be monitored according to the process control section on a regular schedule.

3 Troubleshooting copper sulfate. As with any plating solution it is essential to determine whether the bath constituents are in operating range prior to taking any corrective action to solve a given problem. Erroneous additions, a filter leak, a leak in a tank, or excessive drag-out can easily go undetected but will at once become evident when an analysis of the bath is made.

a. Roughness. Roughness may have any of several causes. Inadequate filtration of foreign matter is a prominent cause; as indicated previously, continuous filtration will eliminate it.

Anode bags should be checked for holes. The copper fines contained by the anodes, if allowed to permeate the solution, will readily cause roughness. Filtration through a 5-μm filter aid will remove the particles.

Nickel and iron in sufficient quantity will cause roughness. Aside from defective plating racks as a source, the chances of a ferrous tool or similar instrument falling into the bath and being subsequently dissolved are always good. The spray from an agitator in a copper sulfate bath is very corrosive and will corrode any ferrous alloys in the immediate area. That can lead to corrosion products falling into the bath.

Chloride content in excess of 200 ppm will manifest itself in the form of roughness. In areas in which a high buildup of chloride is possible because of the local water supply, either deionized or reverse-osmosis water should be employed for bath makeup and replenishment.

b. Organic contamination. Organic contamination can manifest itself in several ways. Dullness, pits, and step-plating are symptoms of organic contamination. The sources of organic contamination include excessive amounts of additives, plating resists, poor-quality plating chemicals, and oil from the air supply. Oil contamination will usually cause very large pits and will be quite distinctive from other forms of contamination.

The appearance of organic contamination will necessitate a carbon and peroxide treatment for removal. It should be noted that the purifying treatment will be likely to remove most additives and that the additives will have to be replaced.

c. Dullness. In a bath employing additives for brightness a dull deposit can indicate organic contamination, low brightness, or low chloride. Low chloride content can be determined by analysis. The addition of brighteners in small increments will determine if low brightness is the problem. Most proprietary additives are added on an ampere-hour basis. It may be necessary in some cases to alter the addition schedule to prevent low brightener content.

d. Stressed plating. As in pyrophosphate copper, the excessive use of additives, particularly for brightness, can cause stress-cracking upon soldering at the orifice of the plated-through hole. The brightener content can be lowered by drenching the bath. In that case the addition schedule of any brighteners should be revised downward.

e. Poor throwing power. As the copper content is increased, the throwing power of the bath is decreased. In a heavily worked bath that will not usually occur, but in baths that are idle for long periods it may be necessary to remove the anodes to prevent the excessive buildup of copper metal from anode corrosion. Copper metal content can also be regulated by the addition or removal of anodes.

As the sulfuric acid content decreases, so will the conductivity of the bath, and with a resultant loss in throwing power. Routine analysis should detect any decrease in the sulfuric acid content.

f. Low metal content. Low metal content is easily detected by analysis and is characterized by a low plating rate. Consistently low metal, excluding excessive drag-out, can usually be traced to the anodes. Dense anode bags, coupled with the formation of copper fines from anode corrosion, can so inhibit further corrosion as to prevent the bath from being replenished with copper as quickly as it is depleted at the cathode. Highly dense anode bags should not be employed, and a periodic wash-out of all bags should be included in routine maintenance. Insufficient anode area also will contribute to low metal. Anodes should be frequently inspected to determine the degree to which they have been corroded.

4 Equipment. Materials of construction for tank linings, filters, and other equipment are given in Table 7.21.

5 Plating rate. Plating rates are given in Table 7.5.

19. Copper Fluoroborate^{19,20} Copper fluoroborate is another of the acid baths. It has been replaced to some extent by copper sulfate for through-hole plating. Its constituents are more expensive than copper sulfate, but it does have several advantages. It is marketed as a liquid concentrate. That greatly facilitates bath makeup and metal replenishment as opposed to the dissolution of metal salts. Fluoroborate can be plated at higher current densities than pyrophosphate copper or copper sulfate, which permits faster buildup. Fine-grained deposits can be achieved without additives, although molasses (1 pt/100 gal) is used to harden the deposit and prevent "treeing."

As can be seen from Table 7.4, the operating parameters of the bath constituents are wide and do not require the razor-edge control inherent in other formulations. Lower metal and higher acid content will increase the throwing power of the bath. For through-hole plating proprietary, high-throwing-power formulations are primarily employed.

1 Function of constituents. Copper fluoroborate is the source of metal ions. Higher metal concentrations allow high current density. Tetrafluoroboric (fluoroboric) acid is added for bath conductivity. Boric acid acts to stabilize the fluoroborate ion and prevent copper salts from precipitating as $Cu(BF_4)_2$. Additives, when used, affect hardness, grain structure, and brightness, depending on the type selected.

TABLE 7.4 Specifications for Acid Copper Fluoroborate Baths

Component	g/L	oz/gal
Copper fluoroborate, $Cu(BF_4)_2$	224-448	30-60
Copper metal, Cu^{+2}	60-120	8-16
Fluoroboric acid, HBF_4 (free)	10.5-30	1.4-4
Boric acid, H_3BO_3	15-30	2-4
Addition agents	As required	
Operating conditions		
Temperature, °F	65-120	
Cathode current density, A/ft ²	75-125 (low copper content) 125-350 (high copper content)	
pH (calorimetric)	1.2-1.7 (low copper) 0.2-0.6 (high copper)	
Anodes	Rolled electrolytic or OFHC	
Anode-to-cathode ratio	1:1	
Agitation	Blower air and cathode bar	

2 Bath maintenance

a. Temperature. Increasing the temperature increases the limiting current density and lowers the bath resistance. When additives are employed, the effect of higher temperatures on their stability should be investigated.

b. Filtration. Continuous filtration is preferred, but occasional filtration is generally sufficient.

c. Analysis. The copper metal content should be determined in accordance with the procedures in the process control section. The fluoroboric acid content can be maintained by keeping the pH below 1.7 by the addition of muriatic acid. An analysis for free fluoroboric acid should also be made periodically.

d. Anode bags. Anode bags should be either Dynel or polypropylene. They should be washed and checked for holes frequently.

3 Troubleshooting copper fluoroborate. Many of the problems encountered with the copper sulfate bath are possible with copper fluoroborate also. The section on troubleshooting copper sulfate should be checked.

a. Nodules on plating resist. The high current densities that can be used with copper fluoroborate can cause nodule formation on plating resist when used for pattern plating. That may necessitate a lower current density or modification or change in image transfer procedures.

b. pH. When the pH is allowed to exceed 1.7, a dark, brittle deposit will result.

4 Equipment. Materials of construction for tank linings, filters, and other equipment are given in Table 7.21.

5 Plating rate. Plating rates are given in Table 7.5.

SOLDER (TIN-LEAD) PLATING²¹⁻²³

20. Function and Properties Solder plating (60 percent tin, 40 percent lead) has a dual purpose. It functions both as a metal etch resist and as a solderable substrate for the subsequent soldering of components. Since the alloy is near the tin-lead eutectic (63 percent, 37 percent Pb, 376°F), it is easily reflowed, and that makes it readily solderable. For through-hole plating, solder plate is employed by the majority of printed circuit fabricators when a constant alloy content is required for consistent soldering. Military specification MIL-P-81728, Plating Tin-Lead (Electrodeposited), states that

Unless otherwise specified, the thickness of tin-lead plating for electronic components (printed circuit boards, especially those with plated through-hole interconnections, terminals and eyelets) shall be a minimum average of 0.0003 inch (0.3 mil) thickness when measured at four points at least 0.10 inch apart.

TABLE 7.5 Plating Rate for Copper Pyrophosphate, Sulfate, and Fluoroborate Based upon 100 Percent Cathode Current Efficiency

Thickness, in	Plating time, min, at varying current densities, A/R ²							
	10	15	20	25	30	40	50	60
0.0001	106	7.1	5.3	4.2	3.5	2.6	2.1	1.8
0.0002	21	14	10.6	8.4	7.1	5.3	4.2	3.5
0.0003	32	21	16	13	10.6	8.0	6.4	5.3
0.0004	42	28	21	17	14	10.6	8.5	7.1
0.0005	53	35	27	21	18	13	10.6	8.8
0.0006	64	42	32	25	21	16	13	10.6
0.0007	74	50	37	30	25	18	15	12
0.0008	85	57	42	34	28	21	17	14
0.0009	95	64	48	38	32	24	19	16
0.001	106	71	53	42	35	26	21	18
0.00125	133	89	67	54	44	33	27	22
0.0015	159	106	80	64	53	40	32	26
0.00175	186	124	93	74	62	46	37	31
0.002	212	142	106	84	71	53	42	35
0.003	318	213	159	126	106	79	64	53

Method 208 of MIL-STD-202 describes a mechanism used to determine the solderability of a plated deposit. To be acceptable, plating should be easily and completely coated with solder when tested.

The tin-lead alloy plating bath consists of tin fluoroborate, lead fluoroborate, fluoroboric acid, and additives. The metal fluoroborates are available as concentrates that are diluted with water to the desired metal content. The contents of the available liquid metal concentrates are listed in Table 7.6. Variations in the bath makeup are listed in Table 7.7; they include a high-throw formulation for through-hole plating.

1 Function of bath constituents. Stannous and lead fluoroborates are the source of metal. Variations in the metal content of the bath will affect the content of the alloy deposited.

Free fluoroboric acid increases bath conductivity and improves throwing power. At higher concentrations of fluoroboric acid the grain size of the deposit becomes finer and smoother. Boric acid is added to maintain bath stability by inhibiting the decomposition of fluoroborate. Peptone promotes smooth, fine-grained "tree-free" deposits. Gelatin has much the same effect and has been used successfully in place of peptone.

Proprietary additives serve the same function as peptone but are marketed as liquid concentrates, and are thus easier to employ. Several brighteners that are available will give a mirror-bright solder. It has been claimed that a bright deposit eliminates the need for a reflow operation.

2 Factors affecting the metal content of the deposit. Tin is the prime constituent subject to variation, and several conditions in the bath can bring about an increase or decrease in its content.

a. Stannous tin will oxidize on standing for a long period of time. An analysis will show if the stannous tin content in a bath is still within the operating parameters.

TABLE 7.6 Composition of Tin and Lead Metal Concentrates

Component	Weight, %	g/L	oz/gal
Lead fluoroborate concentrate			
Lead fluoroborate, Pb(BF ₄) ₂	50.0	877.5	117.0
Lead metal, Pb	27.2	475.5	63.4
Fluoroboric acid, free, HBF ₄	0.6	10.5	1.4
Boric acid, free, H ₃ BO ₃	3.0	48	6.4
Stannous fluoroborate concentrate			
Stannous fluoroborate, Sn(BF ₄) ₂	50.0	800	106.6
Tin metal, Sn	20.3	325	43.3
Fluoroboric acid, free, HBF ₄	3.0	48	6.4
Boric acid, free, H ₃ BO ₃	3.0	48	6.4

- b.* Temperatures higher than 100°F will increase the tin in the deposit slightly.
c. Higher than recommended current densities will increase the tin in the deposit.
d. Excessive agitation will increase the oxidation of stannous tin in the bath and subsequently decrease the tin in the deposit.

TABLE 7.7 Specifications for Solder (60% Tin, 40% Lead) Plating Baths^{10,20}

Makeup solution for 100 gal of standard bath	
Stannous fluoroborate concentrate	17.2 gal
Lead fluoroborate concentrate	5.25 gal
Boric acid	9 lb
Fluoroboric acid, 48%	15 gal
Peptone	4 lb
Water	62.5 gal

Dissolve boric acid in hot water prior to adding to tank. Allow peptone to swell in cold water, and then heat the water while stirring vigorously. The fluoroboric acid should be added to water first, followed by boric acid and the metal concentrates.

Composition of standard bath

Component	g/L	oz/gal
Stannous tin	56.2	7.5
Lead	26.2	3.5
Free fluoroboric acid	100.0	13.3
Free boric acid	26.2	3.5
Peptone	5.2	0.7

Makeup solution for 100 gal of high-throwing-power formulation*

Stannous fluoroborate concentrate	4.3 gal
Lead fluoroborate concentrate	2.2 gal
Fluoroboric acid, 48%	60.0 gal
Boric acid	18 lb
Peptone	4 lb
Water	33 gal

Composition of high-throwing-power formulation*

Component	g/L	oz/gal
Stannous tin	15.0	2.0
Lead	10.0	1.3
Free fluoroboric acid	400.0	53.3
Free boric acid	21.6	2.9
Peptone	5.2	.7

Operating conditions

Temperature, °F	60-100
Cathode current density, A/R ²	10-25 (low metal content) 25-40 (high metal content)
Agitation	Slow cathode bar
Anode-to-cathode ratio	2:1
Anodes	60 Sn, 40 Pb

* U.S. Patent 3,554,878

e. Resorcinol is added to the bath at a concentration of 0.5 to 1.0 g/L to inhibit tin oxidation. The content of the deposit will be increased through the use of resorcinol. This will necessitate a reduction of the stannous tin content of the bath.

3 Bath maintenance

a. Temperature. For baths employing peptone or a similar additive, the bath temperature is not very critical. Baths employing proprietary brighteners require closer control, of the order of 70 to 80°F, to prevent brightener breakdown.

b. Filtration. The bath should be periodically filtered to maintain clarity and prevent roughness. Periodic carbon treatment of the bath is necessary to remove break-

down products from additives. Hydrogen peroxide should not be used in any purifying treatments.

c. Analysis. Frequent analysis should be performed for stannous tin, lead, and fluoroboric acid. Any bath that has been idle should first be analyzed to ensure the constituents are within the operating parameters. In addition, an analysis of the deposit should be made in conjunction with the solution analysis. Analytical procedures are in the process control section.

d. Anode bags. Anode bags should be either Dynel or polypropylene. They should be washed and checked for holes frequently.

4 Troubleshooting tin-lead fluoroborate.

a. Organic contamination. Most of the effects of organic contamination can be avoided by periodically treating with carbon. The symptoms are a streaky deposit, loss of throwing power, and poor solderability of the deposit. The prime sources of organics are additive breakdown and plating resists.

*b. Metallic contamination.*²¹ Since copper plating normally precedes solder plating, it is a prime candidate for drag-in and subsequent metallic contamination. It is characterized by a loss of throwing power and an increase in grain size. Copper should be removed by illuminating at the normal plating current density; corrugated steel cathodes should be used.

c. Chlorides. Chlorides can reduce throwing power, affect grain structure, and throw the alloy deposit out of balance. To avoid chlorides, do not use water that contains chlorides for bath makeup or replenishment. Do not use a hydrochloric acid pickle prior to solder-plating. A fluoroboric acid pickle is recommended.

d. Air. Do not allow air to be introduced into the bath; it will cause rapid oxidation of the stannous tin. Avoid leaky filter pumps. Ensure that the effluent, when filtering, is introduced to the bath beneath the solution level so as to keep air contact to a minimum.

e. Poor solderability. In addition to organic contamination in the bath being a source of poor solderability, inclusions in the deposit can cause severe dewetting. It is a practice in some smaller shops to scrub the solder deposit with pumice or a similar but proprietary compound to remove any etchant by-products and in general improve the aesthetics of the deposit. Although a semibright appearance will be imparted to the solder deposit, the pumice and other materials imbedded in the surface of the deposit by the scrubbing can cause severe soldering problems when the board is assembled. Many bright dips that are available will serve the same function, as well as reflowing, and still preserve solderability.

5 Stripping solder (60 tin, 40 lead) plate. Tin-lead deposits are stripped from copper by immersion in the following solution:

Acetic acid (glacial)	1 gal
Hydrogen peroxide (30 percent)	20 fl oz
Water	3 gal

Room temperature

6 Equipment. Materials of construction for tank linings, filters, and other equipment are given in Table 7.21.

7 Plating rates for solder (60 Sn, 40 Pb) plate. These are given in Table 7.8.

NICKEL ELECTROPLATING

21. Function and Properties Nickel plating is employed on printed boards as an undercoat for precious and nonprecious metals. For surfaces such as switch contacts or tabs that normally receive heavy wear, the use of nickel under a gold or rhodium plate will greatly increase wear resistance. When used as a barrier layer, nickel is effective in preventing diffusion between copper and other plated metals. Nickel-gold combinations are frequently used as metal etch resists. Nickel alone will function as an etch resist against the ammoniacal etchants. Mil-STD-275B calls for a low-stress nickel with a minimum thickness of 0.0005 in followed by gold as one of the options for through-hole plating.

Low-stressed nickel deposits are generally obtained by using modified Watts bath formulations and nickel sulfamate baths in conjunction with additives that function as stress reducers.

22. Nickel Sulfamate Nickel sulfamate is widely used both as undercoat for through-hole plating and on tabs. The deposit obtained has low internal stress, is hard, and has excellent ductility. The addition of a stress reducer to the bath will yield deposits that are slightly compressively stressed. There are several variations of the sulfamate bath; a typical one is given in Table 7.9. The nonchloride formulation is widely used because of the low stress properties of the deposit.

TABLE 7.8 Plating Rate for Solder (60 Sn, 40 Pb) Plate from the Fluoroborate Bath Based upon 100 Percent Cathode Current Efficiency

Current density, A/ft ²	Plating time, min, for varying plate thickness, in.			
	0.0001	0.0003	0.0005	0.001
10	4.5	13.5	22.5	45.0
15	3.0	9.0	15.0	30.0
20	2.3	6.9	11.5	23.0
25	1.8	5.4	9.0	18.0
30	1.5	4.5	7.5	15.0
40	1.1	3.3	5.6	11.2

1 Function of bath constituents. Nickel sulfamate is a primary source of nickel metal ions. The concentration of the liquid may vary somewhat with the vendor.

Boric acid functions as a buffer. It also reduces burning at high current densities. Nickel bromide is used in place of nickel chloride to minimize internal stress and act as an agent for anodic corrosion. Stress reducer is added to give a slightly compressively stressed deposit. It will also impart a semibright appearance to the deposit as opposed to a mat finish without the stress reducer. It is generally added by ampere-hour.

TABLE 7.9 Typical Nickel Sulfamate Plating Bath^{19,20}

Component	g/L	oz/gal
Nickel sulfamate, Ni(SO ₃ NH ₂) ₂ *	300-525	40-70
Nickel metal, Ni	60-120	8-16
Boric acid, H ₃ BO ₃	30-45	4-6
Nickel bromide, NiBr ₂ *	11-19	1.5-2.5
Stress reducer	As needed	
Operating conditions		
Temperature, °F	120-140	
Cathode current density, A/ft ²	15-80	
Agitation	Cathode bar or solution circulation	
Anodes	Rolled, depolarized, or cast carbon. Nickel may be used.	
pH	2.5-4.0	

*Nickel sulfamate and nickel bromide are supplied as liquid concentrates.

2 Bath maintenance.

a. Temperature. The operating temperature is critical and should be maintained within the limits specified.

b. pH. pH is raised by the addition of nickel carbonate and lowered by the addition of sulfamic acid.

c. Anode bags. Anode bags should be made of Dynel or polypropylene. Bags should be periodically washed and checked for holes. New bags should be soaked in boiling water prior to use.

d. Purification. Baths should be carbon-treated when there is evidence of organic contamination.

e. Analysis. The bath should be analyzed periodically in accordance with the procedures outlined in the process control section.

f. Filtration. Constant filtration is ideal.

3 Troubleshooting the nickel sulfamate bath. See Modified Watts Nickel below.

4 Plating rate. See Table 7.11.

23. Modified Watts Nickel The modified Watts formulation, Table 7.10, employs nickel sulfate in conjunction with nickel bromide or chloride. The bromide is preferred because of the internal stress factor. It produces a deposit that is semibright, slightly tensile-stressed, ductile, and readily activated for subsequent plating.

1 Function of bath constituents. Nickel sulfate is the primary source of nickel metal ions. Boric acid functions as a buffer. It also reduces burning at high current densities,

TABLE 7.10 Modified Watts Nickel Plating Bath

Component	g/L	oz/gal
Nickel sulfate, NiSO ₄ ·6H ₂ O	225-375	30-50
Nickel metal, Ni	50-84	6.7-11.15
Boric acid, H ₃ BO ₃	30-45	4-6
Nickel bromide, NiBr ₂ *	11-19	1.5-2.5
Stress reducer	As needed	
Operating conditions		
Temperature, °F	120-140	
Cathode current density, A/ft ²	15-80	
Agitation	Cathode bar or solution circulation.	
Anodes	Rolled, depolarized or cast carbon. Nickel may be used.	
pH	2.5-4.0	

*Nickel bromide is supplied as a liquid concentrate.

Nickel bromide or chloride is used to promote anode corrosion. Stress reducer serves to maintain a low internal stress as well as impart a semibright appearance to the deposit. Most proprietary stress reducers are added by ampere-hour.

2 Bath maintenance.

a. Temperature. The operating temperature is critical and should be maintained within the limits specified.

b. pH. pH is raised by the addition of nickel carbonate and lowered by the addition of sulfuric acid. The pH will tend to rise as the bath is operated. A higher pH will result in a brighter deposit but with some loss of ductility. Low pH values will cause lowered plating efficiencies.

c. Anode bags. Anode bags should be made of Dynel or polypropylene. Bags should be periodically washed and checked for holes. New bags should be soaked in boiling water prior to using them.

d. Purification. Baths should be carbon-treated when there is evidence of organic contamination. This process will normally remove the stress reducer, which will have to be replaced.

e. Analysis. The bath should be analyzed periodically in accordance with the procedures outlined in the process control section.

f. Filtration. Constant filtration is ideal.

3 Troubleshooting the nickel sulfamate and the modified Watts nickel bath.

a. Pitting. Pitting is a result of organic contamination. Large pits will generally indicate oil contamination. Poor agitation will fail to dislodge hydrogen bubbles, which will result in pits. The incidence can be reduced by using a wetting agent.

b. Roughness. Roughness indicates a dirty solution. Adequate filtration will correct the difficulty.

c. Lack of adhesion. Peelers will occur if the copper deposit has not been adequately deoxidized. The bond between copper and nickel will fail. Current interruption will cause the nickel deposit to peel from itself at the point of the interruption.

d. Brittle deposits. Brittle deposits will generally manifest themselves when the deposit is flexed or given any degree of wear. They are indicative of organic contamination. Excessive additives, drag-in, and plating resist are prime sources of organics; they necessitate carbon treatment.

e. Dark nonuniform deposits. Dark, erratic deposits will frequently indicate metallic contamination. Since copper plating generally precedes nickel plating, it is usually the prime candidate for drag-in. It is essential that plating racks be adequately maintained to

minimize trapped solution. To remove metallics, particularly copper, this solution should be dummied at 2 to 5 A/ft² by using corrugated steel cathodes for 5 Al per gallon of solution.

f. Burned deposits. Burned deposits are caused by insufficient boric acid, low metal concentration, too low an operating temperature, too high a current density, or insufficient agitation.

g. Low deposition rate. A low deposition rate will be caused by a low pH or a low current density.

4 Equipment. Materials of construction for tank linings, filters, and other equipment are given in Table 7.21.

5 Plating rate. Plating rates for nickel are given in Table 7.11.

TABLE 7.11 Plating Rate for Nickel Sulfamate and Modified Watts Nickel Based upon 98 Percent Cathode Current Efficiency

Current density, A/ft ²	Plating time, min, for varying plate thickness, in				
	0.0001	0.0003	0.0005	0.0007	0.0010
10	12	36	60	84	120
20	6	18	30	42	60
30	4	12	20	28	40
40	3	9	15	21	30
50	2.4	7.2	12	16.7	24

TIN-NICKEL ELECTROPLATING¹⁹

24. Function and Properties As the cost of precious metal plating has increased, tin-nickel alloy has generated a moderate degree of interest as a metal etch resist. It is deposited as an intermetallic containing 65 percent tin and 35 percent nickel. Tin-nickel has functioned successfully on contact areas where electrical resistance has not been critical. The deposit has excellent corrosion resistance, high hardness, and good conductivity. The solderability of the deposit can vary considerably. Freshly plated surfaces will solder readily with type RA rosin flux, whereas aged deposits will generally require water-soluble (active) fluxes to promote a reliable solder joint.

The bath has excellent throwing power, which makes it suitable for through-hole plating. No brighteners are employed in the bath, and the appearance of the deposit will mirror the base metal. Both fluoride-chloride and fluoroborate electrolytes are used to plate tin-nickel. The electrolytes are extremely corrosive, and the precautions outlined in the safety section should be followed. Subsequent discussion will focus on the fluoride-chloride formulation, Table 7.12.

1 Function of bath constituents. Stannous chloride and nickel chloride are the source of metal ions. Most platers use nickel anodes and continually add stannous chloride to maintain the tin content, but both nickel and tin anodes can be employed

TABLE 7.12 Specifications for a Tin-Nickel Plating Bath^{19,20}

Component	g/L	oz/gal
Stannous chloride, SnCl ₂	48.8	6.5
Tin (Stannous tin), Sn	26-38	3.5-5.0
Nickel chloride, NiCl ₂	300	40.0
Nickel, Ni	60-83	8.0-11.0
Ammonium bifluoride, NH ₄ HF ₂	41.0	7.5
Total fluorine*	34-45	4.5-6.0
Operating conditions		
Temperature, °F	150-160	
Cathode current density, A/ft ²	10-30	
Cathode current efficiency, %	100	
Anode-to-cathode ratio	1:1 to 2:1	
Agitation	Preferably none	
pH	1.5-2.0	

*Total fluorine should always exceed the total tin concentration; i.e., the sum of the stannic tin plus sum of the stannous tin.

simultaneously. Fluorides are added as ammonium bifluoride or hydrofluoric acid to complex the tin and keep it in solution. As the bath ages, stannic tin will increase, and this will necessitate higher fluoride concentration.

2 Bath maintenance.

a. Operating temperature. This is critical and should be maintained within the limits specified.

b. pH. pH is raised by the addition of ammonium hydroxide and lowered by the addition of hydrofluoric acid. The pH will tend to rise as the bath is operated. It is checked colorimetrically.

c. Anode bags. Preferable materials for anode bags are nylon, polypropylene, and Dynel.

d. Purification. Baths should be carbon-treated when there is evidence of organic contamination. Frequent dummying of the bath at low current density (2 to 5 A/ft²) is necessary since the deposit is very sensitive to metallic impurities. Do not use hydrogen peroxide.

e. Analysis. To maintain the metals in proper balance, periodic analysis of the bath, in accordance with the procedures outlined in the process control section, is necessary.

f. Filtration. To maintain solution clarity and cleanliness, filtration with nonsilicated filter aids should be performed when the bath is not operating. There are normally no organic additives, and so it is practical to filter the bath, when not in use, through activated carbon to minimize any organic contamination buildup.

3 Troubleshooting the tin-nickel bath.

a. Dark deposits. Dark deposits, especially in the area of rack contacts, are indicative of metallic contamination. Since the bath will solubilize most metals, it is essential to remove slanks that have fallen off racks from the bottom of the tank. Another symptom of metallic contamination is excessive cracking in the wave-soldering operation. The bath must be frequently dummyed to keep metallic contamination in check.

b. Cloudy deposits. When the fluoride content is exceeded by the total tin content, the deposits will become cloudy. In addition to normal dummying, the fluoride content is lowered by the evolution of hydrogen fluoride fumes. It is essential that the fluoride content be maintained by the addition of either hydrofluoric acid or ammonium bifluoride. An excess fluoride content is not detrimental.

c. Burned deposits. Since tin is not replenished through anodes and must be maintained through the addition of stannous chloride, it is the most likely constituent to drop rapidly in concentration. The drop is characterized by "burning," especially in high-current-density areas.

d. Poor solderability. As indicated previously, tin-nickel baths can be difficult to solder, especially if aged. A cleaning operation including a 50 percent by volume hydrochloric acid pickle followed by thorough rinsing prior to loading should overcome any serious soldering problems.

4 Equipment. Materials of construction for tank linings, filters, and other equipment are given in Table 7.21.

5 Plating rate. The plating rate for tin-nickel is given in Table 7.13.

TIN ELECTROPLATING^{25,26}

25. Function and Properties Tin-plating has been employed in the electronics industry in the past primarily to promote solderability. Until recently its use on printed boards was

TABLE 7.13 Plating Rate for Tin-Nickel Electroplate Based upon 100 Percent Cathode Current Efficiency

Current density A/ft ²	Plating time, min, for varying plate thickness, in				
	0.0001	0.0002	0.0003	0.0005	0.001
10	7.5	15	22.5	37.5	75
15	5.0	10	15	25	50
20	3.8	7.5	11.3	18.8	37.5
25	3.0	6	9	15	30
30	2.5	5	7.5	12.5	25

not very extensive and was confined mostly to the acidic formulations. The development of bright acid tin and the increase in the cost of precious metal plating has stimulated considerable interest in tin as a metal etch resist and for solderability and electrical contact. The bright acid tin deposit does not require a reflowing operation. Tin forms a thin oxide at the surface that meets dual needs: corrosion resistance and maintaining solderability. The deposit is readily soldered in accordance with the requirements called out in MIL-F-14256. MIL-STD-275B states that electrodeposited tin shall be a minimum of 0.0003 in thick and shall be in accordance with MIL-T-10727. For composition and operating conditions of a bright acid tin-plating bath see Table 7.14.

TABLE 7.14 Specifications for a Bright Acid Tin Plating Bath

Component	g/L	oz/gal
Stannous sulfate, SnSO ₄	30	4
Tin metal, Sn	13.5-16.5	1.8-2.2
C.P. sulfuric acid, H ₂ SO ₄	184	24.5
Brighteners	Per vendor instructions	
Operating conditions		
Temperature, °F	55-75	
Cathode current density, A/ft ²	0.5-40	
Anode current density, A/ft ²	Never to exceed 25	
Anodes	Pure tin	
Filtration	Continuous filtration preferred	
Agitation	Cathode bar or solution circulation	

In addition to the foregoing there has been some effort to employ tin as a final edge connector deposit and further eliminate the need for precious-metal plating. There have been reports of problems between the tin-plated edge connectors and gold-plated inserts in the form of battery activity. Other reports have indicated successful results with tin-plated edge connectors and tin-plated inserts.

1 Function of bath constituents. Stannous sulfate is the source of tin metal ions. Sulfuric acid is used both to solubilize the tin salt and for bath conductivity. The proprietary additives function both as levelers and brighteners. They are replenished mostly on an ampero-hour basis. The brightener systems available consist of two parts, one for bath makeup and one for normal replenishment.

2 Bath maintenance.

a. Temperature. Adequate control of the bath operating temperature is very critical. As the temperature decreases, the brightness and throwing power increases. In most instances it will be necessary to cool the bath to maintain it within the prescribed parameters.

b. Anodes. Anodes should be removed if the bath is to be idle for an extended period of time. Anode bags are not necessary. Periodic brushing of anodes should be performed. Sufficient anode area must be maintained to prevent anode passivation.

c. Filtration. Constant filtration is recommended. It can also function as mild agitation while the bath is in operation.

d. Analysis. The bath should be analyzed periodically in accordance with the procedures outlined in the process control section. In conjunction with the analysis, Hull cell techniques are employed to determine brightener concentration and to pinpoint potential problems.

3 Troubleshooting bright acid tin.

a. Brittle deposits. Brittleness is caused by excessive current or metallic contamination. Metallic contamination can be removed by low-current-density dummying. However, tin will also be deposited in the process. The bath should be analyzed when dummying is completed.

b. Poor throwing power. Poor throwing power is generally caused by metal content or low acid.

c. Dullness in low-current-density areas. Dullness such as may be encountered on a large ground plane can be attributed to several factors. Low brightener, passive anodes, high temperature, and chloride contamination can cause this type of dullness. To avoid the chance of chloride drag-in, a sulfuric acid pickle should precede the tin bath.

d. **Pitting.** Pitting will occur in high-current-density areas with excessive current density or inadequate agitation.

e. **Black streaks.** Excessive brightener additions will cause black streaking.

4 **Equipment.** Materials of construction for tank linings, filters, and other equipment are given in Table 7.21.

5 **Plating rate.** The plating rate for bright acid tin is given in Table 7.15.

TABLE 7.15 Plating Rate for Bright Acid Tin Electroplate Based upon 100 Percent Cathode Current Efficiency

Current density, A/ft ²	Plating time, min, for varying plate thickness, in				
	0.0001	0.0002	0.0003	0.0005	0.001
10	4.7	9.4	14.1	23.5	46.9
20	2.4	4.7	7.1	11.8	23.5
30	1.6	3.1	4.7	7.8	15.6

GOLD ELECTROPLATING

26. Function and Properties Gold-plating on printed boards has several functions. As a metal etch resist it will resist all the common etchants. Its electrical conductivity is very high; it has resistivity of 2.44 $\mu\Omega\cdot\text{cm}$. Its negative oxidation potential makes it ideal for tarnish resistance and for contact surfaces with low electrical resistance. The subject of gold as a solderable substrate has been one of controversy for a number of years. Suffice it to say gold has been successfully used under controlled conditions as a soldering aid.²⁷

Many new gold processes, most of them proprietary, have been developed in recent years. They represent an effort to circumvent the classic alkaline cyanide gold and its deleterious effect on plating resists. The marked increase in gold costs has prompted the development of gold alloy baths as a cost-cutting means.²⁸⁻³⁰

27. Acid Gold Acid gold baths operate at a pH of the order of 3.5 to 4.5. The system utilizes potassium gold cyanide in a mild organic acid electrolyte. Stable complexes of cobalt, nickel, and indium can be incorporated into the formulation to increase hardness and wear resistance. To a large extent acid golds are used to fill the requirements of MIL-STD-275B, which states "gold plating shall be a minimum of 0.000080 in thick and shall be in accordance with type II of Specification MIL-G-45204." Type II is 23+ carat (hard) plate.

The acid golds will usually exhibit poor cathode current efficiency, and this must be taken into consideration when calculating plating times. Table 7.16 indicates one of the available acid gold formulations.

TABLE 7.16 Specifications for a Typical Acid Gold Bath

Component	g/L	Troy oz/gal
Gold metal as potassium gold cyanide	4-16	0.5-2
Conductivity salts	As necessary for bath conductivity; Maintain °Bamé within parameters.	
pH-adjusting salts	As necessary to maintain pH	
Operating conditions		
Temperature, °F	70-120	
pH	3.5-4.0	
Agitation	Vigorous	
Cathode current density, A/ft ²	10	
Anode-to-cathode ratio	4:1	
Anodes	Platinized titanium	

28. Alkaline Noncyanide Gold Another type of bath currently being used successfully on printed boards is the mild alkaline noncyanide type. It has a high cathode current efficiency and excellent throwing power. This type of bath is ideal for heavy gold deposits, since very little hydrogen gas is evolved at the cathode to scrub off any plating resist. Trace quantities of other metals may be added as complexes to improve hardness

and wear resistance. Table 7.17 indicates one of the available alkaline noncyanide gold formulations.

29. Gold Strikes There are nearly as many gold strikes available as there are gold baths; each is formulated to be compatible with the gold-plating process that will follow it. All strike baths have a low-metal and a high-complexing-agent content. Since gold is a noble metal and will readily displace metals less noble than itself, it is essential that a thin layer of gold be electrodeposited prior to any immersion gold deposit. Otherwise, bonding problems will result. The high complexing agent present in the gold strike allows the electrodeposition of gold while it prevents an immersion gold deposit. The strike bath is a relatively inefficient bath, and hydrogen at the cathode is an aid in maintaining an oxide-free metallic substrate on which to electroplate. The gold deposit from a strike bath is generally only several millionths of an inch thick.

In addition to the foregoing, a strike solution can function as a sacrificial bath. The dragging of both inorganic and organic contaminants into a gold bath can necessitate a costly carbon treatment or even replacement of the bath. Since the metal concentration of a strike bath is low, replacement is not so expensive and the unused gold can be reclaimed.

TABLE 7.17 Specifications for a Typical Alkaline Noncyanide Gold Bath

Component	g/L	Troy oz/gal
Metallic gold content	6-16	0.7-2.0
Conductivity salts	As necessary to maintain bath conductivity. Maintain °Bamé within given parameters.	
Reagent grade sodium hydroxide	To raise pH	
Reagent grade sulfuric acid	To lower pH	
Operating conditions		
Temperature, °F	90-130	
pH	9.0-10.0	
Agitation	Vigorous	
Anodes	Platinized titanium	
Anode-to-cathode ratio	2:1 minimum	
Cathode current density, A/ft ²	3-10 (lower value preferred)	

The use of a gold strike with the acid gold formulations containing gold as a cyanide salt is not as critical as the use of a strike with a noncyanide formulation. Some acid golds readily function as a strike solution when their metal content is lowered. The alkaline noncyanide golds described previously are typical of baths requiring a strike.

Gold strike baths have been used to flash several millionths of gold over gold alloy deposits to ensure low contact resistance on the surface.

30. Function of Bath Constituents Because insoluble anodes are used in all the gold formulations, the addition of gold salts governs the metal concentration in the bath. Most vendors supply gold or gold alloys in the form of preweighed salts or liquid concentrates that contain a known amount of gold. In some baths pH-adjusting salts are used to raise and lower pH; in other formulations a known acid and base are used. Conductivity salts make the use of lower voltages possible. They are added as necessary to maintain the specific gravity of the bath within prescribed parameters.

31. Bath Maintenance
1 Anodes. Insoluble anodes, such as platinized titanium or tantalum, have a finite life. They must be replaced periodically when the platinum surface is gone. Care must be taken when any insoluble films are removed from the anodes not to use any abrasives that will remove the platinum.

2 Agitation and filtration. Vigorous agitation is recommended under both the anode and cathode areas. It can be accomplished readily by using a sparger on the effluent from the filter. Constant filtration is recommended to maintain bath cleanliness. Filter cartridges that will adsorb organic contaminants without removing any gold are available for many of the gold formulations.

3 Analysis. Metal content is the most critical parameter of the bath. In addition to gold analysis, the ampere-minutes or -hours put through a given solution can be an aid in

estimating the metal content of the bath. The rate of metal consumption will depend largely on a bath's cathode current efficiency. This can vary greatly from one formulation to the next. The cathode current efficiency of a given gold formulation must be established before any amper-hour system for monitoring gold content can be incorporated. It is still essential that the bath be analyzed periodically for gold in accordance with the procedures outlined in the process control section.

4 Metal recovery. In addition to board trimmings and scrap boards, solution drag-out is a source of salvageable gold. Resins that are specific for gold are marketed by several vendors. The resin is placed in a filter chamber. Rinsewater from stagnat rinses following gold plating is recirculated through the filter chamber via a pump. When saturated with gold, the resin is removed and forwarded for reclamation. A new charge of resin is then put in the filter chamber, and the process is repeated.

32. Troubleshooting Gold Baths Many problems associated with gold-plating can be avoided if the baths are kept within their recommended parameters by an active maintenance program. Since many gold formulations are in use, the following observations may not necessarily apply to specific baths.

1 Low deposition rate, excessive hydrogen gassing. These are characteristic of a low metal content, especially in high-efficiency baths. They may be accompanied by some resist breakdown.

2 Gold peeling from nickel. This can be a function of the nickel formulation employed. Many of the bright Watts baths have brighteners codeposited with the nickel. That type of deposit is very difficult to activate. A low-stress nickel is recommended as a base for gold.

The absence of a gold strike can also be responsible for peelers. This is especially true when noncyanide golds are involved.

3 Low current efficiency. Here the indication is low metal, low pH, or low conductivity, all of which are easily corrected with the proper additions.

4 Resist breakdown. Low-efficiency baths will readily cause resist breakdown even when they are within the recommended operating parameters. Resist breakdown is even more prevalent when an effort is made to build heavy deposits (0.000025 in) in low-efficiency baths. For heavier deposits the high-efficiency formulation should be employed.

33. Equipment Materials of construction for tank linings, filters, and other equipment are given in Table 7.21.

34. Plating Rate In view of the wide variety of proprietary and nonproprietary gold baths available, a usable plating table depicting all the baths is not practical. Each formulation will have pretty much its own efficiency. The cathode current efficiency can vary from 25 to nearly 100 percent. That must be taken into consideration when a gold process is selected for a given application.

35. Gold Solderability^{29,27,21,22} As indicated previously, the subject of soldering on gold plate has evoked considerable controversy. Most vendor literature describes gold as a highly solderable substrate capable of being soldered by fluxes that are in accordance with MIL-F-14256. Several investigations have revealed that weaker gold-solder joints (up to 20 percent) can result from dissolution of gold in the solder.

The National Aeronautics and Space Administration (NASA), in its hand-soldering requirements, specifies the removal of gold from solder areas, including plated-through holes and terminal pads, to achieve the highest joint strengths and greatest reliability.

The use of a gold deposit having any degree of porosity can result in oxidation of the base metal. Subsequent soldering will remove the gold plate; and since the base metal has not been fluxed and has not received any other type of deoxidizing action, the result will be solder dewetting.

Although studies have shown the foregoing to be true, gold has some uncontested properties. On switch contact areas and tabs its contact resistance is excellent. Its inherent corrosion resistance makes extended board storage prior to assembly feasible. Many fabricators employ selective plating techniques to take advantage of the low contact resistance on edge connectors but avoid potential solder joint contamination with gold

and circumvent the high cost of gold plating. Although selective plating is readily accomplished, the labor costs incurred in the additional processing necessitated by it can exceed any possible savings. Consideration must be given to circuit areas, masking operations, metal stripping, replating, and any additional image-transfer processing brought about by selective plating.³³

Other investigations have shown that gold can indeed cause soldering problems if it is used under adverse conditions; but if it is used properly, it can decidedly be a soldering aid. The following considerations apply to soldering to gold-plated substrates:

Solder (60 Sn, 40 Pb) under normal operating conditions will scavenge the gold deposit from its base metal, so that the solder bond will be to the base metal. If the gold plate is a dense pour-free deposit, the solder joint will be satisfactory.

Thin gold deposits (10 μ in) will not adversely affect the strength of a solder joint.

Thick gold deposits (50 μ in) will contribute a relatively large quantity of gold to the solder and will readily form the tin-gold intermetallic (AuSn) and thus weaken the solder joint. The formation will be characterized by the appearance of a "sugar" or cold joint.

Thin gold deposits will wet more readily than thick ones.

Alloy golds that are dissolved in solder are likely to cause weakened solder joints.

RHODIUM ELECTROPLATING

36. Function and Properties Rhodium is one of the group of platinum metals. The deposit obtained from a rhodium bath is hard (900-1000 Knoop), highly reflective, extremely corrosion-resistant, and highly conductive (resistivity is 4.51 $\mu\Omega\cdot$ cm). Its use on printed boards is confined to edge connectors and other critical contact areas. Rhodium plate is employed where a low-resistance, long-wearing, oxide-free contact is required; also, it minimizes noise level for moving contacts. That rhodium is corrosion-resistant is illustrated by the fact that it is not attacked even by aqua regia.

MIL-STD-275 B specifies the minimum rhodium thickness as 20 μ in and the maximum as 50 μ in with a barrier layer of low-stress nickel having a minimum thickness of 500 μ in. When nonmilitary boards are involved, a rhodium thickness of 5 μ in has proved adequate. Rhodium plating must not be used where soldering to the conductive pattern is required.

Rhodium is electroplated from both a phosphoric acid and a sulfuric acid bath, but the latter formulation is predominant. Deposits are usually highly stressed, but additives in the proprietary formulation reduce tensile stress and even give a compressively stressed deposit. Makeup and composition of a rhodium sulfate bath are given in Table 7.18.

TABLE 7.18 Specifications of a Rhodium Sulfate Bath

Metallic rhodium	4-10 g/l
Sulfuric acid	25-35 mL/L
Temperature	110-130°F
Agitation	Moderate
Anodes	Platinized titanium
Anode-to-cathode ratio	2:1
Cathode current density	10-30 A/ft ²
Plating rate	At 20 A/ft ² , 10 μ in will deposit in 1.4 min based on 70% cathode current efficiency.

Contaminants. Metallic contaminants will cause dark, streaky deposits. It is essential that adequate rinsing be employed to prevent metallic contamination. To minimize copper contamination, nickel-plated bus bars should be employed. Organic contamination will cause stress and flaking problems. The problem can be overcome with frequent carbon treatments.

ELECTROLESS AND IMMERSION PLATING

37. Function and Properties In addition to the extensive treatment given electroless copper, some mention must be made of other chemical plating processes even though

they are only employed occasionally and to a much lesser degree in printed boards. A distinction must be made between the terms "electroless" and "immersion" plating. As indicated in the electroless copper section, all electroless plating baths contain a reducing agent as a source of electrons to reduce metal ions to the metallic state. On nonmetallic substrates noble metals are used to initiate the electroless reduction. Once begun, the reaction is autocatalytic and theoretically could proceed indefinitely if the constituents were replenished and the by-products removed. On metallic substrates normal preparatory electroplating procedures that will give an active metal surface are adequate for initiating electroless deposition on metals that are less noble than the metal that is being reduced. On metallic substrates that are more noble than the metal being reduced, extraordinary methods of initiating the electroless deposition, such as an immersion deposit of a noble metal on the substrate, must be employed. In both cases the electroless deposit can be used to build heavy deposits and has been used to some extent on printed boards for full additive circuits and for board repair.

Immersion deposits are accomplished via a much more simple mechanism. Briefly stated, it is the displacement of one metal by a metal ion with a lower oxidation potential than the metal displaced in an aqueous medium. The metal displaced is oxidized and solubilized, and the metal ion will gain an electron or electrons from the oxidized metal and be reduced to the metallic state. Immersion plating is strictly a surface phenomenon; and when the displacement on a given substrate is complete, all plating ceases. The deposits from immersion baths are relatively thin and are limited to the thickness of the deposit that can be achieved. They are used on printed boards primarily to inhibit corrosion and promote solderability.

38. Electroless Nickel Electroless nickel is applied to printed boards when it is not practical to use conventional electroplating practices. A typical application is selective plating where there is no electrical continuity for electroplating. The deposit obtained is 85 to 94 percent nickel; the balance is phosphorus. The appearance of the deposit will generally mirror the substrate upon which the deposit is plated. Many formulations, both proprietary and nonproprietary, are available. The plating rates vary from 0.0003 to 0.001 in/h.

Metals that are more noble than nickel must be catalyzed to initiate deposition. The following solution can be used for metal activation:

Palladium chloride, PdCl₂, 1 g
Hydrochloric acid, HCl, 10 mL

Dilute to 1 gal with water when the PdCl₂ has been dissolved in HCl. A 15 to 20-s dip is generally sufficient for activation. It is followed with a rinse. Table 7.19 illustrates the composition of an acid electroless nickel bath. Current evaluations of electroless nickel formulations are being conducted; amine boranes are employed as reducing agents to produce nickel-boron deposits. The baths are run at low temperatures. A low contact resistance is claimed for the nickel-boron deposit.

TABLE 7.19 Electroless Nickel Bath*

Nickel chloride	30 g/L
Sodium glycolate	30 g/L
Sodium hypophosphite	10 g/L
pH	4-6
Temperature, °F	190
Deposition rate	0.0003-0.0005 in/h

*U.S. Patent 2,532,283

39. Immersion and Electroless Gold There are many proprietary immersion and electroless gold solutions. They are used predominantly in areas where electroplating is not practical because of the lack of electrical continuity. A typical application is on printed board edge connectors. The gold is usually deposited on a copper or nickel substrate. Normally, electroplating pre-treatments are sufficient to initiate the gold deposit.

Care must be exercised in selecting an immersion or electroless gold, because many vendors indiscriminately interchange the names "electroless" and "immersion." Immersion gold deposits are generally limited to a maximum thickness of 10 μ m. The true test of an electroless gold bath is to determine if gold buildup can be accomplished on a gold substrate.

Table 7.20 illustrates the composition of an immersion gold bath for copper. Immersion gold baths for nickel are generally acidic.

TABLE 7.20 Immersion Gold Bath

67% Potassium gold cyanide	1/2 oz/gal
Sodium cyanide	4 oz/gal
Soda ash	5 oz/gal
Temperature	150-180°F

40. Immersion Tin Immersion tin is used over both copper and tin-lead plating. On copper it is used as a temporary aid for solderability. It is temporary because the deposit is generally 50 to 100 μ m thick and will oxidize on standing. With a tin-lead plated surface, immersion tin is employed prior to etching to give a tin-rich surface to the tin-lead plate to prevent the oxidation of the lead during processing in ammoniacal etchants. Most of the immersion tin baths available are proprietary.

PLATING EQUIPMENT

41. Selection Prior to the installation of any plating process, consideration must be given to the equipment required to operate the process successfully. Nearly all chemicals are corrosive to some degree, and the appropriate equipment for tanks, heaters, filters, etc. must be so selected as to resist any adverse effects. To that end vendor experience should be utilized to the fullest. Table 7.21 may be used as a guide for material selection.

TABLE 7.21 Materials of Construction for Printed Circuit Plating Processes

Bath	Tank lining	Heaters	Filters	Filter aids
Copper pyrophosphate	PVC, rubber, polypropylene	Stainless steel	Stainless steel, epoxy	Cellulose fiber, asbestos fiber
Acid copper sulfate	PVC, rubber, polypropylene	Carbon	Hard rubber	Nonsilicated filter aid
Acid gold	PVC, Pyrex, Teflon, polypropylene	Quartz	PVC-lined	Asbestos, diatomaceous earth
Alkaline gold	PVC, Pyrex, Teflon, polypropylene	Titanium	PVC-lined	Asbestos fiber
Tin-lead	PVC, rubber	Not heated	PVC- or rubber-lined	Nonsilicated filter aid
Tin-nickel	PVC, polypropylene	Carbon	Teflon-lined	Nonsilicated filter
Modified Watts nickel and nickel sulfamate	PVC, polypropylene	Quartz, carbon	PVC- or rubber-lined	Asbestos fiber, diatomaceous earth
Rhodium sulfate	PVC, polypropylene, Pyrex	Quartz	PVC- or Teflon-lined	Asbestos fiber, diatomaceous earth
Bright acid tin	PVC, polypropylene, rubber	Not heated	316 stainless steel, hard rubber	Asbestos fiber

ADDITIVE CIRCUITS³⁴⁻³⁶

The fabrication of additive printed circuits consists of depositing conductor patterns on a substrate such as G-10 or FR-4 by employing electroless plating alone or in conjunction with electroplating. The need for an insulator with a copper-clad surface is eliminated. The concept of additive circuits is by no means new to printed board fabrication. The problem has been that, although many techniques of fabrication have been successful, they have not been conducive to high-volume production and have not been able to compete successfully with the cost of subtractive circuitry. The introduction of proprietary processes indicates that additive circuits may compete both in volume and cost with subtractive circuits in some applications.

The key to successful additive circuit fabrication is the series of conditioning steps employed to produce a satisfactory bond between the circuitry and insulator. Two methods of conditioning the surface of the insulator are available. Following conditioning, full additive (all electroless) plating or semiadditive (electroless and electrolytic) plating may be used to form the conductor patterns.

42. Swell and Etch "Swell and etch" refers to the technique employed for conditioning the surface of the insulator; it is very similar to the procedures followed for plating on plastics. Swelling of the substrate is accomplished by the use of a solvent system. It is followed by etching in a strong oxidizing medium consisting of a mixture of sulfuric and chromic acids. The final result will be a surface with a very fine etch that will serve for mechanical keying of the subsequent electroless deposit and produce the required bonding properties.

When G-10 or FR-4 is processed, special laminate specifically manufactured for additive circuit processing must be employed. The special laminate has an epoxy-rich surface upon which the actual etching process occurs. Conventional unclad laminate is unsuitable because etching would penetrate the fiberglass and the result would be unacceptable bond degradation.

43. Preconditioned Laminate In an effort to circumvent the problems in handling solvents, strong oxidizing mixtures, and waste disposal, a laminate with a preconditioned surface has been developed. The laminate is produced by pressing aluminum foil that has been anodized on one side into the substrate in the laminating or pressing cycle. The resultant surface of the substrate when the foil is etched away is a topography that mirrors the anodized aluminum foil. The surface simulates a micro-etched surface that will promote a strong mechanical bond with the electroless copper.

44. Full Additive vs. Semiadditive Following the conditioning step the fabricator may elect to use either a full additive cycle (all electroless plating) or a semiadditive cycle (electroless plating plus electrolytic plating) for the formation of conductor patterns. Following the conditioning step, activation of the substrate for subsequent electroless deposition is accomplished by using the same mechanism (colloidal palladium followed by a tin solubilizer) that is employed with conventional through-hole plating. The two cycles diverge following activation; the full additive cycle proceeds to an image-transfer process and the semiadditive cycle to an electroless copper step. A minimum deposit of 100 μin is obtained, and the boards move on to image transfer. Following image transfer, full additive cycle boards are placed in a high-speed electroless copper bath for 8 to 10 h to obtain a copper thickness of roughly 0.0015 in. Following image transfer, conventional pattern electroplating is employed for semiadditive boards. The original 100 μin of electroless copper is readily etched away by a quick dip in an etch such as ammonium persulfate. The following steps indicate the process sequence of full additive and semiadditive fabrication.

1. Blank and drill.
2. Condition surface. (a) Swell and etch in oxidizing mixture, or (b) etch away aluminum in 25 percent hydrochloric acid.
3. Activate in colloidal palladium.
4. Post-activate (solubilize tin from the activator.)

Full Additive	Semiadditive
5a. Dry.	5b. Electroless copper plate (100 μin).
6a. Image transfer.	6b. Dry.
7a. High-speed electroless copper 8-10 h (0.0015 in).	7b. Image transfer.
8a. Strip resist.	8b. Copper electroplate (0.0015 in) 1 h.
9a. Immersion gold or tin (optional).	9b. Electroplate solder, tin, or nickel-gold.
10a. Solder resist (optional).	10b. Strip resist.
	11b. Etch away electroless copper.
	12b. Solder resist (optional).

45. Pros and Cons of Additive Circuit Fabrication The additive circuit has generated its share of controversy. The pros may be summarized as follows.

1. The problem of etchant disposal is greatly reduced.
2. The cost of etchant is greatly reduced.
3. Unrelated problems are eliminated.
4. Dry photo polymers may be employed with the semiadditive process to produce very fine traces and spaces.
5. Copper-to-copper bond problems are eliminated.
6. Rejects other than those due to drilling or blanking can be stripped and reprocessed.

The cons are as follows:

1. The swell-and-etch process exchanges the problems of etchant disposal for the problem of disposing of hexavalent chromium and the solvent.
2. Etching the aluminum-clad material is a hazardous operation. The reaction is strongly exothermic and gives off toxic, corrosive fumes, especially when a large volume of material is involved.
3. The bond strengths obtained with aluminum-clad material are erratic.
4. The epoxy-rich surfaces vary from one laminate vendor to the next, so that the swell and etch process may be satisfactory for one vendor's laminate but cause severe degradation with another vendor's material.
5. The cost of an epoxy-rich surface or aluminum-clad material is more than that of conventional copper-clad laminate.
6. The use of the full additive system (all-electroless plating) is time-consuming, requires sophisticated equipment for electroless bath control, and involves the use of expensive electroless plating solutions. Many platers claim "electricity is the world's cheapest reducing agent."
7. The introduction of thin copper-clad materials ($\frac{1}{2}$, $\frac{1}{4}$, $\frac{1}{8}$ oz) would tend to offset many of the advantages of additive circuits.

CHEMICAL SAFETY IN THE PLATING SHOP

46. Function The purpose of this section is to call attention to the potential hazards and safety requirements of the chemicals that are normally encountered in printed board plating. To operate any plating facility safely, all pertinent individuals should receive training and instructions regarding the application, properties, and storage of the chemicals. The abuse of the basic tenets of chemical safety can result in the loss of life and property.

47. Identification of Chemical Properties by Label The predominant label that is readily visible on certain types of chemicals is the Interstate Commerce Commission label; chemicals having certain properties are required by law to have it. The label is diamond-shaped and color-coded as follows:

1 **Red diamond.** All flammable liquids having a flash point of 90°F or less. Flash point is defined as the temperature at which an open flame or spark will ignite the vapor from a given liquid.

2 **White diamond.** All materials that are corrosive must have a white diamond label. Most acids and strong alkalis fall into this category.

3 **Yellow diamond.** This label indicates oxidizing agents; they are defined as being sources of oxygen on decomposition. This statement does not necessarily match the true chemical definition. Many oxidizing agents are unstable and will decompose with explosive violence.

4 **Green diamond.** Compressed gases.

Materials with dissimilarly colored diamond labels should not be allowed to come in contact and should be stored accordingly. Red- and yellow-label materials are particularly reactive when in contact and will invariably produce a fire or explosion.

48. Solvents Solvents are used to a large extent in plating shops for cleaning and stripping plating and etch resists.

1 **Ketones.** Ketones are present in many thinners and proprietary strippers; acetone and methyl ethyl ketone are the common ones. They are red-diamond-label materials and are regarded as skin irritants.

2 Alcohols. Short-chain alcohols are frequently used in strippers and thinners for drying and as flux carriers. The most frequently used alcohols are methyl, denatured ethyl, and isopropyl. They are red-label materials and will dry out skin on repeated contact. Methyl alcohol vapor is also very toxic.

3 Chlorinated solvents. Chlorinated solvents are used in vapor degreasers and resist strippers. The inhalation of chlorinated solvent vapors in excessive amounts can cause narcotic-like effects and kidney and liver damage. There have been fatalities due to suffocation when large degreasers were cleaned without adequate safety procedures or equipment. Chlorinated solvents will, on contact, remove oil from skin, and continued exposure can result in dermatitis. The principal chlorinated solvents in use are trichloroethylene, 1,1,1-trichloroethane, and methylene chloride; they have threshold limit values of 100, 350, and 500 ppm, respectively.

49. Acids Acids are universal in all plating operations and cause most of the injuries to individuals. Most of them are white-label materials, but some exceptions are classified as yellow label. Acids normally encountered in plating operations are very soluble in water. Contact with an acid should be followed by washing with copious quantities of water. The addition of an acid to water is exothermic in varying degrees, depending upon the acid. It should always be remembered that, to dissipate the heat generated, *acid should always be added to water and never vice versa*; otherwise, an eruption might occur.

1 Sulfuric acid. Concentrated sulfuric acid is a severe dehydrating agent that will char skin or flesh. It is very strongly exothermic when added to water. Sulfuric acid will react violently with many organic materials and cause either an explosion or a fire.

2 Hydrofluoric acid. Hydrofluoric acid is used in tin-nickel formulations. It can cause extremely severe burns. Anyone who is working with it should wash exposed areas frequently. Hydrofluoric acid attacks glass and must be stored accordingly. Its fumes are very pungent and corrosive.

3 Fluoroboric acid. Fluoroboric acid is used in deoxidizing solutions and tin-lead and tin-nickel plating solutions. Its properties are similar to those of hydrofluoric acid, including the corrosive effect on glass.

4 Nitric acid. Nitric acid is used for stripping rack tips and cleaning metallic deposits from tanks. Some proprietary chemical polishing solutions contain quantities of nitric acid. The acid will attack many metals and evolve nitrous oxide (a brown gas), which, in quantities of 200 to 700 ppm, may be lethal. Adequate ventilation must be employed when nitric acid strippers are used.

5 Hydrochloric acid. Hydrochloric acid is used extensively for deoxidizing and as a medium for electroless catalyzing solutions. Hydrogen chloride fumes are very pungent and toxic, and repeated exposure to them can cause ulcers in the respiratory tract. Most ferrous alloys are attacked by hydrochloric acid and will evolve explosive hydrogen gas.

50. Oxidizing Agents All materials classified as strong oxidizing agents are potentially very dangerous. They react violently with reducing agents and red-diamond-label materials. In some cases they will support their own combustion or decomposition once the reaction has been initiated.

1 Chromic acid and chromates. Chromic acid is used in some copper etchants. Ammonium dichromate is used for sensitizing emulsions in image-transfer processing. Physical contact with chromates in concentrated quantities will cause tissue destruction, and extensive contact with dilute quantities will result in dermatitis. All chromates will ignite flammable solvents. In many instances chromate contact with paper or wood will result in a fire.

2 Nitrates and nitrites. Nitrates and nitrites are present as proprietaries in some plating bath additives. They will ignite organic materials, including wood and paper. Ammonium nitrate is especially dangerous, since it can decompose with explosive violence.

3 Hydrogen peroxide. Hydrogen peroxide is employed for purifying plating solutions. It is also used as an oxidizer in some metal-stripping formulations. Physical contact with high-strength solutions (30 percent) will cause rapid tissue destruction. The peroxide will react with flammable vapors or liquids to cause a fire or an explosion.

4 Oxygenated chlorides. These are present in many of the alkaline etchants as sodium chlorite. In addition to the danger of igniting organic materials, there is the

possibility of splashing solution onto clothing because of the large volume of etchant employed. Subsequent drying of the solution on clothing will deposit salts that are readily ignitable by a spark and will decompose with explosive violence.

51. Cyanides In the early 1970s there was a decline in the use of cyanides in printed board plating. Their use is confined mostly to gold plating, as potassium gold cyanide, and to some proprietary gold strippers in which free cyanides are present. Some of the newer alloy gold formulations are free-cyanide-based.

Cyanides are well known for their toxicity. Most plating accidents involving cyanides have been caused by the inadvertent addition of acid to cyanide or vice versa to produce hydrogen cyanide. *Remember: Never add acid to a solution containing free cyanides!* Exposure to hydrogen cyanide can cause death within a few minutes. In addition, cyanides are strong reducing agents and will react violently with oxidizing agents.

52. Caustic Compounds Caustic compounds (sodium hydroxide and potassium hydroxide) are present in many alkaline cleaners. They are classified as being white-diamond-label materials. In addition, there are other alkaline compounds (the aqueous solutions of which have a pH of 10 or over) that can cause burns or tissue destruction. They include silicates and phosphates. The stronger caustic compounds have an exothermic reaction when added to water. To ensure complete dissolution of salts and heat dissipation, caustic compounds should be added to water with vigorous stirring.

Contact with caustic compounds should be treated with thorough rinsing with water followed by rinsing with 3 percent acetic or lactic acid solution followed by more water rinsing.

PROCESS CONTROL

This section outlines procedures for analyzing the major plating solutions. Most vendors of plating solutions using proprietary additives will supply procedures that they feel are most compatible with their processes. Frequently Hull cell or similar plating cell procedures will be given by vendors, along with diagrams outlining a particular problem. Before any plating cell procedures are followed, an analysis of the bath should be made to insure that all bath constituents are in their proper concentrations. In many instances there are alternate methods of standard solution preparation, and analytical procedures are available in references^{10,20,27,36} other than those cited in this section.

53. Copper Pyrophosphate Solution Analysis

a. Determination of Copper. The following reagents are required:

1. 0.2 N EDTA: disodium salt (37.23 g/l.)
2. Ammonium hydroxide (NH₄OH) 29 percent
3. Murexide indicator. Mix 0.2 g with 100 g of sodium chloride. Store this mixture dry. Use 0.2 to 0.4 g of the mixture for each titration.
 1. Pipette a 5-mL sample into a 500-mL flask
 2. Add 10 mL of concentrated ammonium hydroxide.
 3. Add 300 mL of distilled water
 4. Add 0.2 to 0.4 g of murexide indicator. This will impart a yellow-brown color to the solution.
 5. Titrate with 0.2 N EDTA until the end point (color change from yellow-brown to blue-violet) is reached.

Calculation:

$$\begin{aligned} 1 \text{ mL } 0.2 \text{ N EDTA} &= 0.006357 \text{ g Cu} \\ \text{mL } 0.2 \text{ N EDTA} \times 1.271 &= \text{g/L Cu} \\ \text{mL } 0.2 \text{ N EDTA} \times 0.17 &= \text{oz/gal} \end{aligned}$$

b. Determination of Pyrophosphate. The following reagents are required:

1. 30 percent solution of sodium hydroxide
2. 0.5 N sodium hydroxide (20.0 g/L)
3. Bromophenol blue indicator. To prepare the indicator, weigh out 1 g and mix with 15.0 mL of 0.1 N NaOH to form a slurry. Dilute the mixture to 1 L with distilled water in a volumetric flask.

4. 0.5 N sulfuric acid (H_2SO_4) 24.5 g/L
5. 10 percent solution of zinc sulfate
6. 1 N sodium hydroxide (40.0 g/L)

Procedure:

1. Pipette a 5-mL sample into a 500-mL beaker and add 200 mL of distilled water.
2. Heat the solution to a boil and slowly add 30 percent sodium hydroxide to precipitate the copper as copper oxide until further addition produces no more precipitation.
3. After allowing the precipitate to settle, filter the solution through No. 40 Whatman paper or its equivalent and wash the collected precipitate thoroughly with hot water. Collect the filtrate in a 500-mL beaker and discard the precipitate.
4. Add several drops of the bromophenol blue indicator and run in from a burette 0.5 N sulfuric acid until the indicator just turns yellow.
5. Add 30 mL of 10 percent zinc sulfate solution with vigorous stirring.
6. Allow the solution to stand for 5 min and titrate with 1 N sodium hydroxide to the first reappearance of the yellow color.

Calculation:

$$\begin{aligned} 1 \text{ mL } 1 \text{ N NaOH} &= 0.08698 \text{ g } P_2O_7 \\ \text{mL } 1 \text{ N NaOH} \times 17.3 &= \text{g/L } P_2O_7 \\ \text{mL } 1 \text{ N NaOH} \times 2.32 &= \text{oz/gal } 2.32 \end{aligned}$$

c. Determination of Ammonia. The following reagents are required:

1. Methyl red indicator. Dissolve 1 g methyl red indicator powder in 50 mL of isopropyl alcohol and add 50 mL of distilled water.
2. 0.1 N hydrochloric acid standard solution
3. 25 percent sodium hydroxide
4. 0.1 N sodium hydroxide

Procedure:

1. Pipette 5 mL into a 125-mL Kjeldahl flask
2. Add 50 mL of 25 percent sodium hydroxide and attach the flask to a condenser.
3. Distill to one-half (½) the original volume, collecting the distillate in a 400-mL beaker containing 50 mL of 0.1 N hydrochloric acid, 200 mL of distilled water, and 5 drops of methyl red indicator. The outlet of the condenser should be below the solution level.

4. When the solution in the beaker has cooled, titrate with 0.1 N sodium hydroxide to a yellow point.

Calculation:

$$\begin{aligned} 50 \text{ mL } 0.1 \text{ N NaOH} \times 1.17 &= \text{g/L ammonia, 29\%} \\ 50 \text{ mL } 0.1 \text{ N NaOH} \times 0.16 &= \text{oz/gal ammonia, 29\%} \end{aligned}$$

d. Determination of Orthophosphate. The following reagents are required:

1. Phenolphthalein solution 1 percent. Dissolve 1.0 g of phenolphthalein in 50 mL of isopropyl alcohol and add 50 mL of distilled water.
2. Source of hydrogen sulfide.
3. Magnesia mixture. Dissolve 55 g of magnesium chloride and 105 g of ammonium chloride in 800 mL of distilled water. Add 2 mL of hydrochloric acid and dilute to 1 L with distilled water.

Procedure:

1. Determine the pyrophosphate content by the method of 53b.
2. Pipette a 3-mL sample of the copper bath into a 400-mL beaker. Dilute to 200 mL with distilled water and add 5 mL of sulfuric acid.
3. Bubble hydrogen sulfide through the solution for 5 min (under a hood).
4. Filter the solution through No. 40 Whatman paper into a 400-mL beaker.
5. Boil the solution down to a volume of 25 mL.
6. Dilute to 100 mL and add several drops of 1 percent phenolphthalein.
7. Add 29 percent ammonium hydroxide until the solution turns pink.
8. Add concentrated hydrochloric acid slowly until the pink color disappears and then add 10 mL in excess.
9. Add 5 g of ammonium acetate.
10. Add 50 mL of the magnesia mixture.

11. Heat the solution to boiling.
12. Add 29 percent ammonium hydroxide slowly until the solution becomes pink.
13. Stir the solution and cool to room temperature.
14. Add 50 mL of 29 percent ammonium hydroxide.
15. Allow the solution to stand for a minimum of 4 h.
16. Filter the residue through a previously weighed sintered-glass crucible.
17. Wash the precipitate with dilute ammonium hydroxide.
18. Moisten the precipitate with a saturated solution of ammonium nitrate.
19. Dry the residue in an oven.
20. Heat the crucible to a red heat over a Meker (or Tirrill) burner for 30 min.
21. Allow to cool and reweigh the crucible to obtain the weight of the precipitate.

Calculation:

$$\text{Weight of precipitate (g)} \times 37.9 - (\text{oz/gal } P_2O_7) \times 1.09 = \text{oz/gal } PO_4$$

54. Acid Copper Sulfate Analysis

a. Determination of Copper. The following reagents are required:

1. Murexide indicator (see copper pyrophosphate)
2. 0.2 N EDTA:disodium salt (37.23 g/L)
3. Ammonium buffer. Dissolve 68 g of ammonium chloride in 300 mL of distilled water. Add 570 mL of 29 percent ammonium hydroxide and dilute the mixture to 1 L.

Procedure:

1. Pipette 2 mL of solution into 600-mL flask.
2. Add 5 mL of ammonium buffer and dilute to 450 mL with water. The solution should be clear at this point. If it is not, add ammonium buffer until it is.
3. Add 0.2 to 0.4 g of murexide indicator.
4. Titrate with 0.2 N EDTA to blue-violet end point.

Calculation:

$$\begin{aligned} 1 \text{ mL } 0.2 \text{ N EDTA} &= 0.006357 \text{ g Cu} \\ 1 \text{ mL } 0.2 \text{ N EDTA} \times 3.18 &= \text{g/L Cu} \\ 1 \text{ mL } 0.2 \text{ N EDTA} \times 0.424 &= \text{oz/gal Cu} \end{aligned}$$

b. Determination of Sulfuric Acid. The following reagents are required:

1. 0.1 percent methyl orange. Dissolve 0.1 g of the salt in 100 mL of water.
2. 1 N sodium hydroxide (40.0 g/L)

Procedure:

1. Pipette 5 mL of solution into a 250-mL Erlenmeyer flask.
2. Add 150 mL of distilled water and 10 drops of methyl orange indicator.
3. Titrate with 1 N sodium hydroxide to a pale-green end point.

Calculation:

$$\begin{aligned} 1 \text{ mL } 1 \text{ N NaOH} &= 0.0490 \text{ g } H_2SO_4 \\ \text{mL } 1 \text{ N NaOH} \times 9.8 &= \text{g/L } H_2SO_4 \\ \text{mL } 1 \text{ N NaOH} \times 1.31 &= \text{oz/gal } H_2SO_4 \end{aligned}$$

55. Acid Copper Fluoroborate Analysis

a. Determination of Copper. See acid copper sulfate analysis.

b. Determination of Free Fluoroboric Acid. The following reagents are required:

1. 1.0 N sodium hydroxide (40.0 g/L)
2. Cresol red indicator. Weigh out 1 g of the salt. Make a slurry by grinding with 26.2 mL of 0.1 N NaOH. Dilute to 1 L in a volumetric flask.

Procedure:

1. Pipette a 10-mL sample into a 250-mL beaker.
2. Add 150 mL of distilled water and several drops of cresol red indicator.
3. Titrate with 1 N sodium hydroxide to a green-blue end point.

Calculation:

$$\begin{aligned} 1 \text{ mL } 1 \text{ N NaOH} &= 0.0878 \text{ g } HBF_4 \\ \text{mL } 1 \text{ N NaOH} \times 8.78 &= \text{g/L } HBF_4 \\ \text{mL } 1 \text{ N NaOH} \times 1.18 &= \text{oz/gal } HBF_4 \end{aligned}$$

56. Solder (60 Sn, 40 Pb) Acid Fluoroborate Solution Analysis.

a. Determination of Lead. The following reagents are required:

1. 50 percent methyl alcohol

Procedure:

1. Pipette a 5-mL sample into a 400-mL beaker and dilute it to 100 mL with distilled water.
2. Add 20 mL of concentrated sulfuric acid.
3. Heat to boiling for 10 min and allow 1 h to cool to room temperature.
4. Filter through a previously weighed sintered-glass crucible. Wash with dilute sulfuric acid and finally 50 percent methyl alcohol.
5. Dry in an oven for a minimum of 1 h at 100°C.
6. Reweigh the crucible.

Calculation:

$$\begin{aligned} \text{Weight of precipitate (PbSO}_4) \times 136 &= \text{g/L Pb} \\ \text{Weight of precipitate (PbSO}_4) \times 18.2 &= \text{oz/gal Pb} \end{aligned}$$

b. Determination of Stannous Tin. The following reagents are required:

1. 0.1 N iodine (12.7 g/L)
2. Starch indicator. Disperse 1 g of starch in 5 mL of cold water. Pour this suspension into 100 mL of boiling water and allow to cool. Use 2 mL per each titration.

Procedure:

1. Pipette 5 mL of solution into a 500-mL Erlenmeyer flask.
2. Add 50 mL of hydrochloric acid (37%)
3. Add 2 mL of starch indicator.
4. Titrate with 0.1 N iodine solution to first permanent blue color.

Calculation:

$$\begin{aligned} 1 \text{ mL } 0.1 \text{ N I}_2 &= 0.005935 \text{ g Sn} \\ \text{mL } 0.1 \text{ N I}_2 \times 1.187 &= \text{g/L} \\ \text{mL } 0.1 \text{ N I}_2 \times .1584 &= \text{oz/gal} \end{aligned}$$

c. Determination of Free Fluoroboric Acid. The following reagents are required:

1. 1 N sodium hydroxide (40.0 g/L)

Procedure:

1. Pipette a 10-mL sample into a 125-mL Erlenmeyer flask.
2. Against a dark background titrate the undiluted sample with 1 N sodium hydroxide.
3. The end point will be the first turbidity due to the precipitation of lead salts.

Calculation:

$$\begin{aligned} 1 \text{ mL } 1 \text{ N NaOH} &= 0.0878 \text{ g HBF}_4 \\ \text{mL } 1 \text{ N NaOH} \times 8.78 &= \text{g/L HBF}_4 \\ \text{mL } 1 \text{ N NaOH} \times 1.18 &= \text{oz/gal HBF}_4 \end{aligned}$$

d. Assay of the Tin-Lead Deposit⁶⁹

Procedure:

1. Scrape off portions of the deposit with a clean knife.
2. Weigh the sample (0.10 to 0.15 g)
3. Place the sample in a 400-mL flask, add 10 mL concentrated sulfuric acid, and heat until sample dissolves.
4. Cool and add 100 mL of distilled water.
5. Filter through a previously weighed sintered-glass crucible.
6. Wash the precipitate with dilute sulfuric acid, water, and, finally, 50 percent methyl alcohol.
7. Dry for 1 h at 100°C and cool in a desiccator.
8. Reweigh

Calculation:

$$\frac{(\text{Weight of PbSO}_4)(0.683) \times 100}{\text{Weight of sample}} = \text{Pb percent by weight}$$

57. Modified Watts Nickel Analysis

a. Determination of Nickel. The following reagents are required:

1. 0.2 N EDTA: disodium salt (37.2 g/L)
2. Murexide indicator (See copper pyrophosphate)

Procedure:

1. Pipette a 2-mL sample into a 250-mL Erlenmeyer flask.
2. Add 50 mL of distilled water and 10 mL of 29 percent ammonium hydroxide.
3. Add 0.5 g of murexide indicator.
4. Titrate with 0.2 N EDTA to a blue end point.

Calculation:

$$\begin{aligned} 1 \text{ mL } 0.2 \text{ N EDTA} &= 0.00587 \text{ g Ni} \\ \text{mL } 0.2 \text{ N EDTA} \times 2.94 &= \text{g/L Ni} \\ \text{mL } 0.2 \text{ N EDTA} \times 0.392 &= \text{oz/gal Ni} \end{aligned}$$

b. Determination of Chloride or Bromide Content. The following reagents are required:

1. 0.1 N silver nitrate (16.99 g/L)
2. Potassium chromate indicator. Saturate distilled water with potassium chromate.

Procedure:

1. Pipette 5 mL of the solution into a 250-mL Erlenmeyer flask and add 100 mL of distilled water.
2. Add 2 g of sodium bicarbonate and several drops of the potassium chromate indicator.
3. Titrate slowly with 0.1 N silver nitrate until the precipitate formed takes on a faint reddish-brown tint.

Calculation:

$$\begin{aligned} 1 \text{ mL } 0.1 \text{ N AgNO}_3 &= 0.01189 \text{ g NiCl}_2 \cdot 6\text{H}_2\text{O} \\ 1 \text{ mL } 0.1 \text{ N AgNO}_3 &= 0.01363 \text{ g NiBr}_2 \cdot 3\text{H}_2\text{O} \\ \text{mL } 0.1 \text{ N AgNO}_3 \times 2.378 &= \text{g/L NiCl}_2 \cdot 6\text{H}_2\text{O} \\ \text{mL } 0.1 \text{ N AgNO}_3 \times 0.317 &= \text{oz/gal NiCl}_2 \cdot 6\text{H}_2\text{O} \\ \text{mL } 0.1 \text{ N AgNO}_3 \times 2.726 &= \text{g/L NiBr}_2 \cdot 3\text{H}_2\text{O} \\ \text{mL } 0.1 \text{ N AgNO}_3 \times 0.364 &= \text{oz/gal NiBr}_2 \cdot 3\text{H}_2\text{O} \end{aligned}$$

c. Determination of Boric Acid Content. The following reagents are required:

1. 1 N sodium hydroxide (40.0 g/L)
2. Bromocresol purple. Thoroughly mix 1 g of the indicator with 18.6 mL of 0.1 N sodium hydroxide. Dilute to 1L in a volumetric flask.
3. C.P. glycerine

Procedure:

1. Pipette a 5-mL sample into a 250-mL Erlenmeyer flask.
2. Add 25 mL of C.P. glycerine and 0.5 mL of bromocresol purple indicator.
3. Titrate with 1 N sodium hydroxide until the color changes to green to purple as a final end point.

Calculation:

$$\begin{aligned} 1 \text{ mL } 1 \text{ N NaOH} &= 0.0619 \text{ g H}_3\text{BO}_3 \\ \text{mL } 1 \text{ N NaOH} \times 12.38 &= \text{g/L H}_3\text{BO}_3 \\ \text{mL } 1 \text{ N NaOH} \times 1.65 &= \text{oz/gal H}_3\text{BO}_3 \end{aligned}$$

58. Nickel Sulfamate Analysis

a. Determination of Nickel Content. See modified Watts nickel.

b. Determination of Bromide Content. See modified Watts nickel.

c. Determination of Boric Acid Content. See modified Watts nickel.

59. Bright Acid Tin Analysis

a. Determination of Stannous Tin. See stannous tin determination in tin-lead plating.

b. Determination of Sulfuric Acid. The following reagents are required:

1. 1 N sodium hydroxide (40.0 g/L)
2. 4 percent ammonium oxalate. Dissolve 4 g of ammonium oxalate in 96 mL of distilled water.
3. Methyl red indicator. Dissolve 1 g methyl red indicator powder in 50 mL isopropyl alcohol and 50 mL of distilled water.

Procedure:

1. Pipette 5 mL of solution into a 250-mL Erlenmeyer flask.
2. Add 50 mL of 4 percent ammonium oxalate solution
3. Add 20 drops of methyl red indicator.
4. Titrate with 1 N sodium hydroxide to a yellow end point.

Calculation:

$$\begin{aligned} 1 \text{ mL } 1 \text{ N NaOH} &= 0.0490 \text{ g H}_2\text{SO}_4 \\ \text{mL } 1 \text{ N NaOH} \times 9.8 &= \text{g/L H}_2\text{SO}_4 \\ \text{mL } 1 \text{ N NaOH} \times 1.31 &= \text{oz/gal H}_2\text{SO}_4 \end{aligned}$$

60. Tin-Nickel Solution Analysis

- a. Determination of Nickel Content. See nickel solution analysis.
- b. Determination of Stannous Tin Content. See stannous tin in tin-lead analysis.
- c. Determination of Total Tin Content. The following reagents are required:

1. 0.1 N iodine (12.7 g/L)
2. Iron powder
3. Starch indicator. Disperse 1 g of starch in 5 mL of cold water. Pour this suspension into 100 mL of boiling water and allow to cool. Use 2 mL per each titration.

Procedure:

1. Pipette 5 mL of solution into a 500-mL Erlenmeyer flask.
2. Add 100 mL of distilled water.
3. Add 50 mL of concentrated hydrochloric acid and 2 to 3 g of iron powder.
4. Warm the mixture gently until all of the iron is dissolved.
5. Cool the solution rapidly and add 2 mL of starch indicator.
6. Titrate to a permanent blue end point with 0.1 N iodine.

Calculation:

$$\begin{aligned} 1 \text{ mL } 0.1 \text{ N I}_2 &= 0.005935 \text{ g Sn} \\ \text{mL } 0.1 \text{ N I}_2 \times 1.187 &= \text{g/L Sn (total)} \\ \text{mL } 0.1 \text{ N I}_2 \times 0.1584 &= \text{oz/gal Sn (total)} \\ \text{Total tin} - \text{stannous tin} &= \text{stannic tin} \end{aligned}$$

- d. Determination of Fluoride Content. The following reagents are required:

1. 0.1 N potassium chloride (7.45 g/L)
2. Bromophenol blue indicator. See pyrophosphate determination in copper pyrophosphate analysis.
3. 3 percent solution of EDTA magnesium derivative dipotassium salt. Dissolve 3 g of the salt and dilute to 100 mL.
4. Eriochrome black T. Dissolve 0.4 g of indicator in 100 mL of methyl alcohol.
5. 0.2 N EDTA (37.2 g/L)
6. 0.2 N lead nitrate solution (33.12 g/L)
7. Buffer solution. (54 g ammonium chloride, 350 mL of 29 percent ammonium hydroxide, dilute to 1 l.)

Procedure:

1. Pipette a 1-mL sample of solution into a 400-mL beaker.
2. Add 100 mL of 0.1 N potassium chloride and several drops of bromophenol blue indicator.
3. Add dilute ammonium hydroxide to a green color.
4. Transfer the solution to a 250-mL volumetric flask, add 100 mL 0.2 N lead nitrate, and make up to the mark.
5. Allow the lead chlorofluoride to settle.
6. Filter the solution through No. 40 Whatman paper.
7. Pipette 50 mL of the filtrate into a 250-mL Erlenmeyer flask and add 50 mL of distilled water.
8. Add 1 g of Rochelle salts, 10 mL of buffer solution, 2 g of potassium cyanide and 10 drops of the magnesium EDTA complex.
9. Add several drops of eriochrome black T indicator and back-titrate to a blue end point with 0.2 N EDTA.

Calculation:

$$\begin{aligned} 1 \text{ mL } 0.2 \text{ N Pb(NO}_3)_2 &= 0.00190 \text{ g F} \\ \text{mL } 0.2 \text{ N Pb(NO}_3)_2 \times 9.50 &= \text{g/L F} \\ \text{mL } 0.2 \text{ N Pb(NO}_3)_2 \times 1.268 &= \text{oz/gal F} \end{aligned}$$

The residual lead is back-titrated with EDTA:

$$\begin{aligned} (20 \text{ mL } 0.2 \text{ N EDTA}) \times 9.50 &= \text{g/L F} \\ (20 \text{ mL } 0.2 \text{ N EDTA}) \times 1.268 &= \text{oz/gal F} \end{aligned}$$

61. Gold Solution Analysis

a. Determination of Gold. Most gold baths can have their metal content determined gravimetrically by the following method.

Procedure:

1. Pipette a 10-mL sample into a 250-mL Erlenmeyer flask.
2. Under a hood add 10 mL of concentrated sulfuric acid.
3. Heat until dense white fumes evolve and a clear solution is formed.
4. Allow to cool and add 10 mL of concentrated nitric acid.
5. Heat until white fumes evolve and continue for 20 min.
6. Allow the solution to cool and add 100 mL of distilled water.
7. Bring the solution to a boil and allow to cool to room temperature. The precipitate in the flask is gold metal.
8. Filter out the precipitate with ashless filter paper and transfer the paper and precipitate to a previously weighed porcelain crucible.
9. Ignite the paper and allow the crucible to cool.
10. Weigh the gold metal.

Calculation:

$$\begin{aligned} \text{Weight (g)} \times 100 &= \text{g/L Au} \\ \text{Weight (g)} \times 12.2 &= \text{troy oz/gal Au} \end{aligned}$$

62. **Microsectioning**⁶⁰ Microsectioning is an integral part of process control for plating printed boards. A microsection is a special test section cut from a board. The section is mounted in plastic, sanded, polished, and then examined under a metallurgical microscope. The function of microsectioning is to evaluate plating thickness, stress, throwing power, inner layer connections for multilayer boards, undercut from etching, drilling quality, and solderability. Must sectioning of printed boards is performed on holes that are plated through. All of the foregoing evaluations that are applicable can then be conducted.

The sample to be mounted is sheared approximately $\frac{1}{8}$ in from the edge of a hole or line of holes. A large number of mounting plastics, both of the cold-setting and hot-mounting variety, are available. A hot-mounting plastic will necessitate the purchase of a specimen-mounting press. The selection of a mounting plastic will depend to a large extent on the material composition of the specimen, the area to be examined, and the quality of the mount necessary for adequate inspection.

Once the specimen has been mounted, it is processed through several stages of preparation for final examination. These may be roughly described as coarse grinding, fine grinding, rough polishing, and final polishing. A power grinder may be used both for sanding and polishing. An 8-in disk-type power grinder has proved to be very popular. Water should be used during the grinding to carry away all waste material. Power grinders are variable-speed; the lower speeds are used for sanding and the higher speeds for polishing. The mounted specimen is first ground with 60-grit disk sandpaper until the holes begin to break open. Next 180 grit is used to completely break open the holes, 320 grit to sand halfway through the holes, and 600 grit to remove deep scratches.

A medium-nap cotton polishing cloth is used for initial polishing. A series of alumina slurries should be on hand for high-speed polishing. The cloth should be kept wet at all times with the micropolish solution. The initial polish should be with 1- μ m alumina followed by a polish with 0.3- μ m alumina. For the final polish the cloth should be removed and replaced with a micropolish cloth such as AB Microcloth.* 0.05- μ m alumina

*AB Microcloth is a trademark of Buehler, Ltd., Evanston, Ill.

should be used as a polishing medium for the final polish. The section at this point should be smooth and shiny.

To remove from the section any smeared copper that might cause an erroneous reading, the following etch is recommended:

Potassium dichromate	2 g
Water	100 mL
Sodium chloride (saturated solution)	4 mL
Concentrated sulfuric acid	12 mL

REFERENCES

1. Plating 1,000,000 Circuit Boards per Year, *Products Finishing*, January, 1971.
2. T. Fay, Plating PC Boards for Magnavox, *Products Finishing*, May, 1972.
3. G. C. Wilson and R. H. C. Lee, Printed Circuits Plated Through Holes, *Products Finishing*, January, 1970.
4. C. R. Shipley, U.S. Patent 3,011,920, 1961.
5. E. D. D'Ottavio, U.S. Patent 3,650,913, 1972.
6. R. L. Cohen and K. W. West, *J. Electrochemical Society*, 120:502, 1973.
7. A. Rantall and A. Holtzman, Mixed Stannous Chloride-Palladium Chloride Activators: A Study of Their Formation and Nature, *Plating*, April, 1974.
8. D. J. Levy, Aspects of the Anticatalytic Plating Reaction, 50th Annual Technical Proceedings, American Electroplaters Society, 1963.
9. U.S. Patent 2,938,805, 1960.
10. "Metal Finishing Guidebook," Metals and Plastics Publications Inc., Westwood, N.J., yearly.
11. O. R. Cundall and B. F. Rothschild, Trouble-shooting the Multilayer Printed Wiring Board Plating Processes, *Metal Finishing*, February, 1969.
12. J. J. Grunwald and A. J. Siegmund, Adhesion as Applied to Printed Circuits, *Products Finishing*, October, 1970.
13. C. M. Smith, New Printed Circuit Production Technique Achieves Finer Tolerances, Uniformity, *Plating*, April, 1969.
14. How Electroplating Baths Support Methods Electronics High-efficiency Low-cost Production, *Plating*, May, 1972.
15. P. F. Mentone, Pulse Plating, *Products Finishing*, April, 1973.
16. Acceptability of Printed Wiring Boards, IPC-A-600A, Institute of Printed Circuits, 1970.
17. J. B. Mohler, Through Hole Printed Circuit Boards, *Metal Finishing*, September, 1972.
18. M. W. Jawitz, Evaluation of Copper Plating for Printed Circuit Boards, *Metal Finishing*, July, 1973.
19. R. W. Couch and R. G. Bikales, Plating of Printed Circuits with Pyrophosphate Copper and Tin-Nickel, 48th Annual Technical Proceedings, American Electroplaters Society, 1961.
20. A. K. Graham, "Electroplating Engineering Handbook," 3d ed., Metals and Plastics Publications, Inc., 1971.
21. R. Fleming, Bright Solder Plating of Printed Circuits, *Products Finishing*, May, 1972.
22. B. F. Rothschild and D. Sanders, High Throwing Power Solder (Tin-Lead) Plating, December, 1969.
23. J. B. Mohler, Lead-Tin Plating From Fluoborate Baths, *Metal Finishing*, December, 1971.
24. G. Cavanaugh and J. Langan, "Effects of Solution Impurities on Tin-Lead Electrodeposits," Proceedings Manual of the American Electroplaters Second Plating Symposium, February, 1969.
25. U.S. Patent 3,361,652, 1971.
26. K. P. Bellinger and Ularis-Joachim Steeg, Practical Bright Acid Tin Plating, 53rd Annual Technical Proceedings, American Electroplaters Society, 1966.
27. W. B. Harding and H. B. Pressly, Soldering to Gold Plating, 50th Annual Technical Proceedings, American Electroplaters Society, 1963.
28. D. Gardner Foulke and Robert Duva, Gold Alloy Plating Systems, *Insulation/Circuits*, November, 1971.
29. A. M. Weisberg, Nine Ways to Cut the Cost of Gold Plating, *Products Finishing*, September, 1973.
30. F. I. Nobel, D. W. Thomson, and J. M. Leibel, An Evaluation of 18 Karat and 24 Karat Hard Gold Deposits for Contact Applications, *Plating*, July, 1973.
31. A. Koebelak and R. Duva, Precious Metal Plating and Solderability, 48th Annual Technical Proceedings, American Electroplaters Society, 1963.
32. E. W. Wolfgram, Surface Solderability, *Insulation/Circuits*, March, 1972.
33. C. E. Whitfield, U.S. Patent 3,470,043, 1965.
34. I. B. Goldman, A Photoimaging Process for Printed Circuit Manufacture, *Plating*, January, 1974.
35. An Analysis of the Additive Process, Proceedings, April 1971 Meeting, Institute of Printed Circuits.
36. Plating Additive Circuits, *Products Finishing*, February, 1974.
37. K. E. Langford and J. E. Parker, Analysis of Electroplating and Related Solutions, Robert Draper Ltd, 1971.

38. D. Foulke and F. Crane, "Electroplaters Process Control Handbook," Reinhold Publishing Corporation, New York, 1963.
39. A. Wasserman, Rapid Determination of Lead in Tin-Lead Alloy Baths, *Metal Finishing*, April, 1971.
40. G. L. Kehl, "The Principles of Metallographic Laboratory Practice," McGraw-Hill Book Company Inc., New York 1949, pp. 1-59.

Chapter 8

Etching

EDWARD F. DUFFEK

Adlon Engineering Co., Cupertino, California

ERNEST ARMSTRONG

Norcal Tec, Mountain View, California

Introduction	8-2
General Etching Considerations and Procedures	8-2
1. Screened Resists	8-2
2. Hole Plugging	8-4
3. UV-Cured Screen Resists	8-4
4. Photoresists	8-4
5. Plated Etch Resists	8-6
Resist Removal	8-9
6. Screen Resist Removal	8-9
7. Photoresist Removal	8-9
Etching Solutions	8-11
8. Alkaline Ammonia	8-11
9. Cupric Chloride	8-14
10. Persulfates	8-17
11. Ferric Chloride	8-20
12. Chromic-Sulfuric Acids	8-25
13. Other Etchants	8-27
Materials for Board Construction and Their Processing	8-27
14. Thermosetting, Laminates	8-28
15. Thermoplastic Laminates	8-30
16. Metals	8-31
Undercut, Etch Factor, and Overhang	8-37
Equipment and Techniques	8-37
17. Immersion Etching	8-37
18. Bubble Etching	8-38
19. Splash Etching	8-38
20. Spray Etching	8-39

Etching Area Requirements	8-43
21. Equipment Selection	8-43
22. Facility	8-43
References	8-44

INTRODUCTION

The last major step in the chemical processing of the subtractive printed board is metal removal or etching to achieve the desired circuit pattern. Technical, economic, and environmental needs for the practical application of process control have brought about major improvements in etching techniques. Batch-type operations with their variable etching rates and long downtimes have been replaced with continuous, constant-etch-rate processes. Acidic etchants with their generally lower dissolved copper capacities are being replaced by alkaline processes with high copper capacities and potential for regeneration.

Process steps include resist stripping, precleaning, etching, neutralizing, water rinsing, and drying. The technology for etching high-quality fine-line complex circuits in high volume with lower costs is described in this chapter; it includes examination of closed-loop continuous processing, constant etch rates, control at high dissolved copper capacities, regeneration-recovery, reduced pollution, and increased safety. Problems of waste disposal and pollution control have been minimized through adaptation of these principles.

Typical procedures are given in this chapter for etching print-and-etch and metal-plated boards. Strippers and procedures for resist removal are described; they are based on resist selection, board design, cost, and pollution problems.

Etching processors must know the properties of the various etchants available for their use. Factors in addition to the above improvements include finish plate compatibility, control methods, ease of control, and equipment maintenance.

Other considerations include chemical and etchant effects on common PC dielectric laminates, etching of metals other than copper, thin clad copper and semiadditive boards, equipment selection and the techniques involved, production capabilities, quality attained, and facilities.

GENERAL ETCHING CONSIDERATIONS AND PROCEDURES

Good etching results are dependent upon proper image transfer in the case of both print-and-etch and plated-metal etch resists. Etch personnel must be familiar with screened, photosensitive, plated, and other resists being used.

The etching of printed boards must begin with suitable cleaning, inspection, and pre-etch steps to ensure acceptable products. Plated boards also require a careful and complete resist removal process. The steps after etching are important because they are necessary to remove surface contaminations and yield sound surfaces. This discussion will consider the various types of resists and will outline typical procedures used to etch printed boards by the use of organic and plated resist patterns.

1. Screened Resists Screen printing is the most common method for producing standard copper printed circuitry on metal-clad dielectrics and other substrates. The etch-resist material is printed with a positive pattern (circuitry only) for copper etch-only boards or a negative image (field only) when plated-through holes and metal resist are present.

The type of resist material used must meet the requirements for proper image transfer demanded by the printer. From the metal etcher's point of view this same material need only provide good adhesion and etch solution resistance, be free of pinholes, oil, or resist bleed-out, and be readily removable without damage to substrate or circuitry. A description of screen printing is given in Chap. 6.

Oil-base varnishes do not have the greaselike viscosity required for screen printing. They do not dry completely, and they bleed out oils which prevent proper etching. Their chemical resistance to etchants is poor.

Asphalt-base (tar) resists have long been used in etching and plating. They are commercially available in forms suitable for screen and offset-press printing. They can be thinned with petroleum and chlorinated solvents.

Proprietary asphalt-base materials are claimed to provide pinhole-free coverage, fine definition without bleeding, smooth screening, no screen clogging, and resistance to acid etchants. These materials resist the chemical attack of ferric chloride, ammonium persulfate, cupric chloride, chromic-sulfuric acid, and other acid etch solutions at normal room temperatures. As temperatures rise above 76°F, softening, sagging, or running occurs. Required line definition and resolution can be lost. The alkaline etching and plating solutions dissolve the materials at a rate which increases with alkaline concentration and/or temperature.

The dried resist is not hard unless baked, and it can be easily scratched. However, since coatings are usually thick, penetration to base metal is not usually a problem. Boards which have been printed must not be allowed to touch other surfaces, since, even when the resist is dry, there is enough tackiness to cause image damage.

Mechanical cleaning can be done with a soft-bristle brush, but the use of abrasives such as pumice can result in scratches down to the metal laminate. Fine abrasives can be used if they are carefully applied.

Near-neutral alkaline cleaners can be used at temperatures up to 100°F for short duration without ill effects. Low-voltage electrocleaning in mild cleaning solutions at about 85°F is also acceptable practice.

Resist removal is accomplished by utilizing a variety of petroleum solvents or chlorinated hydrocarbons such as trichloroethylene. Ventilation and health protection must be provided.

Cellulose-base lacquers have been used successfully as acid resists. They are not resistant to alkalis, and their high volatility makes them hard to apply. Pattern formations can be attained by spraying through stencils, but coatings tend to be thin. Low viscosity of the lacquers leads to considerable running. Attempts to use higher-solids-content material have failed because of rapid drying, which clogs sprays or stiffens brushes. Dip-coating methods work, but thickness control varies considerably. Slow drying with gradual increase in temperature results in a harder, more durable coating. Unfortunately, this also results in a brittle, poorly adherent film. Resist removal is attained through use of ketones, chlorinated solvents, and commercial solvent strippers.

Lacquer coatings have been successfully used as backup films on sheets of metal which later become printed circuits, without substrates. The backup film prevents parts from falling out during etching.

Vinyl-based screen-printed resists are very suitable for printed circuit manufacture. They are available commercially in consistent form in grades suitable for screen and offset printing. Chemical resistance of the applied coatings is high enough to be excellent protection against the action of acidic etchants including ferric chloride, cupric chloride, sulfuric-chromic acid mixtures, ammonium persulfate, and sulfuric-peroxide, as well as the common low-pH and mildly alkaline (pH 8 to 9) solutions encountered in plating. Alkaline etchants containing high concentrations of ammonium hydroxide are not recommended for use with vinyl-based resists.

When they are properly baked, vinyl-type resists withstand abrasion and yet are adherent but not brittle. (Excessive baking makes them brittle.) Mechanical cleaning can be done by using a soft-bristle brush and fine soft abrasives without damaging the resist, but pumice and other hard abrasives will rapidly cause damage. Commercial alkaline cleaners can be used as soak or electrocleaners at temperatures up to 180°F. The cleaning time should not exceed 30 s. Alkaline treatment over 180°F will loosen the resist.

When current densities used in plating are excessive, some plating may occur on top of the resist. The overplating, plus metal nodules which grow from pinholes in the resist, is usually removed when the resist is removed. If the overplating and nodules are not removed, etching will not be clean and uniform.

A typical cycle for vinyl-based resists used for print-and-etch production of copper printed boards is as follows:

1. Alkali-clean (dip or electroclean), 150°F.
2. Water rinse.
3. HCl 20 percent (by volume).
4. Water rinse.

5. Microsurface etch, i.e., ammonium persulfate, 30 to 60 s.
6. Water rinse.
7. Sulfuric acid 5 to 10 percent (by volume).
8. Water rinse.
9. Examine exposed copper for water breaks. If present, preceding steps should be repeated.
10. Etch under the appropriate conditions for etchant being used.
11. Water rinse.
12. Neutralize etchant.
13. Water rinse.
14. Blow dry, inspect.
15. Strip resist (see section on resist stripping).
16. Rinse in clean stripper to ensure complete removal of resist.
17. Scrub board with soft brush and copper or abrasive wheel in machine scrubber.
18. Water rinse and blow dry.

Adequate cleaning before etching is necessary to ensure a good product. Typical problems are excessive undercoat, unetched areas, and inner-layer shorts in multilayer boards.

Alkali-soluble screening resist is also widely used for print-and-etch applications. Etching is carried out in acidic solution; typically, cupric chloride is used. Abrasion resistance and adhesion are adequate but not as reliable as with vinyl-based materials. Trace edge breakdown occurs with deep etching on panel-plated copper boards. High temperatures, high spray pressures and insufficient preprint cleaning aggravate the problem. Cleaning steps outlined for vinyl resists are suitable except that acid or neutral cleaners must be used. The resists are stripped in hot alkaline solutions.

2. Hole Plugging Plugged-hole, copper-only boards utilize alkaline screen resists in a unique manner. The technique called hole plugging makes possible an all-copper plated through-hole board. The main process steps are as follows:

1. Prefab and drill laminate.
2. Electroless copper-plate.
3. Copper-panel-plate.
4. Hole-plugging; coat plated-through holes with alkali-soluble resist.
5. Cure resist.
6. Sand surface and clean.
7. Screen-print positive pattern with alkali-soluble resist.
8. Cure resist.
9. Etching; acid etchant.
10. Strip resist. Use 2 percent sodium hydroxide.
11. Coat holes and trace with solder.

3. UV-cured Screen Resists UV-cured solventless screen systems are available for print-and-etch and plating applications.*† These resists have good resistance toward commonly used acidic plating and etching solutions. Stripping is carried out in alkaline solutions. Additional information on these materials is given in Chap. 6.

4. Photoresists Photoresists have the same basic functions as screened resists in the manufacture of etched printed boards. Like screened resists, photosensitive resists can be used to print negative or positive patterns on the metal-clad laminate. Liquid and dry film materials are significantly different and will be considered separately.

Liquid photoresist materials, although less durable than screened resists, are capable of finer line definition and resolution. A comparison of photoresists and screened resists, including properties, methods of applying, and advantages of each, is given in Chap. 6. This section deals with the properties and applications of photoresists as they pertain to metal-etching practice.

In general, both positive- and negative-acting resists offer the greatest protection in acidic rather than alkaline solutions. The negative-acting types are more tolerant of

alkaline solutions. The negative resists, once exposed and developed, are no longer light-sensitive, and hence they can be processed and stored in normal white light. The positive resists remain light-sensitive even after developing and must therefore be protected from white light.

Increasing the temperatures of processing solutions also increases the chemical and swelling action on photoresists. Photoresists which can be baked onto the copper at high temperatures are least affected by higher solution temperatures. Certain negative-acting photoresists,* when not baked at a high temperature, lose their adhesion as solution temperatures increase. Thicker coatings of these materials offer greater resistance, but care must be taken not to sacrifice fine-line definition or acuteness. The positive-acting resists are generally affected by high temperature, particularly in alkaline etching and other solutions. Thick AZ† or PRS‡ is least affected. Chapter 6 gives additional details on the various photoresist properties.

With the following differences, the steps for etching copper with photoresist films are similar to those given for screened resists. Scrub cleaning must be done carefully. Photoresist films, even in a baked condition, will not hold up to abrasive cleaners. Here, as with chemical resistance, greater thicknesses help simply by providing a thicker blanket for protection. Electrocleaning is not preferred, since it can lift the resist films.

A light etch followed by a thorough water spray rinse, 10 percent sulfuric acid, and again a thorough water rinsing prior to either etching or plating has been very successful and does not adversely affect any of the common photoresists. When the positive-acting resists are used, it is good practice to give the boards an acid dip just prior to plating or etching, since the resist developer is an alkaline solution and may cause basic copper salts to be present on the surface. Scrubbing prior to plating ensures adhesion.

Problems which may occur with the negative-acting resists include the following:

1 Resist breakdown in cleaning. This is most often caused by applying resist over an oxidized board or one which has been bright-copper-plated but not abrasive- and brush-cleaned. Resist may also not have been baked long enough. Treat boards as follows to improve adhesion: mild etch, H₂O rinse, acid-dip, rinse, dry thoroughly, cool, coat with resist. Mat- or satin-type finishes are preferred for the best adhesion. Abrasive cleaning prior to chemical cleaning is useful, particularly when bright copper or other plated surfaces are to be printed. Chemical cleaning is not as effective as mechanical cleaning with respect to increasing adhesion. Make sure resist is baked properly.

Photoresist breakdown may also occur during plating, especially in cyanide processes. The reasons for breakdown are similar to those given above.

2 Pinholes in resist. This is frequently caused by solids, water, or oil in the air line used to dry or spray-coat boards. The cure is to use positive filters and liquid traps. Do not shake air into the resist. Make sure all absorbed moisture has been removed from boards by baking prior to coating them. Keeping air out of the resist during coating also helps to reduce rejects due to pinholes.

3 Excessive undercut when deep etching. Excessively baked photoresists are brittle and may break away during etching to expose more metal. The result is excessive undercut. The opposite effect can be attained by the use of flexible photoresists which protect the edges as etching progresses downward.

4 Plated-through holes etched away. This difficulty may be caused by a variety of factors such as resist in holes during plating, rough drilling, rough plating, insufficient resist metal plating, and excessive time in etch solution.

Resist remaining in the holes is due to light exposure down into the holes. Film out of registration or targets in pads can allow light to expose resist in holes. Prevention is necessary because exposed resist in holes cannot be removed by normal cleaning. Dip- or whirl-coated photoresist may also stay in the holes even if not exposed, simply because the developing operation will not dissolve such a bulk amount.

Rough drilling also can result in voids and opens in plating because of the ability of the etch solution to attack the copper not protected by plated resist metal. Insufficient resist metal plate is first observed in holes, since the holes receive a smaller portion of the total resist plate. When the difficulty occurs, all holes, rather than an isolated few, are affected.

*For example, KOR, KMER, KTRF, products of Eastman Kodak Company, Rochester, N.Y.

†Product of Shipley Co., Newton, Mass.

‡Products of Dynachem Corporation, Santa Fe Springs, Calif.

*DYNACURE, a product of Dynachem Corporation, Santa Fe Springs, Calif.

†SPR Resist, a product of W. R. Grace Co., Columbia, Md.

Electroplating conditions should be reviewed to improve resist-metal-plating results. See Chap. 7 for additional suggestions.

Excessive time in etch must be avoided. The reasons for exceeding the normal etching time must be determined. For example, it may be that etch solution is spent, the temperature is too low, or excessive copper, if panel-plated has been deposited. Still another reason could be improper cleaning. All but a small area may etch clear, but the operator may leave the board in the etching solution longer to get complete metal removal. That invariably results in excessive etching and undercutting of the normally etched areas, and it may allow penetration of the etch-resist coating. Complete, proper cleaning prior to etching must be standard practice.

The positive-acting resists are subject to the same problems as the negative-acting resists, although they are somewhat easier to remove cleanly, after exposure, from areas to be etched. Resist-breakdown problems are due mainly to undesired light exposure, excessive temperature, and alkalinity during processing. Alkaline etchants will destroy positive-acting resists. Excessive current densities during cleaning and plating, which result in gassing, may also lift the resist. Problems related to plating photoresist-coated boards are reviewed in Chap. 7.

Other materials also are used as photosensitive etch resist.^{1,2} Naturally occurring colloidal organic materials (albumin, fish, glue, gelatine, gum arabic) sensitized with ammonium dichromate have been referred to as "hot-top enamels." Noncolloidal organic products (shellac), also sensitized with ammonium dichromate, are known as "cold-top enamels"; they are so named because somewhat lower baking temperatures are involved. These systems have been used in the preparation of lithographic plate printed circuitry and recently for etched lead frames for microelectronics. They are not used extensively because they must be heated at temperatures above degradation limits of dielectric substrates (greater than 300°F) to achieve etchant resistance. Also, they may have dimensional instability, scum during development, and/or have poor aging properties. However, they are attractive from the viewpoint of cost, and in certain instances they have very fine line definition capability. For example, cold-top enamels in electroforming practices have been used to produce 0.010-in metallization masks, 500-mesh screen size with 0.002-in centers.

Dry film photoresists such as Laminar* and Ristout† are well suited for plating and etching processing. Chemical resistance of both the solvent- and alkali-soluble resists in most etchants is good.

Resist degradation is a function of time and temperature of exposure to alkaline solutions. Hot strongly alkaline soaks and electrolytic cleaners degrade the resists and may lead to contaminations in plating solutions. Stripping of dry film resists is considered in later sections.

Electrophotographic processes for printing prior to etching are available, but they had not been utilized on a production level at the time of writing. They appear to be particularly attractive because a finished prototype can be made in a few minutes from a master drawing without involving photo film.

5. Plated Etch Resists Plated-through-hole circuit boards represent the most extensive use of metal-plated resists. Solder plate (60 Sn, 40 Pb), gold, tin, and tin-nickel are the most used resists. Silver is used to some extent for light-emitting and liquid crystal applications. Details concerning the deposition of these metals are given in Chap. 7. The following discussion pertains to the use of the metals as etch resists.

a. Solder Plate. Tin-lead solder 0.0003 to 0.001 in thick is the most common plated etch resist. The 60 Sn-40 Pb alloy offers good etchant resistance with few problems. Increased reliability is achieved by the use of solder plate over tin-nickel.³

Alkaline ammonia, chromic-sulfuric acid, and ammonium persulfate-phosphoric acid are the most suitable etchants. Ferric and cupric chloride acid etchants cannot be used. Post-etch neutralization rinses are needed, especially with alkaline systems, to rinse away etchant residues and maintain good surface properties.

*Registered trade name of photopolymer film resist made by the Dynachem Corporation, Santa Fe Springs, Calif.

†Registered trade name of photopolymer film resist made by E. I. du Pont de Nemours & Company, Wilmington, Delaware.

A typical procedure used to etch a solder-plated board is as follows:

1. Resist stripping, bath 1.
2. Resist stripping, bath 2.
3. Water rinse.
4. Examine board after stripping. Make sure all resist has been removed. Return to stripping if not complete.
5. Examine for solder or other materials in the copper area which is to be removed. All such materials are to be scraped off. Care must be taken not to damage circuit lines.
6. Examine circuit area including inside of holes for exposed copper. All exposed copper in circuit lines is to be painted over with screen ink resist (touch-up).
7. Oven-dry touch-up ink for 10 min at 110°F.
8. Alkali cleaning (optional). For boards heavily stained or oxidized, 160 to 180°F, 3-min soak.
9. Water rinse.
10. H₂SO₄, 10 percent (by volume), 30 s (optional).
11. Water rinse.
12. Blow dry.
13. Inspect.
14. Pre-etch (optional).
15. Inspect for resist residues.
16. Place on conveyor or etching rack.
17. Etch boards (alkaline etchant).
18. Water rinse.
19. Neutralize.
20. Water rinse.
21. Blow dry.
22. Proceed to contact plating or solder reflow.

b. Gold. Gold plated upon a smooth pore-free base metal such as copper or copper with an underplate of nickel or tin-nickel provides excellent resistance to all the common copper etchants. Ferric chloride, cupric chloride, sulfuric-chromic acid mixtures, and alkaline ammonia can be used successfully. Some etchants have a slight dissolving effect on gold and hence necessitate avoidance of the long exposures which may be required for deep etching.

Process steps for etching a gold-plated board for high-reliability applications are as follows. Steps 1 to 9 are the same as those for solder-plated board.

10. HCl 20 percent (by volume) at room temperature for 30 s.
 11. Spray water-rinse, proceed with step 10 for etching.
 12. Blow dry with air and rack in tote boards if etching is not done immediately.
 13. Pre-etch for 30 to 60 s.
 14. Examine for resist remaining in letters, etc. Preclean as needed.
 15. Place in etching rack or conveyor. Etch as required.
 16. Remove boards from etcher and rinse thoroughly in running water.
 17. Examine for copper not removed.
 18. If desired, copper not removed may at this point be etched off with solution etchant by using a paint brush. This reduces undercoat from areas already etched clean (optional).
 19. Water rinse.
 20. Neutralize rinse.
 21. Spray water rinse.
 22. Hot-water dip; 180°F for 5 to 10 s.
 23. Scrib with alkali cleaner and a soft brass brush to remove burrs. (Optional) Ultrasonic clean.
 24. Water rinse.
 25. HCl-dip, 20 percent by volume for 2 to 5 s.
 26. Water rinse.
 27. Blow dry with filtered air.
 28. Remove boards from etch area.
- c. Tin Plating.** Bright tin plating directly over barrier layers of nickel or tin-nickel is used because of its good solderability properties. Chromic-sulfuric acid, alkaline ammo-

nia, and ammonium persulfate phosphoric acid etchants have been formulated especially for bright tin.

d. Tin-Nickel. Tin-nickel (65 percent Sn, 35 percent Ni) is generally overplated with gold, solder, or tin for best board reliability. Alkaline ammonia, ammonium persulfate and CuCl_2 etchants are well suited for this alloy.

e. Precious Metals and Alloys. Rhodium has been described as a suitable etch resist for contact tabs on boards, but it may not be entirely suitable because the thicknesses are often only 5 to 10 μm (0.000050 in.). Also, even when plated over nickel, it tends to be porous and to lift during etching. Because of the varying surface properties, 18k alloy gold must be evaluated carefully when it is used as a substitute for higher-karat or pure-gold systems.

f. Silver. Although silver is not used on most printed boards (MIL-STD-275 states that it shall not be used), it has found some application for camera, light-emitting, and liquid crystal devices. Copper etching using silver as resist can be done with alkaline ammonia solutions. About 0.0001 in./min per minute is dissolved during etching.

g. Etching Procedure: Recommendations for Plated-metal Resists. Etching of the organic resist and metal-plated boards begins with removal of the resist. Commercial solvents and strippers are available. The section on resist removal given later should be consulted for additional details. Gold, solder, and tin resists must be handled carefully, because they scratch very easily. Tin-nickel alloy plate, however, is very hard and is abrasion-resistant.

The oily residues left by resist strippers can be removed by alkali cleaners, heated to 180 to 190°F and used as soak, or by electrocleaning solutions. After alkali cleaning and rinsing, it is easy to detect organic resist residue by looking at the edges of circuit lines and the centers of letters and numbers. The most difficult materials to remove are the negative-acting photoresists. They do not dissolve in their stripper; instead, they soften and form gel-like films which lift off the metal surface. Soft-nylon-bristle brushing with alkali cleaners helps, but nylon brushes must be kept out of organic stripper. Soft brass and stainless steel is suitable for use on boards plated with tin-nickel alloy. Gold (type II) is somewhat harder and can be brushed to some extent with a soft brass brush. Brushing must be done with discretion, however, since it is possible to scratch through the gold plating. Solder plate should not be brushed. The use of pumice or similar hard abrasives to aid in cleaning plated boards prior to etching is not recommended.

Once the organic resists have been removed, surface oxides and other films are removed by rinsing the boards in an acid solution, e.g., sulfuric acid (5 to 10 percent by volume). After thorough water rinsing the boards can proceed directly to etching without drying. If short storage is required before etching, the boards should be dried completely with clean air and stored in a way that will prevent contamination.

Some circuit board manufacturers use mild etching as an additional step after the sulfuric acid rinse. The mild etching is followed by a water rinse, an additional sulfuric acid dip, and another water rinse. Areas not completely clean become very evident at this point, since they do not have the matte etched appearance of the rest of the surface. The board is ready for etching when the areas to be etched are water-break-free.

When pattern (selective) copper and/or other metal plating has been done, small metal nodules may appear because of pinholes in the resist. Preferred practice is to leave the nodules alone, since they can scratch the plated resist if they are removed physically. Their base is very small and they usually etch off without additional effort. The best solution, however, is to improve the imaging techniques.

The procedure after etching includes thorough water rinsing and acid neutralizing to ensure removal of etchant residues on the board surface. Alkaline etchants are followed by treatment with proprietary neutralizing solutions; 5 to 10 percent solutions of HCl or oxalic acid are used after FeCl_3 etching; H_2SO_4 is used after ammonium persulfate; and alkaline cleaning is used for tin-lead boards etched in chromic-sulfuric acid. Etchant residues, particularly if they are not removed before drying, result in lowered electrical resistance of the dielectric substrate surfaces as well as electrical and soldering problems on the conductive wiring surfaces. Surface contamination studies are covered in the literature.⁴⁻⁶

Problems which occur in etching when a plated metal is used as the etch resist are for the most part traceable to the photographic film (which may have pinholes in it), thin or porous organic resist, or plating which is thin or porous. In the etching process itself, improper cleaning, excessive etch solution temperatures, excessive time in etch solution,

wrong etchant, uneven etching due to lack of motion during etching, excessive spray pressures, inadequate rinsing and neutralizing, rough handling, and many other faulty practices may result in rejected circuit boards.

A problem common to etching printed boards is that of having the entire area etch clean at the same time. That rarely happens, however, since etch action is more rapid at the edges of printed areas than in broad expanses of copper. Spray etching and paddle etching have lessened the differential effect, yet it remains. When very fine patterns and lines are required, the result can be loss of the pattern due to underetch, especially when the board is left in the etcher until all the field copper has etched away. To prevent that, fine-line boards should be removed from the etcher as soon as possible. Dry film photoresist, thin clad laminate, and fully additive or semiadditive processing should be used for fine-line boards in high-volume production.

RESIST REMOVAL

The method used for resist stripping is an important consideration when a resist is selected. Effect on board materials, cost and production requirements, and compliance with safety and pollution standards must be taken into account. Both organic and aqueous stripping systems are in use.

6. Screen Resist Removal Screened vinyl- and asphalt-based resists are removed by a dissolving action. In the case of solvent-soluble resists, removal is readily effected by chlorinated and petroleum solvents, xylene, mineral spirits, and laquer thinner. Methylene chloride is used extensively in "cold" stripper formulations. General makeup has been described.⁷⁻⁹ Commercial cold strippers are classified according to their pH in a 10 percent stripper-water mixture. The most common strippers are acidic (lower-pH) formulations which contain copper brighteners and water-rinsing agents. The usual procedure for static tank stripping involves soaking the coated boards in at least two tanks of stripper followed by a water spray-rinse. Excessive time in strippers is to be avoided for reason of attack on the "butter" (epoxy) coat, especially on print-and-etch or single-sided boards. See General Considerations for complete cycles.

Solvent stripping machines are commercially available. High-volume users realize considerable savings through the use of conveyORIZED systems equipped with reclaimation facilities.

Distillable cold strippers generally contain methylene chloride or trichloroethylene. Some heating is helpful, but it can be tolerated only in closed systems for reason of health and fire hazards. Water is a contaminant in most cold strippers. In all cases adequate ventilation and solvent waste disposal must be provided.

Alkali-soluble resist inks are generally selected for copper print-and-etch boards, and they are used extensively in consumer applications. See General Considerations for the cycle. Stripping in the case of air-dry and UV-curable resists is accomplished in 2 to 10 percent solutions of sodium hydroxide at 140°F. The resist is loosened and rinsed off with a water spray. Advantages are low cost and easier compliance with disposal requirements. Adequate safety precautions must be taken, since caustics are harmful.

ConveyORIZED etchers and resist-stripping machines employ high-pressure pumping systems which spray heated alkaline solutions on both sides of the boards and through the holes. Single-sided boards and certain laminate materials such as the polyimides are attacked by alkaline strippers. Measling, staining, or other degradation is noted when strippers attack epoxy or other substrates.

7. Photoresist Removal

a. Dry Film Dry film resists have been formulated for removal in solvent and aqueous-alkaline solutions. Strippers of each type are available for static tank and conveyORIZED systems. Cold stripper solvent-type formulations are similar to those described for use on screen inks and liquid negative-acting photoresists.⁸ The primary solvent is methylene chloride with added ingredients for enhancing dissolution, swelling, copper brightening, and water rinsability. Refer to the section on dry film resist for additional information.

b. **Negative-acting Liquid Photoresist Removal.** Negative-acting, liquid-applied photoresists, such as the KPR* series and DCR† series, can be removed readily from printed boards if they have not been baked excessively. Baking is critical to removal because it relates to the degree of polymerization. Since overbaking is also damaging to the insulating substrates, processes should stress minimal baking—only enough to withstand the operations involved. See Image Transfer, Chap. 6.

The negative-acting resists are removed by using TCE, Stoddard solvent, methylene chloride, and commercial strippers. In this case, the resist does not dissolve; instead, it softens and swells, breaking the adhesive bond to the substrate. Once that has taken place over the entire coated area, a light tap water spray is used to flush away the film. With the more difficult-to-remove resists (baked, for example) a 3-min soak (depending on substrate resistance to the stripper) followed by scrubbing with a soft cotton swab soaked in the stripper has been used effectively. Thin coatings, when properly baked, are generally removed more readily than thick coatings.

Cold-soak commercial strippers are composed mainly of methylene chloride and chemical additives. Methyl alcohol as cosolvent 10 percent by volume, furfural (cyclic aldehyde), and phenol (carboic acid) up to 1 percent by volume aid in causing the resist to swell. Phenol also acts as a weak acid in solution; it dissolves copper oxidation products and brightens the copper. Formic, sulfonic, and other organic acids are added as activators for copper brightening, emulsification and ease of water rinsing.⁸ Paraffin or a higher ketone (Bertone) is used to retard evaporation.

Dwell times in cold strippers is limited by the possibility of attack on the epoxy laminate. Although not too effective, ketones are added to inhibit attack on epoxy boards.

Soaking followed by mechanical brushing or spraying aids considerably in removing the jellylike resist. If the soaking time is not long enough, resist will remain under plated-over trace edges and cause bridging, jagged traces, and possible soldering problems.

Resists of the Kwik KMER and KTFR series, Dynachem's CMR 5000, and Waycoat's PF series are not generally used on conventional printed boards. They are most often used on solid metal, ceramics, and silicon substrates. Their baking may reach up to 650°F, which makes them extremely adherent and resistant to attack. Removal calls for the use of hot solvents or strong oxidizers. Since these resists are used most commonly on metal masks or other solid metal parts, substrate chemical resistance is not a problem. A good practice is to allow the chemicals to do the removing, rather than try to scrub and half-soften resist. Soak, swab, and water spray will remove the swelled film.

With pattern plating it is extremely difficult to remove the photoresist from under the overplate or overhang. Trichloroethylene liquid- and vapor-degreaser treatment helps soften the film, which is then effectively removed during mechanical cleaning by using a cold stripper and a soft brush. Electrocleaning, which results in considerable gassing, also helps to free the film.

Plasma oxidation (vapor burn-off) may damage the common PC board substrate, but it is very effective in complete removal of photoresists from metals, ceramics, and silicon substrates. Photoresists may also be removed by high-temperature firing (750 to 930°F), mechanical abrasion, or boiling in sulfuric or benzene sulfonic acids. None of these methods is practical for conventional printed boards, however.

Cleaning after resist stripping of plated metal-resist boards is essential for good complete etching. Photoresist left at the edge of circuit traces or in the centers of letters results in a sawtooth effect and incomplete etching. In some cases oil residues in the strippers cause random unetched areas. The metal to be etched must be cleaned as though in preparation for plating. Hot alkali electrocleaning (cathodic) with soft-brush scrubbing (no abrasives) followed by a water rinse, 20 percent sulfuric acid, water, inspection, and final etching will result in a suitable product. Attempts to strip negative-acting photoresists by using hot alkaline solutions have also been made. Other references are cited.^{8,9}

c. **Positive-acting Photoresist Removal.** Photoresists of the AZ§ type and other positive-acting photoresists are removed by dissolving in acetone, ketone, cellulose acetate, and other organic solvents. Commercial organic and inorganic strippers are suitable when baking has not been excessive. Removal by exposure to ultraviolet light and subsequent

*Product of Eastman Kodak, Rochester, N.Y.

†Product of Dynachem Corp., Santa Fe Springs, Ca.

‡Waycoat, Product of Hmit Chemical Company, New Jersey.

§Product of Shipley Co., Newton, Mass.

dipping in alkaline developer, sodium hydroxide, TSP, and other strongly alkaline solutions is effective. Overexposure is required to ensure against the shielding of metal overplate at trace edges on pattern-plated boards. Overbaking makes removal very difficult.

Machine stripping is carried out in a solution of 0.5 N sodium hydroxide, nonionic surfactants, and defoamers.

ETCHING SOLUTIONS

This section is a survey of the technology and chemistry of the etching systems in common use. Changes from batch-type operation to continuous constant-rate systems and greater use of alkaline systems as compared with acidic processes represent major innovations in etching practices. The various factors that account for the changes are as follows:

1. Printed board design
2. Compatibility with resist
3. Control of etch rate
4. Etching rate (speed)
5. Dissolved copper capacity
6. Ease of process control
7. Ease of equipment maintenance
8. Costs and economics
9. High yield
10. Regeneration and replenishment
11. Material recovery
12. Disposal and pollution control

Copper etchants are evaluated in view of the above factors. Application, chemistry, chemical makeup, and comparison of etchants are given, along with suggestions for selection and control.

Etching solutions are not well understood by many of their users. Equipment and reagent solution control problems are frequently difficult to separate. Although selection of an etching process is not difficult, the utmost in performance and life is not always attained. Additional work and understanding are required, especially in regard to full regeneration and recycling of materials. An earlier survey of those considerations has been reported.¹⁰

8. **Alkaline Ammonia** Alkaline etching with NH_4OH complexing has found increased acceptance owing to continuous operation, compatibility with most metallic and organic resists and especially solder, minimum undercut, high capacity for dissolved copper, and fast etch rates. Both batch and continuous (closed-loop) operation are in use. Continuous operation provides constant etch rates, high work output, ease of control and replenishment, and improved pollution control. Costs are relatively high; neutralization after etching is critical; and the ammonium ion introduced to the rinses is a difficult waste-treatment problem. Complete regeneration with chemical recycling is not widely practical.

Early versions of alkaline etchants were batch-operated. They had a low copper capacity, and the etch rates dropped off rapidly as copper content increased.^{10,12} It was found necessary to include controlled amounts of dissolved oxidizing agents to speed up the rate and increase copper capacity at a constant temperature. Batch operation is still practical for low-volume production.

Etching solutions are operated at 120 to 130°F and are well suited to spray etching. Good exhaust systems are required, because ammonia fumes are released during operation. (Maximum acceptable concentration of NH_3 is 100 ppm.)¹³ Etching machines must have a slight negative pressure and moderate exhausting to retain the ammonia necessary for holding the dissolved copper. Currently available solutions offer constant etching of 1-in. copper in 1 to 1.5 min with a dissolved copper content of 18 to 24 oz/gal.

u. **Chemistry.** Compositions of alkaline solutions are proprietary, but general ideas regarding them can be gleaned from patent literature as illustrated in Table 8.1. The functions of the basic constituents are as follows:

NH₄OH acts as a complexing agent and holds copper in solution. NH₄Cl increases etch rate, copper-holding capacity, and solution stability. Copper ion, Cu²⁺, is an oxidizing agent; it reacts with and dissolves metallic copper. NaClO₂ also is an oxidizing agent that reacts with and dissolves metallic copper. NH₂HCO₃ is a buffer; it retains clean solder holes and surface. (NH₄)₂(PO₄) retains clean solder and plated-through holes. NH₄NO₃ increases etch rate and retains clean solder.

Some batch formulations require the mixing of two solutions just before operation. Oxidizer and complexer are separated for shelf stability. Processes available for continuous operation are single-solution makeup buffered to a pH of 7.5 to 9.5.

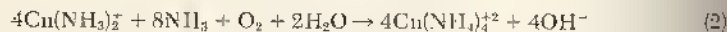
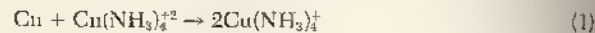
TABLE 8.1 Composition of Alkaline Etchants

Component	1 st	2 nd	3 rd
NH ₄ OH	3.0 mol/L	6.0 mol/L	2-6 mol/L
NH ₄ Cl	1.5-0	5.0	1-4.0
Cu (as metal)		2.0*	0.1-6
NaClO ₂	10.375		
NH ₂ HCO ₃	0-1.5		
(NH ₄) ₂ PO ₄		0.01	0.05-5
NH ₄ NO ₃	0-1.5		

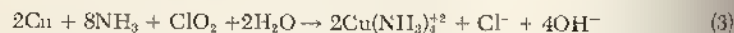
*Starter solution only

Alkaline etching solutions dissolve the exposed field copper on printed boards by a chemical process of oxidation, solubilizing, and complexing. Ammonium hydroxide and ammonium salts combine with copper ions to form cupric ammonium complex ions [Cu(NH₃)₄²⁺], which hold the etched and dissolved copper in solution at 18 to 30 oz/gal.

Typical oxidation reactions for closed-loop systems are shown by the reaction of cupric ion on copper and air (O₂) oxidation of the cuprous complex ion:



and for batch-operating systems by chlorite ion on copper:



Six ounces per gallon of dissolved copper is the calculated copper capacity achieved in the case of batch etching with chlorite (Table 8.1, No. 1). Air oxidation (O₂) or a temperature increase is needed for higher copper capacity.

A constant etch rate of 1.5 min with 1-oz copper is achieved by gradually increasing the temperature from 90 to 120°F.

It can also be seen that closed-loop etching can be continued with the formation of Cu(NH₃)₄²⁺ oxidizer from air during spray etching and as long as the copper-holding capacity is not exceeded.

b. Closed-loop Systems. A practical method of maintaining a constant etch rate with minimal pollution utilizes specific-gravity- (density-) controlled automatic feeding.¹⁶ The process, which is generally referred to as flow-through, is illustrated in Fig. 8.1. As the printed boards are etched, copper is dissolved and the density of the etching solution increases. The density of the etchant in the etcher sump is sensed to determine the amount of copper in solution. When the density sensor records an upper limit, a switch activates a pump which automatically feeds replenishing solution to the etcher and simultaneously removes etchant until a lower density is reached. Typical operating conditions are as follows:

Temperature	120-130°F (49-50°C)
pH	7.8-9.2
Specific gravity at 130°F	1.169-1.199
Baumé, Be°	21-24
Copper concentration, oz/gal	18-24
Etch rate, 0.001 in/min	1.6-2.0

Etching rate versus copper content is shown in Fig. 8.2.

A study of the etching rate of copper vs. dissolved copper content shows the following effects: at 0 to 11 oz/gal copper, long etching times; at 11 to 16 oz/gal copper, lower etch rates but difficult solution control; at 18 to 22 oz/gal, etch rates are high and solution is stable; at 22 to 30 oz/gal, solution is unstable and tends toward sludging.

All work must be thoroughly rinsed with water immediately as it leaves the etching chamber. Do not allow the boards to dry before rinsing. Etched circuitry with plated tin-lead solder resist also requires an acid neutralization and solder-brightening application. The main purposes are to remove etchant from under circuit edges and clean and brighten the wirenet surfaces and plated-through holes. The materials are proprietary. This operation is critical for good solder reflow results. Thorough deionized water rinsing and clean-knife drying result in clean, stain-free surfaces.

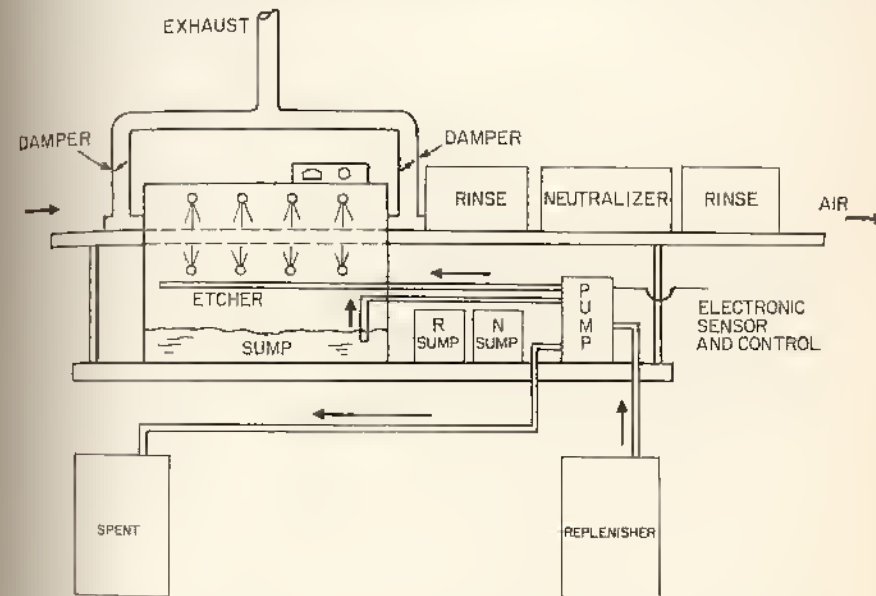


Fig. 8.1 Automated flow-through alkaline etching system.

c. Regeneration. True regeneration requires the following:

1. Removal of portions of the spent etching solution from the etcher sump under controlled conditions.
2. Chemical restoration of spent etchant, i.e., removal of excess copper by-products and adjustment of solution parameters for reuse.
3. Replenishing of etchant in the etching machine.

Constant etching conditions are achieved when regeneration is continuous. The principal methods of regeneration are crystallization, acidification and liquid-liquid extraction.

a. Recrystallization reduces the copper level in the etchant by chilling and filtering the precipitated salts. That is followed by re-fortification and adjustment of operating parameters.

b. Acidification precipitates copper hydroxide, which is filtered to leave a clean solution for re-fortification and pH adjustment. The pH must be reduced very carefully because excess acid (HCl) will produce poisonous and explosive gases of ClO₂ and chlorine. The danger of overacidification because of faulty equipment, miscalculation, or accident must be given careful consideration. Dependable exhausts and other safety equipment must be used.

c. Liquid-liquid extraction.^{18,19} This method is meeting acceptance because of its continuous and safe nature. The process involves mixing spent etchant with an organic

solvent (i.e., hydroxy oximes) capable of extracting copper. The organic layer containing copper is subsequently mixed with aqueous sulfuric acid, which extracts the copper to form copper sulfate. The copper-free etchant is restored, and the copper sulfate is available for electrolytic copper, acid copper plating, or copper recovery.

Closed-loop regeneration systems reduce chemical costs, sewer contamination, and production downtime. However, regeneration by these methods is expensive and is limited to large printed circuit facilities.

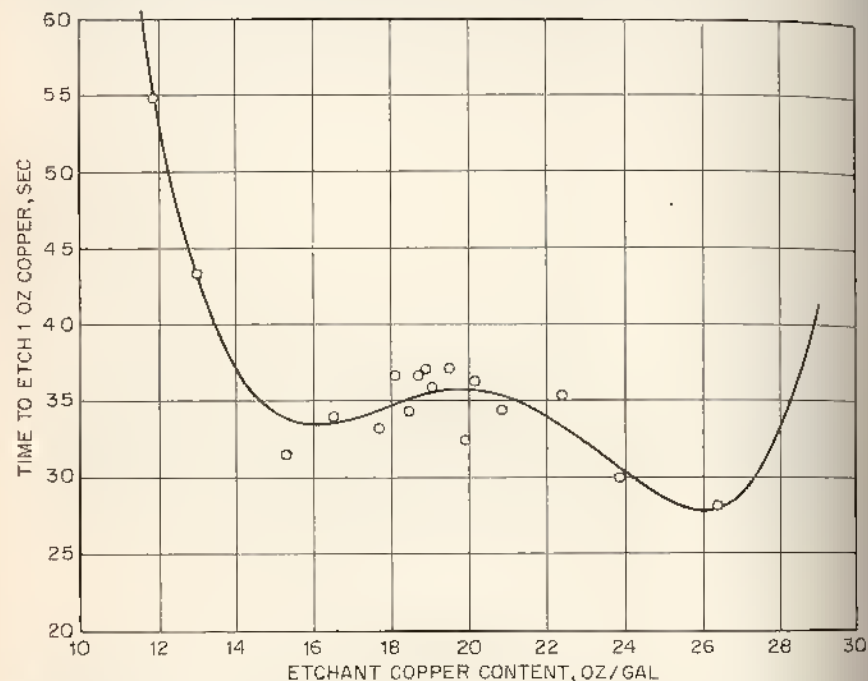


Fig. 8.2 Etching rate vs. etchant copper content (oz/gal). (Courtesy Hewlett-Packard Company, California.)

d. Problems

1 **Reduced etch rates.** This problem can be related to a number of factors. First check etching conditions and solution balance, for example, temperature, spray pressure, density, and pH. A low pH may be due to excessive ventilation, heating, or spray pressure. Adjust pH upwards with aqua ammonia. Check automatic replenishing equipment.

2 **Sludging.** Loss of ammonia (reduced pH), dilution, or equipment malfunction. A white foam on the etchant indicates water introduction.

3 **Solder attack.** Review etching conditions and solution balance. Excess chloride (Cl^-) may cause solder attack.

4 **Residues on solder plate, holes, and traces.** Dark or contaminated holes require control of etching conditions and neutralizer rinses.

5 **Under- or overetching.** Examine etching conditions and solution control.

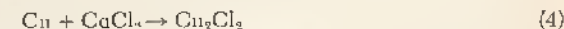
6 **Pollution.** Rinse water coming from etchers must be free of dissolved copper; if not it must be chemically treated and separated from ammonia-bearing rinses.

Thin clad copper laminates present a further problem because the faster throughput of the etcher increases the carryover of etchant into the rinses.

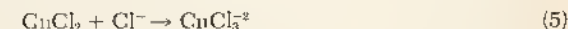
9. **Cupric Chloride** Cupric chloride systems are typical of the innovations in etching practice to achieve closed-loop regeneration, lower costs, and a constant, predictable etch rate. Steady-state etching with acidic cupric chloride permits high throughput, material recovery, and reduced pollution. Regeneration in this case is somewhat complex but is

readily maintained. Dissolved copper capacity is high compared with that of batch operation. Cupric chloride (CuCl_2) solutions are used primarily for multilayer inner details and print-and-etch boards. Resists are screened inks, dry-film, gold, and tin-nickel. Solder (lead-tin) and tin boards are not compatible with cupric chloride etchant.

a. **Chemistry.** The etching reaction is as follows:

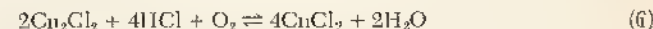


Chloride ions, when added in excess as HCl, NaCl, or NH_4Cl , act to solubilize the relatively insoluble Cu_2Cl_2 and thereby maintain stable etch rates. The complex formation can be shown as follows:



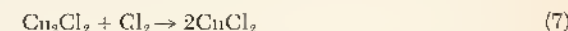
The etchant can be regenerated by reoxidizing the cuprous chloride to cupric chloride as illustrated by the following reactions:

1. Air:

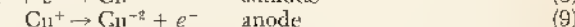
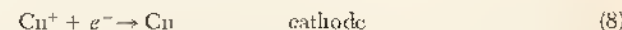


This method of regeneration is not used because the oxygen reaction rate in acids is slow and the solubility of CuCl_2 in hot solution is low (4 to 8 ppm). Spray etching induces air oxidation.

2. Chlorination:



3. Electrolytic:



Typical formulations are given in Table 8.2. Other formulations can be found in the literature.^{9,125}

b. **Properties and Control.** Early cupric chloride formulations had slow etch rates and low copper capacity and were limited to batch operation.²⁰⁻²⁸ Regenerable continuous operation with modified formulations (Table 8.2) has brought useful improvements. Etch rates

TABLE 8.2 Cupric Chloride Etching Solutions

Component	1 ^o	2 ^o	3 ^o	4 ^o
$\text{CuCl}_2 \cdot 2\text{H}_2\text{O}$	1.42 lb	2.2 M	2.2 M	0.5-2.5 M
HCl (20% Bc)	0.6 gal	30 mL/gal	0.5 N	0.2-0.6 M
NaCl		4 M	3 M	
NH_4Cl				2.4-0.5 M
H_2O	*	*	*	*

*Add water to make up 1 gal.

of 50 to 55 s for 1-oz copper are obtained from cupric chloride-sodium chloride systems operated at 130°F with conventional spray-etching equipment. Copper capacities are maintained at 15 to 20 oz/gal.

c. **Closed-loop Etching and Regeneration.** Two systems are in common use; chlorination and electrolytic regeneration.

1 **Chlorination.** Direct chlorination is a preferred technique for regeneration of cupric etchant because of its low cost, high rate, and efficiency. Recovery of copper and pollution control are also considered. The CuCl_2 -NaCl system (Table 8.2, No. 3) is a suitable formulation. Etching temperature is 130°F. Figure 8.3 illustrates the process. Oxygen can be used as an alternative to chlorine.

Chlorine, HCl, and NaCl solutions are automatically fed into the system as required. Sensing devices include redox and colorimeter (Cu oxidation state), density (Cu concentration), etch rate monitor, level sensor, and thermostats. Chlorination is a reliable and controllable chemical reaction. Additional considerations are as follows:

a. **Safety.** Use of chlorine gas requires adequate ventilation and leak detection equipment.

b. **Solution control.** An increase in pH will cause the copper colorimeter to give

erroneous readings that are due to turbidity in the solution. Excess NaCl at 18 to 20 oz/gal copper causes coprecipitation of salts when the solution is cooled. The reservoir contains 3 M HCl and 1 N HCl.

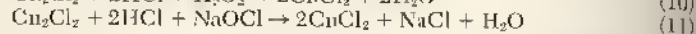
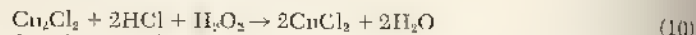
c. Maintain temperature at 125 to 130°F.

2 Electrolytic regeneration. The electrochemical reversal of the etching of copper by CuCl_2 reaction (4) is claimed to be effective and economical. Considerations of the system are given in the literature.²⁰ On a large scale, electrolytic regeneration requires a high investment in equipment and materials and high power consumption.

The etchant is a solution of cupric chloride and HCl (Table 8.2, No. 1). Etchant flows continuously between spray-etching machines and a plating tank. In the plating machine two processes take place simultaneously; copper is plated on the cathode and regeneration of the spent etchant takes place at the anodes.

d. Batch Regeneration

1 Oxidizing agent treatment. As shown by reaction (6), chemical regeneration is possible with O_2 . Similarly, it is possible with faster, more active oxidizing agents in batch processing as, for example, carried out by Clark²⁷ using H_2O_2 or sodium hypochlorite. Reactions (10) and (11) illustrate the oxidation processes:



The peroxide treatment must be slow and with vigorous stirring. Ventilation must be good to carry away the HCl fumes. Each gram of dissolved copper requires 1.75 mL of 30 percent H_2O_2 (100 volume) or 14 mL of 5 percent available Cl_2 sodium hypochlorite solution.

Common practice is to remove about half of the solution, add sufficient acid, and then slowly add, with vigorous stirring, the amount of oxidizing agent required. The solution volume is then adjusted. This method is justified for small operations only.

2 Solution replacement. Black and Cutler²⁸ have also shown that reaction (6) can be used in principle to regenerate a CuCl_2 etching solution. In their case a solution at 100°F in a spray etch operation was used. After etching 18×18 -in 1-oz copper boards they replace (by means of a proportioning pump) 3 L of the used etchant with 2 L HCl (37 percent) and 1 L H_2O . Solution replacement must be in keeping with the low copper capacity of the batch-type etchant, i.e., below 4 oz/gal copper.

Further data on etchant regeneration systems is available in the literature.¹⁷

e. Etching Practice. Any of the common techniques, immersion, huddle, splash, or spray etching, can be used with CuCl_2 . Spray, splash (paddle), and air huddle etching are the most effective. Increasing temperatures and agitation speed up the etch action, although higher temperatures cause excessive fumes with HCl-type etchants and attack on some resists. As with all other etchants, final rinsing must be thorough. A 5 percent HCl (by volume) dip followed by thorough water rinsing is recommended.

Spray equipment, tanks, and linings are most commonly made of PVC or other chemical-resistant plastic. Pottery and glass can be used but are subject to shock cracking. Temperature controls and safety devices must be used to prevent plastic-lined tanks from overheating and possibly burning. Quartz immersion heaters are suitable and are in common use. Titanium pump parts, heaters, racks, etc., are also very suitable. Stainless steel should not be used. Parts such as door hinges which do not come into direct contact with the etchant can be made of Monel.

f. Problems Common to Cupric Chloride Systems

1 Prolonged etching time. This is frequently due to low temperatures, insufficient agitation, or lack of chemical control of etch composition. When temperature and agitation are under control, an increase of etching time may result from a decrease in the cupric ion content. When time becomes prohibitive, renew or regenerate. Acid must also be added to clarify solutions. In regenerative systems the source of oxidation may be depleted.

2 Sludging. Sludging may occur in absence of the complexing agent. Another cause of sludging may be dilution with water.

3 Breakdown of photoresists. This happens when excess acid is present, particularly at elevated temperatures. Correction is readily made by either neutralizing with NaOH or replacing part of the solution with water. If the acid content and photoresist conditions have been used successfully in the past and lifting occurs, the source of the trouble may be in the cleaning prior to resist coating, inadequate exposure, or baking. Thick film photoresists are more resistant to breakdown.

4 Yellow residue on copper surface. The residue is generally cuprous hydroxide. It is water-insoluble and is left as a residue when boards are etched and alkali-cleaned. The solution in which the board is rinsed just prior to final water-spray rinsing should be acidic. A white precipitate may also be present. It is probably cuprous chloride, which can remain after etching in cupric chloride etchants that are low in Cl ion and acid. A 5 percent (by volume) HCl solution rinse prior to final water-spray rinsing will remove the precipitate.

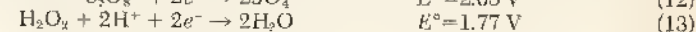
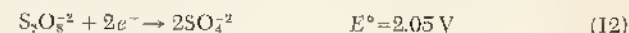
5 Waste disposal. Spent etchant can be sold for its copper content. Resins must be chemically treated to precipitate copper. Sewer disposal must not be used. See Chap. 9 on pollution control.

10. Persulfates Ammonium, sodium, and potassium persulfates modified by certain catalysts have been adopted for the etching of copper in printed circuit manufacturing. A regenerative-continuous system using ammonium persulfate* and a hatch system using sodium persulfate are in use.³¹ Wide use is made of persulfates as a copper pre-etch in Cu electroless and plating processes. Persulfate solutions will allow all common types of resists on printed boards including tin-lead (solder), tin, tin-nickel, screened inks and photosensitive films. Persulfate solutions are not suitable etchants for gold for reason of excess undercut and low etch factors.

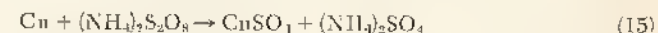
Formulations of ammonium persulfate catalyzed with mercuric chloride have etch rates comparable to those of the chloride etchants and were preferred for solder, print-and-etch, and Sn-Ni boards. Formulations with proprietary additives other than mercury catalysts are available³¹ and have been improved to give good etch factors. Regenerative systems have made possible higher copper capacities and constant etch rates.

In general, persulfate etchants are unstable and will exhibit decomposition, lower etch rates vs. copper content, and lower useful copper capacity. The use of persulfate etching systems has declined in recent years for reason of costs and developments in alkaline ammonia etchants.

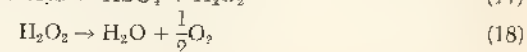
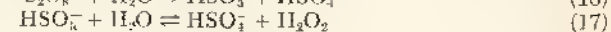
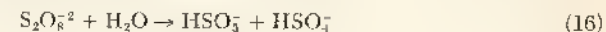
a. Chemistry. Ammonium, potassium, and sodium persulfates are stable salts of persulfuric acid ($\text{H}_2\text{S}_2\text{O}_8$). When these salts are dissolved in water, the persulfate ion ($\text{S}_2\text{O}_8^{2-}$) is formed. It is the most powerful oxidant of the commonly used peroxy compounds, as shown by the following standard oxidation potentials:



During copper etching, persulfate oxidizes metallic copper to cupric ion as shown in (15).



Persulfate solutions hydrolyze to form peroxy monosulfate ion (HSO_5^-) and, subsequently, hydrogen peroxide (H_2O_2) and O_2 .



*Cuper, FMC Chemicals, Princeton, N.J. U.S. Patent 3,399,090.

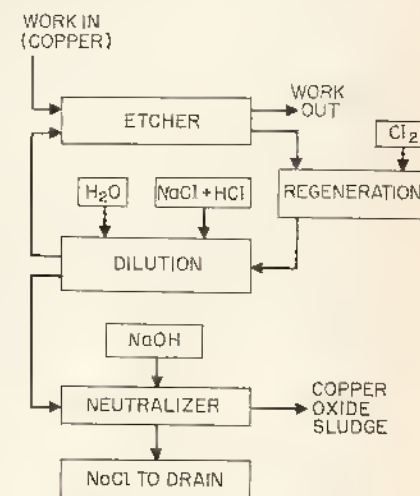


Fig. 8.3 Cupric chloride chlorination regeneration system.^{22,29}

This hydrolysis is acid-catalyzed; it accounts for the instability of acidic persulfate etching solutions.

Ammonium persulfate solution, normally made up at 20 percent, is acidic. Hydrolysis reactions and etchant use cause a reduction of the pH from 4 to 2. The persulfate concentration is lowered and hydrated cupric ammonium sulfate, $[\text{CuSO}_4 \cdot (\text{NH}_4)_2\text{SO}_4 \cdot 6\text{H}_2\text{O}]$ is formed. The precipitate may interfere with etching.

Solid persulfate compounds are stable and do not deteriorate if stored dry in closed containers. Solution composition is catalyzed by various agents including organic matter and transition metals (Fe, Cr, Cu, Pb, Ag, etc.). Materials for storage must be chosen carefully. Persulfates should not be mixed with reducing agents or oxidizable organics. Avoid contact with solvents, oils, grease, nitric acid, HCl, halide salt solutions, strong caustics, or ammoniacal solutions. Supplier literature should be consulted for additional details.²⁷

b. Properties and Control. Etchant compositions are given in Table 8.3.

Solution 1, used for organic resist and tin-nickel, is made up as follows:

Dissolve 2 lb ammonium persulfate for each gallon of deionized water preheated to about 140°F. Stir until dissolved. Do not store heated or in closed containers. The catalyst is made up by dissolving 27 g of mercuric chloride in 1 L of deionized water (3.6 oz/gal). One milliliter of the concentrate is added for each gallon of solution.³¹⁻³³ When the concentration is more than 1 mL/gal (5 ppm), a white film of Hg may appear on the copper or plated surface.

TABLE 8.3 Composition of Persulfate Etch Solutions

	1 ³¹	2 ³¹	3 ³¹
$(\text{NH}_4)_2\text{S}_2\text{O}_8$	2 lb/gal	2 lb/gal	
$\text{Na}_2\text{S}_2\text{O}_8$			3 lb/gal
HgCl ₂	5 ppm	5 ppm	15 ppm
X-134* (optional)	1 g/gal	1 g/gal	
H ₂ CO ₃		57 mL/gal	57 mL/gal

*TMC Chemical Corp., Princeton, N.J.

Composition 2 is used for solder-plated circuits. H₃PO₄ (1 to ½ percent by volume) is added to eliminate the incomplete etching of copper next to the solder conductors ("runoff") and minimize solder darkening.³⁴ Occasionally, improper plating conditions give rise to solder deposits of the wrong composition. In those cases the solder will darken during etching and may cause portions of the copper around the circuit to remain unetched. When the lead content is high, a white coating of lead sulfate forms; when tin is too high, a film of blue tin oxide forms. Conditions for electroplating must be corrected to give 60 percent tin and 40% lead. Processing temperatures range from 75 to 140°F, but the preferred temperature is about 115°F to balance etch rate, copper capacity, and solution decomposition. All common types of etching systems can be used, but spray etching in conveyorized equipment is preferred for final etching. Cooling coils are required to maintain a constant etching temperature and prevent runaway reactions.

The useful capacity of the etchant is about 7 oz/gal copper at 100 to 130°F. Above 5 oz of copper per gallon it is necessary to keep the solutions at 130°F to prevent salt crystallization. The etch rate of a solution containing 7 oz/gal of dissolved copper is 0.00027 in/min at 118°C.

Copper concentration is readily controlled by specific gravity or colorimetric measurements. Table 8.4 shows the dependence of specific gravity on copper concentration in an etchant containing 2 lb/gal of ammonium persulfate.

TABLE 8.4 Density of Ammonium Persulfate Etching Solutions

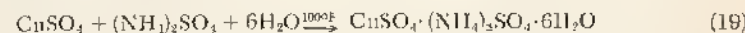
Copper concentration, oz/gal	Specific gravity at 120°F	Copper concentration, oz/gal	Specific gravity at 120°F
0	1.115	4	1.171
1	1.128	5	1.186
2	1.142	6	1.201
3	1.158		

c. Batch Operation. Composition 3 is used for batch-type spray etching.³¹ Sodium persulfate is often preferred because it has minimal disposal problems and somewhat higher copper capacity and etch rates. Etch rates vary through bath life and range from 0.0018 to 0.0006 in/min for copper content of 0 to 7 oz/gal. Prepared solutions must be aged for 16 to 72 h before etching when proprietary additives are used.

d. Continuous Regeneration.^{31,35,36} The method for regeneration of ammonium persulfate etchants involves etching, crystallization, and refertilization. The steps are as follows:

1. Ammonium persulfate etches copper to form CuSO_4 and $(\text{NH}_4)_2\text{SO}_4$ as shown in equation (15).

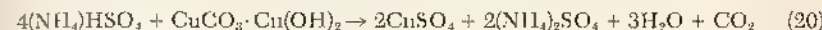
2. Spent etchant containing unreacted ammonium persulfate is pumped from the etching machine into a crystallizer, where it is chilled to 40°F. The products of reaction are precipitated as crystals.¹⁹



3. The solution is passed through filters to remove the precipitated crystals.

4. The filtered liquid, which is essentially ammonium persulfate, is pumped into a feeder tank, refertilized with new etchant, and heated. The cycle is repeated.

Control procedures include specific gravity, pH, flow rate, and additions of catalyst and phosphoric acid. Additions of basic copper carbonate, $\text{CuCO}_3 \cdot \text{Cu}(\text{OH})_2$, are made to neutralize bisulfate and control the pH at 1.2.³⁶ See equation (20).



Advantages of continuous ammonium persulfate etching and recovery are constant etch rate, high persulfate utilization, and less pollution. However, the necessary equipment is expensive and requires complex control, so application is generally limited to high-volume producers. Persulfate decomposition becomes a problem for intermittent low work loads.

e. Problems. 1 **Salt crystallization.** Salts crystallize out on the board and cause streaked etch action, damaged solder plate, and plugged spray nozzles or filters. When copper content is high, blue salts may precipitate out. White salts of ammonium persulfate also may precipitate out when the solution becomes highly concentrated because of excess evaporation of water. The etching temperature should be maintained at over 95°F to prevent crystallization. Plastic tube filters installed at the sump inlet will protect the spray nozzles and protect pump impellers.

2 **White films formed on solder surface.** This may occur normally or when the lead content in the solder plate is too high. It can be removed with a soft metal brush and by rinsing with 20 percent (by weight) solution of ammonium acetate or fluoroboric acid-thiourea.³⁷

3 **Black film on solder.** This condition can result when the solder alloy is high in tin. A dip in 20 percent oxalic acid helps, but brushing is generally required. Holes always remain darkened. When solder reflow or soldering is to follow, a coating of tin can be applied from one of several immersion tin solutions. In both cases adjust H₃PO₄ and solder plating conditions.

4 **Spontaneous decomposition of etch solution.** The decomposition is due to contaminated or overheated solutions. Also, when ammonium persulfate etchants are placed in etching machines or tanks that were previously used for ferric chloride or other etchants, it is possible for the persulfate to be catalyzed and to spontaneously decompose. Complete cleaning with acid solutions, thorough water rinsing, and final rinsing with ammonium persulfate will prevent that. Ammonium persulfate etchants are rather unstable, especially at higher temperatures. At about 150°F the solution decomposes quickly. Use soon after mixing.

5 **Disposal.** The exhausted etchant consists essentially of ammonium or sodium and copper sulfate with a pH of about 2. As with other etchants, direct sewer discharge is not allowed. Two methods are suggested here.

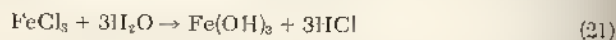
a Electrolytic deposition of the copper on the surface of passivated 300 series stainless steel. The spent etchant is acidified with H₂SO₄ prior to electrolysis. Once the copper has been removed, the remaining solution can be diluted, neutralized, and discarded. The copper can be removed from the cathode. Spent sodium persulfate can be treated with caustic soda.

b Addition of aluminum or iron machine turnings to a slightly acidified solution is

another practical but possibly more difficult means of removing the dissolved copper. The presence of chloride ions catalyzes the reactions with aluminum. The reaction, especially in the presence of chloride ions, will be violent, and considerable heat will be given off if the solutions are not diluted. Because of problems in handling, the electrodeposition method and scavenger service are recommended means of disposal. A guide to those methods and decomposition cost estimates are given in the literature.³¹ See also Chap. 9, Pollution Control.

11. Ferric Chloride Ferric chloride (FeCl_3) solutions find widespread use as etchants for copper, copper alloys, Ni-Fe alloys, and steel in printed circuit applications, electronics, photoengraving arts, and metal finishing. FeCl_3 is used with screen inks, photoresist, and gold patterns, but it cannot be used on solder- or tin-plated boards. Kovar and similar glass-to-metal sealing alloys are etched with FeCl_3 by using photoresist techniques. Typical applications include leads on solid-state microcircuit ceramic flat packages and fine-line metallization masks. At one time FeCl_3 was the most accepted and used etchant.³² That is because all the etchants were not commonly regenerated and FeCl_3 had a high tolerance for dissolved copper as well as being low in cost. Ferric chloride still remains as an attractive batch etchant because of those factors.

The composition of the etchant is principally FeCl_3 in water, with concentrations ranging from 28 to 42 percent by weight (see Table 8.5). Free acid is present because of the hydrolysis reaction.



This HCl is usually supplemented by additional amounts of HCl (up to 5 percent) to hold back the formation of insoluble precipitates of $\text{Fe}(\text{OH})_3$. Commercial formulations also contain wetting and antifoam agents. The effects of FeCl_3 concentration, dissolved copper content, temperature, and agitation on the rate and quality of etching have been reported in the literature.³³⁻⁴⁵

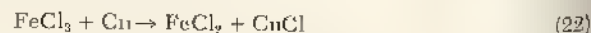
TABLE 8.5 Composition of FeCl_3 Solutions*

	Low strength	Optimum		High strength
Percent by weight	28	34	38	42
Specific gravity	1.275	1.353	1.402	1.450
Baumé	31.5	38	42	45
lb/gal	3.07	3.9	4.45	5.11
g/L	365	452	530	608
Molarity	2.25	2.79	3.27	3.75

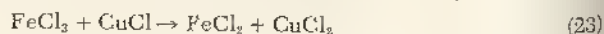
*Data taken at 68 to 77°F (20 to 25°C). Photoengraving FeCl_3 42° Baumé has 0.2 to 0.4% free HCl. Proprietary etchants contain up to 5% HCl.

Commercial availability includes lump $\text{FeCl}_3 \cdot 6\text{H}_2\text{O}$ and aqueous solutions with and without additives. Ferric chloride with additives has the advantage of low foaming (reduced odor and fuming), fast and even etching (due partly to added strong oxidizers and surface-wetting properties), and reduced iron hydroxide precipitate formation owing to slight additional acidity and chelating additives. The useful life of ferric chloride etchants and uniformity of etching rates have been greatly extended by proprietary solution manufacturers.

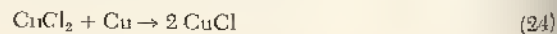
a. Chemistry. At the copper surface, ferric ion oxidizes copper to cuprous chloride with formation of green ferrous chloride.



In the body of the solution, cuprous chloride is further oxidized to CuCl_2 .



As cupric chloride builds up in the etching solution, a disproportionation reaction takes over.



Saubestre³³ has determined that, at 50 percent exhaustion (14 oz/gal copper) of 42° Baumé

FeCl_3 solution, 84 percent of the copper is etched according to reaction (23). In practice when a 42° Baumé solution contains 8 oz/gal (60 g/L) or more of dissolved copper, the etch time becomes longer than desired so that the etchant is replaced. The importance of reactions (22) to (24) in practical etching is evident in Fig. 8.4. The time to etch a given amount of copper stays relatively constant at first and then increases slowly up to 40 percent solution exhaustion (11 oz/gal of copper). Some commercial companies discard their solution before this point. Sometimes that action is taken unnecessarily because the conditions to compensate for changes in the solution are not well understood. Other problems related to poor etch factors—foaming, contamination, Baumé gravity changes, solid residues, and slow etch rates—are reasons why little-used solutions are discarded.

The etching time increases rapidly beyond 11 oz/gal of copper, but the solution still has

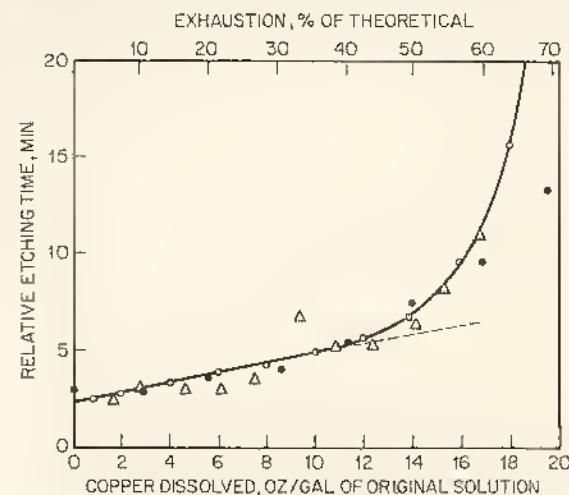


Fig. 8.4 Relative etching time vs. dissolved copper for a 42° Baumé FeCl_3 solution.³³

a large remaining capacity to dissolve copper. Many etching plants make use of partially depleted solutions by adding HCl at this point and using the solution to 70 percent of theoretical exhaustion in counterflow operations and preliminary plate cleaning. See regeneration section for other corrective action that can be taken to extend the useful life of the etchant.

A knowledge of the effects of varying conditions of composition, solution exhaustion, acidity, temperature, agitation, dilution, etc., can be used by the etcher to determine the best manner in which to use FeCl_3 .

b. Properties and Control. One of the most important considerations in the etching of copper is what criteria should be used to determine exhaustion and usefulness of partially depleted FeCl_3 . Because of the complex relations arising from FeCl_3 - CuCl_2 composition changes, evaporation, dilution, material introduction, etc., methods other than chemical analysis are not particularly helpful in determining the useful properties of partially exhausted etching solutions. Thus it is worthwhile to consider additional procedures to evaluate the performance of FeCl_3 as an etchant and the effect or desirability of changing operating conditions.

Two criteria are proposed.³⁶

Etching time (etch rate)

Degree of etching solution exhaustion

The etching time, or rate, is defined differently by each user, but it should include a measurement of the exact time required to dissolve a known weight of copper when all conditions are constant or specified, i.e., temperature, concentration, agitation, copper area exposed, solution volume, etc. If the quality of etch is also to be judged, a number of other conditions must be given: procedures for cleaning, positions in etch machine, size of board, etc.

The etching rate will vary with FeCl_3 concentration, temperature, chemical additives, and mode of agitation. A 0.0014-in nominal (1-oz) copper thickness should etch clean in 1 to 1½ min with fresh 42° Baumé FeCl_3 proprietary etchant at 100 ψ^2 in a high-pressure oscillating spray etcher. When the etch time exceeds 7 min under those conditions, the solution is considered spent.

The degree of solution exhaustion is commonly expressed as ounces of copper dissolved per gallon of original solution used. It is determined most accurately by chemical analysis for copper content. Various methods are described in the literature.¹

For less accurate comparisons, standard solutions can be made with, for example, 2, 4, 6, or 10 oz/gal of copper diluted 2:1 and used without the aid of an instrument to determine whether a bath is sufficiently depleted to be discarded. Greer¹⁶ has also described an electrochemical method for measurement and control of the etching strength of FeCl_3 .

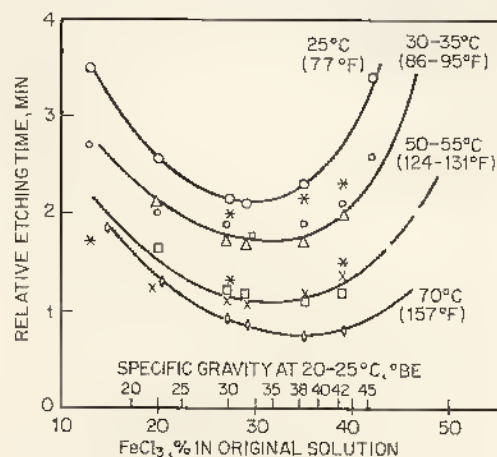


Fig. 8.5 Relative etching time vs. FeCl_3 concentration at various temperatures.³⁹

c. Effect of Variables

1 Concentration of reagents. Concentration of FeCl_3 (fresh solution) versus relative etch times at different temperatures is shown in Fig. 8.5.

The best concentration for minimum etching time with fresh solutions varies from 30 percent FeCl_3 at room temperature to 35 percent FeCl_3 at 70°C (158°F). As would be expected, an increase in the temperature causes an increase in the etch rate. The maximum temperature tolerable is that which can be used without harming the etch resist being used, and/or one at which excessive fumes are produced. Spray etching is generally operated at about 100 to 120°F.

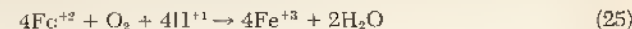
The advantage in etching rate shown in Fig. 8.5 is not retained beyond a certain level of copper. Beyond 8 oz/gal of copper, etch rates became intolerably high (Fig. 8.6). The 30 percent solutions are frequently used for reason of cost advantage and because solutions are discarded after a concentration of 8 oz/gal of copper is reached.²

2 Addition of HCl. Figure 8.6 illustrates how additional HCl improves the relative etching time for FeCl_3 solutions. Above 6 oz/gal of copper, the HCl depresses the formation of $\text{Fe}(\text{OH})_3$ and aids in the etching reactions. In this case, 19.3 fluid oz of 20° Baumé HCl was added per gallon of original 39 percent FeCl_3 . In effect, it increased the useful capacity by 2 oz/gal of dissolved copper.

3 Agitation. The type of solution agitation has a considerable effect on the etching time. The time required to etch copper in a still solution (immersion) is intolerably high from the start and rapidly increases, as does the dissolved copper. Time to etch a given amount of copper in a still bath which has 8 oz/gal (60 g/L) of dissolved copper is just about one-third more than that required for an agitated solution containing 16 oz/gal (120 g/L) of dissolved copper at the same temperature.

Etch action is increased considerably when air is used for agitation or is introduced with

a spray or splash operation. Increased etch rates are due in part to the oxidation of Fe^{+2} and Cu^{+1} to Fe^{+3} and Cu^{+2} .



Dark green to black precipitates form in this solution or on the work as it is being etched. The precipitate formation has been described as being due to a reaction between ferrous chloride and air. That is shown by reaction (27).

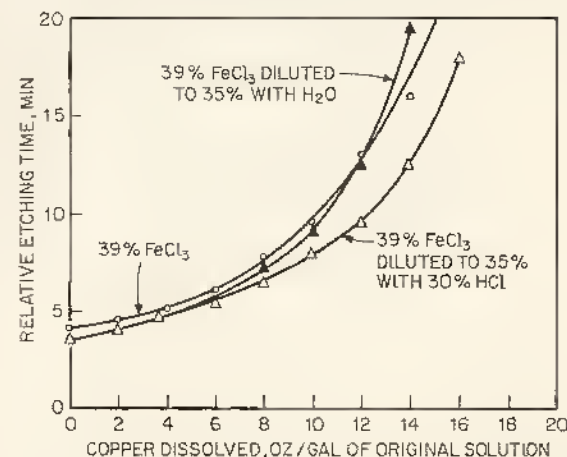
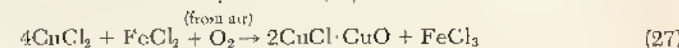
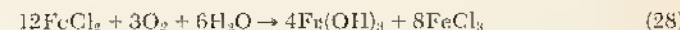


Fig. 8.6 Relative etching time vs. dissolved copper for 42° Baumé FeCl_3 diluted with H_2O and HCl (30%).³⁹

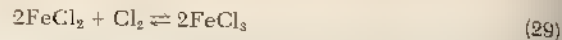
The double-salt crystals are only slightly soluble in hydrochloric acid. Another reaction product, hydrated ferric oxide, may also be formed on etchant surfaces. Reaction (28) shows the action.



Ferric hydroxide also forms when the acid concentration is depleted by stepwise hydrolysis. Hydrochloric acid is added in small amounts to suppress precipitation. Ammonium chloride is added occasionally to increase the etch rate.

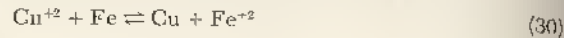
High etch factors are also obtained when air is introduced into the spray or other etching processes. Additional studies on control of undercutting of patterns are given in the literature.⁴²⁻⁵¹ Methods require a careful use of solution makeup and agitation of the etchants. Powderless etching methods are used to gain increased etch factors, and the ratio of depth of etch to lateral etch is increased by the use of additives. Examples of additives are thiurea and disulfides, which form gelatinous films on the lateral shoulders and sidewalls of etched characters.^{48,56} Phosphoric acid esters also have been proposed for this purpose.⁵⁷ The practical value of these agents is limited by the instability of the gelatinous film under the conditions of hard etching operations. The changeover to the newer technologies of thin clad laminates and additive and semiadditive systems has reduced the need for this approach.

d. Regeneration. FeCl_3 is not readily regenerated. Methods had been proposed^{1,27} but were not widely used at the time of writing. Regeneration systems are based upon the need to separate copper chloride from ferric chloride. When ferrous metals are etched, as in microelectronic applications, the dual method is practical. In such cases chlorine gas and HCl are used to regenerate ferric chloride from ferrous chloride.



Two methods are advanced as reasonable approaches for circuit boards. One method (Fig. 8.7) adds Cl_2 and recovers copper from the ferric etchant by crystallization. The addition of NH_4Cl (2 M) and cooling to ambient temperature permits crystallization of a chloride double salt $\text{CuCl}_2 \cdot 2\text{NH}_4\text{Cl} \cdot 2\text{H}_2\text{O}$.

Ferric chloride is maintained at 6 oz/gal of copper. A problem is the disposal of the copper precipitate. A second method for copper recovery involves passing spent etchant over iron plates that reduce dissolved copper to metal as in (30) and Fig. 8.8.



By-products are copper sludge and dilute iron chloride solutions.

e. Etching Problems

1 **Rapid decrease in the Baumé value.** This is generally due to introduction of water into the etcher sump. It is commonly encountered when automatic equipment includes a rinse cycle. Frequent determination of the Baumé and scheduled inspection for worn in-

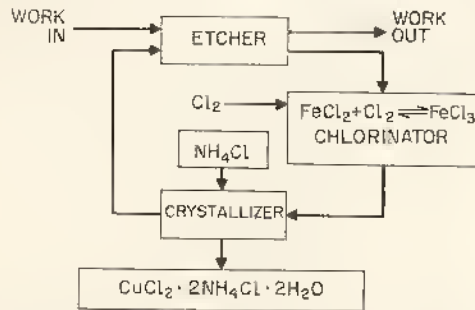


Fig. 8.7 Ferric chloride closed-loop regeneration and recovery process using recrystallization.⁵⁴

broken gaskets, plugged drains, and excessive water pressures must be standard procedure. Baumé gravity by itself is not recommended for etch-solution control, because additional factors are involved.

2 **Excessive sludge formation.** Excessive temperature causing HCl evaporation, or presence of materials that react with the free HCl or hard water—particularly hard water with a high carbonate content, are generally to be suspected.

When sludge is encountered, it can be dissolved by the addition of 1.2 mL of concentrated HCl for each gallon of etching solution at 42° Baumé. There is an etching-time advantage in diluting the etchant to 38 to 39° Baumé; and since HCl has the added advantage of reducing sludge, it would be preferred to water for this dilution. The operation must be approached with care, however. Since thin photo and etch resists are attacked, the fumes are toxic and corrosive.

3 **Insulating substrate contamination.** Water rinsing does not always easily remove the ferric chloride. Rinsing accompanied by a heavy spray is effective while the salts are in solution. Manufacturing processes and equipment must be such that clean rinsing is consistently attained. A 5 percent by volume solution of hydrochloric acid, oxalic acid, or EDTA is effective in removing iron and copper salts and is strongly recommended for use after the first rinse; it is followed by a vigorous final water spray. Simple conductivity devices can be installed in critical rinse tanks to indicate solution contamination. Salts absorbed on the laminar substrate are present as complex forms of FeCl_3 . Subsequent washing, hydrolysis, and drying converts these salts to insoluble forms of ferric oxide. Insulation resistance of the substrate readily degrades under such conditions.

Effective removal treatments involve fine pumice abrasion, a rinse of a 5 to 10 percent solution of HCl or oxalic acid, and a spray- or flowing-water rinse. The reader is referred to various studies on contamination of printed boards by FeCl_3 .⁵⁵

4 **Clogged drains.** Rinse tank drains, particularly small-opening hose types, easily become stopped up by precipitated salt formation (crust) on the walls. Large drain

openings and frequent cleaning with acids prevent the difficulty. Disposal of spent etchant into sewers with automatic systems to increase the pH or simultaneous caustic dumping results in heavy sludge formation. Sewer disposal of spent FeCl_3 is not allowed, and it may be illegal.

5 **Corrosion.** Steel, cast iron, and other metal pipes, decking, racks, roofs, drains, etc., are quickly attacked to complete degradation by FeCl_3 and its fumes. Carbon or glass pipes leading to tile-lined dilution sumps are effective drain systems. PVC, polyvinyl chloride, lined tanks, pipes, metal paints, etc., also offer good protection from the etchant. Epoxy and other plastics as coatings are useful. Titanium is suitable when strength and structural properties are required. Pump parts, cooling coils, etc., which are kept constantly in contact with the solution may be made of titanium. Quartz, carbon, and titanium heaters are often used. Disadvantages of quartz and carbon include cracking and poor heat transfer. Nickel- and cobalt-steel alloys may be utilized to a limited extent.

6 **Disposal.** Disposal of FeCl_3 is becoming an increasingly difficult problem as states and cities become more concerned about sewage effluents. In addition to the severe corrosion problem, metals in the solution, particularly copper, when dumped in natural waters, will disturb the bacterial growth required for sewage breakdown and will destroy plants and fish. Use of a local scavenger service is a possible solution to the disposal problem, but it is usually more costly and not the full answer. Refer to Chap. 9 on pollution control.

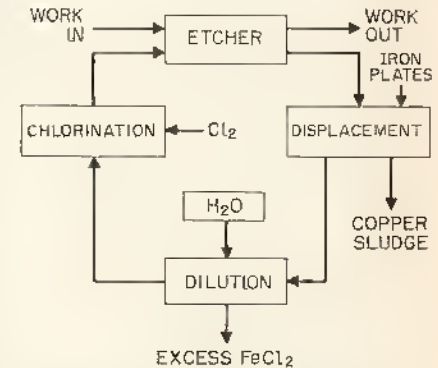
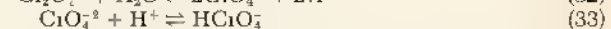
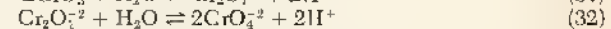
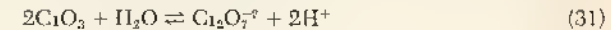


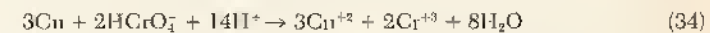
Fig. 8.8 Ferric chloride closed-loop regeneration and recovery by using iron displacement.⁵⁴

12. **Chromic-Sulfuric Acids** Chromic-sulfuric acid systems were the preferred etchant for solder- and tin-plated boards for many years. The availability of alkaline and modified ammonium persulfate etchants has reduced this preference. Also responsible are pollution restrictions and the fact that economic regenerative systems for chromic-sulfuric etchants are not available. In addition, the etch rate for the system is not constant, and the practical limit of dissolved copper is only 8 oz/gal.

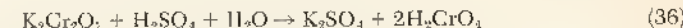
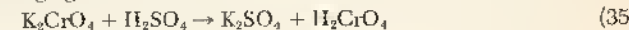
Although chromic acid etchants are strong oxidizing agents, solder is not attacked owing to the formation of insoluble lead sulfate. Undercut is generally less with solder-plated patterns as compared with gold and organic resists. Proprietary etchants are made by using chromic acid and sulfuric acid. Chromic acid and water form chromates according to the following reactions:



The reaction between copper and chromate can then be written as:



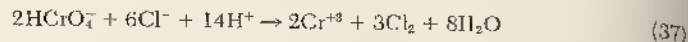
Potassium chromate (K_2CrO_4) and potassium dichromate ($\text{K}_2\text{Cr}_2\text{O}_7$) may also be used as starting materials. Equations (35) and (36) show the reaction with sulfuric acid to produce the hexavalent Cr^{+6} oxidizing agent.



Compositions of chromic acid etchants are given in Table 8.6.

Sulfuric acid is normally used as the source of hydrogen ions. Concentrations of 10 to 13 percent by volume are used to achieve optimum etchants⁵⁵ and maximum holding capacity for copper. The use of other acids has been considered, but problems are

observed: HNO_3 increases the etch rate but causes attack on solder plate; H_3PO_4 decreases the etch rate; acid fluorides attack titanium etcher parts and solder; and organic acids are degraded by chromates. Hydrochloric acid must not be added to this etchant because of poisonous chlorine gas formation:



Chromic-sulfuric mixtures etch copper slowly, and additives are necessary to increase the etch rate. For example, Na_2SO_4 is used in formulation 1 of Table 8.6 and iodine compounds are utilized in formulation 2.⁵⁶ Other additives reported are NaCl , AgNO_3 , and mercuric chloride.⁵⁷

TABLE 8.6 Composition of Chromic-Sulfuric Acid Etch Solutions*

	1 ⁵⁶	2 ⁵⁵
CrO_3	240 g/L	480 g/L
Na_2SO_4	40.5 g/L	
H_2SO_4 (96%)	180 g/L	31 ml/L
Copper		4.9 g/L

*Proprietary agents contain wetting, antifoaming, and chelating agents, and catalysts.

Other formulations are suitable not only for solder but also for tin-nickel-plated boards. Chromic acid etchant is suitable for use with gold-plated resists, screened vinyl lacquer, dry film, and liquid photoresists. However, there is a tendency for etchant mixtures to be aggressive on thin photoresists, particularly at high concentrations and temperature.

Chromic-sulfuric etchants with additions of NaCl are used for Kovar photoetching.⁵⁷ Microscopic cross sections of 0.004-in-thick Kovar foil etched in chromic-sulfuric acid at 100°F in a vigorous spray etcher showed nearly perfect flat walls with little or no undercut.

a. Properties and Control. Increase of temperature and agitation increases the etch rate considerably. However, air agitation of a solution in a tank results in a much slower etch rate than vibration, spray, and splash etching. Air also produces a mist or spray which is not only obnoxious but also poisonous. The fumes are very corrosive and cause disastrous results to plating solutions. Proprietary solutions contain antimist agents. It is a practical approach to consider the use of floating plastic chips to keep mist down on tank etchers. Rubber hoses, gloves, and electrical insulating materials will not hold up to this solution or its fumes.

A hydrometer is used to control the Baumé value of the solution. The density should be maintained at 30° Baumé at 68°F and 28° Baumé at 80°F. Water is used to decrease high-Baumé solutions. Colorimetric standards made up by adding known quantities of copper to fresh 30° Baumé etchant are used to estimate copper content.

b. Regeneration. Commercial systems for the regeneration of chromic-sulfuric acids are hampered by the complications of the processes, the corrosive nature of the products, and the handling hazards and pollution restrictions. Methods proposed include electrolytic deposition of copper⁵⁸ and electro dialysis.⁵⁹

c. Problems

1 Solder attack. The protective value of solder is dependent on the formation of insoluble compounds on the surface. If the sulfate content of the bath becomes very low, it is possible for the etchant to attack the solder. The presence in this etchant of chloride or nitrate ions from improper acid addition or water with exceptionally high chloride content can also cause attack. When deionized water is piped in, it is common practice to purge the line periodically with a chlorine solution in order to destroy algae and other organisms. If the chlorine solution is not completely rinsed away, it can be the source of chloride ion contamination in the etch of other solutions. The solder plate composition can also be the cause of etchant attack. When the lead content becomes very low, the sulfate film protection is diminished and protection is lost. Excess catalyst is also a cause.

2 Slow or no etching of copper. This can be caused by low chromic content (high or low Baumé gravity), low temperature, insufficient acid, or high copper content (above 5.5 oz/gal). The solution should be maintained as close as possible to 30° Baumé, pH about

0.1, and temperature between 80 and 90°F. The solution should be discarded when copper metal content exceeds 5.5 oz/gal.

3 Staining of board materials. The surfaces of dielectric substrates such as paper-based phenolics are attacked by chromic acid etchants. Removal is difficult or impossible, and the boards are generally rejected.

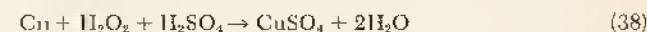
4 Disposal. Spent chromic acid etchants also present a serious disposal problem. Both chromic and copper ions are harmful to the biological processes of sewage-treatment plants. The selected method of disposal must be based on pollution standards and approved practice. See Chap. 9 on pollution control.

5 Safety hazards. Chromic acid is an extremely strong oxidizing agent. It will attack clothing, plastics, and many metals. Safety involves providing good ventilation to keep fumes out of room air, acid-protection gloves, aprons, and face and eye shields, and storing away from combustible materials. Dermatitis and nasal membrane damage are common ailments of chromium salt handlers. Rubber gloves, shoes, and aprons recommended by some manufacturers are preferably synthetic rubber such as Neoprene rather than natural rubber, which may be attacked and/or become brittle.

6 Equipment failure. Plastic piping and etcher structures may be catastrophically attacked or stressed, which will cause leaks to occur. Use only tested materials.

13. Other Etchants Solutions of hydrogen peroxide-sulfuric acid are used to clean and pre-etch copper in electroless copper and preplate processes.^{59,60} Stability of peroxide has been accomplished for these operations. The solutions would also be attractive as primary etchants because of their low cost, closed-loop regeneration, and copper recovery potential. Additional features are high copper capacity and reduced waste disposal problems. Reported processes show a wide variation in peroxide concentration.⁵⁹⁻⁶¹

The etching reaction is as follows:

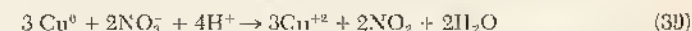


Technical problems in achieving application of this process for spray etching of copper are etching rate and peroxide stability. Titanium parts in spray etching machines may have to be replaced with 304C or 316L stainless steel. Cooling coils for control of the exothermic reaction must also be provided.

Nitric acid, although potentially an attractive etchant for copper, has not found successful application in printed circuit manufacture. It has a strong oxidizing capability which leads to rapid etching, a high dissolved copper capacity, and very soluble nonsludging products. It is readily available and economical.

That potential has not been realized because of control, fuming, and attack on resists and substrates. Copper etching is exothermic, and violent runaway reactions occur. The evolution of toxic and corrosive fumes requires exceptionally good ventilation.

a. Chemistry. The overall reaction in strong acids is shown by equation (39):



Solid film photoresists may hold up in this etchant. Etching is improved by addition of sulfamate or urea complexing compounds.⁶² The commercial value of nitric acid etchant will not be realized until the reactions can be controlled and substrates are resistant to attack.

Copper and its alloys can also be etched with $\text{NaCN-H}_2\text{O}_2$ and sodium chlorate- HCl solutions.⁶³ In addition, metals other than copper are etched with a variety of special etchants and techniques information about which is proprietary in many cases. Certain etching processes are described in the next sections. The reader is also referred to the literature for additional recommendations for etching systems.

b. Electroetching. This method of etching printed boards is limited by problems involved with continued current flow once breakthrough to substrate occurs.

The technology for such processes has been discussed in the literature.⁶⁴ Noteworthy in its utilization of this principle is the Morosi process.⁶⁵

MATERIALS FOR BOARD CONSTRUCTION AND THEIR PROCESSING

Printed board laminates are generally composed of copper bonded to a dielectric organic material. Metals other than copper and substrates of ceramic material are also used.

Organic dielectrics are thermosetting or thermoplastic resinous materials usually combined with a selected reinforcing filler. The need for overall stability, chemical resistance, and good dielectric properties has resulted in the major use of thermoset-reinforced materials for rigid and flexible printed boards. Thermoplastic materials are made with or without reinforcing fillers and are generally used for flexible circuit applications. One of the many factors in material selection is the effect of process solutions, etchants, and solvents.

Although the danger of solution and solvent contamination exists with organic dielectric systems, extensive tests and experience have shown that a highly reliable and electrically sound system can be manufactured. The type of substrate present may limit the choice of organic resist, stripper, etch solution, and conditions. Epoxies, other thermosetting plastics, ceramics, and glass clad with copper or other metals will not be adversely affected by higher operating temperatures, unless the time in solution is excessive. When the metal cladding does not completely cover the dielectric substrate, organic solvents and long periods in acids and alkalis should be avoided. Temperatures of hot cleaners will soften and deform thermoplastic materials.

In addition, the adhesives used in laminating the metal to the substrate can be softened, loosened, and even attacked by some solutions used. If the laminate lifts from the substrate during normal cleaning or etching, the material manufacturer should be consulted.

The substrate chemical resistance limits the etchants and other solutions which may be used. The chemical resistance of these materials will vary considerably even for a specific material (such as epoxy-glass laminate), since each manufacturer varies his formulation.

14. Thermosetting Laminates

a. Amino. Printed circuit laminate resins are derived as reaction products of formaldehyde and urea or melamine. They are hard, rigid, abrasion-resistant, and fairly resistant to compression; they resist deformation, withstand zero temperatures without becoming brittle, are excellent electrical insulators, and are unaffected by common organic solvents, greases and oils, and weak acids and alkalis. Operating temperatures are -70 to 210°F .

High mold shrinkage, age shrinking, cracking on wet-dry cycling, loss of physical strength, and electrical property changes limit the use of the material. Melamines are not resistant to strong oxidizing acids and strong alkalis. Weak sulfuric acid solutions show little effect on alpha-cellulose-filled melamine even on long exposure. Visible discoloration is evidenced when melamine laminates are boiled in 1 percent sodium hydroxide solutions or exposed to bleach. The common etchants used for printed circuit manufacturing can be used. Some discoloration and loss of surface properties may result in some cases.

Melamine printed circuit laminates can be hot-punched and cold-sheared. Cracking, however, is common with some types when cold sheared.

b. Epoxy. These thermosetting resins are the most commonly used plastics in printed board manufacture. They are combined most often with glass or paper, and they possess exceptionally good electrical, physical, and chemical properties. Properties can be varied over a wide range by changing resin type and/or filler. Most manufacturers have their own formulations. Manufacture with well-controlled conditions has shown that a high-quality and high-reliability product can be made on a consistent basis.

Chemical resistance is good. Strong acids and bases have no effect at normal temperatures even when exposure is well in excess of that required for printed board processing. Long-term exposure to strong alkalis, especially if heated, will result in swelling and some resin breakdown. Care should be taken when hot alkaline strippers are used.

Commercial organic resist strippers are, for the most part, not detrimental to laminates which are covered with copper. Damage may be substantial when boards are produced by the print-and-etch method, which results in exposure of the unclad substrate. Time in and temperature of strippers and vapor degreasers must be held to a minimum. The practice of leaving drilled boards with unplated holes in strippers, TCE, acetone, xylene, and other solvents for extended periods will result in swelling and delamination. Cracks around holes on both sides are evidence of excessive exposure to solvents or strong alkalis.

Flame-retardant grades are available for constant use at temperatures over 125°C . High-temperature-stabilized grades operate continuously at 200°C and are used for semiconductor chip and wire attachment applications.

Epoxy resins are attacked by concentrated sulfuric acid. This process, sometimes in combination with hydrofluoric acid, has been used for etchback of plated-through-hole multilayer boards.

c. Polyester. These thermosetting resins preceded the epoxies. To a considerable extent the base polyester resin in combination with glass has been replaced by the epoxies because of the polyester's lower dimensional stability and chemical resistance. Cost is lower than that of epoxies; and because its chemical, electrical, and physical properties are good, polyester is still used in applications from which epoxies are excluded because of cost.

The most important film-type polyester as far as printed circuit manufacturing is concerned is Mylar.* Copper-clad laminates of this material and similar films are readily available in a variety of thicknesses. It provides a very strong substrate. The thinner films tend to wrinkle when the circuit pattern is etched. This results in a need for side-frame-type supports. The high dielectric strength of Mylar makes it possible for use as very thin films having high electrical resistance breakdown.

Chemical resistance is generally good. Resistance to glacial acetic acid, 20 percent sulfuric acid, 10 percent sodium hydroxide, acetone, ethyl acetate, trichloroethylene, xylene, motor oil, silicone oil, asphalt, glyptal resin, and silicone resin is rated as excellent. "Good" chemical resistance rating is given for resistance to concentrated HCl, 35 percent HNO_3 , and 12 percent NH_4OH . "Poor" rating was given after exposure to chromic acid, MEK, and concentrated NH_4OH .

Concentrated sulfuric-chromic acid and hot NaOH are used as conditioners to render the surface hydrophilic for electroless plating processes.^{69,70} Hot concentrated sulfuric acid can be used to etch holes through Mylar by using copper as an etch resist. An acceptable technique is to coat double-sided copper-clad board with photoresist, expose and develop hole pattern, etch away copper with ferric chloride, rinse, and dry. The holes are then etched through with hot concentrated sulfuric acid. This is a dangerous operation which must be done under a very effective exhaust system and with adequate safety protection. The H_2SO_4 must be kept dry and free from water. Once etching is complete, thorough rinsing and neutralization are required to remove all traces of acid. High-velocity xylene or TCE spray after rinsing helps to remove webbing. Light abrasive vapor blasting is also effective in removal of the resist and cleaning out of the holes.

Common etchants can be used. Organic resist strippers can be used; but tests should be made on samples first to determine their effect, since they may be attacked by certain chlorinated solvents.

One major disadvantage of polyester material is that it can easily melt, scorch, delaminate, or warp during soldering, although drilling, punching, and shearing operations can readily be done with proper techniques. When drilling is done, care must be taken not to overheat and melt the resin. When the resin melts, it smears over the copper and causes plating, etching, soldering, and electrical problems. Service temperatures of Mylar film range from -51 to 150°C , and the melting point is between 250 and 265°C .

Mylar has been widely used for flat flexible wire harness, but it has been replaced in many flexible circuit applications by heat-resistant polyimide materials.

d. Phenolic. These thermosetting plastics are made by combining phenol and formaldehyde. They have high dielectric and impact strength, and the heat, acid and alkali, and water resistance of the resin is good. Phenolics are lightweight and are easily machined. Printed circuit laminates have been made since their inception by using the phenolics with various fillers. The sheets have been standardized by the National Electrical Manufacturing Association (NEMA) into various grades and classes each of which has its own set of electrical and physical parameters.

Copper-clad phenolic paper is often used for commercial electronics equipment because it is lower in cost than epoxy-glass laminates and yet is adequate for many applications. Phenolic materials will burn, but at a rate which varies with the filler used. Weak acids have little effect at room temperature. Phenolics are decomposed by concentrated oxidizing acids, acetic acid, and strong alkali solutions. Concentrated organic and reducing acids have little effect at room temperature. Weak alkalis cause slow degradation at room temperature. Hot strong alkali cleaners must be used very carefully with these materials.

*Registered trademark of E. I. du Pont de Nemours & Company.

Although phenolics are rated as fairly resistant to solvents, considerable care must be taken when resist strippers are used. Danger is lessened when the laminate is protected by copper on both sides and in the holes, or when no holes exist, since only the cut edges are exposed. On paint-and-etch boards, the stripping must be done very carefully and quickly to prevent attack on the exposed phenolic. Concentrated, hot chromic-sulfuric acid mixtures used for etching will attack phenolic. Ferric chloride is the safest etchant to use. Laminate manufacturers rate their materials suitable and resistant for use in ferric chloride, chromic acid, and ammonium persulfate for as long as 30 min. On solder-plated boards, alkaline etchant or ammonium persulfate used at room temperature or only slightly elevated temperatures is suitable. Standard commercial materials with paper filler are suitable for continuous use at temperatures up to 250°F (121°C). Glass, mica, or asbestos fillers may raise the temperature of use to as high as 500°F (260°C).

e. Silicone. These thermosetting plastics are higher in cost compared with the organic (carbon) family of plastics. Resins combined with glass fibers or other fillers are used in laminates suitable for temperatures greater than 600°F (315°C). Silicone laminates which are available commercially are suitable for continuous use at 250°C temperatures. Acceptance as printed circuit laminates has been slow.

Weak acids and alkalis have little or no effect on the materials; strong acids have some slight effect. Strong alkalis and organics will attack the silicones at a rate dependent on temperature, composition, etc. Organic strippers should be used with care. Ferric chloride, ammonium persulfate, and alkaline etchant can be used to etch copper on silicone substrates.

f. Teflon. Fluorocarbon thermosetting resins including, for example, FEP film and TFE- or FEP-impregnated glass cloth have found wide acceptance in industry. Applications include aerospace, aircraft, automotive, computer, communications, electrical appliance, and instrumentation. Copper-clad and unclad film and sheet are available for standard and flexible printed circuitry.

The excellent chemical resistance of fluorocarbon resins is widely known. Thermal stability up to 200°C is good. Soldering irons, resistance soldering, or solder preforms, as well as wave and flow soldering, can be employed without degradation or deformation.

Because of its superior chemical resistance, few chemical agents are available for etching or surface conditioning prior to plating. The most suitable are mixtures of alkali metals with liquid NH_3 or naphthalene. Halogenated hydrocarbons, with a preconditioning treatment, also are effective etchants at elevated temperature. Boards with plated-through holes in Teflon* can be made by using a preconditioning treatment with Na-liquid NH_3 .

g. Polyimides. Resins of this class, although thermoplastic, decompose rather than melt. Temperature and dimensional stability are outstanding; in the form of either rigid laminate or flexible film, the polyimides are recognized for high-temperature applications. The Kapton† (H film) temperature range is -269 to 400°C, and use can be continuous at 200°C. Polyimide copper circuitry finds use in silicon chip-tape packaging and standard chip and wire attachment. It is suitable for use in all of the common etchants, but it will be attacked by concentrated acids and bases. It is available with a Teflon overcoating which allows it to be heat-sealed or joined. Adhesives are available for use with it, and resistance to solder damage is good.

Other useful thermoplastics include diallyl phthalate (DAP) and such high-temperature space age materials as phenylsilane, polybenzimidazole, and diphenyl oxide.

15. Thermoplastic Laminates Laminates with reinforcements of glass or asbestos and thermoplastic resins find use in flexible circuit applications. Chemical resistance to these materials is generally less than that of the thermoset resins. In some cases solvents will dissolve or etch the thermoplastic resins. Materials are arbitrarily divided as follows:

1. Low cost and general utility: polyethylene, polypropylene, polycarbonate, and polyvinyl chloride.
2. Microwave strip-line: polystyrene, polyphenylene oxide, PTFE, polysulfone, and irradiated polyethylene.
3. High-temperature: polyimide and stabilized epoxy.

*Registered trademark of E. I. du Pont de Nemours & Company.

†Registered trade name of E. I. du Pont polyimide film.

Additional information on all materials discussed can be found in the literature^{71-73,81} and NEMA and military designations.

16. Metals

a. Copper (1-oz). Copper, on the various substrates described above, accounts for the major portion of the printed board cladding metal today. The trend has been toward thin clad, $\frac{1}{2}$ oz/ft² or less. This section contains metallographic cross sections of conductor circuit lines prepared by the methods described in earlier chapters and etched according to the procedures described under General Etching Considerations. Standard 1-oz copper material was used except in cases noted. Circuit boards were spray-etched under the conditions described. Every attempt was made to remove the boards as "etch through" was observed. Etchants were alkaline ammonia, cupric chloride, $FeCl_3$, ammonium persulfate, and chromic-sulfuric acids.



Fig. 8.9 Dry film photoresist print and etch over 0.002-in panel-plated copper. Conditions: alkaline ammonia, 125°F. (Original magnification, 300 \times .)



Fig. 8.10 Dry film photoresist print and etch over 0.002-in panel-plated copper. Conditions: cupric chloride, 120°F, 16 oz/gal dissolved copper. (Original magnification, 300 \times .)



Fig. 8.11 Solder plate. As-plated panel-plated copper 0.002 in, solder 0.001 in. Conditions: chromic-sulfuric acid, 120°F. (Original magnification, 300 \times .)



Fig. 8.12 Solder plate. Pattern-plated copper and solder, as-plated. Conditions: chromic-sulfuric acid, 120°F. (Original magnification, 300 \times .)

1 Dry film. Figures 8.9 and 8.10* show typical trace edges produced in alkaline and cupric chloride etchants.

2 Solder plate. Alkaline ammonia, ammonium persulfate, and chromic-sulfuric acid etchants on solder-plated copper circuitry show the typical etch structures in Figs. 8.11 and 8.12. The overhang of undercut copper and solder is ductile and difficult to remove. Hard brushing or scrubbing is not recommended because of the resultant scratches, wear, and exposure of copper. Fusion reflow of the solder plate is the most effective treatment and results in the overhang being fused over the copper edge. See Figs. 8.13

*Microsection material courtesy of Datalab, Santa Clara, Calif.

and 8.14. These etchants result in similar circuit line and pad edges after etching, and tin-lead reflow operation.

3 Solder over Tin-Nickel. This metal system shows increased reliability when compared with solder over copper boards.³ Typical trace edges for alkaline and ammonium persulfate etching are shown in Figs. 8.15 and 8.16. Solder reflow is effective in this case.



Fig. 8.13 Solder plate. Pattern-plated copper and lead-tin solder reflowed at 400°F. Conditions: alkaline ammonia etchant, 125°F. (Original magnification, 300×.)



Fig. 8.14 Solder plate. Pattern-plated copper and lead-tin solder reflowed at 400°F. Conditions: Chromic-sulfuric acid, 120°F. (Original magnification, 300×.)

4 Gold over Tin-Nickel and Tin-Nickel. Figure 8.17 shows a typical overhang produced on a panel-plated board when spray-etched with $FeCl_3$. Figure 8.18 shows a typical circuit trace edge after $FeCl_3$ etching. The board was pattern-plated with copper, tin-nickel, and gold. In actual practice the gold thickness may be reduced substantially (4 to 10 μ in). These results are obtained when boards are produced with tin-nickel (no gold) or when chromic acid etchant, which requires a longer etch time, is used. Typical examples of tin-nickel and gold over tin-nickel boards etched with ammonium persulfate are shown in Figs. 8.19 to 8.22. In these cases considerable undercutting was observed even though time in the etchant was not excessive.

5 Tin Plate. The results of alkaline and ammonium persulfate etching on tin over nickel are shown in Figs. 8.23 and 8.24. Similar results are achieved with Sn over Sn-Ni. Reflow is not feasible for bright tin plating.

6 Gold. Figures 8.25 and 8.26 are typical examples of cross sections of gold-plated printed circuit traces etched in either $FeCl_3$ or chromic-sulfuric acid.

Pattern- and panel-gold-plated boards etched in ammonium persulfate are shown in Figs. 8.27 and 8.28. As before, undercut

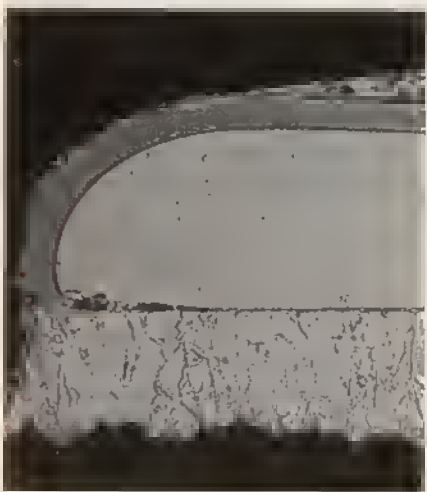


Fig. 8.15 Solder over tin-nickel. Pattern-plated copper, tin-nickel, and lead-tin alloy. Conditions: alkaline ammonia, 125°F. (Original magnification, 300×.)



Fig. 8.16 Solder over tin-nickel. Pattern-plated copper, tin-nickel, and lead-tin alloy. Conditions: ammonium persulfate, 120°F. (Original magnification, 300×.)



Fig. 8.17 Gold over tin-nickel. Panel-plated copper, 0.002 in; tin-nickel, 0.00015 in; gold, 0.000080 in. Conditions: 42° Baumé $FeCl_3$ (product of P. A. Hunt Co., N.J.), 100°F, 2 min. (Original magnification, 300×.)



Fig. 8.18 Gold over tin-nickel. Pattern-plated copper, 0.003 in; tin-nickel, 0.00018 in; gold, 0.00008 in. Conditions: 42° Baumé $FeCl_3$ (product of P. A. Hunt Co., N.J.), 100°F. (Original magnification, 300×.)

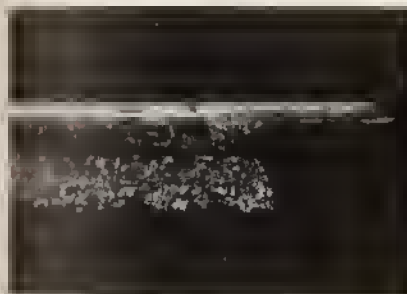


Fig. 8.19 Gold over tin-nickel. Panel-plated copper, tin-nickel, and gold. Conditions: ammonium persulfate, 120°F. (Original magnification, 300×.)



Fig. 8.20 Gold over tin-nickel. Pattern-plated copper, tin-nickel, and gold. Conditions: ammonium persulfate, 120°F. (Original magnification, 300×.)



Fig. 8.21 Tin-nickel. Panel-plated copper and tin-nickel. Conditions: ammonium persulfate, 120°F. (Original magnification, 300×.)



Fig. 8.22 Tin-nickel. Pattern-plated copper and tin-nickel. Conditions: ammonium persulfate, 120°F. (Original magnification, 300×.)



Fig. 8.23 Tin over nickel. Pattern-plated copper, nickel, and tin. Conditions: alkaline ammonia, 120°F. (Original magnification, 300×.)



Fig. 8.24 Tin over nickel. Pattern-plated copper, nickel, and tin. Conditions: ammonium persulfate, 120°F. (Original magnification, 300×.)

and gold overhang are pronounced in each case. As noted elsewhere, the overhang or featheredge must be removed in all cases to prevent operating electrical shorting. Ultrasonic agitation and cleaning after etching is effective provided a harder brittle alloy gold (Type II) has been used.

7 Thin Clad Copper. Etched printed boards with 1/4-oz copper-epoxy laminate show minimal overhang and slivers. Metallographic cross sections of solder mask over copper



Fig. 8.25 Gold. Panel-plated copper, 0.002 in. gold, 0.000100 in. Conditions: 42° Baumé FeCl_3 (product of P. A. Hunt Co., N.J.), 100°F. (Original magnification, 300×.)



Fig. 8.26 Gold. Pattern-plated copper, 0.0032 in. gold, 0.001 in. Conditions: 42° Baumé FeCl_3 (product of P. A. Hunt Co., N.J.), 100°F. (Original magnification, 300×.)



Fig. 8.27 Gold. Panel-plated copper and gold. Conditions: ammonium persulfate, 120°F. (Original magnification, 300×.)



Fig. 8.28 Gold. Pattern-plated copper and gold. Conditions: ammonium persulfate, 120°F. (Original magnification, 300×.)

and solder-plated boards trace-processed in alkaline etchant are shown in Figs. 8.29 and 8.30.

8 Semiadditive copper. A 0.00004- to 0.000100-in copper thickness with subsequent copper and tin plating shows no overhang or silver formation. Figures 8.31 and 8.32 illustrate results.

b. Aluminum. Aluminum-clad flexible circuits find use in microwave strip-line⁶⁵ and radiation-resistance applications. Aluminum and its alloys have good electrical conductivity, are light in weight, and can be plated, soldered, brazed, chem-milled, and anodized with reliable results. Special procedures are required. Laminate dielectrics include PPO⁶⁶, polyimide⁷⁰, epoxy-glass, and polyester.

Pre-cleaning for resist application includes nonetch alkaline soak, water rinsing, 5 to 10 s in chromic-sulfuric acid, rinsing and drying. Photoresist or screen inks are applied for circuit etching.

Ferric chloride (12 to 18° Baumé), NaOH (5 to 10 percent), inhibited HCl, and phosphoric acid-based acids are preferred etchants. Solutions of HCl-HF and FeCl_3 -HCl also etch aluminum. The cross section of a circuit trace is shown in Fig. 8.33.

For deep etching or chem-milling, screen-printed vinyl resists are the most durable, although photoresists also are suitable. Thorough water rinsing (spray deionized water preferred) follows etching. A dip in a 10 percent by volume HNO_3 solution will remove possible residues (dark copper smut) from the surface or edges of conductor lines which may be left on some alloys. Dilute chromic acid has also been used for this purpose. Thorough deionized water spray rinsing after etch is recommended. Pure aluminum is inert to the chromic-sulfuric acid etchant.

c. Nickel and Nickel-based Alloys. Nickel is finding increased usage as a metal cladding and as an electroplated or electroformed structure for printed wiring applications because of its suitable welding characteristics. Nichrome- and nickel-based magnetic alloys are further examples of this type of material requiring special etching techniques.

The standard methods described in Chap. 6 are readily adapted for image transfer on nickel-base materials. Etching is carried out by using FeCl_3 , 42° Baumé, at about 100°F. Other etchants include solutions made from 1 part HNO_3 , 1 part HCl, and 3 parts H_2O or 1 part HNO_3 , 4 parts HCl, and 1 part H_2O . The HNO_3 and HCl are 70 and 37 percent, respectively.

d. Stainless Steel. Alloys of stainless steel are used where there is need for resistive elements or high tensile strength. Flexible circuit dielectrics include Kapton* and epoxy-glass. Etching of the common 300 to 400 series can be carried out with the following solutions:

1. FeCl_3 (38 to 42° Baumé) with 3 percent HCl (optional).
2. 1 part HCl (70 percent by volume), 1 part HNO_3 (70 percent by volume), 1 to 3 parts by volume. Etch rate is about 0.003 in/min at 175°F. This etchant is useful for high 300 to 400 series.
3. FeCl_3 + HNO_3 solutions.
4. 100 parts HCl (37 percent) by weight, 6.5 parts HgNO_3 by weight, 100 parts H_2O by weight.

An example of a stainless-steel flexible printed circuit is shown in Fig. 8.34.

e. Silver. Silver, the least expensive of the precious metals, possesses electrical and thermal conductivity superior to all other metals and ductility second only to gold, high reflectivity of visible light, a high melting point, and relatively good chemical resistance. It finds wide uses through the electronics industry. Flexible circuit structures silver clad to epoxy-glass are used in electronic cameras and LED products.

Standard image-transfer methods are suitable. Pre-etch cleaning should include a rinse with dilute HNO_3 . Etching can be done in nitric acid (70 percent), but considerable heat is evolved by the rapid attack. Mixtures of nitric and sulfuric acids are effective etchants. With silver on brass or copper substrates a mixture of 1 part HNO_3 (70 percent) by volume and 19 parts H_2SO_4 (90 percent) by volume will dissolve the silver without adversely attacking the substrate. Water must be kept out of these solutions, and the solutions should be changed frequently to prevent immersion silver formation on the copper.

Controlled etching can be done with a solution containing chromic acid, 40 g; H_2SO_4 ,

*Kapton, product of E. I. du Pont de Nemours & Company.



Fig. 8.29 Thin clad copper. Solder mask over 1/4-oz copper. Conditions: alkaline ammonia, 120°F. (Original magnification, 300×.)



Fig. 8.30 Thin clad copper. Tin-lead plating over 1/4-oz copper. Conditions: alkaline ammonia, 120°F. (Original magnification, 300×.)



Fig. 8.31 Semiadditive copper. Pattern-plated copper and tin over electroless copper (0.000040-0.000100 in). Conditions: alkaline ammonia, 120°F. (Original magnification, 300×.)



Fig. 8.32 Semiadditive copper. Solder mask and pattern-plated copper over electroless copper (0.000040-0.000100 in). Conditions: alkaline ammonia, 120°F. (Original magnification, 300×.)



Fig. 8.33 Aluminum-clad flexible circuit. Aluminum 1100 series, 0.001 in on Pyralux (product of E. I. du Pont de Nemours & Company). Conditions: ferric chloride, 100°F. (Original magnification, 300×.) (Courtesy Buckbee-Mears Company, Minn.)



Fig. 8.34 Stainless steel flexible circuit. 316 full hard stainless steel, 0.002 in on Pyralux (product of E. I. du Pont de Nemours & Company, Delaware). Conditions: ferric chloride, 100°F. (Original magnification, 300×.) (Courtesy Buckbee-Mears Company, Minn.)

(96 percent), 20 mL; and H₂O, 2000 mL.¹⁷ This is followed by a rinse in 25 percent NH₄OH. Thin films of silver are etched in 55 percent (by weight) Fe(NO₃)₂ made up with H₂O or ethylene glycol. Solutions of an alkaline cyanide and H₂O₂ will also dissolve silver. Electrolytic etching is possible with 15 percent (by volume) HNO₃ (70 percent) at 2 V and a stainless-steel cathode.

Examples of other laminates include beryllium-copper and brass on polyimide. Etchants are identical with those for copper, Fig. 8.35.

UNDERCUT, ETCH FACTOR, AND OVERHANG

During etching, as the depth of etch proceeds vertically, the sidewalls tend to etch sideways. This produces an undercut action. The degree to which it occurs is known as the etch factor. The etch factor is defined as the ratio of depth to side attack (Fig. 8.36).

In practice, controlled spray etching vertical to the copper surfaces with selected etchants leads to high etch factors. Immersion etching generally results in lower etch factors. As suggested earlier, fine-line etching with a minimum of undercut is best carried out by removing the board from the etching machine exactly at the time of completion (etch through). The undercutting action must be considered and compensated for or designed into the artwork, especially for panel-plated boards, thick metal cladding, and high-definition and fine-line patterns. The etch factor can be minimized on metal parts by putting the resist pattern in register on both sides and etching the two sides together.

Addition agents, referred to in the photoengraving arts as powderless etching and banking agents, promote protective films on the bare sidewalls.¹⁴⁸ Other references on fine-line etching and undercutting may be found in the literature.⁴¹⁻⁵¹

Undercut and overhang for plated resists are illustrated in Fig. 8.37. Overhang is especially dangerous in a high-vibration or airborne system, since it may fall loose as metallic slivers and cause electrical shorting. Removal after etching is by soft-brass brushing, ultrasonic agitation, rinsing, adhesive-tape application (impractical and also subjects metal to unnecessary peel forces), or, in the case of solder-plated resists, by the preferred method of fusion.

EQUIPMENT AND TECHNIQUES

Etching techniques and the equipment used today have evolved from four basic methods: immersion etching, bubble etching, splash (paddle) etching, and spray etching. Spray etching is the commonest method, since it is fast, well suited to high production, and capable of very fine line definition.

17. Immersion Etching Deep-tank, or immersion, etching processing represents the simplest method of etching. Pieces are immersed in the solution until etching is complete. Immersion etch techniques require long process times, an intolerable factor in large-quantity production.

Heating the etch solution and agitation of some form speed up the action. Uniform etch action is attained when heat and stirring action are uniform over the entire surface area. The method is particularly suitable for laboratory etching of small or prototype boards.



Fig. 8.35 Beryllium-copper flexible circuit. Beryllium-copper 0.0014 in on Pyralux (product of E. I. du Pont de Nemours & Company). Conditions: ferric chloride, 100°F. (Original magnification, 300×.) (Courtesy Buckbee-Mears Company, Minn.)

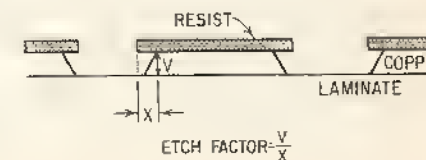


Fig. 8.36 Etch factor in printed board etching.

18. Bubble Etching Bubble etching is a modified form of immersion etching. Air is bubbled through the solution and past the work being etched. Air passing through the solution has two functions: (1) It ensures the availability of fresh etchant at the surface and rinses away dissolved metal. (2) It provides O_2 , which increases the effectiveness of the etchant by providing additional oxidizing power and regenerating the etchant. This is more thoroughly described under the particular etching solutions.

Air pressures up to 2 psi are used. A 0.5-in PVC pipe leading down and across the bottom of the tank, capped at the end and drilled with $\frac{1}{2}$ -in holes randomly spaced on the underside of the pipe at about a 45° angle works satisfactorily. Larger tanks utilize plastic filter cartridges which spread the air throughout the tank. Adequate ventilation is a requirement. Chromic-sulfuric acid etchant used on sulfur-plated copper boards is perhaps the greatest application of bubble etching. Ammonium persulfate may also be used, but cooling coils are required.

In operation, cleaned boards are placed in a rack about 0.5 in apart. Provision must be made for holding them down and yet allowing the solution to flow freely across the surfaces. In some cases the rack is merely a rod placed through a hole at the top and bottom of panels, with spacing made by a bead insulator between each pair of panels. Close-fitting pieces of rubber tubing can be used to secure the ends of the rods. During

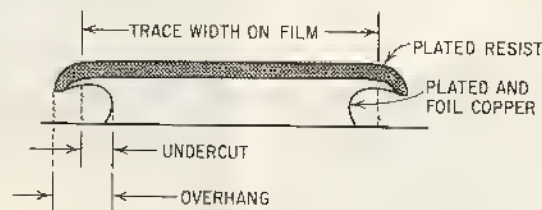


Fig. 8.37 Undercut and overhang for a plated-metal resist pattern.

etching the operator takes the rack out and turns it end for end. This results in more even etching.

Adjustable metal rod or strap racks so designed as to allow end-for-end rotation can be made of titanium. Plastic may also be used. More uniform etching is attained when rack position also is changed during the etch cycle.

Since boards are heavy, racks have to be rugged and built of materials not affected by the etching solutions. PVC has been found to be a very suitable plastic because of its chemical resistance when used below $175^\circ F$ and when not subjected to harmful organic solvents.

Water immersion into a triple-cascade flowing tank works well. Spray rinsing is not effective on racked panels. After thorough rinsing and neutralizing, unstacking can be done while boards are submerged in the last cold-water rinse and blown dry. Alternatively, the entire rack may be alcohol-rinsed and blown dry.

19. Splash Etching Paddle or splash etching offers distinct advantages over bubble or still etching with regard to evenness of etch and minimization of undercut. It has, however, the limitation of being able to handle only a few pieces at one time even when double paddles are used. It is widely used in the photoengraving arts.

A large-volume reservoir is built into the bottom of the tank to minimize solution replacement. The etch solution is picked up in a cup attached to a motor-driven shaft and is thrown by centrifugal force toward boards being etched. Speed of shaft rotation and cup design (shape, volume) determine the amount and force of solution coming in contact with the board. Parallel-speed adjustments are available; they allow tuning for the best action on a variety of metals by different etchants.

A variety of paddle construction materials such as stainless steels, hard rubber, plastic, and ceramic are available. Until recently the main tank was cast ceramic. Ventilation and exhaust fans are highly recommended. Ferric chloride, cupric chloride, and chromic-sulfuric acids may be used in this type of etcher. Ammonium persulfate is not normally used in paddle etching because of the danger of overheating, although cooling coils could be installed.

Disadvantages of slowness, even when construction is designed for double-sided etching, accounts for gradual obsolescence of this method in favor of more automatic and faster etching machines. The quality of etching, undercut, and evenness of etch is generally good. As with several other etching machines, boards may have to be turned and tilted during etching to produce a uniformly etched pattern. Figure 8.38 shows splash-type etcher design.

20. Spray Etching Spray techniques include single- and double-sided etching with either horizontal or vertical positioning of the work.

The fact that spray techniques yield high etch factors and short etching times is well known. The conditions of high impact on the surface, solution replenishment, and oxidizing power are factors contributing to that effect. As in all etching procedures, however, the highest definition (fine-line patterns) or use of thicker copper foils requires (1) a control of the undercutting action by taking care not to prolong the etching action, (2) compensation in artwork designs, and (3) etchant selection.

Spray-etching machines have evolved simultaneously with the chemical-resistant metals and plastics essential to their construction. Materials of the PVC type in sheet and cast form are used extensively in present machine constructions. Support rods, heater sheaths, molding coils, nuts and bolts, shafts, and other items requiring greater strength, heat resistance, or conductivity are most commonly made of titanium. That metal is suitable for constant immersion in all solutions of alkaline-ammonia, cupric chloride, ferric chloride, ammonium persulfate, and dilute and concentrated nitric and sulfuric acids. Sulfuric acid-hydrogen peroxide etchants would require stainless-steel materials. Polycarbonate plastics and Hastelloy C alloys are also commonly used. Polyethylene filter cartridges are used in certain machines and are suitable for use with common etchants.

Main parts, spray nozzles, and other items requiring good dimensional stability along with chemical resistance at somewhat elevated temperatures are made economically from polycarbonate resins. It is very important that organic solvents, oils, and chemical resist strippers be kept away from these plastics, which might be dissolved or swelled by the contact.

Another important caution is that PVC will distort when heated. Distortion will occur with temperatures as low as $130^\circ F$. PVC pipes used to transport hot solutions may expand and stretch to the point that moving parts in certain spray systems become bound up and unworkable.

A simple etching machine has a box-type chamber with a lid at the top and sump below. The solution is pumped from the sump and discharged out of the spray nozzle toward the board surface. In that way fresh solution is constantly impinged against the surface, which results in rapid etching. Evenness of etch is dependent upon uniformity of spray pattern, force, drainage, and factors peculiar to the material being etched, as well as the pattern configuration. Double-sided etching machines are well suited to production.

Both sides of the board must be at equal distance from the spray nozzles for double-sided etching. Also, the pressure distributed to the nozzles should be equal on both sides. If it is not, front-to-back etching will be uneven.

The intake line to the pump should have a filter on it to prevent crystallized salts or undissolved pieces of metal from getting into the pump and causing damage or plugging up nozzles. A 5 percent HCl solution is effective in removing dried spots of ferric chloride or chromic acid from the machine surfaces. Ammonium hydroxide and water are effective on alkaline etchant residues. A heat variance of 5 percent should be expected on production equipment, but closer control can be attained by controlling exhaust and having a better heating system. Spray patterns and pressure also vary with nozzle and pump wear. Maintenance and repair are relatively easy, since machines are well built and are simple in construction.

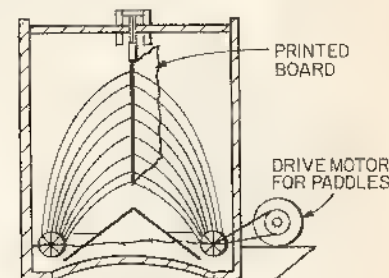


Fig. 8.38 Splash etcher design.

Pressure gauges and regulators for each nozzle bank are available and are essential for even etching. Also, rotary racks which vary the position of the article being etched so that runoff effect is minimized can be used. Modifications to standard, off-the-shelf models can be made by the use of well-positioned, matching spray nozzles and provisions for direct-temperature-readout thermostats which regulate current to heaters.

Ammoniated alkaline etchants, if allowed to crystallize excessively, may cause problems with plugged nozzles, damage, and excessive wear of working parts of the machines.

a. Vertical Etching. Vertical etching is carried out by placing panels in a rack that is similar to a drawer frame and which seats tightly when lowered into the spray-box area. Design of a vertical spray etcher is shown in Fig. 8.39.

More even etch is obtained when the nozzles are oscillated during the etch cycle. That is evidenced by the pattern of the spray breaking through the metal cladding (copper) sooner than at areas where the spray is less direct. By increasing the number of nozzles, the effect may be lessened, but it is not completely eliminated. A combination of nozzle movement and oscillation up and down or sideways, with a large number of nozzles with broad, solid patterns, should provide the best attainable results. Always provide maintenance.

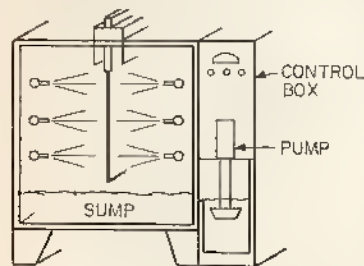


Fig. 8.39 Vertical spray etcher design.

Machines of this type do not have built-in rinsing or neutralizing capability. A low-expense effective rinsing setup for these machines is a narrow tap water spray-only tank into which the entire drawer frame (rack) can be easily transferred. The tank should be so laid out that the rack need not be turned for entry. An acid or other suitable neutralizing solution can also be located in the area.

Although shorter etching times are possible than with the bubble etchers, only a few boards or panels can be done at one time. As compared with batch bubble etching, this slows down the total output considerably.

When sprays are mechanized, it is very common for wear or loosening of locking devices to result in slippage and consequent loss of nozzle movement. Spare seals, O rings, packings, hardware, spray nozzles, pipe fittings, tubing, motor brushes, and fuses should be available for maintenance.

Most vertical etching machines are filled with etchant from the top. An all-plastic transfer pump should be used to transport etchant into the sump.

b. Automatic Vertical Etching. The automatic vertical etching machine is designed with racks which are loaded outside the machine and then placed into the machine. A continuously turning screw or other mechanism carries the rack through the etch chamber, where it is single- or double-side sprayed by oscillating banks of spray nozzles. As the travel continues, the rack goes through a water-spray-rinse chamber and out of the machine. Time in etch is controlled by a feed screw. Built-in heaters and cooling coils are available. Control of pressure to each bank of spray nozzles and on-off valves provide additional versatility.

The etchant sump has a larger volume of etchant than drawer-type vertical etchers have, which allows more boards to be etched before the etchant is spent. Production rates are higher than with drawer-type units. Potential problems of this machine should be pointed out.

1. Rinse-water cycle may cause dilution of the etchant.
2. The lead screw used to feed the loaded racks has considerable wear on it and has a short life.
3. The feed screw will be damaged if the loaded racks are not placed correctly into the machine and therefore cause jamming.

4. Adequate ventilation is required, since etchant fumes come out of the etcher when racks are entering or leaving.

Fine-line etching is attainable when spray nozzles, pressure, speed, and other variables are optimized. Setting the machine time with two complete cycles and turning the racks around between cycles also increases etching uniformity.

c. Horizontal Etching. The single-sided variety of horizontal etching was used for many years in the photoengraving arts prior to its application to printed circuit manufacture. The principal use was for fine-line and dot-printing plates. The principles of the upward vigorous spray and the use of additives to reduce undercut and increase etch factors were developed in the photoengraving trade and are now of use in printed board processing.

Double-sided horizontal etchers are generally preferred in printed circuit manufacturing, since the majority of the boards are two-sided. The etcher is available with a drawer-type holding rack, and this type also has a built-in sink at the left of the etch chamber. Boards are placed in the rack, which is pushed into the etch chamber. The etch operation is carried out and automatically pushes the rack into the rinse area after a timed cycle. Etching is done from independently controlled spray-nozzle banks at the top and bottom. Desirable features in horizontal etching machines include the following:

1. Double-sided etch capability.
 2. Single-side (face down) etching for very finest line work.
 3. Built-in rinse area and automatic control; top and bottom water spray which flushes both sides at once to avoid drying of etchant on board or overetching. Handling is eliminated.
 4. Positive drawer seals from sink to prevent water from getting into rinse area during etching.
 5. Sealed-in etch chamber. This is preferred because it requires no venting even when heated to 100°F.
 6. Automatic cooling coils that prevent overheating and maintain constant temperature (for persulfates).
 7. Independent gauges, pressure valves, and switching that allow compensation for differences between top and bottom etching.
 8. Oscillating action and a large number of spray nozzles that result in even etching over the entire area.
 9. Built-in transfer pump with intake going down into the etchant sump and output located in rinse tank. This allows filling or emptying without spillage, damage, or hazard. The pump should be electrically reversible for both filling and emptying.
 10. Minutes and seconds timer switches that provide careful control of etch time.
 11. Titanium heaters which are not subject to shock cracking.
 12. Electrical controls that are well protected physically and electrically.
 13. Accessibility to all parts of machine.
 14. Simple yet effective mechanical linkages.
 15. Filters at intake side of spray pumps. These prevent solids from damaging or clogging pump and spray nozzles.
 16. Screen insert tray which allows placing thin metal sheet on it for etching. Two screens are sometimes desired.
 17. Structural soundness which allows moving or shipping yet permits easy dismantling for repair.
 18. Chemical and temperature stability of construction materials.
- Limitations and/or undesirable features found in these machines include:
1. Maximum size of panel which can be handled may be smaller than required.
 2. Units with built-in rinse cycle remove mass of etchant but do not neutralize board.
 3. Initial cost of machines having all or most of the desirable features mentioned is high.
 4. Although etch times for 1-oz double-sided copper using fresh ferric chloride is as low as 1 min, production capabilities are limited.
 5. Drains for rinse water must be of sufficient size to allow quick drainage. Drains may not carry water away fast enough when short etch cycles are used in rapid succession. That causes trouble with the bottom spray rinsing and may result in overflowing into the etch-solution sump. The problem is emphasized when cooling water also empties into the rinse tank.
 6. Some machines have only a thermostat bulb for heat sensing. For that reason it is possible for the machine to be drained of etching solution with the heater still in

operation. That represents a dangerous situation. When built-in transfer pumps are used, danger of the occurrence can be minimized by electrically wiring the machine in such a way that, when the transfer pump circuit is on, the heater circuit is turned off. Even that approach is not 100 percent foolproof; for it is possible to turn the transfer pump circuit off and the heater circuit back on when the sump is drained. A heat-sensing device located near the heating element is desirable. Once such a device shuts the heater off, manual resetting is required to turn it back on.

d. **Automatic Horizontal Etching Machines.** The automatic machines should have the desirable features listed for horizontal drawer-type machines. In addition, they have the advantage of conveyORIZED loading and handling of board sizes up to 36 in wide and of indefinite length, as well as built-in complete rinsing and neutralizing. Board racking is not required. In operation, machines are loaded by laying boards flat on an open horizontal conveyor belt which carries them progressively through the etch chamber and subsequent rinses. Rollers on conveyor belts are alternately spaced to allow the bottom spray to reach the board. These machines are made for high-volume production. Large etchant sumps minimize downtime due to etchant replacement. Automatic equipment is available

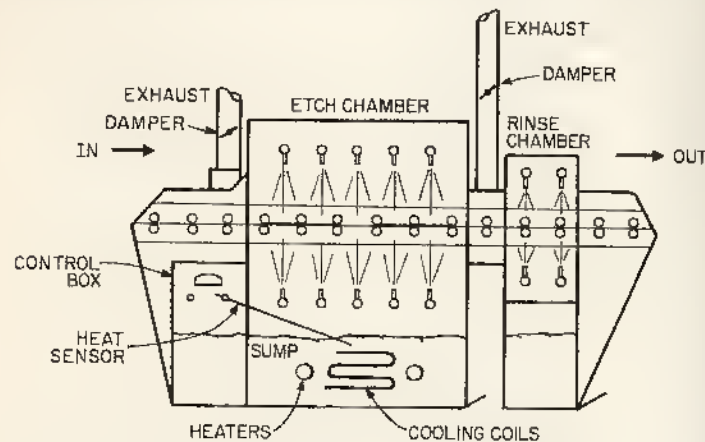


Fig. 8.40 Horizontal conveyORIZED spray etcher design.

for flow-through solution replenishment, which results in constant etch rates. Figure 8.40 is an example of an automatic horizontal spray-etching machine design.

The limitations and undesirable features of this type include the following:

1. The machines represent a considerable investment. Installation, plumbing, electrical work, and venting also are costly. Considerable floor space is required, and the machines are economical only when they can be kept busy.
2. Many moving parts, connections, gaskets and seals, hardware, etc., require competent maintenance.
3. Action of oscillating sprays, when included, can be stopped or get out of phase if connecting sleeves are not properly tightened. The sprays can also leak at the ends, which reduces spray pressure. Upper and lower sprays should hit the same area on opposite sides together. If they do not, small lightweight boards can be tilted or even cocked in the conveyor belt, which can result in considerable damage due to belt binding.
4. When etching solutions are heated, vapors come out on both sides. Exhaust covers and proper venting over those areas are required. Exhaust ducts must be provided when ammoniated alkaline etchants are used. Venting must be balanced with enough exhaust to keep fumes out of the etching room but without causing excessive loss of ammonia.
5. Upper and lower pump pressures should have gauges; otherwise, adjustment to compensate for the differences between top and bottom etch action becomes a constant trial-and-error procedure.
6. Water drains must have a good drop to allow drainage of rinse waters. Water lines to spray units must have vacuum-breaking valves in their connections to prevent the etchant from getting into water lines.

7. First water rinse coming out of etch chamber should be recirculated to keep etchant out of sewer and help reduce waste treatment. Water from this sump can be used to make up evaporation losses. Similarly, the neutralizer compartment may also drain into a recirculating sump. One or more final water rinse chambers can be included. These drain into sewer effluent treatment systems.

e. **Problems.** Most mechanical problems are due to wear of moving parts. The use of low-temperature PVC instead of PVDC or other materials suitable for high temperature is still a problem.

Sludging and crystallization causes wear on all mechanical parts. Keep solution hot, circulating, and in balance.

ETCHING AREA REQUIREMENTS

Basic functions must be provided in the etch area. These are:

1. Pre- and post-etch cleaning
2. Etching and rinsing
3. Neutralizing
4. Resist stripping
5. Regeneration
6. Solder fusing (optional)

21. **Equipment Selection** The following factors must be considered when equipment is being selected:

1. Maximum board size to be handled.
2. Quantities of board to be processed each day.
3. Space available for etching.
4. Type of etching to be done, e.g., fine-line, wide-line, or print-and-etch only.
5. Type of boards to be etched: metal-resist-plated, print-and-etch, etc. Maximum board size to be handled will determine the size of tanks for etching, rinsing, cleaning, and neutralizing. It will also govern the holding-rack sizes or conveyor. A 12 × 18-in board, for example, will require an etch rack or conveyor size larger than that in order to hold it during etching. Enough extra room for easy loading (racking) and unloading should also be provided. Spray etchers require considerable walk-around space.

The quantity of boards to be handled also will determine the type of etching equipment. The equipment must be able to complete the quantity easily in a normal workday, taking into consideration solution makeup time, machine maintenance, and actual man-hours doing the etching. When large quantities of one type of board are being processed, automatic equipment should be seriously considered. Elimination of as much hand labor as possible reduces costs, and final results are more consistent. An automatic spray etcher can be used when quantities permit. Two smaller machines may offer the advantage of having two different etch solutions available.

Storage racks, bins, tables, and shelves are to be provided in each area where boards may accumulate, as from plating, silk screening, photoresist coating, and etching. Separate receiving and holding racks are to be provided for work in such stages of completion as after etching and before and after stripping. Boards must be dried prior to being placed in racks for storage.

Adequate electric power, exhaust systems, storage areas, and safety provisions are particularly important in an etching facility.

22. **Facility** Floor construction should be chemical-resistant tile and mortar or acid-resistant epoxy-coated. An epoxy-fiberglass coating $\frac{1}{8}$ to $\frac{1}{4}$ in thick applied to cured and dry cement may be suitable. Subfloors of wood can be coated with similar plastic coatings. The slope of the floors must provide fast, free drainage of water rinses and other solutions. Tanks, pipes, ducts, etc., must be kept off the floor to allow free drainage and to keep them as dry as possible. Tanks supported well above the base floor, with wooden duckboard walkways are suitable. When pipes, etc., are run under the tank and aisles, they must be constructed of chemical-resistant materials or be well coated to prevent corrosive attack.

Spent solutions should never be sewerer directly. A three-compartment sump is mandatory. pH-controlled automatic systems are connected to the sump which feeds neutralizing chemicals in from a storage tank. Sodium hydroxide is commonly used to

neutralize acids. Ammonia should not be used. Spent chemicals containing metal ions or other objectionable materials should be treated according to accepted pollution control practice or hauled away for treatment elsewhere. Rinses must be treated to discharge as acceptable effluents; see Chap. 9.

REFERENCES

1. P. Fislei (ed.), "The Technology of Printed Circuits," Academic Press Inc., New York, 1959, chap. 7.
2. T. O. Schlabbach and D. D. Rider, "Printed and Integrated Circuitry," McGraw-Hill Book Company, New York, 1963.
3. E. Armstrong and E. F. Duffek, *Electronic Packaging and Production*, vol. 14, no. 10, pp. 125-130, October, 1974.
4. Electronic Industries Association, *Components Bull.*, 2, New York, June, 1959.
5. S. W. Chaikin, C. W. McClelland, J. Janney, and S. Landsman, *Ind. Eng. Chem.*, vol. 51, pp. 305-308, 1959.
6. R. W. Beattie, *ATE J.*, vol. 15, no. 3, pp. 272-276, 1959.
7. M. C. Mound, NEPCON Proceedings, *Electronic Packaging and Production*, Chicago, 1974.
8. W. S. DeForest, "Photoresist Materials and Processes," McGraw-Hill Book Company, New York, 1975.
9. L. Fullwood, California Circuits Association Symposium, San Francisco, Calif., May, 1971.
10. W. S. DeForest, California Circuits Association Symposium, pp. 135-150, November, 1971.
11. E. Lane, U.S. Patent 3,231,503, 1966.
12. L. J. Slominski, U.S. Patent 3,466,208, 1969.
13. N. I. Sax, "Dangerous Properties of Industrial Materials," rev. ed. Reinhold Publishing Corp., New York, 1957, p. 464.
14. D. J. Sykes, E. Papaconstantinou, and K. Mushi, U.S. Patent 3,868,485, February, 1975.
15. J. C. Pooi and C. F. Hsu, U.S. Patent 3,753,818, August, 1973.
16. F. King, U.S. Patent 3,705,061, 1972.
17. M. C. Mound, California Circuits Association Symposium, November, 1971.
18. R. W. Spilmeier, U.S. Patent 3,440,036, 1966.
19. "Solvent Extraction Technology," Center for Professional Advancement, Somerville, N.J., 1975.
20. G. Parikh, E. C. Cayer, and W. Willard, *Western Electric Engineer*, vol. XVI, no. 2, pp. 2-8, April, 1972. *Metal Finishing*, pp. 42,43, March, 1972.
21. L. Missel and F. D. Murphy, *Metal Finishing*, pp. 47-52, 58, December, 1969.
22. F. Cornman, Regenerative Cupric Chloride Copper Etchant, California Circuits Association meeting, 1973. *Electronic Packaging and Production*, pp. 43-46, January, 1974.
23. W. Thurnal, U.S. Patent 3,306,792, 1963.
24. P. D. Gam and L. H. Sharpe, U.S. Patent 2,964,453, 1960.
25. A. Jones and F. J. Haskins, U.S. Patent 3,083,129, 1963.
26. L. H. Sharpe and P. D. Gam, *Ind. Eng. Chem.*, vol. 51, pp. 293-298, 1959.
27. J. O. E. Clark, *Marconi Rev.*, vol. 24, no. 142, pp. 134-152, 1961.
28. O. D. Black and L. H. Critch, *Ind. Eng. Chem.*, vol. 50, pp. 1539-1540, 1958.
29. Anonymous, *Circuits Manufacturing Magazine*, pp. 55-57, April, 1972.
30. J. A. Dumkeberger and R. L. Erwin, Regenerated Cupric Chloride as an Etchant for Copper, *Tech. Bull.* 769, Chancet Corp., Pa., 1968.
31. "Etching Metals with Ammonium Persulfate," Food Machinery and Chemical Corporation, Princeton, N.J. Also see *Tech. Bulletins* 111, 52, 54, 55; 112, 4, 29, 1, 35.
32. *Ibid.*, *Tech Bulletin* 110.
33. P. A. Margulies and J. E. Krossbach, U.S. Patent 2,978,301, 1961.
34. B. J. Hegya and W. J. Tillis, U.S. Patent 3,476,624, 1969.
35. F. E. Capropreso, K. J. Badimeri, and B. J. Hogya, U.S. Patent 3,399,090, 1968.
36. J. S. C. Chiang, U.S. Patent 3,671,344, 1972.
37. W. J. Tillis, U.S. Patent 3,181,984, 1965.
38. E. B. Saubestre, *Ind. Eng. Chem.*, vol. 51, pp. 288-290, 1959.
39. W. F. Nekervis, "The Use of Ferric Chloride in the Etching of Copper," Dow Chemical Co., Midland, Mich., 1962.
40. B. M. Schaffert, "Ferric Chloride Etching of Copper for Photoengravings," Photo-Engravers Research, Inc., Columbus, Ohio, 1959.
41. L. E. Vaaler, *Photoengravers Bull.*, vol. 37, no. 10 pp. 41-44, 1948.
42. Anonymous, *Photoengravers Bull.*, vol. 36, no. 11, pp. 19-24, 1947.
43. G. MacDougall, *Patra J.*, vol. 8, pp. 35-40, 1944.
44. B. Biers and L. S. Ornstein, *J. Electrodepositors Tech. Soc.*, vol. 13, pp. 1-8, 1937.
45. Anonymous, *Phot. J.*, pp. 166-175, April, 1915.
46. W. N. Greer, *Plating*, vol. 48, pp. 1095-1098, 1961.
47. D. B. Ahutt, *Photoengravers Bull.*, vol. 47, no. 7, pp. 10-18, 1958.
48. P. M. Daugherty, U.S. Patent 3,144,368, August, 1964.
49. J. R. Snyers and J. Smit, *Plating*, vol. 48, pp. 789-793, 1961.
50. E. C. Jubb, *Plating*, vol. 51, pp. 311-316, 1964.
51. J. S. Hoffman, Methods of Producing Fine Lines in Printed Circuits, Proceedings of the 1960 Fall Meeting of the Institute of Printed Circuits, Inc., Chicago, Ill.
52. A. Shere and R. M. McClanahan, U.S. Patent 3,367,875, 1968.
53. T. D. Schlabbach and B. A. Diggory, *Electrochem. Tech.*, vol. 2, pp. 118-121, 1964.
54. R. T. Lindstrom, U.S. Patent 3,505,135, 1966.
55. S. Gewri, K. S. Indira, and B. T. Shemol, *Metal Finishing*, vol. 64, pp. 54-59, 1966.
56. L. J. Slominski, U.S. Patent 3,322,673, 1967.
57. R. D. Byers and J. G. Poor, U.S. Patent 3,351,555, 1967, 3,362,911, 1968.
58. F. Steward, California Circuits Association Symposium, San Francisco, Calif., October, 1974.
59. E. Lande, Institute of Metal Finishing Symposium, Bristol, England, September, 1964.
60. R. S. Baumish, U.S. Patent 3,483,050, 1969.
61. R. H. Jenkes, U.S. Patent 3,412,032, 1968.
62. H. F. Jones, C. L. Alderuccio, J. C. Pooi, and L. P. Gould, U.S. Patent 3,293,093, 1966.
63. British Patent 1,055,009, 1967.
64. P. H. Margulies and W. J. Tillis, U.S. Patent 3,137,600, 1964.
65. J. M. Haviland and D. J. Masters, U.S. Patent 3,367,874, 1968.
66. B. Miller and T. D. Schlabbach, *Ind. Eng. Chem. Process Design Develop.*, vol. 4, no. 3, pp. 259-263, July, 1965. U.S. Patent 3,361,674, 1968.
67. W. B. Kleiner, AES, *Technical Proceedings*, pp. 149, 1960.
68. S. L. Marosi, U.S. Patent 3,239,441, 1966.
69. J. S. Judge, J. R. Morrison, and D. E. Spiliotis, *J. Appl. Phys.*, vol. 36, pp. 948,949, 1965.
70. F. Saubestre, *Metal Finishing*, vol. 60, no. 6, pp. 67-73; no. 7, pp. 49-53; no. 8, pp. 45-49, 525; no. 9, pp. 59-63, 1962.
71. C. A. Harper (ed.), "Handbook of Electronic Packaging," McGraw-Hill Book Company, New York, 1969.
72. C. S. Harper (ed.), "Handbook of Materials and Processes for Electronics," McGraw-Hill Book Company, New York, 1970.
73. J. M. Bunsch, *Electronic Packaging and Production*, pp. 91-98, vol. 15, no. 5, May, 1975.
74. J. R. Cannizzaro, *Solid State Technology*, pp. 31-39, vol. 12, no. 11, November, 1969.
75. F. T. Mansini and R. G. Antiello, *Insulation*, pp. 58-61, March, 1968.
76. H. R. Johnson and J. W. Dini, *Insulation*, p. 31, August, 1975.
77. P. F. Kury, *J. Electrochem. Soc.*, vol. 103, p. 257, 1956.
78. P. M. Daugherty and H. C. Vangh, U.S. Patent 3,033,725, 1962; J. Braday, L. Elston, and W. Burrows, U.S. Patent 3,033,793, 1962; L. Elston, U.S. Patent 3,148,100, 1964. P. Borth and J. McKeone, U.S. Patent 3,340,195, 1967.
79. M. F. Good, Surface Preparation of Circuit Boards for Plating—A New Approach, Proceeding of the 1975 Spring meeting of the Institute of Printed Circuits, Inc. Washington, D.C.
80. H. Holden, California Circuits Association Symposium, Irvine, Calif., October, 1975.
81. G. F. Jacky, Paper presented at NepCon, Electronic Packaging and Production Conference, Anaheim, Calif., February 1976.

Chapter 9

Pollution Control

R. GLENN AFFLECK

Hewlett-Packard Company, Palo Alto, California

Introduction	9-1
Water Pollution Control	9-2
1. Need for Treatment	9-2
2. Industrial Waste Survey	9-2
3. Picking a Consultant	9-3
4. Getting the Most from Your Consultant	9-6
Typical Types of Waste Treatment	9-6
5. Process Changes	9-6
6. Treatment of Chemical Wastes	9-6
7. Contract Disposal of Sludge and Spent Chemicals	9-12
8. Safety Notes	9-13
9. Costs of Waste Water Treatment	9-14
Air Pollution Control	9-14
10. Dust Removal	9-14
11. Wet Scrubbers	9-14
12. Carbon Adsorption of Chlorinated Solvents	9-15
13. Equipment Costs	9-15
Dealing with Regulatory Agencies	9-16
Recycling, Recovery, and Conservation	9-17
14. Water Conservation	9-17
15. Gold Recovery	9-20
16. Recovery of Chlorinated Solvents	9-21
17. Copper Recovery	9-23
18. Palladium Recovery	9-23
Keeping Current	9-23
References	9-23

INTRODUCTION

Pollution control is an integral part of any printed circuit manufacturing operation. The public and the regulatory authorities are now aware of the potential effects on the environment of indiscriminate discharge of chemicals to the sewer, to the air, and to land.

At one time the limits on toxic material discharges were primarily related to the *immediate* detrimental effect on the sewage treatment plant or on personnel working in the sewer or near the exhaust stack of a facility. Now the chronic *long-term* effects on the air, water, and land are being evaluated. Discharge limits are continually being tightened as studies establish the relations between traces of toxic materials and the health of plants, animals, and, most importantly, human beings.

Regardless of whether plant managers are concerned about the potential effect of chemical discharges on the environment, they are interested in the law. Accelerated activity in enforcement of regulations will force all printed circuit shops to segregate and treat their waste discharges.

WATER POLLUTION CONTROL

1. Need for Treatment Nearly all municipal and industrial sewage treatment plants are biological. Bacteria in the sewage treatment plant feed on the dissolved and suspended materials carried by the combined discharges to the sewer. The bacteria biodegrade, or consume, those materials. Some of the toxic materials, such as copper, nickel, lead, and chlorinated hydrocarbons, are digested by the bacteria and end up in the sludge from the treatment plant. The rest of those toxic materials flow through the biological sanitary plant and enter the receiving waters.

If the concentration of toxic materials gets too high, it will kill the bacteria in the sewage treatment plant. That can result in an upset of the bacteriological process and major pollution of the receiving waters.

The deleterious effects of very minute amounts of toxic materials on the aquatic environment has been conclusively demonstrated by biological research.¹ Small plants and shellfish are very sensitive to particular toxic materials. Large fish and animals can tolerate somewhat higher toxic concentrations in water. It is interesting to note that humans can drink water in which fish cannot survive.

If the planned use of the receiving water is for swimming or for industrial cooling water, the concentrations of toxicants can be higher than if the water must sustain aquatic life. Rising treatment costs have pushed up the effective price of water, and recycling and reuse of water is becoming more economically attractive. This has added new standards that create a need for special treatment. For example, fluoride must be kept at or below drinking water standards if waste water is to be used for groundwater makeup. Also, any added heat must be removed in order to prevent a rise in groundwater temperature that would be experienced by all well owners. The results of all those requirements are rapidly making their way into enforced regulations. Table 9.1 shows some typical mid-1970s requirements for a discharge into a sanitary sewer and directly into a stream or ocean. Most of the parameters in the table are expressed in units of concentration: milligrams per liter. The Environmental Protection Agency (EPA) wants to change the standards to units related to production, i.e., milligrams per square meter of material processed, and current EPA requirements must be reviewed before proceeding with a disposal program. Some of the requirements were written into law before feasible technology was available to achieve the standards. Either standards will need to be relaxed or better treatment techniques will have to be developed if the legal requirements are to be met.

2. Industrial Waste Survey The first step in meeting any discharge requirements is to define problems. Waste problems are usually determined by making a survey of plant effluents. Often the person who makes the survey also designs any required treatment system. The survey can be made and the preliminary engineering can be done by a printed circuit process engineer, a member of your corporate staff, an equipment supplier, or an outside consultant.

In all cases, the printed circuit process engineer will need to collect the data necessary to design a waste treatment system. Water pollution is the principal problem associated with a printed circuit shop.

a. Data Required. Data usually required to design a waste water treatment facility include:

1. Rinse tank flow rates.

2. Process flowcharts showing the sequence in which parts travel through cleaners, plating tanks, treatment tanks, and rinses.
3. The volume, contents, and dumping schedule of each tank.
4. Incoming water quality.
5. A plant layout showing the location of all process tanks.
6. Prints and flowcharts of existing waste treatment systems. Some indication of the treatment effectiveness of the existing system may be helpful to the new-system designer.

TABLE 9.1 Typical Mid-1970s Effluent Requirements for Printed Circuits

Substance	Maximum concentration allowable	
	Discharge to sanitary sewer,* mg/L	Direct discharge to stream or ocean,† mg/L
Aluminum	10‡	0.5
Arsenic	0.1	
Barium	5.0	2.0
Beryllium	1.0	
Boron	1.0	
Cadmium	0.1	0.2
Chromium, hexavalent	1.0	0.1
Chromium, total	2.0	0.5
Cobalt	1.0	
Copper	1.0	0.5
Cyanide	1.0	0.1
Fluorides	1.5§	18
Hydrocarbon solvents	50	
Iron	5.0‡	1.0
Lead	0.5	0.1
Manganese	1.0	
Mercury	0.05	
Nickel	1.0	2.0
Phenols and derivatives	1.0	
Selenium	2.0	
Silver	5.0	0.05
Total suspended solids	300	50
Zinc	5.0	1.0
pH	5.5-10	6-9

*Except where noted, from Palo Alto, California, regulations for discharge to secondary (biological) sewage treatment plant.

†From EPA proposed discharge limits for the metal finishing industry, revised January 31, 1973.

‡From San Jose, California, regulations.

§Limit set at drinking water level so water could be used for groundwater recharge.

7. An itemized list of all major chemicals used, the consumption rate, and the disposal method.
8. The facility water and sewer flow rates and costs.
9. Flow rate and analyses of each plant discharge and the nature of the receiving stream, i.e., municipal sewer, stream, bay, etc.
10. Copies of all applicable government regulations and standards.

Part of the above data is necessary to size pipes, pumps, treatment tanks, sumps, and treatment chemical delivery systems. Justification of the need and type of equipment is calculated from part of the data. Plant layout and prints of the existing treatment system and facilities are used to prepare layouts of the new waste treatment system and interface it with existing treatment and process equipment. Figure 9.1 is a sample survey form used by one waste treatment consultant to ask potential clients for this information.

3. Picking a Consultant As a rule, an electronics company or an independent printed circuit fabricator will not have, within the company, the necessary talent to design its own waste treatment system and will instead turn to an outside consultant. Consultants can be either independent of any treatment equipment manufacturer or specialists in application of their company's equipment to pollution problems. When a consultant is to be chosen, the primary considerations are track record, staff qualifications, and specialization.

Salesman _____
Date _____

PRELIMINARY SURVEY: WASTE TREATMENT

Company name _____ Phone no. _____

Street address _____

City _____ State _____ Zip code _____

Persons to be contacted _____

A. Work treated

- Steel
- Die castings
- Others (explain) _____
- Aluminum
- Brass or bronze
- Plastic

B. Regulatory agencies

Give address of pertinent waste treatment regulatory agencies and representatives to be contacted. _____

Please furnish copy of existing regulations.

Has client been cited for pollution? Yes No

If yes, please furnish all citation correspondence.

C. Where is waste now discharged?

Sanitary System of _____

Storm Sewer _____

Watercourse _____

Sewer rental charge _____

D. Process rinsewater

Source _____ Cost/1000 Gal _____

Water consumption, total _____ Cooling _____

Process rinsing _____

If water supply is metered, please furnish average reading per 8-hour shift.

If incoming water analysis is available, please furnish.

E. Plant layout

1. A site plan is required and should show the following items:

- (a) Property boundaries
- (b) Physical location of plant
- (c) Topography
- (d) Location of sewer lines
- (e) Outside space available for location of waste-treatment equipment
- (f) Location of parking lots, driveways, walks, etc.

2. An equipment layout is required and should show the following items:

- (a) Arrangement of metal-finishing equipment (all tanks should be numbered to correspond to tank numbers used in process cycle information—See pages 3 & 4)
- (b) Location of sewer lines
- (c) Line drawings of exit water trenches and piping
- (d) Space available for location of waste-treatment equipment
- (e) Roof-truss height and plant power
- (f) Location of fume scrubbers, degreasers, and other waste-producing equipment

3. If future plant, please include dates for:

(a) Start of construction _____

(b) Occupancy _____

(c) Start of production _____

F. Mechanical finishing operations

- Buffing
- Polishing
- Wet tumbling
- BBI, Cyanide stripping
- Vibratory finishing
- Sand blasting
- Shot blasting
- Burnishing

Wet separation of solids from exhaust

Hydrofilter exhaust _____

Sludge handling _____

G. Phosphating operations

Method of application

- Spray Dip Roll coat
- Paint Oil Drawing compound

Final finish _____

H. Painting operations

Method of application

- Dip Spray Electrostatic
- Electropaint By hand

Is chromate dip used? _____

I. Rust-preventive operations

- Water-displacing dips (Solvent emulsions)
- Solvent oil rust preventives

Give tank volumes and dumping schedules _____

Line no. _____ Type _____ Production _____ Hours/Day _____

Days/Week _____

Racks/Hour _____ Barrels/Hour _____ Type barrel _____

Sq ft/Rack _____ Perfs. size _____

List tanks in correct sequence of use.
Indicate all tank volumes in gallons.
Include all rinse tanks and indicate when counterflowed... Also, include drag-out tanks and fume scrubbers.

Tank no.	Process	Volume, gallons	Estimated drag-out (gal/h)	Rinse flow (gal/h)	Contents in oz/gal or % (acid, alkali, cyanide, or chromium)	Name (if proprietary compound)	Dumping schedule

The consultant should be able to show you several operating treatment plants similar to the one proposed for your plant. Should there be a claim of great breakthroughs that have not yet been installed in an operating system, you share the risk that the system will not operate to your satisfaction. The consultant's staff should include professional engineers who have demonstrable experience with mechanical, electrical, and chemical engineering problems.

Usually consultants have specialized in a certain phase of pollution control. Some may design wet scrubbers; others handle biological waste water treatment systems. It is very important to pick a consultant with a specialty that matches your requirements. For cleaning the general exhaust from a printed circuit shop a consultant with experience in wet scrubbing of acid fumes and ammonia would be appropriate. Likewise, a consultant who specialized in treatment of metal-finishing wastes would be best suited to design heavy-metal removal systems for a printed circuit operation.

Fig. 9.1 Sample industrial waste treatment survey form.

4. Getting the Most from Your Consultant The best pollution control system cannot be designed if the consultant has only part of the data necessary. Your anticipated changes in types of processes and the growth of your operations are the most critical data inputs and the ones most often overlooked. Give your consultant the best and most complete information you can collect.

Some of your past and present effluent data may be politically sensitive. In that case give them to the consultant, who will be held responsible for keeping them confidential. In many cases working closely with regulatory agencies will be necessary to get approval of the proposed system. The consultant needs to know what can be repeated and what information is "classified." Plant operating personnel should be asked to review even the earliest proposals from the consultant. Often they will have some valid objections to the way systems are engineered. If they are permitted to make their comments early, the consultant will be able to make accommodating changes in the system before expensive detailed engineering has to be redone. That review, together with the accommodating changes that make sense, will help in gaining acceptance of the final system by the operating personnel. Without that acceptance the most perfectly engineered system will usually fail to function properly.

A consultant needs a contact in a client's facility who speaks waste treatment language and can get any requested data. A process engineer or a plant engineer who understands the production processes should be assigned as the consultant's contact. The engineer should have this liaison role as a first-priority project until after start-up of the system, and should then become your internal consultant on the system.

Most consultants include, in their services, training of the operator of the treatment system and some guides to maintenance of the equipment. The operator and maintenance supervisor will get much more from the consultant if they are involved very early with the design and construction of the system. As discharge standards get tighter, treatment systems get more and more complex and contain more and more equipment. By the time a system is installed and ready to operate, the selected operator and maintenance supervisor should understand the function and the role in the overall system of each piece of equipment.

TYPICAL TYPES OF WASTE TREATMENT

Given the need to dispose of a particular waste chemical from a printed circuit operation, there are at least three alternatives to choose from. You can change to a different process; you can install a treatment system; or you can have the waste chemical hauled away and disposed of by a contractor. The following discussion looks at each of the three alternatives.

5. Process Changes Often it will be easier to change a process than to install a special treatment system. A change to additive circuit processes involves a major change in waste treatment requirements by eliminating the etching operation and its associated copper-bearing waste. Replacement of 1-oz copper foil with $\frac{1}{4}$ -oz foil reduces the quantity of waste etchant proportionately. Switching from ammonium persulfate to sodium persulfate or peroxide-sulfuric acid for pre-etching in the electroless copper line can greatly simplify waste treatment. To switch from chromic-sulfuric etchant to an ammoniated etchant is to substitute a chrome treatment requirement for an ammonia treatment requirement.

6. Treatment of Chemical Wastes Table 9.2 lists the chemical wastes usually generated by printed circuit processes. Each type of waste in a printed circuit shop requires a specific set of chemical reactions to remove dissolved pollutants. Treatment is further complicated by the need for batch treatment of concentrated dumps and continuous treatment of flowing rinses. Sometimes the treatment processes are very much the same. A typical treatment system is the following:

1. Final pH control for all rinsewaters
2. Continuous and batch treatment of electroless copper rinses and batch dumps to precipitate copper
3. Continuous treatment of rinses containing heavy metals to lower the metal concentration to acceptable levels
4. Neutralization of acid and caustic batch dumps

5. Reduction and precipitation of dissolved chromium
 6. Precipitation of heavy metals from wastes as insoluble sludge
 7. Fluoride precipitation and removal as a sludge
 8. Special trap for chlorinated solvent; recovery and recycling of spent chlorinated solvents
 9. Haul-away of spent solvents and spent baths to an approved dump site or an incinerator if available
- Often, as with processes requiring heating a solution, economics encourages different

TABLE 9.2 Chemical Wastes Typical of Printed Circuit Manufacture

Process	Wastes generated
Board cleaning	Spent acid and alkaline solutions and rinses; some contain dissolved copper and ammonia
Electroless copper plating	Spent electroless copper solution and rinsewater Rinses and occasional bath dumps containing copper, nickel, lead, and fluoride
Etching	Spent etchant and rinsewater containing copper and perhaps ammonia, chromium, or iron depending on the type of etchant
Plating resist stripping and photoresist developing	Water rinses containing chlorinated hydrocarbons and batch dumps of spent chlorinated solvent
Laminate treatment for additive circuits	Chromium-bearing rinses and batch dumps
Multilayer etchback	Fluoride-bearing rinses and batch dumps
Carbon treating of plating solutions	Wash-up water bearing high quantities of suspended carbon and dissolved heavy metals.

schemes for batch and continuous treatment of specific chemicals. Without giving all the engineering details, the following treatment methods are typical of those needed by a printed circuit shop.

a. pH Control. Figure 9.2 shows a schematic of two typical pH control systems for

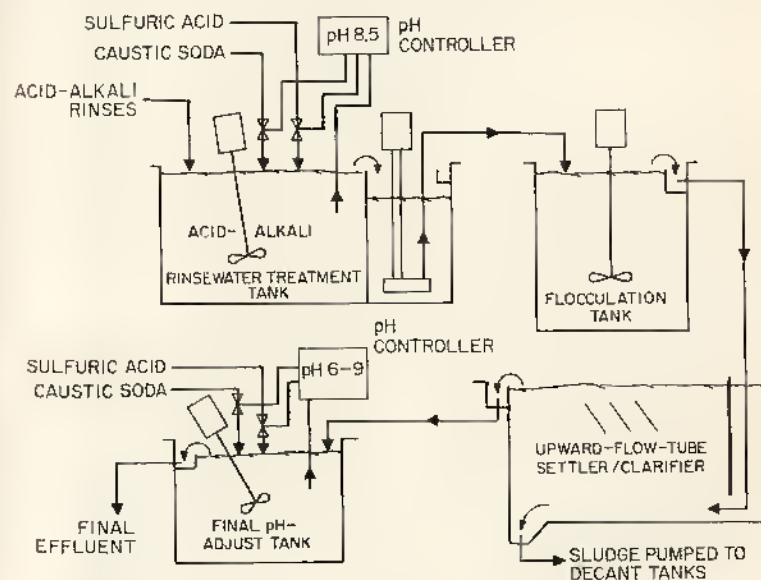


Fig. 9.2 pH control system for metal-bearing acid-alkali rinsewater.

metal-bearing acid-alkali rinsewater, one in the first and one in the last tank. Normally a printed circuit shop uses much more acid than it does base and there is little possibility of having a high-pH discharge. (pH 1.0 = strongly acid; pH 7.0 = neutral; pH 14.0 = strongly basic.) The pH control system can then be simplified to include only the addition of base to raise the pH of the effluent. Control of pH becomes an integral part of several other treatment systems, since many of the chemical reactions involved are pH-sensitive. The design of a good pH control system is much more complex than the simple diagram would indicate. Factors that must be considered include sizing and design of base and acid delivery systems, type of mixing, size of reaction tank, and position of pH probe relative to treatment chemical input point. Of particular importance to an understanding

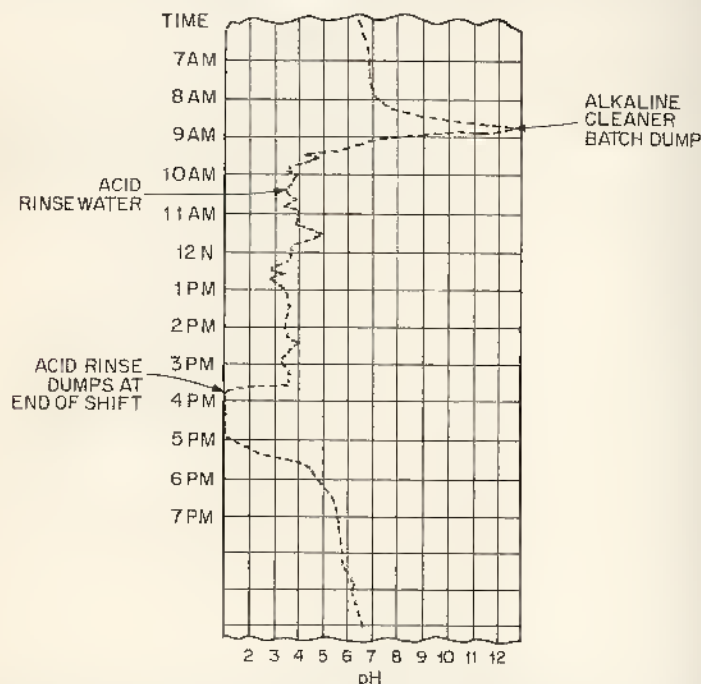


Fig. 9.3 Typical pH of printed circuit operation with no pH control system (8-h workday).

of a pH control system is the recognition that pH is a logarithmic function. To change the pH of an effluent from 2.0 to 3.0 takes 10 times the amount of base that is needed to change the pH from 3.0 to 4.0 and 100 times that needed to change the pH from 4.0 to 5.0. Figure 9.3 shows an example of a pH recorder output for the effluent from a printed circuit operation with no pH control.

b. Heavy-Metal Precipitation. Heavy metals include copper, nickel, lead, chromium and iron. If those metals are not complexed with ammonia, EDTA, or another agents, they can be precipitated as hydroxides by raising the pH. Figure 9.2 also shows the treatment steps necessary to remove those hydroxides. First the pH is adjusted to about 8.5. Then the overflow is transferred to a flocculation tank, where slow, steady agitation promotes the growth of precipitated solids (floc particles). The precipitated solids then settle out of solution in a clarifier, and the clarifier water is discharged through a monitoring and final pH-adjust tank. Following precipitation, sludge is pumped from the clarifier, concentrated, and hauled away. Figure 9.4 shows a sludge concentration system designed by Lancy Laboratories for this process. In this system a sludge decant tank receives insoluble settled solids from the clarifier or batch treatment tanks. The decant panels act as filters when they are lowered into the tanks; they remove clear water and produce a sludge that has been dewatered to a concentration of about 8 to 10 percent solids. That greatly

reduces the volume of sludge that must be hauled away. Other systems use a filter or a centrifuge to concentrate the sludge.

c. Electroless Copper Batch Treatment. The copper in an electroless copper bath is highly complexed and will not be removed by raising the pH and precipitation. Of the several treatment methods that have been reported, four are outlined here:

1. Addition of a small amount of catalyst to the spent solution will initiate plating out of the remaining copper in the form of a heavy copper sludge. Figure 9.5 shows the system schematically.
2. Heating the electroless copper bath will have the same effect as adding catalyst (see Fig. 9.5).

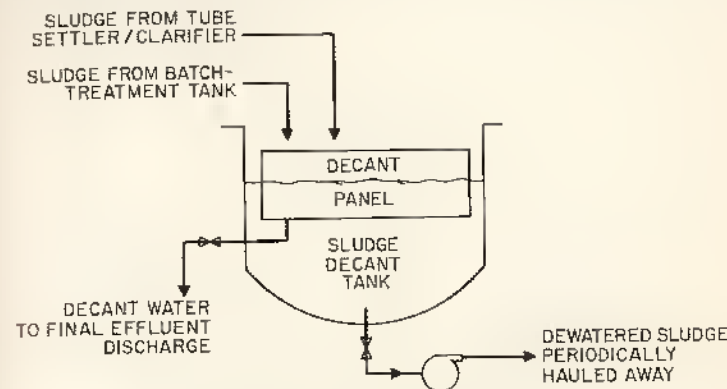


Fig. 9.4 Sludge decant and disposal system.

3. Lowering the pH of the electroless copper solution to a pH of 5 to 8 by adding acid also will drop the copper out of solution (see Fig. 9.5).
4. A combination of methods 2 and 3 followed by the addition of caustic and sodium hydrosulfite will precipitate copper oxide and reduce dissolved copper to a very low level (Figure 9.6).

The choice of system for electroless copper depends upon the size of your operation and how completely you need to remove the copper in order to meet your effluent limits.

d. Treatment of Ammonium Persulfate. Several methods for treatment of ammonium persulfate have been reported. Three of them are listed here:

1. Removal of copper by electrodeposition. The solution is heated in an agitated tank with lead anodes and stainless-steel cathodes. Prior to electrodeposition, the reduction of persulfate with sulfur dioxide will increase plating efficiencies. Figure 9.7 is a diagram of the process.

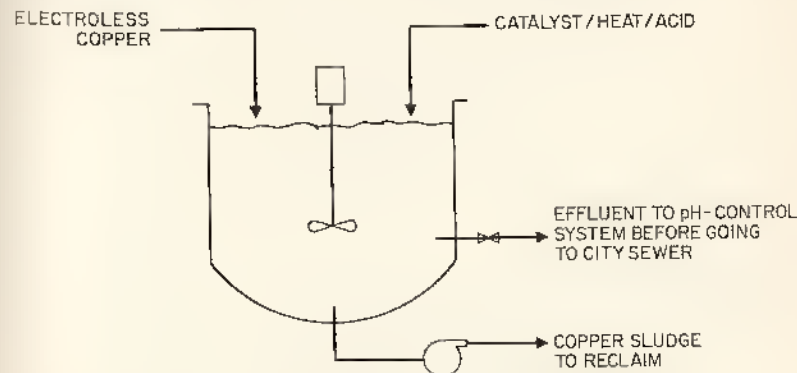


Fig. 9.5 Electroless copper batch treatment by catalyst/heat/acid addition.

2. Precipitation of copper by reaction with scrap aluminum and sodium chloride. Special care must be taken to prevent this exothermic reaction from getting out of control (Fig. 9.8).
3. Precipitation of copper hydroxide after removal of ammonia with alkali and heat. If the temperature of this reaction is near boiling, copper will precipitate as copper oxide rather than the hydroxide (Fig. 9.9).

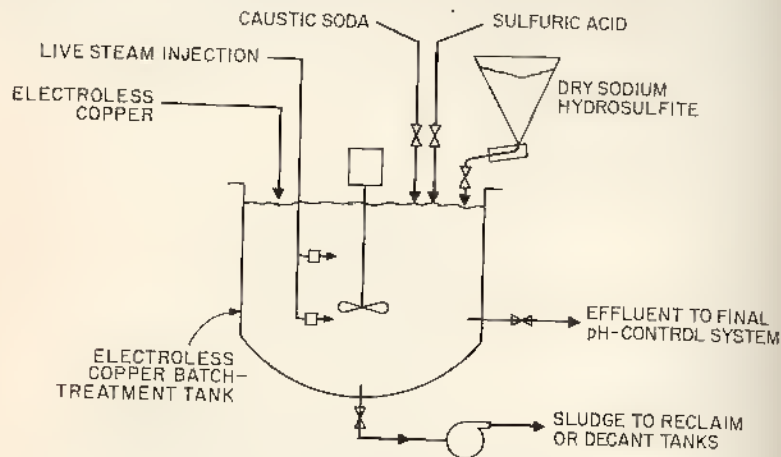


Fig. 9.6 Electroless copper batch treatment.

e. Ammoniacal Rinse Treatment. As mentioned above, in connection with treatment of ammonium persulfate, ammonia can be removed by heating a solution that has been made highly alkaline by the addition of caustic.

In the case of continuous rinses containing ammonia and complexed metals, it is not practical to heat all the water to near boiling. A system that has been demonstrated by Lancy Laboratories to be successful for continuous rinses involves the addition of phos-

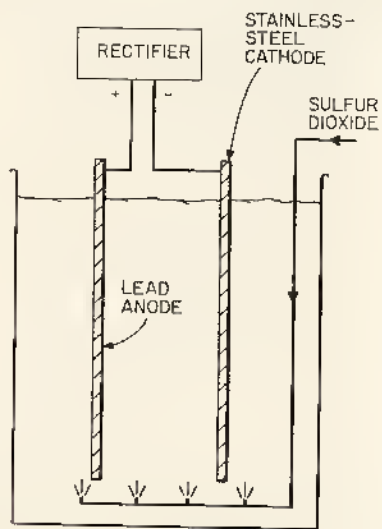


Fig. 9.7 Ammonium persulfate treatment by electrodeposition.

phoric acid prior to raising the pH of the solution to precipitate heavy-metal ammonium phosphates. Figure 9.10 shows such a system schematically. In the first tank, phosphoric acid is added, usually by sensing and adjusting pH, to assure the presence of phosphate ion. Lime is added to the second tank to raise the pH to above 10, which assures precipitation of heavy-metal (i.e., copper and nickel) ammonium phosphates, hydroxides, and other insoluble compounds. The precipitated compounds are subsequently removed from the treated rinsewater by flocculation and settling (see Fig. 9.2).

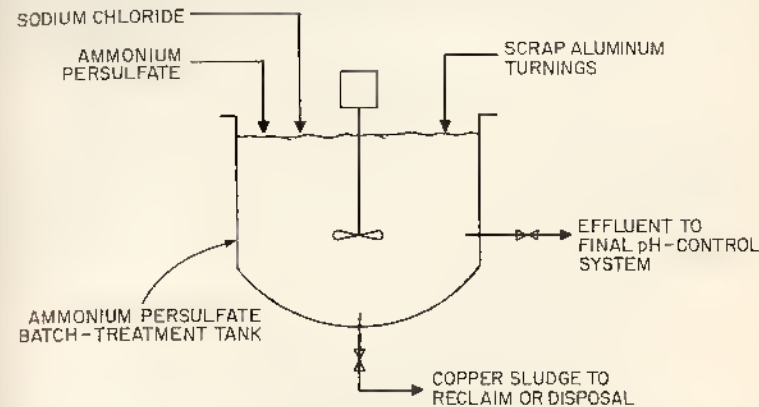


Fig. 9.8 Ammonium persulfate treatment with aluminum scrap.

f. Fluoride Treatment. Since several plating baths, some cleaners, and etchback solutions contain high fluoride concentrations, it may be necessary to remove some fluoride. Using lime for final pH control will precipitate some of the fluoride as calcium fluoride. But since even calcium fluoride is relatively soluble, it may be necessary to segregate the fluoride waste streams and put in a separate treatment system using lime and a solids removal clarifier, filter, or centrifuge. Figure 9.11 shows one such scheme.

g. Chromium Treatment. Chromium is usually present in epoxy pretreatment baths for additive circuits, in some chemical solutions for cleaning multilayer boards, and in

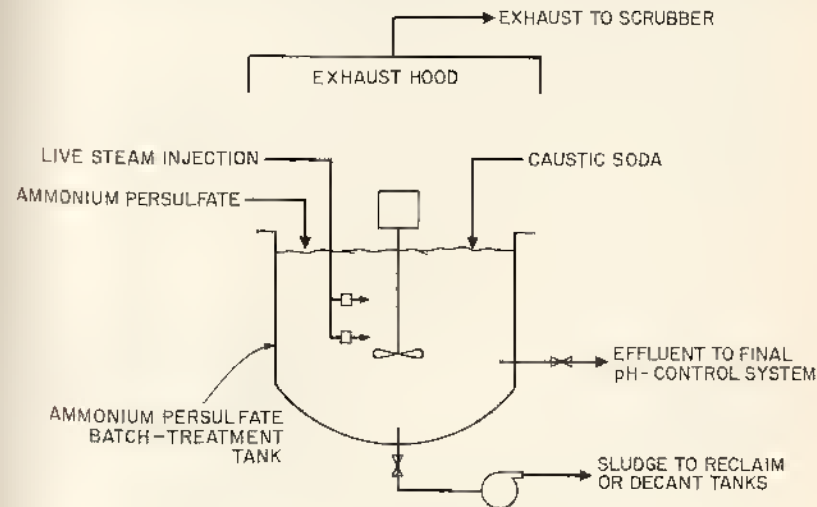


Fig. 9.9 Ammonium persulfate batch treatment.

chromic-sulfuric etchant. Chromium treatment is done in two steps starting with a reduction of all hexavalent chromium to trivalent with any one of a variety of reducing chemicals, i.e., sulfur dioxide, sodium bisulfite, or hydrazine. The reduction reaction takes place much more rapidly at a low pH, so the first phase of treatment involves pH adjustment. Figure 9.12 is a schematic of the first stage of chromium treatment.

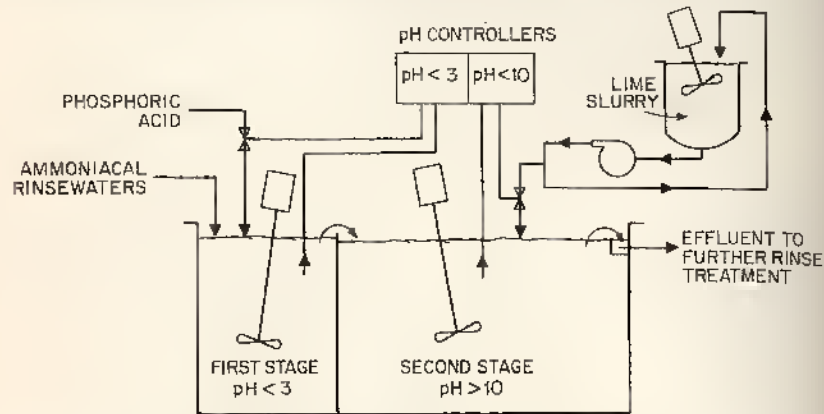


Fig. 9.10 Ammoniacal rinsewater treatment.

The second step in chromium removal is the same as the treatment for the other heavy metals. The pH is raised by addition of caustic or lime, and chromium hydroxide is precipitated. This second stage of treatment is identical with the pH control system shown in Fig. 9.2.

7. Contract Disposal of Sludge and Spent Chemicals All printed circuit shops will have some spent chemicals and sludges to dispose of rather than treat. The following list includes many of the materials that either are the final product of a waste treatment system or are often more economically disposed of in some way besides in-house chemical treatment.

1. Heavy-metal sludges from a waste treatment system.
2. Spent copper etchant—alkaline, chromic-sulfuric, ferric chloride, ammonium persulfate, cupric chloride.

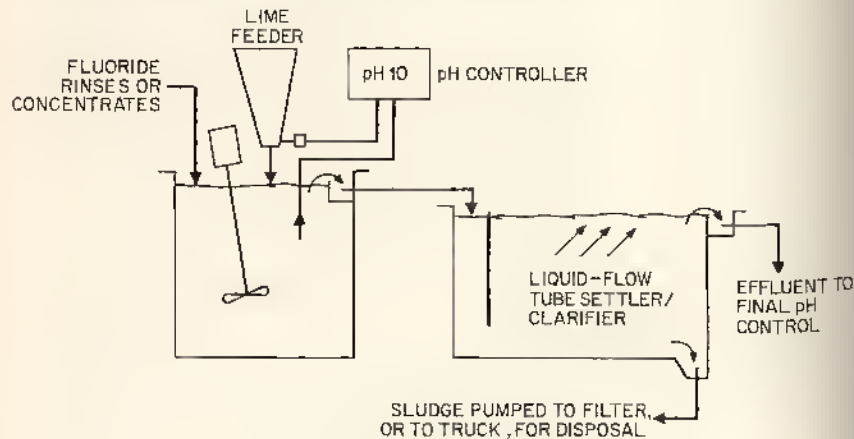


Fig. 9.11 Fluoride treatment system.

3. Photoresist stripper usually containing methylene chloride.
4. Contaminated plating baths that cannot be cleaned up to operating standards.
5. "Chemical cleaning" solutions used to remove epoxy from the holes and surface of multilayer boards. These solutions are usually chromic acid or concentrated sulfuric acid.
6. Etchback solutions containing concentrated sulfuric acid, concentrated hydrofluoric acid, and dissolved epoxy-glass laminate.

8. Safety Notes: Many of the spent concentrated chemical baths cannot be mixed without some very hazardous reactions taking place. Some mixings result in violent generation of

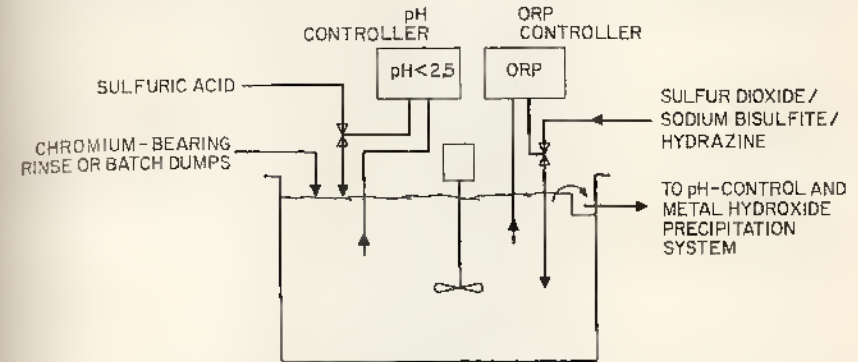


Fig. 9.12 Chromium treatment first-stage reduction.

heat, and others in the evolution of toxic gases. Before mixing any of these chemicals, be sure to have the proposal reviewed by someone with an excellent understanding of the chemistry involved.

Disposal of spent chemicals and sludge into special landfill operations has been shown to be an economic alternative. Critics argue that these landfill operations have a potential for environmental harm. California has established strict regulations for dump sites, called Class 1 dumps, which prevent groundwater contamination. Other states will probably follow, and it is important to review local law before proceeding with a disposal plan.

Hazard in case of earthquake or flood is also used as a criticism of landfill disposal. The counterargument is that most of the chemicals react with each other and form insoluble compounds practically unaffected by flood or earthquake. For example, hexavalent chromium poured into a special dump pond would soon oxidize other waste and be reduced to trivalent chromium. If the dump is required to neutralize wastes before disposal, the trivalent chromium will precipitate in the pond as chromium hydroxide. The hydroxide has little potential for environmental harm short of being dissolved by a strong acid.

A second method of disposal is to hire a disposal company to pick up the spent chemicals and to contract with it for proper disposal. This involves some sticky legal questions about whose waste it is if it is disposed of illegally. However, if the disposal company uses legal methods of disposal, it offers great economic advantage. A large central disposal facility has many economies of scale, can sometimes combine two wastes that treat each other, and can often economically reclaim chemicals collected from many small sources.

Incineration is a third method of disposal for spent chemicals. You can eliminate any dependence on outside firms by buying your own incinerator. Several plants report this as an economic alternative, especially where there is also a source of waste material that has a high Btu value and reduces fuel requirements. Disadvantages include the problems of forming HCl by burning chlorides, volatilization of metals like lead, mercury, and arsenic, and ultimate disposal of a highly water-soluble ash.

A fourth alternative disposal method is to mix the waste with portland cement and make a solid that has very low solubility and can be disposed of with demolition wastes or landfilled. A process that handles waste in a similar way but uses a proprietary process is marketed by ChemFix Corp.

9. Costs of Waste Water Treatment Costs for treatment equipment and installation and operation vary widely with size, degree of automation, customization of the system to fit into rigid space available, and the quality of the equipment used.

a. Equipment Costs. A report from the Illinois Institute for Environmental Quality contains many graphs useful in estimating capital and operating costs for waste treatment systems.² For a rough estimate the following rule of thumb may be helpful. The cost of any one system such as chromate reduction, electroless copper batch treatment, or heavy metal precipitation will usually be between \$30,000 and \$60,000 depending upon system size and not counting buildings or land. Simply decide whether your shop is small, medium, or large and multiply the number of treatment systems that you think you need by \$30,000, \$45,000, or \$60,000, respectively. About half of the total cost is used for solids removal and handling.

For example, a small printed circuits shop (50 gal/min of rinsewater) may have decided to have a contract waste hauler take all concentrates away. It has almost no ammonia in the system. It may need only pH control with heavy-metal precipitation. A rough estimate of \$60,000 should cover the system. On the other hand, a large printed circuits shop (440 gal/min of rinsewater) would require eight different systems costing $8 \times \$60,000 = \$480,000$.

Most printed circuit facilities are in areas with fairly limited space for waste treatment systems. If your plant is fortunate enough to have lots of available land, some solids removal equipment can be replaced by large settling lagoons.

b. Operating Costs. Chemicals, operator labor, and maintenance cost will usually range from \$0.15/1000 gal for simple pH control to \$0.640/1000 gal for complex chemical treatment and up to \$0.80/1000 gal for an ion-exchange water-recycling system.

c. Hauling Costs. Contract hauling and disposal cost are influenced mostly by quantity and method of handling. The round-trip distance for the haul and the treatment required are also major factors. Costs range from \$0.08/gal for 4000 gal hauled in half-day trips up to \$0.35/gal for chromic acid in a small number of 55-gal drums hauled on a one-day trip.

AIR POLLUTION CONTROL

10. Dust Removal Printed circuit fabrication involves many processes that create airborne particulates better known as dust. Drilling, sanding, routing, sawing, beveling, and slotting operations all require dust collection to protect the employees and plant equipment. The dust cannot be simply blown out into the atmosphere without polluting the neighboring areas.

A cyclone separator, which is an inertial separator without moving parts, can be used to remove dust typical of printed circuit manufacture. Figure 9.13 shows how a cyclone separates particulate matter from a carrier gas by transforming the velocity of an inlet stream into a double vortex confined within the cyclone. In the double vortex the entering gas spirals downward at the outside and upward at the inside of the cyclone outlet. The particulates, because of their inertia, tend to move toward the outside wall and then fall into a disposal box placed under the cyclone.

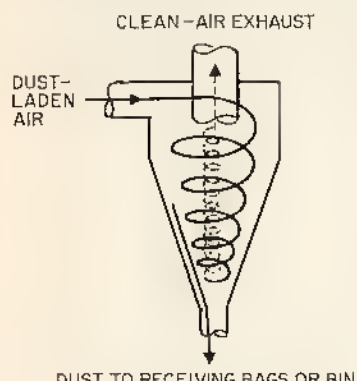


Fig. 9.13 Double-vortex path of the gas stream in a cyclone separator.

A baghouse can also be used to separate dust from the exhaust air from a printed circuit manufacturing operation. Figure 9.14 shows the construction of a Dustube fabric filter which operates like a giant vacuum cleaner with many small bags. Dust is filtered out of the airstream inside the bags. Periodic shaking of the bags knocks the dust down into the hopper for later disposal.

11. Wet Scrubbers Acid fumes, chlorides, and ammonia are exhausted from the plating shop as gases or very small airborne liquid particles. To remove them from the air requires technology most often referred to as wet scrubbing. The design of the scrubber varies, but the principle is the same, i.e., contacting the airstream with water or an aqueous solution and transferring those fumes into the water from the air prior to releasing the air into

the atmosphere. Techniques include (1) pulling or pushing the air upward through a wetted column packbed with surface-multiplying rings or balls (Fig. 9.15) and (2) passing the air through a high-velocity water spray or fog (Fig. 9.16).

12. Carbon Adsorption of Chlorinated Solvents Air pollution limits and environmental health limits have been set by regulatory agencies on several chlorinated solvents used in printed circuit manufacture. Chlorinated solvents often used in printed circuit processes include trichloroethylene, 1,1,1-trichloroethane, and methylene chloride.

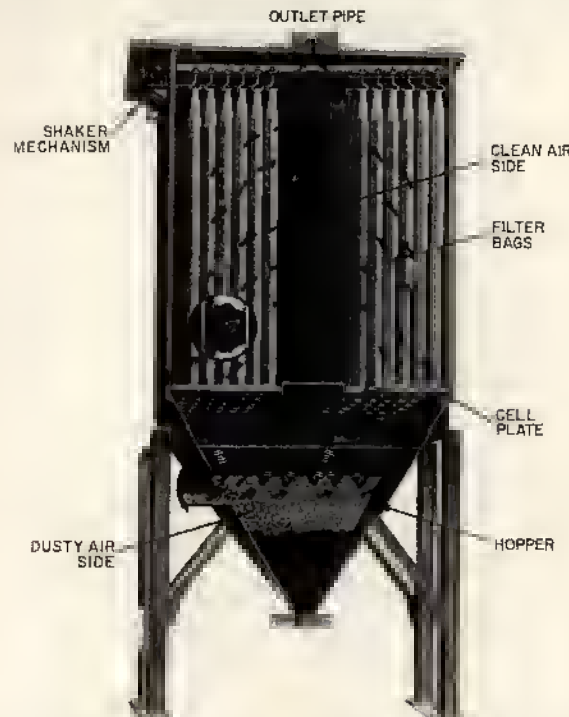


Fig. 9.14 Typical simple baghouse with mechanical shaking (Wheelabrator-Frye, Inc.).

Trichloroethylene now is regulated in most states to limits which make it easier to change solvents than continue to use it. Typically, it has been used in vapor degreasers and as a stripper for silk-screened plating resist. Several air pollution regulations limit the evaporative loss per operation to less than 40 lb/day; alternatively, a system that removes at least 85 percent of the exhausted solvent must be installed. Even a small degreasing operation or a conveyorized resist stripper will lose over 100 lb/day of trichloroethylene. A fairly well designed exhaust system with carbon adsorption columns will recover typically about 50 to 60 percent of the trichloroethylene. If 85 percent recovery is required, it will be nearly impossible to meet the regulation without changing solvent.

The decision may be made to remove chlorinated solvents from exhausted air. The reason for removal may be to meet pollution control regulations and/or to conserve these expensive materials. Chlorinated solvent fumes can be removed by passing the exhaust through a column of activated carbon. Figure 9.17 is a flow diagram of a carbon adsorption system for 1,1,1-trichloroethane. When the carbon is later regenerated with steam, the recovered chlorinated solvent can be returned to service.

13. Equipment Costs The cost of air pollution control equipment is primarily a function of size and type of material required to resist the pollutants in the airstream involved. Dust removal requires fairly inexpensive materials of construction. A good central system

serving a medium-size printed circuit shop (3600 ft²/min) would cost approximately \$9000. Wet scrubbing of acid fumes requires more expensive materials and much greater air capacity to exhaust large open tanks. A wet scrubber for a medium-sized printed circuit shop (40,000 ft²/min) might cost about \$25,000.

DEALING WITH REGULATORY AGENCIES

The best way to deal with most regulatory agencies is with an open hand. If you show them your problems and proclaim your sincere desire to do what is best for the environment, they will generally try to help your company comply with their regulations. That does not mean that they will not expect timely action from you. It means that they will

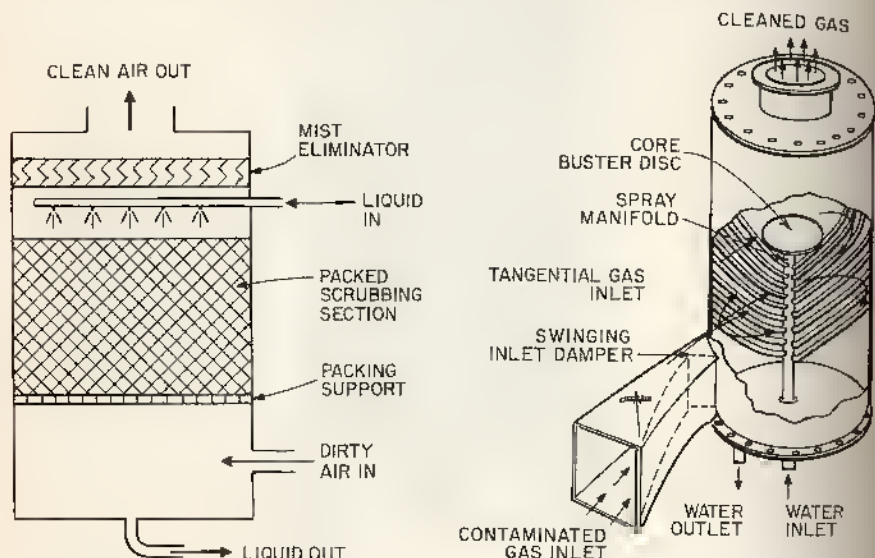


Fig. 9.15 Schematic diagram of a packed counter-current scrubber.

Fig. 9.16 Cyclone scrubber (Chemico).

grant you a realistic time schedule and believe your explanation of any problems. On the contrary, dealing with the agencies like a tax evader can give you nothing but grief. Revealing your records only after a court order, making the agencies resort to legal processes to force you to put in pollution control equipment, and playing cat and mouse with the waste inspectors or trying to buy them off are risky measures. If your company's reputation is of any value, such tactics will prove to be very costly.

When new ordinances are being developed or proposed, companies affected will find it very worthwhile to get involved. Industry often has access to better data on which to base effluent standards than the regulatory agencies have. Many old municipal ordinances that were passed were impossible for industry to meet, were then unenforceable, and resulted in a general disregard for the law. Now the climate is different. Federal funds for construction and expansion of municipal sewage treatment plants are contingent on enforcement of industrial discharge requirements. The law is or soon will be strictly enforced. Minimum fines of \$2500 per day for discharges not meeting the standard are a common part of the enforcement action.

For competitive reasons it is important that the law be uniformly enforced. Companies should have good liaison with their regulatory agencies and neighboring companies and promote uniform enforcement of the law. Several legal suits brought by regulatory agencies against a polluter have been lost because the agencies were not uniformly enforcing the law. The courts would not allow an agency to make an example of one particular company.

RECYCLING, RECOVERY, AND CONSERVATION

The most desirable way to dispose of metals and spent chemicals is to clean them up and recycle them. When chemicals cannot be cleaned up for reuse in a process, it may still be possible to reuse them in a less critical process. If recycling is not feasible, recovery of metals or certain components may be possible. As the costs of pollution control and chemicals increase, the economics of recycling and recovery look better and better.

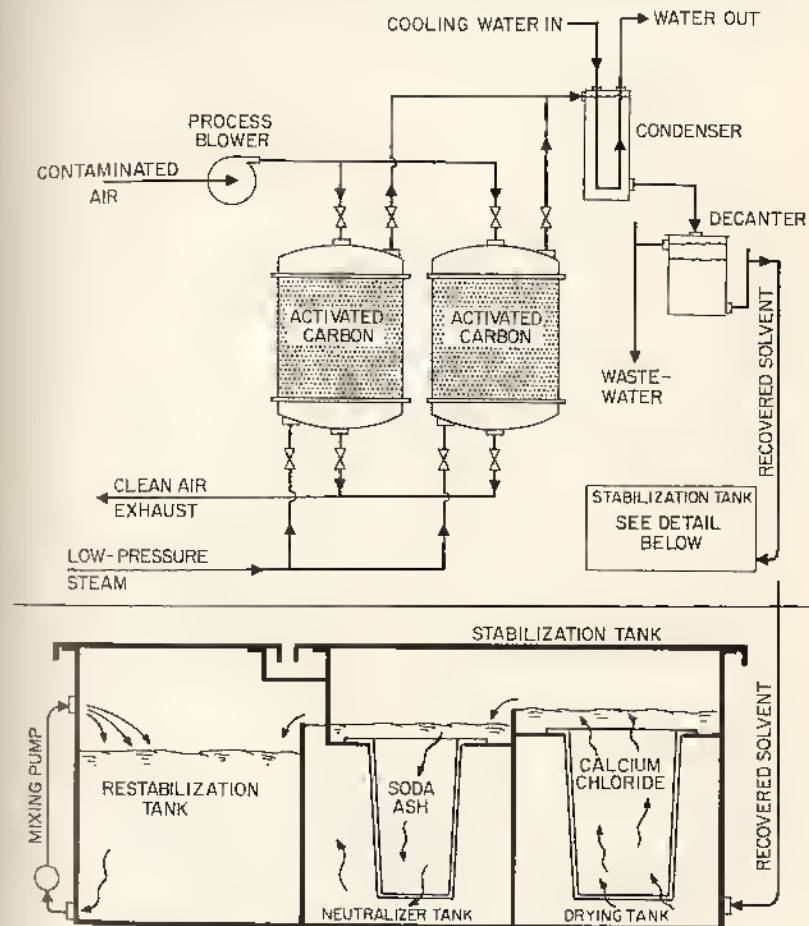


Fig. 9.17 1,1,1-trichloroethane vapor adsorption system.

When recycling and recovery are not possible or economically feasible, conservation of chemicals can greatly reduce waste treatment costs. Using smaller tanks, allowing parts to drip longer over plating baths, and controlling rinsewater flow are examples of conservation measures.

14. **Water Conservation** Water is required in great quantities to manufacture printed circuits. The cost of incoming water and the charges for sewer use make water a significant material cost that is often lumped with other utility costs. Efforts to conserve water can pay for themselves in a very short time. Methods of conserving water include those in the following discussion.

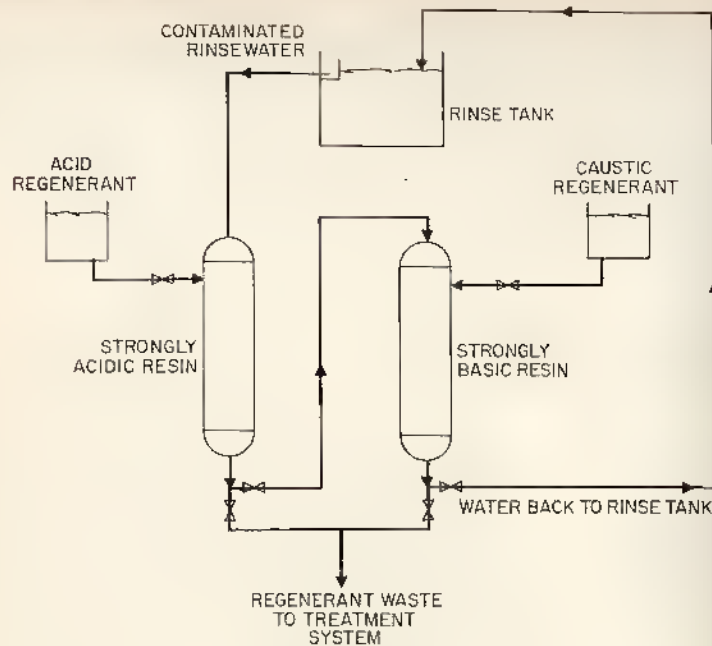


Fig. 9.18 Two-bed ion-exchange system.

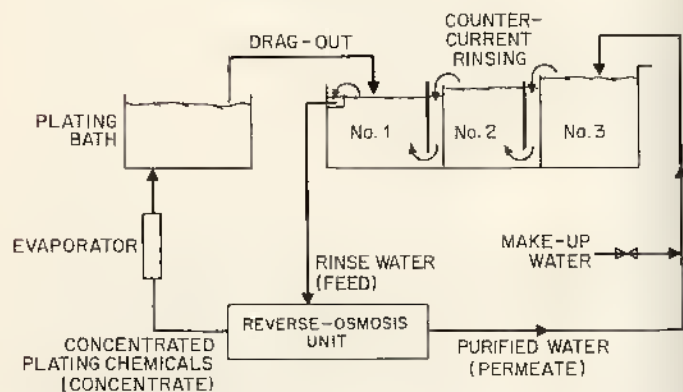


Fig. 9.19 Flowchart of typical reverse-osmosis system for treating and reusing rinsewater.

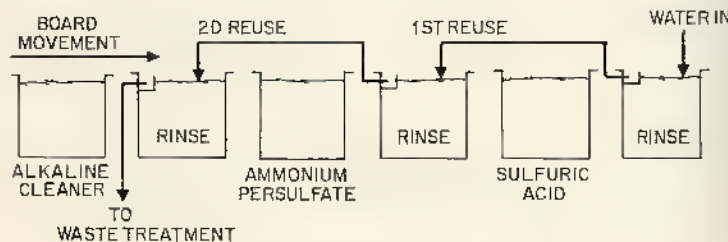
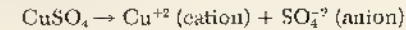
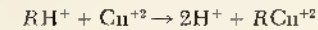


Fig. 9.20 Triple use of rinsewater in electroless copper line.

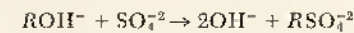
a. Ion Exchange. Pollutants can be removed from rinsewater by ion exchange and the water is then recycled. In special cases the metals removed by the resin can be regenerated and reused in the process. A typical flowchart of an ion-exchange system is shown in Fig. 9.18. Most water-based chemicals used in a printed circuit shop ionize in water to form anions and cations:



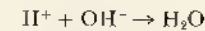
When the material is passed through ion-exchange columns, the cations replace hydrogen ions in the cation resin:



Similarly, the anions replace OH ions in the anion resin:



Then the hydrogen ions, H^+ , and the hydroxyl ions, OH^- , combine to form water:



The ion-exchange resin is regenerated by adding a strong acid to the cation resin and a strong base to the anion resin. The regenerants reverse the above reactions, and the resin is returned to its original state. The metals and anions are removed in concentrated form with the regenerant solutions when the solutions are washed out of the columns.

b. Reverse Osmosis. Water can also be conserved by using a process called reverse osmosis (RO). Under high pressure most of the salts dissolved in water can be removed in a separate concentrate stream. By using two or three RO modules in series, enough water can be removed from the concentrate to return it to a plating bath or similar process tank. Figure 9.19 shows a flowchart of the process.

c. Cascade Rinses. When sufficient room is made available in a plating or cleaning line, rinses can be cascaded to conserve water. Figure 9.20 shows a triple-cascade rinse. Makeup water is added to the final rinse tank and gets progressively more contaminated as it cascades to the preceding rinse tank. By cascading, the amount of rinsewater can be reduced to a small fraction of the water required with a single rinse tank, and the contaminant concentration can be increased correspondingly.

d. Multiple Use. A simple one-for-two conservation of water can be achieved by double use of water. Cooling water can be reused as rinsewater where any rise in temperature improves rinsing. Some treated industrial water is suitable for landscape irrigation. Sometimes the rinsewater from one process can be reused as the rinsewater for another process.

Double or triple use has been demonstrated in the electroless copper line as shown in Fig. 9.20. Notice in this case that none of the impurities from the rinses are deleterious to the tanks following the rinse.

e. Flow Regulation. Flow controllers can be used in conjunction with single-rinse, double-rinse, or cascade rinse to conserve water effectively. In general, rinses run at too high a flow rate. A simple, unadjustable flow regulator can be used to set an upper limit on the flow of water to a rinse tank.

An automatic flow controller opens and closes the rinse line valve on the basis of the

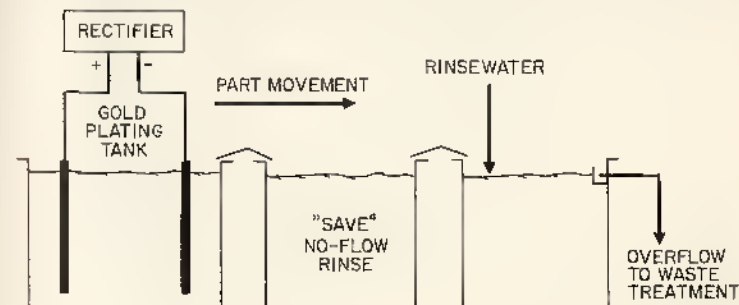


Fig. 9.21 Gold "save" rinse system.

conductivity of the water in the rinse tank. This type of control is particularly useful in a sporadically operated plating or cleaning line.

15. Gold Recovery Some printed boards are gold-plated either over the entire plated area or selectively on contact points. When the boards are removed from the plating bath, they carry out some of the gold-bearing solution. The high value of gold has made this seemingly insignificant loss very important.

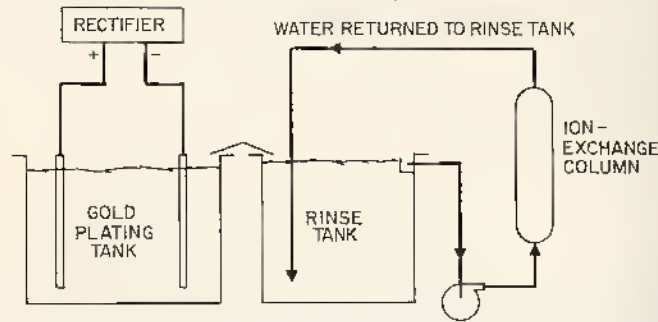


Fig. 9.22 Ion-exchange gold-recovery system.

a. Save Rinse. If the first rinse after gold plating is a still, save rinse, about 90 percent of the gold dragged out can be recovered. When the gold concentration builds up to a point at which the loss from the save rinse is significant, the rinse is taken out for reclaiming the gold (Fig. 9.21).

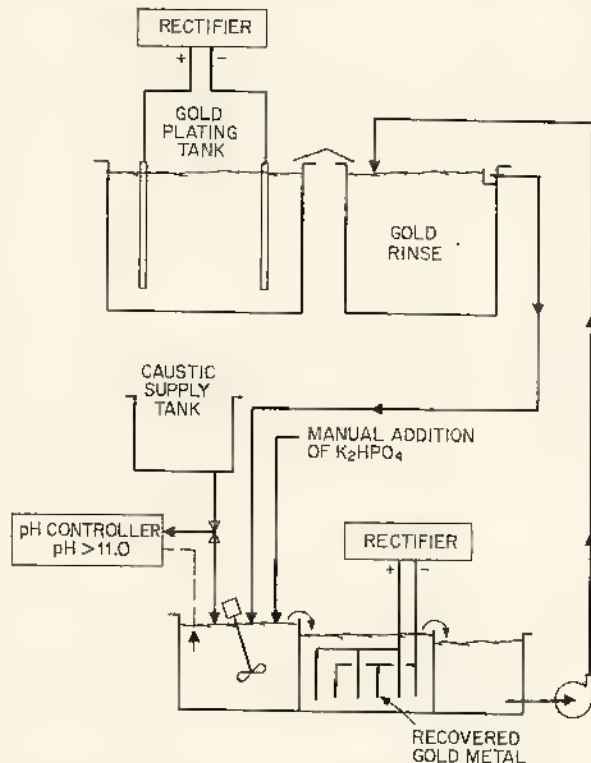


Fig. 9.23 Electrolytic gold-recovery system (Lancy Laboratories).

b. Ion Exchange. Another approach to reducing gold loss is to use an ion-exchange process. The gold rinse can be passed through one or two ion-exchange columns and the gold collected in the resin. A typical cation resin has a capacity of about 250 troy ounces of gold per cubic foot. When the resin is near its capacity, it can be sent out to have the gold reclaimed. A sketch of such a system is shown in Fig. 9.22.

c. Electrodeposition. Gold can also be electrodeposited from the rinsewater. Lancy Laboratories has patented a system that recirculates the first gold rinse, which is a special chemical rinse, through an electrodeposition cell. The scheme is illustrated in Fig. 9.23.

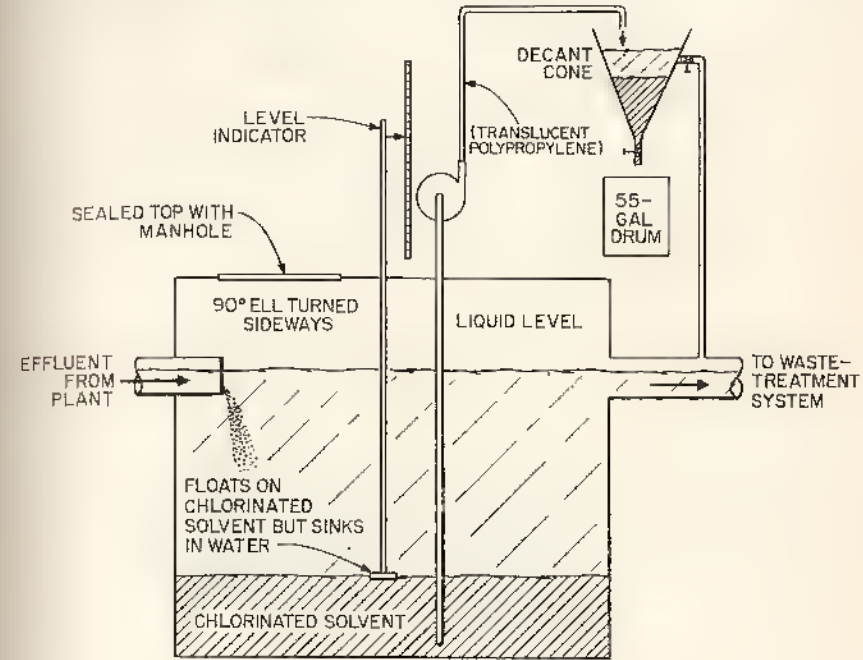


Fig. 9.24 Chlorinated solvent trap.

The advantage claimed for the system is that the amount of gold recovered is readily measurable. The major disadvantage is the security necessary to prevent gold losses.

d. Evaporation. Continuously or periodically, gold rinsewater can be evaporated down to a concentrate or gold sludge. Evaporators for continuous operation are very expensive on the small scale that is usually involved in printed circuit gold plating. One inexpensive application involves the use of a heated tank to which makeup water is periodically added from the gold-save rinse tank. The concentrated gold solution or gold sludge in this evaporation tank can be sent out for gold reclamation.

Spent gold solutions are an obvious source of gold scrap that can be sent out for reclamation. The biggest single source of gold waste is scrap printed boards, board trim, routing dust, and gold-plating-rack buildup. The amount of gold loss via any one of those wastes is very dependent on the shop processes and board design. Usually gold conservation measures will rapidly more than pay for themselves.

16. Recovery of Chlorinated Solvents Spent chlorinated solvents are easily recovered for recycling, and their high cost makes this the preferred disposal alternative. Trichloroethylene, 1,1,1-trichloroethane, and methylene chloride are the three primary chlorinated solvents used to strip silk-screened resist, develop photoresist, and strip photoresist. The solvents can be recovered in-house with a small still, or they can be sold or sent out for reclaiming in 55-gal quantities or large tank loads. Most industrial waste water ordinances put very low limits on chlorinated solvents because of the high toxicity of the solvents to

the bacteria in the municipal treatment plants. Usually resist stripping and developing steps are followed by a water rinse. Small quantities of chlorinated solvent are rinsed off the boards and go down the drain. The solvents are immiscible with water and are heavier than water. They will sink to the bottom of pipes and tanks, attack epoxy linings, and soften many plastics.

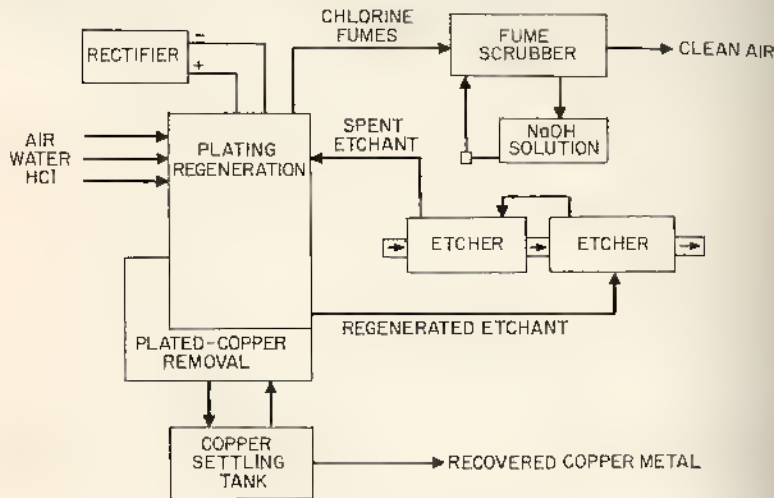


Fig. 9.25 Cupric chloride etchant regeneration system.

An effective scheme for trapping chlorinated solvents and keeping them out of the sewer is shown in Fig. 9.24. A simplified trap could be made with a translucent polypropylene tank with a weir overflow. The chlorinated solvent would be seen as a dark layer in the bottom of the tank that could be periodically pumped or poured out.

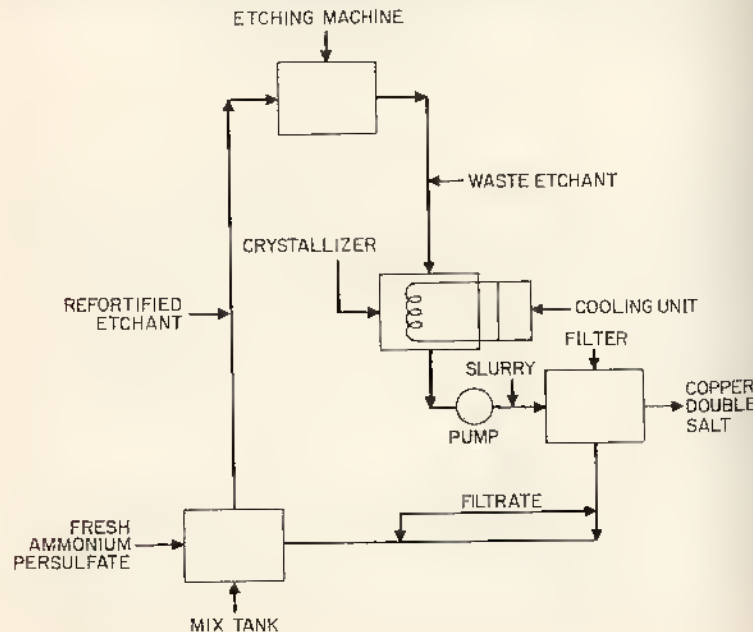


Fig. 9.26 FMC's ammonium persulfate etchant-recovery system (CAPER).

17. Copper Recovery Spent copper etchant containing from 2 to 30 oz/gal of copper is a great potential resource. Most printed circuit shops have been anxious to get rid of spent etchant even at a cost of more than \$0.10/gal, but now several processes make possible the in-house recovery of copper for reuse. Cupric chloride has been used as a regenerable etchant with a by-product of copper. Figure 9.25 is a schematic diagram of a process patented by Bell Telephone Laboratories.

Ammonium persulfate also has been regenerated and recycled with copper ammonium sulfate as a by-product. This process was developed by FMC Corporation and given the trade name CAPER. It is shown in Fig. 9.26.

Recovery of copper from alkaline etchants using liquid-liquid ion exchange shows great promise in the developmental stages. General Mills Chemicals, Inc., has been involved in some of this developmental work.

The copper removed from electroless copper discussed earlier is also in a readily recoverable form (Fig. 9.5).

17. Palladium Recovery The value of palladium has risen at a very rapid rate somewhat in the shadow of the price of gold. Palladium is the main ingredient in the catalyst tank in the electroless copper deposition line. Large plating shops may be able to justify a recovery system for palladium. All the systems used for gold could be modified for recovery of palladium.

KEEPING CURRENT

In this chapter most of the pollution control techniques described have been engineered to varying degrees of success over a period of many years. Innovation in this area of pollution control has in the past been slight; there has been a low frequency of significant developments. Pollution control developments in other industries, however, often have application to the printed circuit industry. Several publications on pollution control have compiled information from many industries.³

With the strong emphasis now being placed on cleaning up the environment, a large amount of research has been and is being carried on. The results of the research will hopefully bring the printed circuit industry more effective, less costly, and more easily operated pollution control systems than have been available in the past.

REFERENCES

1. W. Brung, Effect of Heavy Metal Discharge upon the Aquatic Environment, Environmental Protection Agency Technology Transfer Seminar, Portland, Ore., May 16, 1973.
2. J. W. Patterson et al., Waste Water Treatment Technology, Illinois Institute for Environmental Quality, Chicago, Ill., August 1971.
3. Herbert F. Lund, "Industrial Pollution Control Handbook," McGraw-Hill Book Company, New York, 1971.

Section **3**

Assembly

Chapter 10

Manual Assembly

WAYNE L. JOSLIN

Hewlett-Packard Company, Palo Alto, California

Introduction	10-4
1. Reference Board	10-4
2. Assembly Process	10-4
3. Economic Justification	10-5
Basic Considerations	10-5
4. Product Design	10-5
5. Cost	10-6
Small-Volume Process Planning	10-6
6. Visual Aids	10-6
7. Learning Curve	10-6
8. Typical Right- and Left-Hand Analysis	10-7
9. Preforming Leads	10-7
10. Assembly-Station Layout	10-7
Assembly Process	10-7
11. Lead Clinching	10-7
12. Presoldering Inspection	10-8
13. Soldering	10-9
14. Special Components	10-9
15. Touch-up	10-9
16. Final Inspection	10-9
17. Test and Repair	10-11
18. Conformal Coatings	10-11
19. Stocking	10-11
Prototype Process	10-11
Large-Volume Process	10-12
20. Planning	10-12
21. Assembly-Station Layout	10-12
22. Soldering	10-13
In-line vs. Rotary Work Station	10-13
Summary	10-13

INTRODUCTION

Various types and configurations of components are attached to a printed board to make the board a functional electronic device. The manner in which the components are added has several parameters. The relative weight of each parameter in the consideration of the chosen method will depend upon the ultimate use of the parent electronic assembly. Printed boards used in airborne computers will certainly have different relative values placed on the constraints than will those used in low-cost transistorized radios. In between the extremes of concern for reliability and cost are the majority of printed circuit assembly requirements.

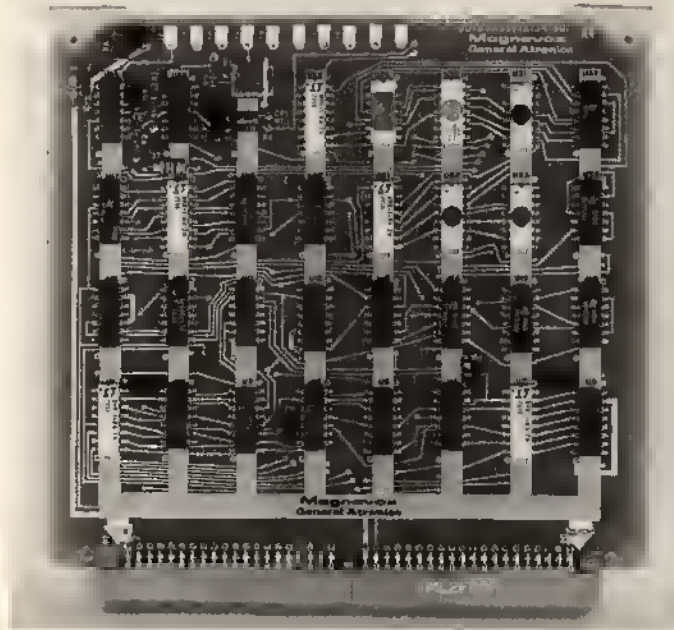


Fig. 10.1 Typical printed wiring assembly, (General Atomics Corp.).

All factors should be reviewed when the assembly process is chosen. That will ensure that changing needs will not have been overlooked. It is not the intent of this chapter to supply a perfect pattern for use in all possible situations. It is intended that the data given will be useful in determining the course of action once the factors have been identified.

1. Reference Board In order to indicate differences in approach, a basic reference must be chosen. A typical board is shown in Fig. 10.1. It has TO-5 transistors, $\frac{1}{4}$ -W carbon resistors, tubular diodes, and dipped mica capacitors; and it is a plug-in board. Although this board has uniform hole spacing for the resistors and diodes, transistors and capacitors, and does not have clinched leads, it can be shown what differences would occur if that were not the case. The board is double-sided with plated-through holes. It is to be produced for commercial use in an industrial instrument. Shock and vibration reliability is no more than that encountered during shipment. Quality levels are those of good general workmanship. Specific assumptions will be made to match the process.

2. Assembly Process Printed board assembly can be described as a process with a number of steps. A typical process consists of these steps:

1. Job put-up (the gathering of all parts required)
2. Component preparation (cutting off and bending of components before assembly)

3. Board cleaning (washing in a commercial cleaner or active flux and rinsing in water, etc., to prepare circuits for soldering)
4. Inspection (examining components and boards for proper dimensions and damage)
5. Kit components (placing of parts in trays in assembly sequence)
6. Board assembly
7. Inspection (inspecting sample for component placement and value)
8. Soldering (hand-solder, dip-solder, machine-solder)
9. Assembly cleaning (removal of all solder splashes and flux)
10. Special-components addition (components that might be damaged by normal cleaning or heat of soldering)
11. One hundred percent visual inspection (a check on solder, raised components, and aesthetic board appearance)
12. Solder touch-up and correction of errors
13. Electrical testing (this will obviate the need to check components in step 10)
14. Defects repair and assembly retest
15. Humidity sealing (if applicable)
16. Stocking

All the above steps must be taken regardless of the details of the actual process, but some may be combined for small volume.

3. Economic Justification No two assembly facilities are identical. Decisions as to the approach of assembly techniques must be made at the manufacturing location. The data submitted will aid in making those decisions. The figures in Table 10.1 are stated in hours. Multiplying by the applicable labor rate plus variable overhead will give the proper order of magnitude. The hour figures are meant for reference; their source is a predetermined time system verified by actual use. They may not be the same for any facility, but the order of magnitude is correct.

The equipment costs are close approximations made to determine if acquisition is feasible.

Equipment	Cost
Rotary table with spare trays	\$ 125
Lead former—axial components	1200
Lead former—transistor	2000
Lead former—radial lead	200
Power unit for items 3 and 4	150
Push line (per station)	80
Hydraulic press for lead forming	800
Dies for press (axial lead only, per set)	150

BASIC CONSIDERATIONS

The constraints on facility organization have two major divisions, design and cost. Each of these has a number of subdivisions.

4. Product Design The "engineered package" is the first subdivision, and it will be the most confining of all factors. Component type, variety, location on the board, and interface with the rest of the instrument will confine the assembly process. It is well to remember that review of a design before release by a skilled manufacturing engineer will usually remove unnecessary restrictions from the package. That will allow lower cost and normally better quality to be achieved.

Quality is the second subdivision of design. A board assembly must meet the required quality levels. An exact concept of quality is difficult to convey. Most firms identify their quality goals by means of definite phrases regarding workmanship, standard practices, aesthetic appearance of the end product, and the duration of time or number of repeated cycles the unit will operate, i.e., its effective lifetime.

Reliability is the third division of design, and it is often confused with quality. It is the degree of certainty that the device will function when required and will continue to operate for a specified period under given environmental conditions. Military specifications for reliability, if required, will probably alter the assembly process.

5. **Cost** The other major constraint is cost. Regardless of all other requirements, the method chosen will need to achieve the considered ends at the lowest possible cost. It is understood that a board that requires special techniques will cost more than one that does not, but it should be manufactured with all available economies consistent with that need.

The subdivisions of cost are direct labor, volume, and equipment. The combination of the three will define the approach outlined by the design constraint.

TABLE 10.1 Typical Assembly Times Per Component
(Figures in hours)

Component type	Hand form		Machine form		Hydraulic form	
	Hand bend, insert, trim after solder	Form	Hand insert	Form	Hand insert	Form
Axial lead (resistor, diode)	0.0037	0.0005	0.0010	0.0005	0.0007	
Radial lead (not transistors)	0.0037	0.0010	0.0010			
Transistor	0.0032	0.0012	0.0015			

For each observation to check location from print or prototype, add 0.0010

For hand lead clinch (per lead), add 0.0006

Example: 100 axial lead, hand bent, clinched leads,
100 observations $(100)(0.0037 + 0.0010 + 0.0012) = 0.590$
100 axial lead, machine formed, four observations $(100)(0.0005 + 0.0010) + (4)(0.0010) = 0.154$
100 axial lead, die formed,
four observations $(100)(0.0005 + 0.0007) + (4)(0.0010) = 0.124$

NOTE: Source of values is predetermined time verified by use.

SMALL-VOLUME PROCESS PLANNING

Generally, "small volume" is considered to mean lots of fewer than 100 in a period of a month. Equipment available should be as follows:

1. Lead former and trimmer for axial lead components
2. Lead former and trimmer for radial lead components
3. Lead former and trimmer for transistors
4. Die and press for cut and trim axial lead components
5. Rotary table and bins with vertical rotary part carriers

6. **Visual Aids** A sepia print is made of the board assembly. Each component is numbered to indicate assembly sequence. (Low-level components, such as diodes and resistors, are installed first; larger components are placed on the board later.) The next step is to list the component part numbers in assembly sequence and note hole spacing for preparation equipment. Copies of the board assembly and component list are made. The first copy is used as the assembly reference document. The second is the guide for arranging component trays, which are numbered according to the list at the staging area.

7. **Learning Curve** The major problem in direct-labor assembly cost is the learning curve. One factor in line assembly is the operator's memorizing what to do. It is not possible to memorize large portions of an operation in a short period (e.g., the time required to build 100 boards), but it is possible to learn after one observation where a component should be placed. If an operator had only to place one component in each board, the learning curve would be almost eliminated. The problem, then, is to allow the boards to flow past the operator so that only one component is placed at a time; in that way learning time and the cycle for one board are minimized. The rotary table ("Lazy Susan") does exactly that. Any number of boards are placed on the table. As each component is placed, the table rotates and offers a new board for assembly. After one completed revolution, a new component is selected and the operation is repeated. If the boards are so small that 10 or more can be placed in front of the operator, the rotary table is not required.

8. **Typical Right- and Left-Hand Analysis** Here a typical right- and left-hand analysis is given for prebent and cut $\frac{1}{4}$ -W carbon resistors. Compare it with the second analysis for die-formed parts.

Left hand	Right hand
1. Get handful of parts from bin (once for each component type)	1. Hold pliers
2. Move parts to work area	2. Hold pliers
3. Move one part to fingertips	3. Advance table
4. Check for orientation	4. Hold pliers
5. Orient, if necessary	5. Hold pliers
6. Hold part	6. Grasp part with pliers at right-hand leads near body of part
7. Release part	7. Hold part
8. Move hand to table	8. Move part to board
9. Hold table	9. Insert component
10. Clear hand	10. Remove pliers
11. Repeat steps 1 to 10 for all boards on table	
12. After all positions of same part are filled, replace remaining parts in bin	12. Hold pliers

Analysis for die-formed and cut parts on vertical parts carriers (see Fig. 10.7):

Left hand	Right hand
1. Index parts wheel (once per part value)	1. Hold pliers
2. Move to work area	2. Grasp part with pliers at right-hand leads near body of part
3. Hold table	3. Move part to table
4. Hold table	4. Insert part
5. Hold table	5. Remove pliers
6. Advance table	6. Reach to next part

The second analysis is obviously faster, but it has the limitation of requiring all boards made for different instruments to have the same component hole spacing. Otherwise, separate die costs become uneconomical.

9. **Preforming Leads** The first analysis for formed leads allows for greater hole-spacing flexibility. However, if volume approaches the levels for machine insertion, the die-form technique is an excellent preliminary stage that prepares for eventual machine insertion.

Die forming is accomplished by applying double-faced pressure-sensitive tape to resistors and diodes before those components are removed from the commercial package. The strip is placed into a die of a hydraulic press that cuts and forms to proper dimensions (Fig. 10.2). (Suggested dimensions are 0.50-in hole spacing for $\frac{1}{4}$ -W carbon resistors and diodes with 0.045 in sticking through the board. See Chap. 1.) The taped components are placed on a circular carrier that rotates vertically. The rows are numbered, as are the bins for rotary tables. Where applicable, combining vertical part carriers with rotary-table bins offers the least variable cost short of mechanical insertion. See Figs. 10.3, 10.4, and 10.5 for other preforming techniques.

10. **Assembly-Station Layout** The assembly station is a 4-ft-diameter rotary table with one layer of bins and one vertical parts carrier. An assembly document has a numbered sequence of components which corresponds to the bin numbers or row of parts on the carrier. Boards are held in place on 1-in-thick Styrofoam pads or other appropriate device. The only hand tool required is a pair of long-nose pliers. The finished parts are transported on the Styrofoam pads (Figs. 10.6 and 10.7).

ASSEMBLY PROCESS

11. **Lead Clinching** If the leads must be clinched to meet a design specification, the boards will be placed into a special fixture after assembly. This fixture consists of a frame nest for the board. A metal plate with $\frac{3}{8}$ -in-thick foam rubber is placed on top. The fixture can now be inverted for clinching.

The operator uses a pair of flush-cutting diagonal wire cutters. The lead is grasped



Fig. 10.2 Preforming axial-lead component leads with die set (Hewlett-Packard Co.).

where it will be cut. The lead is bent against the circuitry and clipped in a single motion. The work begins at the lower left-hand corner and progresses across and upward.

Parts which have been present will have to have excess stick-through length to allow for clinching. The normal dies mentioned earlier do not have this lead-clinching capability unless so specified.

12. Presoldering Inspection One board of the assembled group per table load is 100 percent visually inspected for component placement and value. The purpose of the check is to find consistent errors. It is likely that if one component is misplaced on one board, it



Fig. 10.3 Cutting and preforming transistor leads by machine (Hewlett-Packard Co.).

will be misplaced on other boards. Random errors are found in testing. Under normal operating conditions it is not economical to inspect further at this point.

Overlays made of Mylar* or similar material can be used to check for component placement. They are made by drawing or gluing the appropriate component outline and/or color coding on the transparent material. Another approach is to cut the component outline from heavier material. All components of a given type can be on one overlay. The series of overlays will indicate which components are missing.

13. Soldering The assembly is now ready for soldering. Normal commercial work can avail itself of the water-soluble active fluxes to get better and more consistent solder results. When rosin fluxes are required, the technique varies slightly. For military work refer to MIL-STD-275.

A dip pot is the most common form of soldering at the time of writing. The board has a rubber channel fitted to the plug-in fingers to serve as a solder mask. It is then placed into a tonglike device held in one hand. The board is passed over a foot control spray device which applies the flux, and it is then placed into a skimmed pot of solder that is 490°F at the surface. One edge is placed first, and with a careful rolling motion of the wrist and forearm the solder is applied (Fig. 10.8). The board is set aside and allowed to cool.

The rubber channel is removed. The board is placed in a standard dishwasher and cleaned with a detergent. If the water is very hard, a softener may be desirable to avoid deposition of a film on the board.

If rosin flux is used, a bath in Freon* or similar solvent is substituted for the washing step. Extreme care must be used in choosing the solvent, because some solvents will remove the inks used for component identification. A number of manufacturers make equipment that will distill and recover the solvents. Equipment using ultrasonic transducers to clean is considered unacceptable because of a history of damage to semiconductor devices.

14. Special Components If there are components that will not stand washing or the heat of the dip pot, they are installed now. Some examples are connectors, trim pots, and some tunnel diodes. A normal bench station is used, because the number of components is usually small.

15. Touch-up This is a rework station. Its main purpose will be solder touch-up. The best iron for this purpose is not in excess of 50 W. The main danger here is the lifting of circuitry with excess heat. A suction device that operates with the iron is quite useful in removing excess solder. If it is necessary to remove a component, this device is quite effective.

Since rosin flux is used here, a solvent will be required to remove the flux at touch-up points. It is normally applied with a small brush or cotton swab. It is easily removed with a toothbrush.

16. Final Inspection The boards are now 100 percent visually inspected for soldering, proper seating of components, and other visual defects. Each defect is noted and passed to

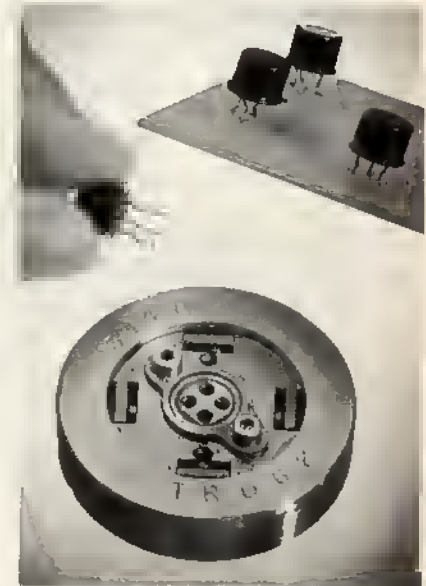


Fig. 10.4 Cutting and forming die for transistor leads, manual method (Hewlett-Packard Co.).

*Registered trademark of E. I. du Pont de Nemours & Company.



Fig. 10.5 Preforming axial-lead component leads with individual handling machine (*Develop-Amatic Engineering*).

the rework operator. It is essential that solder touch-up not be done by the inspector. The tendency is for the inspector to "dress" a board that does not require it. That results in higher touch-up costs and potential board damage. Identification of touch-up points with a dyed flux is preferable. The main requirement is adequate light. A 4× lighted magnifier is excellent for the purpose.



Fig. 10.6 Rotary work station (*Hewlett-Packard Co.*).

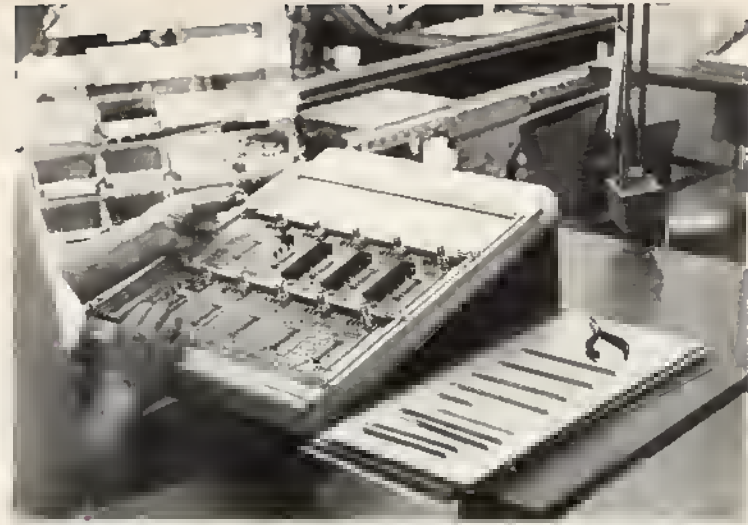


Fig. 10.7 Insertion operation with preformed parts (*Hewlett-Packard Co.*).

17. Test and Repair This station is essential to minimize checkout time of the final instrument. It is obvious that fewer errors going into a system will mean fewer considerations necessary when malfunction of the assembled system occurs. Accessibility of the malfunction is an important factor too. The specific test equipment used will depend on the facility and its volume and the available personnel.

The station may also be used to install selected value components. Most small malfunctions are corrected here and the recheck made. If the facility is large, it may be better utilization to have the troubleshooting done separately. Again the decision is best reached at the facility.

18. Conformal Coatings Humidity seals are applied by spraying or dipping. Spraying is preferable for production because of ease of application. If sealing is a requirement, the boards are usually dried in an oven prior to coating to ensure low initial moisture content.

Required equipment includes an oven, spray booth and spray gun, or vented hood and drying area for dipping. A wire rack is usually sufficient for drying sprayed boards. Dipping requires a drip-off area. Use of clips on a traveling clothesline is very effective in this application.

19. Stocking The assembled boards are now ready for the stockroom or final assembly area. Special containers for boards may seem to be an unnecessary expense, but replacement costs for damaged boards will pay for proper containers. Commercial containers are available.

PROTOTYPE PROCESS

The major difference here is that components will not be preformed and precut. All boards can be placed in front of the opera-



Fig. 10.8 Dip soldering (*Hewlett-Packard Co.*).

tor on 1-in-thick Styrofoam. If the boards are large and have many components, it will be worthwhile to build as though they were small-volume. A check of the justification sheet will confirm that statement.

If dip-solder facilities are not available, hand soldering will suffice. A low-wattage (not to exceed 50 W) iron is a must to avoid circuit damage.

LARGE-VOLUME PROCESS

The main difference between small volume and large volume is the use of the assembly line. Calculations will show that there is a slight theoretical advantage to the line assembly even when loss to line balance has been considered.



Fig. 10.9 Soldering-machine console. (Courtesy Electrobert, Inc.)

20. Planning Additional planning is required for line assembly. All operators must have approximately equal portions of work if the total unit time is to be minimized. Unequal work content will allow wastage of time in the lighter-loaded stations when the heavier-loaded stations are working at a normal pace.

Line balancing must be done for various levels of production, since the number of operators on the line will affect the production rate per unit of time. Production requirements may vary widely over a period of time, or sufficient operators may not be available to operate at a given level.

A document indicating which components are to be assembled by each operator is necessary for consistent level of productivity.

21. Assembly-Station Layout The assembly station will consist of a track on which the board slides, if push-type, or is moved by conveyor, if powered. A quantity of numbered

trays will be evident, as in rotary-table assembly, except that they are stationary. The vertical parts carrier is also of advantage here.

22. Soldering Soldering may be dip pot, but more likely a wave-soldering machine will be used. Either the boards are transferred from the assembly line to fixtures for soldering or the track extends so the soldering is an extension of the assembly stations. In either case the board is automatically fluxed and soldered (see Fig. 10.9 and Chap. 14). Two main advantages of such equipment are reproducibility of process and lower unit cost. Uniform quality is assumed, and unit cost is lower because initial soldering is better and less rework is required.

The rest of the operations are similar to small-volume. They are often connected by conveyors to handle material flow.

IN-LINE VS. ROTARY WORK STATION

When should the rotary table assembly be replaced by line assembly? That is a good question that is difficult to answer. However, as a rough rule of thumb, unless the calculated time values weigh *extremely* in favor of line assembly, the rotary table is preferable. Table 10.1 will aid in the necessary calculations. After the totals are compared, look at the comparative benefits.

Line assembly will provide operator pacing, reduce the time of the first completed piece, allow quick checks of completed-parts status, use less floor space per operator, and give the impression of mass-production savings. Also, it requires more service time for initial start-up and maintenance from manufacturing engineering, requires a skilled labor pool to offset absenteeism and production changes, and is held to the slowest operator. It does not allow job identification as does the rotary table.

Again this is an analysis best made at the facility, by a review of the facts.

SUMMARY

The small-volume printed board assembly process has been detailed with some alternatives. The prototype and large-volume processes were compared with the small-volume process. The constraints of design and cost were explored. Data were given to aid in reaching a decision as to a specific process. To develop a process:

1. Review the constraints, item by item.
2. Make the cost analysis.
3. Add good judgment.

This procedure will yield the optimum process for any facility.

Automatic Assembly

D. W. ACKERMAN

Universal Instruments Corp. Binghamton, New York

Introduction	11-1
Variables Affecting Feasibility of Automatic Insertion	11-2
1. Uniformity of Board Layout	11-2
2. Hole Location and Sizes	11-3
3. Variations in Insertion Spacing	11-3
4. Component Spacing	11-5
Variation in Components	11-8
5. Axial-Lead Components	11-8
6. Radial-Lead Components	11-10
7. Integrated Circuits	11-10
8. Other Insertable Components	11-10
Component Preparation and Input to the Insertion Machine	11-10
9. Axial-Lead Components	11-10
10. Radial-Lead Components	11-15
11. Integrated Circuits	11-15
Insertion Machine Approaches	11-15
12. Single-Component Type	11-15
13. Multiplicity of Types of Components	11-18
Summary	11-21

INTRODUCTION

The primary reason for employing semiautomatic or automatic insertion of components into printed boards is to reduce costs by eliminating the labor otherwise involved. There are other significant advantages in the use of mechanically assembled boards. In view of the increasing emphasis on reliability, good mechanical assembly assists materially in achieving circuits free of open connections in the operating environment. When uniformity of circuit performance is a factor, the mechanically assembled board can be superior.

Properly designed insertion and clenching equipment offers still another advantage in that it makes possible a high component density with minimum copper land area around the hole on the circuit side of the board and a minimum distance between terminal area and adjacent conductor on the underside of the board because lead length is limited. The latter advantage, of course, is particularly useful when miniaturization is an objective.

Most users of insertion equipment are influenced primarily by the cost factor. The savings to be realized, although fundamentally dependent on the efficiency of manual methods previously employed, are influenced by the following factors:

1. Board volume
2. Board configuration and tolerances
3. Product mix
4. Component mix
5. Hole pattern and component location
6. The insertion technique employed

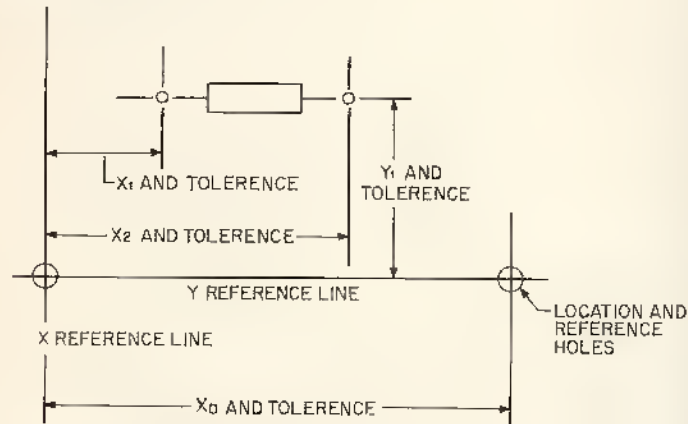


Fig. 11.1 Typical reference hole usage. Locating holes—recommended: $0.125 + 0.002$ in; locating pins—recommended: $0.125 - 0.001$ in.

This chapter considers the effect of board and component variations on the desirability of mechanical assembly, the various machine configurations available, and the factors involved in the selection of the best machine approach.

VARIABLES AFFECTING FEASIBILITY OF AUTOMATIC INSERTION

1. Uniformity of Board Layout The board designer influences the success or failure of a particular program in many ways by the methods he uses in the original board layout. A design can be highly adaptable to automatic component insertion, or it can be made impractical for such insertion by relatively minor variations in orientation (axis) and span choices.

Of prime importance in board construction is the establishment of accurate locating points with respect to which all holes are drilled or punched. The locating references may take the form of two or more holes, slots, or straight surfaces. By one of the references, the board is positioned on a board holder and readied for component insertion. The accuracy of hole placement with respect to the locating points will determine the insertion reliability. Accurate location will lessen the clearance required between lead and hole diameters.

To obtain the least amount of tolerance buildup for inserting, care must be exercised in the initial board drawing to ensure that the component holes and the board locating holes are only "one tolerance" apart. That can be easily accomplished by having the dimensional reference lines go through the board locating holes. Thus, once the holes are made, they may be used as the board reference for screening, drilling (or punching), and insertion (Fig. 11.1).

2. Hole Location and Sizes Just as location of holes on a standard grid pattern offers important advantages in board fabrication, so does it in automatic insertion. Design and construction of fixtures, indexing mechanisms, pantograph-table locating templates, or numerical control systems is simplified and the function is made more reliable. Cost of engineering time at the outset can be recovered many times over in savings in cost of insertion equipment and the labor involved in its operation. For example, a master template for the grid involved with a manual pantograph insertion machine costing \$1500 may be used for all component patterns in combination with an appropriate mask of nominal cost (Fig. 11.2). If a grid system were not employed, individual jig-bored templates costing an average of \$250 each would have to be fabricated (Fig. 11.3). In this example, the master template made possible by use of the grid system pays for itself when as few as ten different component-location patterns are involved.

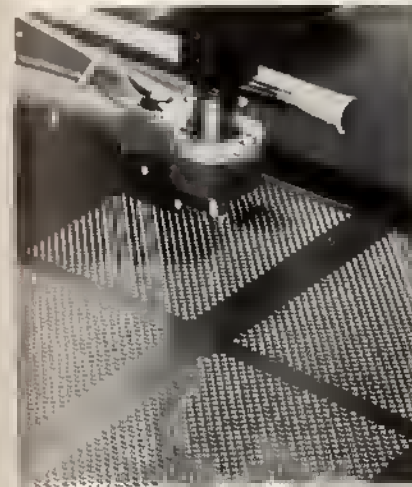


Fig. 11.2 Master grid template.

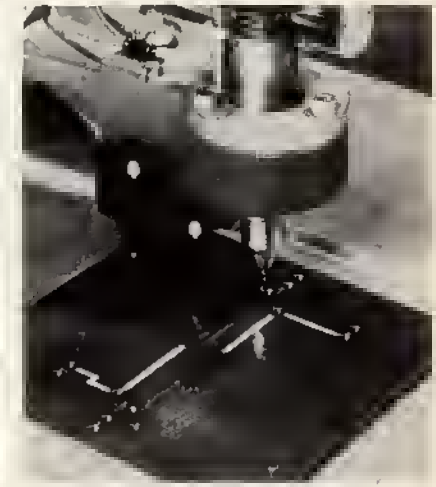


Fig. 11.3 Custom-fabricated template.

The accuracy of board fabrication is a major factor in determining final success of an automation program for board assembly. Compromise places burdens upon machinery manufacturers and usually results in missed insertions, rejected boards, and machine downtime. In general, the hole size necessary for component insertion is a function of (1) component lead diameter and all machine tolerances involved in guiding the leads in the lead for insertion, (2) the table positioning accuracy, (3) the board locating accuracy (on the holder), and (4) the drilled hole pattern accuracy with respect to the board locating reference.

For most pantograph and numerical control (NC) machines, maximum machine tolerance for head, table, and board locating accuracy is $+0.004$ in, so a starting hole diameter 0.008 in larger than the wire diameter being inserted is required. To that hole diameter must be added the manufacturing tolerance of the printed circuit insertion hole pattern with respect to the locating references. Generally, components can be inserted into holes only 0.010 in larger than the wire diameter, assuming high-accuracy boards. In some instances, it may be necessary to allow as much as 0.020 in clearance for less accurate boards. Therefore, the formula for determining hole diameter becomes:

$$\text{Hole diameter} = \text{wire diameter} + 0.008 \text{ in} + \text{location tolerance}$$

As an example take a $\frac{1}{4}$ -W resistor with 0.025 -in wire diameter, boards with hole location accuracy of $+0.003$ in, and the standard machine tolerance of 0.008 in.

$$\text{Hole diameter} = 0.025 + 0.008 + 0.006 = 0.039 \text{ in}$$

3. Variations in Insertion Spacing Relatively few users of printed boards have realized the advantages of standardizing on, or grouping, insertion spacings. When spacings are

standardized, automatic insertion is most practical because the basic mechanisms also can be standardized. That results in excellent advantages in cost savings due to minimizing of tooling and setup changes and maximizing reliability of operation of the mechanisms involved.

Standardization of manufacture naturally limits flexibility of circuit design; therefore, a careful analysis of planned production quantities, design and manufacturing schedules, and circuit function is required before standards can be established. Variations in component size and configuration present an interesting complication, and the design engineer must possess a thorough understanding of assembly techniques and equipment limitations if he is to take advantage of the cost-saving potential.

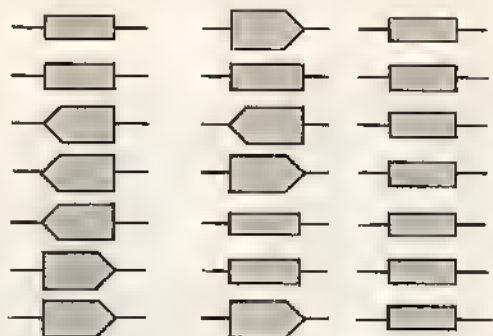


Fig. 11.4 Component placement planning: All one axis, all one span; sequencing shown. A fixed-center-distance head can be used.

There are now three ways in which axial-lead components can be inserted with regard to span (distance between hole centers): (1) fixed span using a standard, fixed-center-distance insertion head with a given set of fixed tooling, (2) two spans using a dual-center-distance (DCD) insertion head which can be switched manually on a pantograph or automatically on an NC machine between any two preset center distances within its range and capability, and (3) more than two spans using a variable-center-distance (VCD) insertion head which can be obtained in a mini-computer-controlled configuration. The VCD can attain any span required to 0.001 in within its range and capability. Equipment to be used for a particular application depends on quantity of boards, desirability of eliminating multiple passes through the machine, and board and component quality.

Proper planning for general-purpose insertion machines involves the selection of a single axis for component mounting. When a single axis is not possible, the x and y axes are preferable to random-angle location for mounting components. Savings in fixtures, setup time, and board handling can compensate many times for the costs involved in

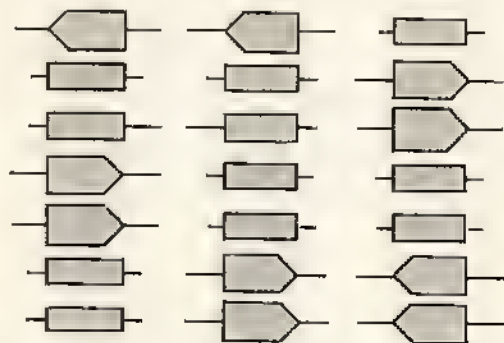


Fig. 11.5 Component placement planning: All one axis, two or more spans; sequencing can be used. Requires multiple passes with a standard head, but may be completed in one pass with a DCD or VCD insertion head.

initial engineering (Figs. 11.4 to 11.6). Components mounted at various angles and insertion spans usually require multiple-head insertion machines with automatic board handling. Each head places a specific component in the board at any given location (Fig. 11.7).

4. Component Spacing

a. Axial-Lead. Since conventional axial-lead insertion heads are equipped with outside formers which guide the leads to the point of insertion in the board, the clearance area around a given hole must be taken into consideration if the equipment is to function properly. Figure 11.8 shows a top view of an outside former at the point of insertion; it

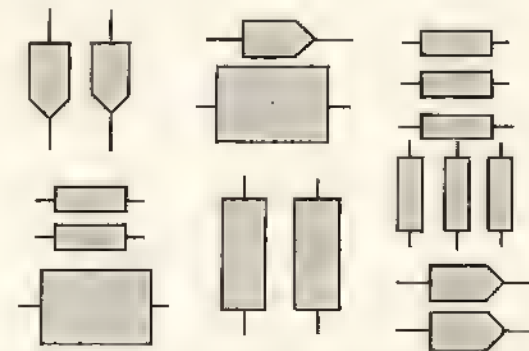


Fig. 11.6 Component placement planning: Two axes, two or more spans; requires sequencing and VCD insertion or multiple-head system.

illustrates the clearances required between the lead being inserted and any adjacent component body or lead. The table in Fig. 11.8 lists typical dimensions used in the construction of outside formers.

A standard blank is generally used for all outside formers. As a result, a progressively larger V is ground to suit larger wire diameters. The centerline of the wire moves deeper into the ground V and decreases the x dimension. For applications involving dense component assembly of small components (diodes or $1/4$ -W or smaller resistors), considerable material can be removed from the outside edges of the former without a loss of durability to the tool, x_{min} and y_{min} are given as practical minimums.

Figure 11.9 shows the clearances required for various component locations and the sequence of assembly.

b. Radial-Lead. Location of radial leads (transistors in general) is probably less defined for tooling requirements than for any other insertable component. The nonuniformity of

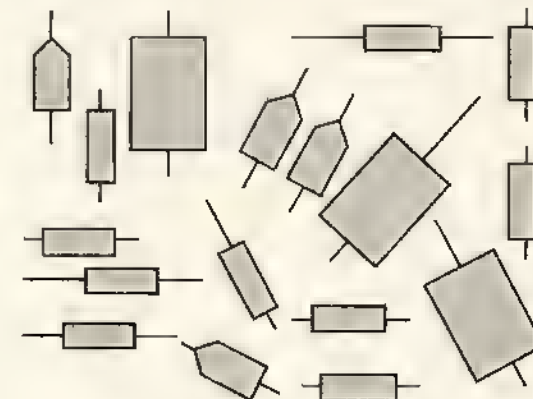


Fig. 11.7 Component placement planning: All directions, many spans, many types.

the component itself and of the packaging method leads to many forms of insertion tooling.

There are two general methods for automatically inserting transistors into printed boards: (1) radial insertion, in which the part is held by the body and the prepared leads are inserted straight in, and (2) axial insertion, in which the leads are formed outward, two one way and one the other, and the lead tips are taped much as in the case of an axial component. The reel-packaged transistors are then placed on an insertion machine with a modified axial component head and inserted in much the same manner as an axial lead component is.

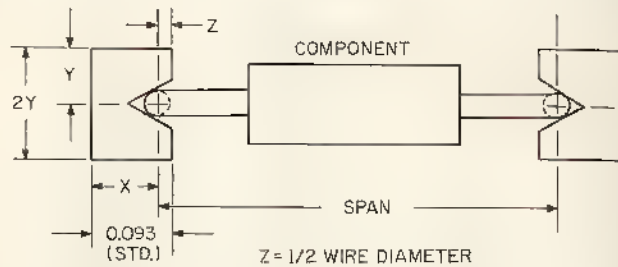


Fig. 11.8 Outside formers. Top view of formal component lead in tooling of insertion head.

Wire diameter, in	X Std., in	X Min., in	Y Std., in	Y Min., in
0.015	0.085	0.030	0.062	.030
0.020	0.083	0.035	0.062	.035
0.025	0.080	0.040	0.062	.040
0.030	0.078	0.045	0.062	.045
0.040	0.073	std.	0.062	std.
0.050	0.068	std.	0.062	std.

In radial insertion, the part is handled by the body; hence each different physical part requires different tooling. In axial insertion the part is handled by the leads, and so long as the part is within a maximum outline, one standard head will handle all physical types.

1. Axial version. Figure 11.10 shows the taping specification for the TO-18 family of transistors formed as an axial device. It also shows the insertion characteristics.

2. Radial version. Figure 11.11 shows a typical specification for the straight-in radial lead form of insertion.

For both types of insertion, tooling clearances must be provided. Above-the-board clearances similar to those shown in Fig. 11.10 must be used in the axial method. The radial technique requires a collet around the body diameter approximately 0.080 in larger than the body.

Radial insertion also requires larger holes for reliable assembly. Since the component body is located by a collet, the leads are left unguided and their accuracy of location adds to the necessary hole size. Generally, transistors can be prepped, depending on lead length and form, to within ± 0.005 in from true body centerline by adding 0.010 in to the hole size requirements of Sec. 2. Tooling can be provided to guide the leads of radial components into the boards, but its cost and the room necessary for operating on the surface of a PC board limits the application.

c. Integrated Circuits (DIPs). Integrated circuits (generally dual in-line packages or DIPs) for automatic assembly, although still much in their infancy, do have some established guidelines. The clearances for DIPs are exactly the same as those for a standard axial component, since the principles of insertion are identical. The general equation is given in Sec. 2, and the only difference is that, for the lead diameter, the effective lead diameter of the DIP—the lead diagonal—is used. For example, DIP leads of 0.010 ×

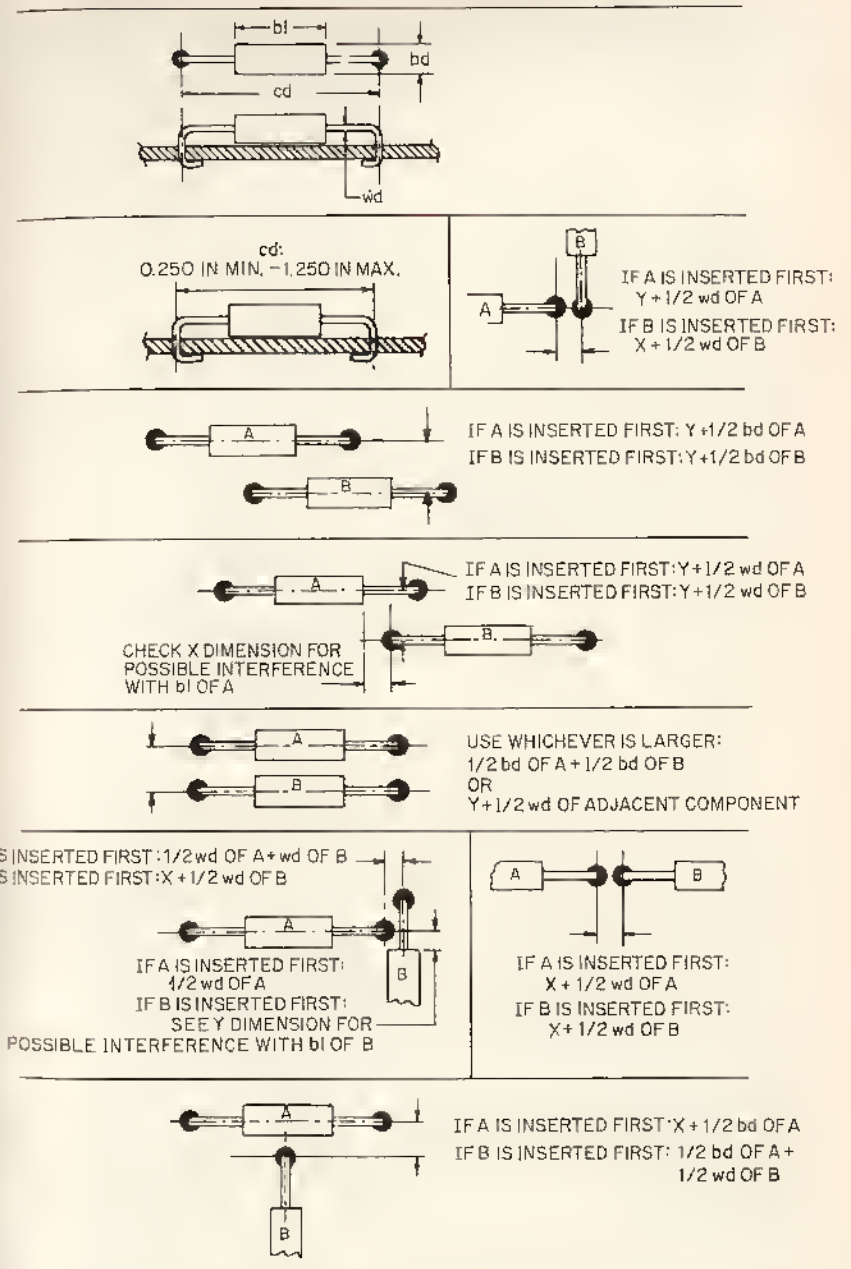


Fig. 11.9 Clearances required for various component locations and sequences of assembly. Key: bl = body length; bd = body diameter; wd = wire diameter; cd = center distance. Dimensions indicated are minimum tolerance unless otherwise indicated.

0.020 in max have an effective diameter of 0.023 in. When they are used with a board that has a hole accuracy of ± 0.003 in (or 0.006 in absolute), and with a machine allowance of ± 0.004 in (0.008 in absolute), the minimum hole diameter should be $0.023 + 0.006 + 0.008 = 0.037$ in. Pointed leads on the DIPs would nominally reduce the effective diameter by 0.003 to 0.006 in depending on the physical configuration used.

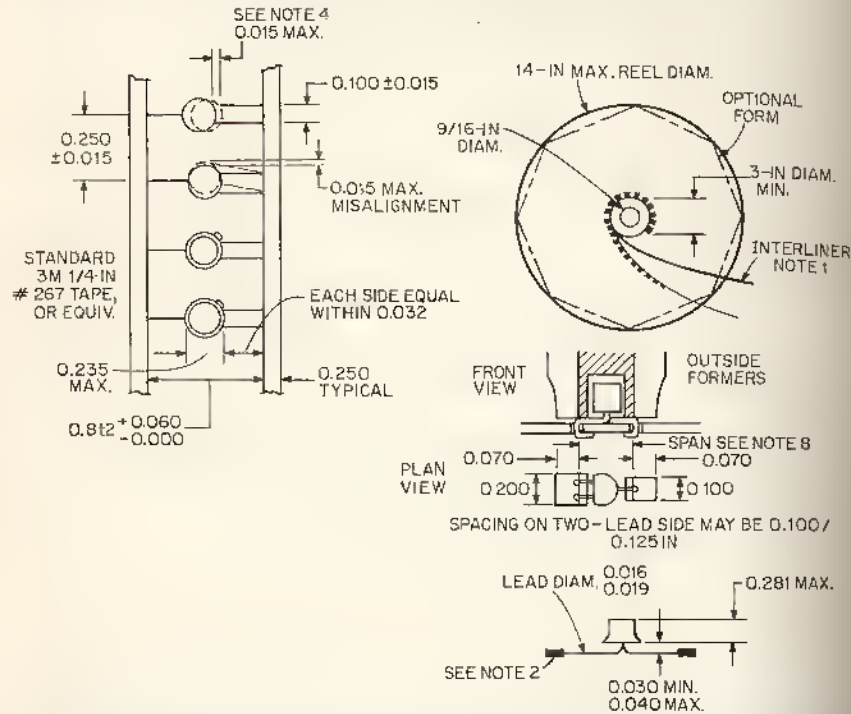


Fig. 11.10 Taping specification for machine insertion of TO-18, TO-92, TO-98 and similar transistors. NOTES: 1. Heavy kraft or single-ply A type corrugated interliner to be used. 2. Lead length in contact with tape, each side: 0.120 in minimum. 3. Leads must be straight within 0.015 in between body and tape. 4. Component bodies must be in line within 0.015 in. 5. No more than one component may be missing at a gap. 6. Distance between tapes may not go below minimum shown. Variation of 0.060 in over reel permissible. 7. Both in-line leads (TO-92, TO-98, etc.) and TO-18 pin configurations suitable for this process. 8. Insertion spacing is a function of part width. For example: for 0.235 in width, it is 0.400 in. Certain plastic devices can be handled down to 0.300 in.

It is recommended that all DIPs be arranged in a single orientation. The 180° orientation is acceptable if sequencing is used on the insertion machine, but it counts as an additional component requiring a separate input on the sequencer. Machines that can perform the sequence-and-insert operation are available with up to 72 separate input magazines.

Figure 11.12 illustrates the end-to-end and side-to-side spacing for DIPs. In general, if axials and DIPs are mixed on the same board, the same spacing rules could apply, provided the axial component height were no greater than 0.250 in at the point of clearance.

VARIATION IN COMPONENTS

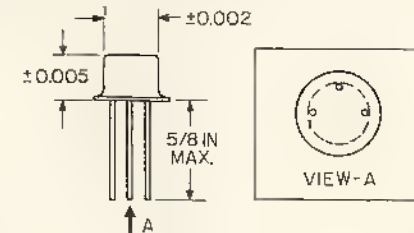
5. Axial-Lead Components Variations in lead diameter put the principal limitation on ability to utilize a given fixed-center insertion mechanism, or station, for a variety of

components. When variation exceeds 0.010 in in diameter, it has generally been found desirable that a different mechanism, or at least some change in tooling, be employed.

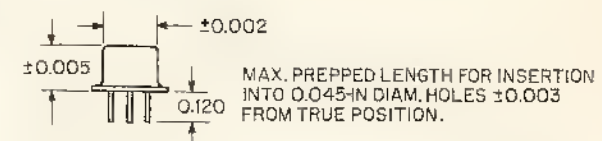
Minimum possible insertion spacing for a given component is determined by lead diameter and body length. The forming mechanism must have room to work between the end of the body with some clearance and be of sufficient thickness to withstand the

I. TO-18, ETC.

A. PREPPER INPUT

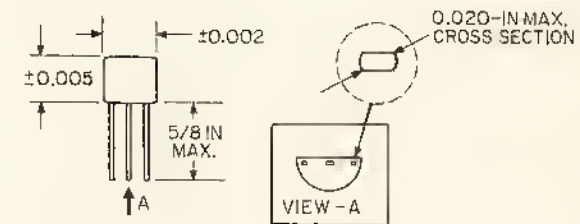


B. INSERTER INPUT



II. TO-92, ETC.

A. PREPPER INPUT



B. INSERTER INPUT

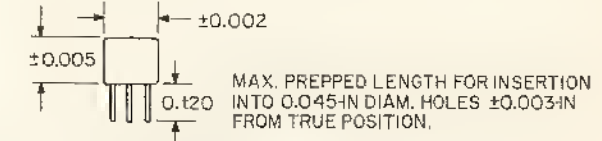


Fig. 11.11 Transistor prepping and insertion specifications.

forming action of the wire size involved. In some cases, lead material also may affect lead spacing. In general, Fig. 11.13 shows the requirements for axial component insertion.

Body diameter is important when a variety of components are considered. Most insertion mechanisms compensate for difference in diameter by having a number of downstops that can be manually or automatically set to limit the stroke of the driver so that the component body is gently pushed against the board surface. Other mechanisms employ springs to compensate for difference in body diameter.

Figure 11.14 shows a typical application utilizing the recently developed technique of varying the component lead spacing within the insertion head (VCD) at the time of insertion. The system not only allows for insertion spans from 0.300 to 1.300 in but can also make fine adjustment for different lead diameters to provide exact centering for critical insertions. All components and jumper wires shown in Fig. 11.14 were inserted in one sequence.

6. Radial-Lead Components Satisfactory operation of radial-lead component insertion equipment is substantially independent upon continuous reliable feed of components into the insertion nest. Accordingly, slight variations in body size, irregularities in configuration, and solder or weld beads cannot be tolerated if the automation effort is to be successful. Change from one type of component to another requires, at the minimum, extensive tooling change. An exception is shown in the example illustrated in Fig. 11.10. The radial-lead component (transistor) has been packaged by the leads, so that variation in the body does not affect insertion reliability. However, variations will affect the packaging equipment and must therefore ultimately be analyzed for tooling change and cost. As a total packaging and insertion system, this approach has proved to be economical in high-volume applications.

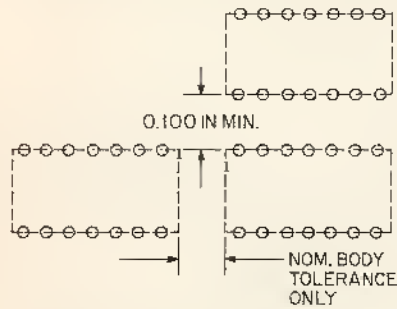


Fig. 11.12 End-to-end and side-to-side spacing for DIPs.

7. Integrated Circuits Generally, multi-leaded devices present problems because of lack of uniformity of body size and shape, difficulty of maintaining desired lead condition, and feed problems at the input to insertion mechanisms. Careful control of component configuration and packaging and planning for automatic insertion from the outset are prerequisites for successful assembly. The component manufacturer, component user, and equipment supplier must work closely together if a suitable solution to the overall problem is to be found.

Successful mechanized assembly for integrated circuits is being achieved by using the relatively standard DIP package. Although standardization is not complete, systems for excellent labor savings are available. Figure 11.15 shows some configuration requirements to make mechanized assembly practical.

8. Other Insertable Components The disciplines required for automatic assembly of active components hold true, as they do for all board "hardware." If an item can be uniformly and economically packaged for presentation to an assembly head, and if the item itself has the characteristics necessary for presentation to its mounting on the board, it can be considered for automation. The hardware manufacturer, user, and equipment supplier must work closely together to ensure success.

COMPONENT PREPARATION AND INPUT TO THE INSERTION MACHINE

9. Axial-Lead Components For insertion equipment to function reliably, components must be presented in a uniform manner and always in the same position with respect to the insertion mechanism. The leads must be straight, and the component must be contained in some way at input to the insertion mechanism. The containment methods commonly employed are manual, chute feed, magazine feed, plastic-strip packaging, and body or lead taping. The latter is perhaps most commonly used. (Figs. 11.16 to 11.18).

Manual input offers the advantage of providing for very short run insertion of components but is slow; the operator paces the machine. Presentation of components to the operator, particularly in sequence, can be troublesome. Chute feed requires manual or mechanized chute loading and is susceptible to jamming owing to component irregularity or the bending of leads in handling. Magazine feed offers advantages in protecting the component prior to input to the insertion machine, but it also is susceptible to jamming when the component condition is not uniform. Some special components, or very small components which can not be lead-taped, are packaged in continuous plastic strip. Although the packaging is excellent for such components, it creates tricky component-removal problems at the insertion mechanism.

Body taping is generally used for large, heavy components but it has the disadvantage of allowing component leads to be bent to such point that the insertion mechanism cannot function properly. Lead taping is in general use because it provides a way of obtaining

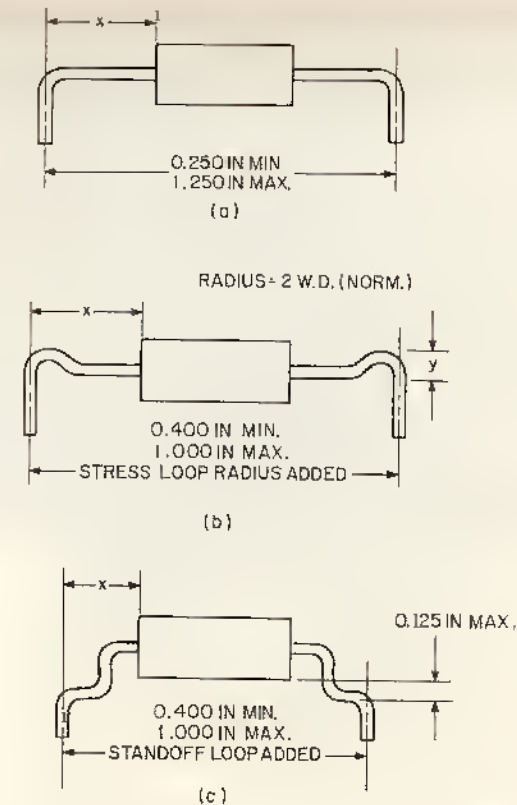


Fig. 11.13 Requirements for axial component insertion.

(a) Standard

Wire diam., in	x min., in
0.015	0.050
0.020	0.075
0.030	0.090
0.040	0.115
0.050	0.120

(b) Stress loop

Wire diam., in	x min., in	y, in
0.020	0.125	0.060
0.030	0.150	0.090

(c) Standoff

Wire diam., in	x min., in
0.020	0.125
0.030	0.150

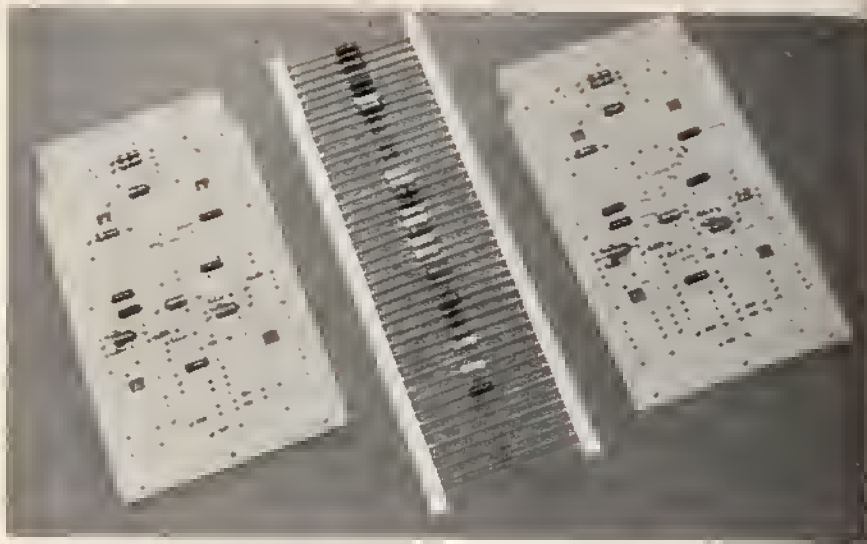


Fig. 11.14 Example of inserted components.

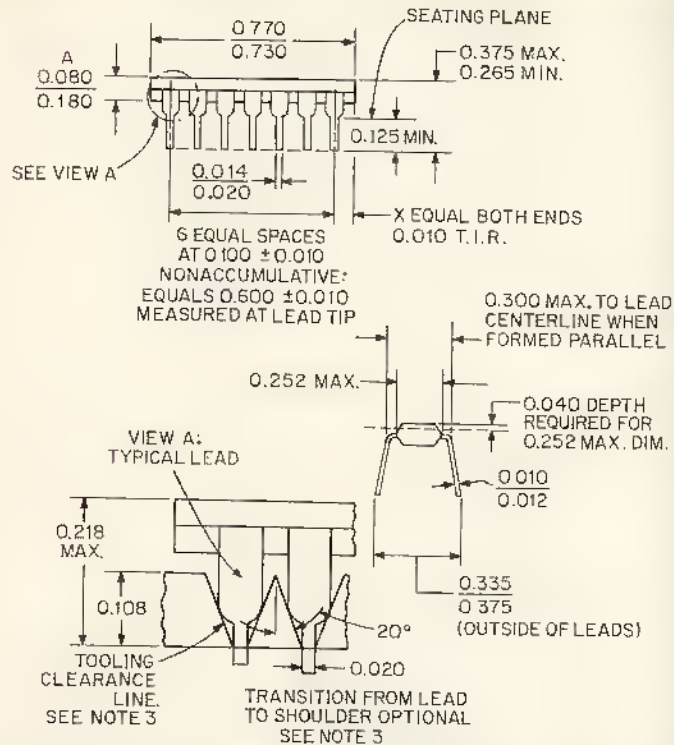


Fig. 11.15 Preferred dual-in-line package configuration. NOTES: 1. Components may vary ± 0.010 in dimension A without insertion adjustment for runs or sequences involving various components from single or multiple vendors. 2. When dimension A varies more than 0.020, either manual or automatic adjustments must be made to compensate for component configuration up to the minimum and maximum as shown. Input magazines must also be changed feeding the insertion head. 3. Lead profile must be within tooling clearance line at all points (see View A). 4. When two-piece bodies are used, top half to be within ± 0.010 of true centerline position at all points. 5. Lead tip position errors to be within ± 0.010 of true location of inside view. 6. Leads to be straight with no inward bends. See end view.

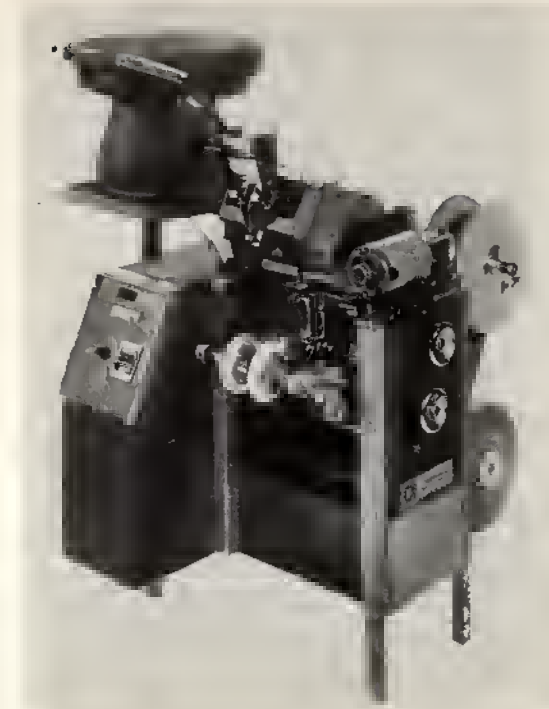


Fig. 11.16 Typical system for feeding, straightening, and lead taping axial-lead components.

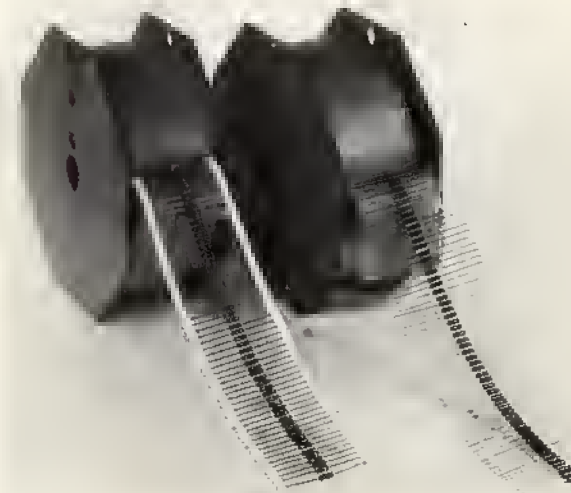


Fig. 11.17 Axial-lead components, lead-taped and body-taped, placed on reels with interliner ready for feeding into an insertion mechanism.

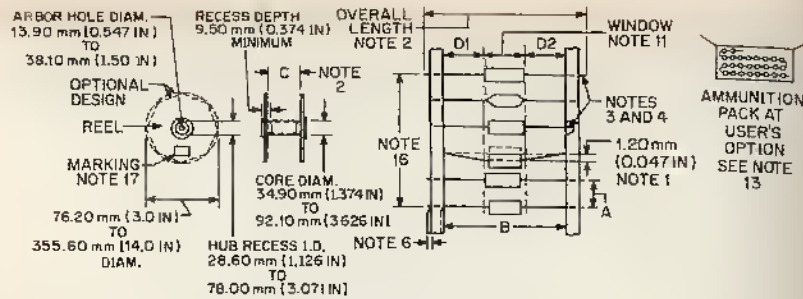


Fig. 11.18 Lead tape reel packaging of axial-lead components (from EIA Standard RS-296).

TABULATION

Component type	Component body diam., in	Component spacing A	Tape spacing	
			B min.	B max.
Resistors, diodes, capacitors,	0.000-200	0.200 ± 0.15	1 ³ / ₃₂	3 ¹ / ₁₆
Inductors, insulated diodes	0.201-.375	.375 ± .015		

Notes: (1) Leads shall not be bent beyond 1.20mm (0.047in) from their nominal position when measured from the leading edge of the component lead at the inside tape edge and at the lead egress from the component. (2) The C dimension shall be governed by the overall length of the taped component. The distance between the flanges shall be 1.50mm (0.059in) to 8.00mm (0.315in) greater than the overall component. (3) Lead ends shall not extend beyond the tape more than a maximum of 1.6mm (0.063in) (See note 15.) (4) Each component lead shall be sandwiched between tapes for a minimum of 3.20mm (0.126in). (6) Exposed adhesive shall be no greater than 0.80mm (0.031in) maximum. (11) The window, which is equal to the maximum component clean-lead-to-clean-lead length, shall be centered between lead tapes such that the difference between D1 and D2 does not exceed 1.4mm (0.055in). (13) Quantity of parts shall be controlled so that the tape components and final cover shall not extend beyond the smallest dimension of the flange (either across the flats or diameter). Once the quantity per reel for each part number has been established, future orders for that part shall be reeled in that quantity. When reeling quantity is less than the established quantity, a suitable size reel or ammunition pack may be used at user's option. (16) Cumulative pitch tolerance A shall not exceed 1.50mm (0.059in) over six consecutive components. (17) Minimum reel markings shall consist of:

- Customer part #
- Purchase order #
- Quantity
- Manufacturer's name or trademark
- Reeling date
- Date codes (when applicable)
- Vendor type or part number (when applicable)
- Electrical value (when applicable)
- Tolerance (where applicable)

(18) Unless otherwise specified—EIA recognizes this revision as a significant change—Deviations may be necessary to accommodate existing equipment. In this regard, specific note is taken of the previous 0.375-in component pitch.

positive feed into the machine at the same time that the leads are firmly held and thus ensures pickup by the insertion mechanism.

10. Radial-Lead Components Packaging for the insertion of radial-lead components has yet to be standardized. Figure 11.19 shows an example of transistor packaging being employed in some installations. Methods based on bowl feeding or magazines have been utilized on a custom basis where very tight control over the dimensions and tolerances of the component was possible.

11. Integrated Circuits The DIP is the only integrated circuit being automatically inserted throughout the industry. The preparation of the device takes two basic forms: the device is

packaged loosely in plastic slide sticks or assembled in a unit carrier. The unit carrier is then handled either in bulk or out of magazines at the insertion station (Fig. 11.20). Each method has its advantages, and which one offers the greatest potential labor savings depends on the needs of the customer.

INSERTION MACHINE APPROACHES

12. Single-Component Type

a. Manual Positioning. A basic single-component single-board manual-positioning insertion machine is shown in Fig. 11.21. It involves a work-positioning table that provides machine balance and support for the board-holding fixture and crimping or

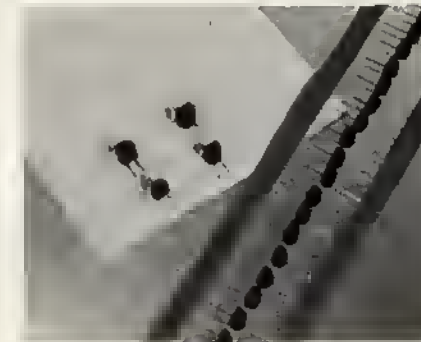


Fig. 11.19 Lead-taped transistors ready for high-speed, high-reliability insertion. Inserted components are also shown. This technique is recommended, since it is not dependent on body configuration or tolerance.



Fig. 11.20 The machine shown here receives loose DIPs out of a plastic stick, performs a spreading, straightening operation on the leads, and assembles the DIP into a unit carrier fed from the vibratory bowl. The loaded carrier is then fed into magazines holding 25 each.



Fig. 11.21 Single-station machine for inserting a specific component into boards which have been manually positioned in the locating fixture.



Fig. 11.22 Single-station manual pantograph board-locating insertion machine. Best economy is obtained when several units of a specific component are inserted into a given board in one setup.

clenching unit, a C-frame assembly, and an insertion head. Use of such a machine should generally be confined to applications involving a variety of boards, components, and component locations, usually short-run. Separate tooling or locating devices are employed for each board and/or component position. A single type of component is inserted, and the boards are processed through such machines as many times as there are components in the board. Insertion rates obtainable are of the order of 500 h^{-1} .

b. Single Board—Pantograph Board Positioning. The machine shown in Fig. 11.22 provides for insertion of multiple units of the same component in a given board in a single



Fig. 11.23 Pantograph system for insertion of a single type of DIP from a magazine input.

setup. The operator locates the board for insertion by positioning the stylus in the template as desired, and insertion is triggered by depressing the stylus. Insertion rates obtainable are as high as 5400 h^{-1} , depending upon component density and spacing, but they are typically of the order of 1500 to 2000 h^{-1} . The mechanical cycle time of the insertion and clenching mechanism shown is approximately 0.5 s . Beyond that, insertion rate depends upon board handling time and the distance between components on the board.

The pantograph type of machine is regarded as the general-purpose workhorse of the insertion machine family. Its usefulness extends to all types of boards, component combinations, component positions, and production runs. A significant advantage of this type of machine, compared with more complex possibilities, is that it is rugged, reliable, and subject to a minimum of downtime. The latter feature is particularly important in comparison with multiple-station machines that can go out of production when a single station is down. Sequenced components also can be inserted, on a limited basis, with the equipment. Semiautomatic depth stop selection for various body diameters and insertion heads with dual center distances (DCD) can be provided, but they put an additional burden on the operator to keep in sequence and select the mode of insertion for each component. Fig. 11.23 shows a pantograph insertion system for DIP.

c. Numerically Controlled Insertion Machines. With the advent of numerical control systems, additional labor savings have been sought through the elimination of full-time operators and also the elimination of human error in board positioning prior to insertion. The point at which a numerically controlled machine represents the best approach from the cost standpoint is not easy to determine. It depends upon all the variables that affect the feasibility of automatic insertion. Use of numerical control is on the increase as the reliability of insertion machines and control systems improves. In some cases, two machines are being serviced by a single operator. Even when a full-time operator is



Fig. 11.24 NC fixed-head insertion machine.

utilized, rapid traverse of the control system—coupled with almost foolproof board positioning—makes the equipment economically practical. Insertion rates up to 4200 in⁻¹ per machine are being obtained (Fig. 11.24). The additional automatic command available with NC also allows sequenced components to be inserted with complete reliability (Fig. 11.25).

13. Multiplicity of Types of Components

a. In-line Machines. When high volume of a specific board involving the same components is encountered, an in-line system with a series of individual work stations through which the board is indexed is often utilized. Shown in Fig. 11.26 is a typical line for assembly of TV boards.

The in-line approach is particularly well suited to circuit designs requiring many angles for component locations. The advantages of such a system include:

1. Simultaneous insertion in many boards during each machine cycle.
 2. Servicing of many stations by one operator or a few operators.
 3. Compensation for board irregularities through use of carriers or pallets into or out of which the boards are loaded prior to processing through all stations.
- Disadvantages relate to complete machine downtime due to malfunction of any one



Fig. 11.25 Sequenced components inserted with a fixed-center-distance head. Use of a dual-center distance head would increase the insertion yield by providing more than one component span per setup (Universal Instruments Corporation).

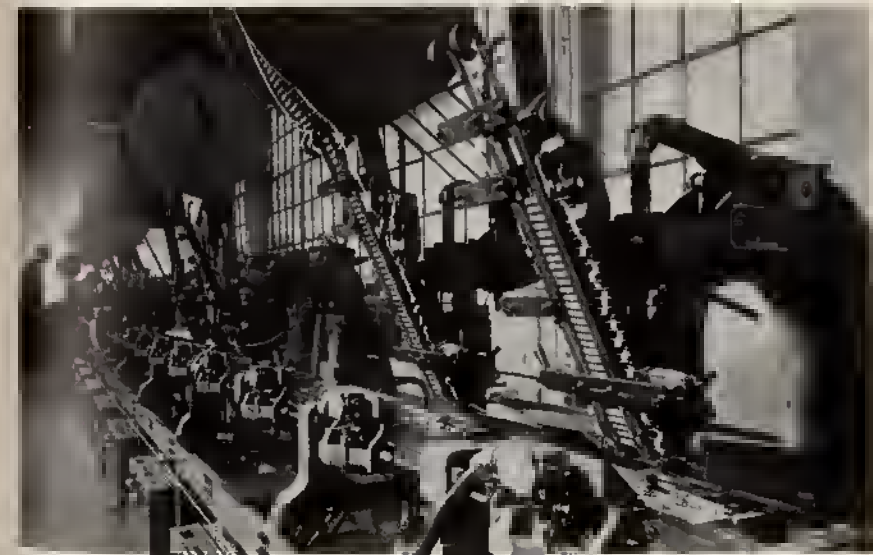


Fig. 11.26 Multiple-station in-line insertion machine with boards indexed through successive stations for insertion of a wide variety of components. Boards are pallet-located to provide for proper location in the insertion mechanism with respect to appropriate board holes.

station and the tendency toward excessive setup time. Normal machine cycle rates are on the order of once every 3 s.

b. Variable-center-distance insertion with Sequencing. The latest in the art of axial-lead component insertion, variable-center-distance (VCD) insertion permits components of many different types and sizes to be presequenced on tape and then inserted into a given board in a single setup. The insertion machine is a completely automatic system providing

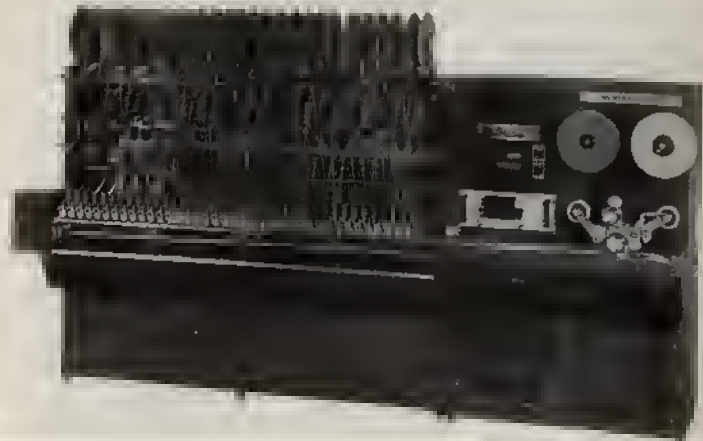


Fig. 11.27 Typical component-sequencing system.

board positioning, nearly 1000 different combinations of center span and wire diameter, and the acceptance of a wide range of body diameters.

A typical sequence system is shown in Fig. 11.27. The machine allows random selection up to 160 different inputs for a sequence of components. It is controlled by a punched tape, and it provides constant checking for missing components or other out-of-sequence conditions. Typical rate is approximately 15,000 to 20,000 components per hour.

The output package of a sequencer for use in a VCD insertion system is shown in Fig. 11.28. Shown in Fig. 11.29 is the VCD insertion system. Table positioning speed is 600 in/min. The head and clinch is programmed for insertion span in 0.001-in increments from 0.300- to 1.300-in centers. The system includes a mini computer control that provides management data and also accomplishes the computer control function. Insertion rates of over 8000 components per hour are obtained. Additional systems can be placed on the same computer to provide significant savings when production quantities warrant an equipment "complex." Complexes can be configured to include other types of insertion machines and sequencers to take full advantage of the computer's capabilities.

c. Magazine Sequencing. In another approach to the sequencing and insertion of axial lead components the components (usually $\frac{1}{4}$ - or $\frac{1}{2}$ -W resistors) are loaded into magazines. The approach is useful when components are received in card packs, rather than lead tape, prior to sequencing. The output of the sequencer can be either into magazines or onto lead tape depending on the insertion mechanism used.

d. Sequencing and Insertion Combination Machines. There are some instances in which the most economical approach is to tie the sequencing and insertion systems together. High-production boards with a maximum of 10 different components may be assembled by machine. Automatic positioning and insertion heads with programmable center span also are available.

The combination sequencer-insertion approach can be used for integrated circuit (DIP) board assembly. Since the DIP is a "higher echelon" device as compared with discrete components, fewer parts of high cost are usually assembled on any one board. Sequencing at the time of insertion is desirable to complete assembly of the board in one setup when

different types or circuit configurations are used. Figure 11.30 shows a machine with 48 DIP stick inputs to provide random selection during insertion.

In another machine available for DIP insertion DIPs which have been previously loaded into plastic carriers are randomly sequenced and inserted. The unit-type plastic carrier is often used when many different body configurations must be handled for insertion and other operations such as incoming test.

SUMMARY

The methods described and types of equipment presented in this chapter are a basic guide for those who are considering automatic assembly. There are other combinations

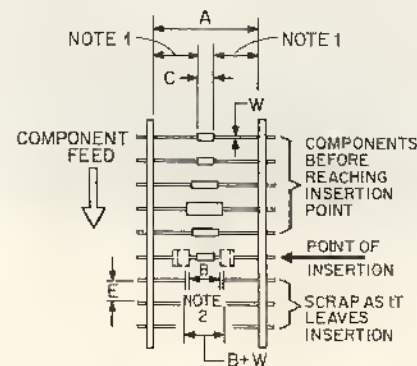


Fig. 11.28 Typical component specifications for lead taping prior to machine loading.

Tape spacing recommended for sequenced components	— A	minimum = 1.562 in maximum = 2.125 in
Note 1		
Body Length	— C	Bodies must be centered within $\frac{1}{2}$ in minimum = 0 maximum = B—0.150 in
Wire Diameter	— W	minimum = 0.015 in for no. 1 former tooling; 0.020 in for no. 2 former tooling
	— W	maximum = 0.032 in for no. 1 former tooling 0.037 in for no. 2 former tooling
Insertion center	— B	minimum = 0.300 in maximum = A—0.787 in (but not greater than 1.260 in)
Component spacing	— E	may be either 0.200 in or 0.375 in
Note 2	—	The variations of B (insertion centers) with a group of 5 consecutive components cannot be greater than 0.400 in if the variation is from the smallest insertion centers to the largest. This one restriction can easily be overcome by programming the sequence path.
B + W	—	The number which is required to program the variable distance of the insertion head. The programmed insertion dimension must, in all cases, be the sum of the insertion center (distance from the center line of one formed lead to that of the other formed lead) and the lead wire diameter (one diameter only).

and types of machines to meet specific requirements. The selection of a total system for a board assembly project is not an easy matter. All factors must be weighed, and they should include component procurement or preparation, inplant handling prior to assembly, and the combination of part usage, volume, and mix. Savings realized in soldering and test as a result of reliable and accurate assembly should also be considered in making a complete mechanization analysis. Thousands of installations in all types of industry and large and small plants have demonstrated the labor-saving advantages of automatic assembly.



Fig. 11.29 VCD insertion system with computer control. A typical assembled board is shown in Fig. 11.14.



Fig. 11.30 Sequencer-Inserter combination machine. Input capacity is 48 magazines of DIP sticks.

Chapter 12

Conformal Coatings

JOHN WARYOLD

HumiSeal Division, Columbia Chase Corporation, Woodside,
New York

Introduction	12-2
1. Acrylic Coatings	12-2
2. Polyurethane Coatings	12-2
3. Epoxy Resins	12-2
4. Silicone Resins	12-3
5. Polyimide Coatings	12-3
6. Diallyl Phthalate Coatings	12-3
The Purpose of Coating	12-3
Selection of the Coating	12-4
7. Application Characteristics	12-4
8. Operating Characteristics	12-6
Preparation of Circuit Assemblies	12-7
9. Design	12-7
10. Cleaning	12-7
11. Storage	12-7
Application of Coatings	12-7
12. Dipping	12-8
13. Brushing	12-8
14. Flow Coating	12-8
15. Spraying	12-8
Inspection	12-8
Repair of Coatings	12-8
Storage of Coating Materials	12-9
Safety Precautions	12-9

INTRODUCTION

Conformal coatings are systems of synthetic resins that are usually dissolved in balanced volatile solvent vehicles. The resins are selected with particular attention to their electrical and thermal properties and their ability to form, when properly cured, a thin but tough film capable of providing protection against various environmental stresses.

Of the vast number of synthetic resins now available, a selected few types of acrylics, polyurethanes, and epoxies have been found to have the best all-around properties and are currently being used as the basis for most general-purpose conformal coatings. For special applications, however, other resins are used occasionally. For instance, to meet high-temperature requirements, silicones and polyimides may be indicated; and when high dielectric properties are needed, polystyrenes are used.

Basic resins, plus accessory chemicals and solvent vehicles, are blended to achieve optimum desired operating characteristics in the cured film. Many variations within each resin group are made possible by the addition of chemicals. For instance, plasticizers may be added for increased flexibility, dyes for identification or inspection, and wetting agents to improve adhesion to the substrate.

1. Acrylic Coatings Acrylics are excellent coating systems from a production standpoint because they are relatively easy to apply. Furthermore, application mistakes can be corrected readily, because the cured film can be removed by soaking the printed circuit assembly in a chlorinated solvent such as trichloroethane or methylene chloride. Spot removal of the coating from isolated areas to replace a component can also be accomplished easily by saturating a cloth with a chlorinated solvent and gently soaking the area until the cured film is dissolved.

Since most acrylic films are formed by solvent evaporation, their application is simple and is easily adaptable to manufacturing processes. Also, they reach optimum physical characteristics during cure in minutes instead of hours.

Acrylic films have desirable electrical and physical properties, and they are fungus-resistant. Further advantages include long potlife which permits a wide choice of application procedures; low or no exotherm during cure, which avoids damage to heat-sensitive components; and no shrinkage during cure. The most obvious disadvantage of the acrylics is poor solvent resistance, especially to chlorinated solvents.

2. Polyurethane Coatings Polyurethane coatings are available as either single- or two-component systems. They offer excellent humidity and chemical resistance and good dielectric properties for extended periods of time.

In some instances the chemical resistance property is a major drawback because rework becomes more costly and difficult. To repair or replace a component, a stripper compound must be used to remove effectively all traces of the film. Extreme caution must be exercised when the strippers are used, because any residue from the stripper may corrode metallic surfaces.

In addition to the rework problem, possible instability or reversion of the cured film to a liquid state under high humidity and temperature is another phenomenon which might be a consideration. However, polyurethane compounds are available to eliminate that problem.

Although polyurethane coating systems can be soldered through, the end result usually involves a slightly brownish residue which could affect the aesthetics of the board. Care in surface preparation is most important, because a minute quantity of moisture on the substrate could produce severe blistering under humid conditions. Blisters, in turn, lead to electrical failures and make costly rework mandatory.

Single-component polyurethanes, although fairly easy to apply, require anywhere from 3 to 10 days at room temperature to reach optimum properties. Two-component polyurethanes, on the other hand, provide optimum cure at elevated temperatures within 1 to 3 h and usually give working potlives of 30 min to 3 h.

3. Epoxy Resins Epoxy systems are available, as two-component compounds only, for coating electronic systems. Epoxy coatings provide good humidity resistance and high abrasive and chemical resistance. They are virtually impossible to remove chemically for rework, because any stripper that will attack the coating will also attack epoxy-coating or

potted components and the epoxy-glass printed board as well. That means that the only effective way to repair a board or replace a component is to burn through the epoxy coating with a knife or soldering iron.

When epoxy is applied, a buffer material must be used around fragile components to prevent fracturing from shrinkage during the polymerization process. Curing of epoxy systems can be accomplished either in 1 to 3 h at elevated temperature or in 4 to 7 days at room temperature. Since epoxies are two-component materials, a short potlife creates an additional limitation in their use.

The limitations with epoxies are short potlife, elevated temperature cure, poor repairability, and the need to use a buffer material around fragile components.

4. Silicone Resins Silicone coatings are especially useful for high-temperature service (approximately 200°C). The coatings provide high humidity and corrosion resistance along with good thermal endurance, which makes them highly desirable for printed circuit assemblies that contain high heat-dissipating components such as power resistors.

Repairability, which is a prime prerequisite in conformal coating, is difficult with silicones. Because silicone resins are not soluble and do not vaporize with the heat of a soldering iron, mechanical removal is the only effective way to approach spot repair. That means the cured film must be cut away to remove or rework a component or assembly. In spite of some limitations, silicone coatings fill a real need because they are among the few coating systems capable of withstanding temperatures of 200°C.

5. Polyimide Coatings Polyimide coating compounds provide high-temperature resistance and also excellent humidity and chemical resistance over extended periods of time. Their superior humidity resistance and thermal range qualities are offset by the need for high-temperature cure (from 1 to 3 h at 200 to 250°C). High cure temperatures limit the use of these coating systems on most printed circuit assemblies. Because the polyimides were designed for high-temperature and chemical resistance, chemical removal and burn-through soldering cannot be successful.

6. Dialkyl Phthalate Coatings Dialkyl phthalate (DAP) varnishes also require high-temperature cure (approximately 150°C), which limits their use on printed circuit assemblies. Furthermore, their removal with solvents or with a soldering iron is difficult, owing to their excellent resistance to chemicals and high temperatures (350°F).

THE PURPOSE OF COATING

Printed boards and components are coated with a protective film primarily to avoid, or at least to minimize, degradation in electrical performance when subjected to environmental stresses encountered in specific operating conditions. There are no coatings that will totally resist the effects of all ambient stresses; most of the stresses are cumulative and will ultimately overcome the protection afforded by the coating. A coating is considered to have served its purpose if it has contained the performance degradation within acceptable levels for an acceptable period of time.

The most damaging, and usually the most prevalent, environmental stress is generally recognized to be humidity. Excessive humidity will drastically lower insulation resistance between conductors, accelerate high-voltage breakdown, lower Q , and corrode conductors.

Contaminants, of which any of a few hundred different types may be found on printed boards, are equally damaging. They can cause the same electrical degradation as humidity, corrode conductors, and even cause dead shorts. The contaminants most frequently found in electronic systems are various chemicals which may be residues of manufacturing processes. A few of them are fluxes, solvents, release agents, metal particles, and marking inks. An important group of contaminants are those inadvertently deposited by human handling, such as body greases, fingerprints, cosmetics, and food stains. Ambient operating conditions may also contribute a variety of contaminants such as salt spray, sand and dust, fuel, acid and other corrosive vapors, and fungi.

Although the list of possible contaminants is almost endless, it is a consolation to know that, in all but the most severe cases, the destructive action can be effectively eliminated with a good conformal coating.

Coatings are applied in a film thickness that rarely exceeds 0.005 in. Such a film will withstand the effects of mechanical shock and vibration as well as of thermal shock and operations at temperature extremes. However, it is a fallacy that such a light coating can be relied upon to provide mechanical strength or adequate thermal insulation for individual components mounted on the printed board. Components must be anchored by mechanical means and must have a suitable sealant of their own.

SELECTION OF THE COATING

Selection of a suitable coating is a major task, considering the vast array of available materials. The user without some working knowledge of coatings would be well advised to turn for assistance to one of several manufacturers specializing in formulating coatings specifically for dielectrical and electronic applications. The manufacturers' printed information is responsive to the needs of the electronics engineer and contains extensive data, including electrical performance characteristics, from which the engineer can determine whether a particular coating meets the requirements. Characteristics of a coating can usually be classified into two groups: application characteristics, which deal primarily with the physical and chemical characteristics of the liquid coating, and operating characteristics, which define the physical and electrical performance of the cured film.

In the selection of the proper type of conformal coating for a specific application, the basic characteristics of the five coating types described previously should be considered. Furthermore, considering the main environmental stresses anticipated during the life of the electronic assembly, some characteristics could be traded off against others to assure high performance reliability. Tables 12.1 and 12.2 give a comparison of the average characteristics of various resin groups.

TABLE 12.1 Coating Selection Chart

	Acrylic	Urethane	Epoxy	Silicone	Polyimide	DAP
Application	A	B	C	C	C	C
Removal (chemically)	A	B		C		
Removal (burn through)	A	B	C			
Abrasion resistance	C	B	A	B	A	B
Mechanical strength	C	B	A	B	B	B
Temperature resistance	D	D	D	B	A	C
Humidity resistance	A	A	B	A	A	A
Humidity resistance (extended periods)	B	A	C	B	A	A
Potlife	A	B	D	D	C	C
Optimum cure	A	B	B	C	C	C
Room-temperature curing	A	B	B	C		
Elevated-temperature curing	A	B	B	C	C	C

Property ratings (A-D) are in descending order; A is optimum.

7. Application Characteristics

a. Potlife. Potlife is the length of time during which a coating material can be used before curing sets in. It is an important consideration in planning the coating procedure, and it has an important impact on the economics of coating in large-scale production. Short potlife leads to a substantial waste of materials and to some lack of uniformity in results owing to rapid buildup in viscosity. An acceptable minimum potlife for most applications is about 30 min at room temperature.

Short potlife is common to most two-component coating systems. Single-component systems, on the other hand, have a very extended potlife that almost approaches their shelf life. Note that potlife must not be confused with shelf life. The latter is the length of time a material can be stored in unopened containers without degradation.

b. Viscosity. Viscosity, or resistance to flow, must be low enough (less than 200 cSt) to permit adequate flowing of the coating over and around components to assure complete coverage. It is particularly important when a high-density board with many components packed tightly together with low clearances is to be coated.

Viscosity will determine the thickness of coating per application. A relatively higher viscosity is desirable when the coating must cover components with sharp corners or

TABLE 12.2 Typical Characteristics of Various Coatings Materials

Properties	Acrylic	Urethane	Epoxy	Silicone	Polyimide	DAP
Volume resistivity, $\Omega \cdot \text{cm}$ (50%RH, 23°C)	10^{15}	2×10^{11}	$10^{12}-10^{17}$	2×10^{15}	10^{16}	1.8×10^{16}
Dielectric constant, 60 cycles	3-4	11×10^{13}	$3.5-5.0$	2.7-3.1	3.4	3.6
Dielectric constant, 10 ⁶ cycles	2.5-3.5	5.4-7.5	3.5-5.0	2.7-3.1	3.4	3.6
Dielectric constant, 10 ⁸ cycles	2.2-3.2	5.5-7.6	3.3-4.5	2.6-2.7	3.4	3.4
Dissipation (power) factor, 60 cycles	0.02-0.04	4.2-5.1	3.3-4.0	0.007-0.001	0.002	0.010
Dissipation (power) factor, 10 ⁶ cycles	0.02-0.04	0.015-0.048	0.002-0.010	0.007-0.001	0.002	0.009
Dissipation (power) factor, 10 ⁸ cycles	2.5-3.5	0.043-0.060	0.002-0.02	0.001-0.002	0.005	0.011
Thermal conductivity, $10^{-4} \text{ cal/(s)} (\text{cm}^2) (\text{°C})$	3-6	1.7-7.4	4-5	3.5-7.5	4.0-5.0	4-5
Thermal expansion, $10^{-6}/\text{°C}$	5-9	10-20	4.5-6.5	6-9	500	350
Resistance to heat (°F) continuous	250	250	250	400	500	350
Effect of weak acids	None	Slight to dissolve	None	Little or none	Resistant	None
Effect of weak alkalis	None	Slight to dissolve	None	Little or none	Slow attack	None
Effect of organic solvents	Attacked by ketones, aromatics and chlorinated hydrocarbons	Resists most	Generally resistant	Attacked by some	Very resistant	Resistant

protruding leads from which it tends to roll off. Coating viscosity must be established by trial and error to achieve optimum performance for a specific board configuration. The viscosity of any solvent-type coating system can be reduced by the addition of the proper thinner.

c. Solids Content. The solids content represents the portion of a coating which will cure into a film; it is opposed to the solvent content, which will evaporate. The value has a bearing on coating thickness per application, and consequently it has some impact on the economics of the coating operation.

Solvent-type coatings are usually supplied with a solids content ranging from 10 to 50 percent. Solventless systems are, of course, rated as "100 percent solids" coatings.

d. The Chemical Components. The chemical components of a coating, particularly the solvent vehicle, may adversely affect the materials used in some of the components such as styrenes, acrylics, some marking inks, and adhesives.

The chemical effects of the selected coating must be ascertained beforehand, and the components designed into a printed wiring circuit must be chemically compatible with the coating.

e. Ease of Application. If the number of boards to coat is substantial, it is naturally more advantageous to use a coating that does not require specialized equipment, skilled labor, or involved production controls for its application. In this respect, room-temperature cure is preferable to elevated-temperature cure, which requires an oven, and single-component systems are preferable to two-component coatings, since they do not need metering and mixing equipment and thus eliminate a frequent source of operator error.

To the greatest extent possible, the coating should be procured at a viscosity at or near the viscosity desired for application. Thus the operations required to thin or reduce the coating are minimized or even eliminated. Some consideration should also be given to toxicity and flammability of the coating, since those characteristics will often require that the material be given special handling.

f. Curing Temperature. Room-curing materials usually require up to 24 h to cure fully. However, oven-cured materials (mostly two-component systems) may be cured in as little as ½ h. Generally, the higher the curing temperature the faster the cure. Although the shortest cure time is desirable to expedite the production process, the curing temperature is limited to the operating temperature rating of the board and the components mounted on it.

8. Operating Characteristics

a. Electrical Properties. One of the principal purposes of the coating applied to a printed board is to provide insulation to the otherwise bare printed wiring. The coating, therefore, must when cured provide insulation resistance and dielectric strength which satisfy design requirements over the entire temperature operating range and under adverse environmental conditions. In certain applications the dielectric constant and loss factor (Q) may also be significant values which should be considered when the coating is selected.

b. Thermal Properties. Operating temperature characteristics of plastics vary substantially. The coating selected must, naturally, have an operating temperature range at least equal to that of the system in which it is used. Within that range the coating must, in addition to meeting minimum electrical performance requirements, remain free from physical degradation such as embrittlement, cracking, or shrinking. A coating characterized by good flexibility will usually resist such physical deterioration.

Particular attention must be given to the hydrolytic stability of the coating; under conditions of high operating temperature in a high-humidity environment, certain coatings may, after a period of exposure, "revert" to a liquid state.

c. Humidity Resistance. Coatings should have low moisture absorption as well as low water vapor permeability. Most coatings marketed for printed board applications have more than adequate resistance to humidity. Yet, in special applications in which unusual and extended conditions of humidity are anticipated, the determination of minimum insulation resistance under humidity should be a guiding factor.

d. Resistance to Chemicals and Fungus. When operating conditions expose the coating to possible contact with chemicals, salt spray, or fungus, it is naturally necessary to determine the coating's ability to withstand the destructive effects of such external agents. Urethanes and epoxies, as a rule, have good chemical resistance. On the other hand, some

of those coatings may contain nutrient inorganic additives which, unless neutralized by a fungicide, will promote fungus propagation. Acrylic coatings usually contain only non-nutrient organic components, but their resistance to solvents is generally low.

PREPARATION OF CIRCUIT ASSEMBLIES

9. Design A major factor in the ability of a coated printed circuit assembly to function under severe adverse environmental conditions is the basic circuit design. Assemblies that have sharp corners, sharp pins, or variable components reduce the effectiveness of a smooth monolithic film, and component density and geometric positioning on the assembly also are major factors. The minimal vertical clearances and interface between dual in-line packages, ICs, etc., create coating penetration problems.

The composition of various components and markings, as well as various insulations such as Teflon, creates problems of compatibility. When choosing materials and components, we must consider whether they possess enough chemical resistance to withstand a variety of cleaning solutions. Too often we choose a type of insulation (TFE, vinyl, etc.) without anticipating receptiveness for coatings.

10. Cleaning The foremost consideration to remember is that coatings seal in as well as out. With that in mind, the cleaning procedure adopted should ensure an absolutely clean substrate. Listed below are a few simple steps which will produce a clean substrate and reduce the possibility of residual contamination (flux, fingerprints, grease, plating salts, moisture, release agents, etc.) being entrapped beneath a cured film. These steps should be followed after all the visual, physical, and electrical tests have been made:

1. Vapor-degrease (a suitable solvent would be trichloroethane, or Freon,) to remove residual greases and fluxes.
2. Rinse in deionized or distilled water, ethyl alcohol, or isopropyl alcohol to dissolve any salts which are not readily soluble in other solvents.
3. Oven-bake for 2 h minimum at 150°F to remove any residual traces of solvent and moisture which may still remain on the assembly. After cleaning and baking, the printed assembly should be handled with rubber or lintless gloves.

11. Storage If the clean assemblies are not to be coated immediately, they should be stored in a desiccator cabinet or in sealed polyethylene bags. If any masking is required, a polyester tape, rather than the conventional pulp-type masking tape, is recommended. The polyester tape is less porous and less likely to be attacked by solvents during the coating procedure. Another tip which may help in masking is to use either RTV silicone or a xitotropic latex mask which can be applied to hardware such as the tuning screws of potentiometers. After the conformal coating has become tack-free, the tape or RTV can be pulled off.

APPLICATION OF COATINGS

There are four major methods for applying conformal coatings: spraying, dipping, brushing, and flow coating. Also, there are variations of each method. The order given is probably the order of preference, but it is not necessarily the order of efficiency.

12. Dipping Dipping is by far the most efficient way to apply a conformal coating. It is the only effective way to insure even deposits of coating and uniform coverage. For best results the temperature of the coating should be kept within the 70 to 90°F range. However, the most important factor influencing the results of the dip-coating application is the speed of the substrate's immersion; the coating should be allowed to seep into all the voids on the printed circuit assembly. Typical immersion speeds are between 2 and 12 in./min. Slow speed of immersion will allow the coating to displace the air surrounding the components. Too rapid immersion may result in trapping the air and thereby forming bubbles. The assembly should be left in the coating until all bubbling has ceased.

The immersion and withdrawal speeds depend, of course, on the size and complexity of the assembly. In most cases, they should not exceed 1 ft/min. Those considerations are especially important for the first dip. Subsequent dips may be made safely at higher speeds.

When conformal coatings are applied by dip, the evaporation of solvents may occur at such high rates that the viscosity of the bath will increase at a similar rate. In such cases, it is desirable to monitor the viscosity of the coating and, if necessary, add the proper thinner to restore the original viscosity.

13. Brushing Brushing is the least effective application method because of the difficulty of getting uniform coverage, controlling bubbling, etc. It is neither a practical nor an efficient way to coat large quantities of printed circuit assemblies.

14. Flow Coating Flow coating consists of pouring or flowing the coating as a curtain as the assembly passes through the coating. The method is especially effective if printed circuit assemblies have minimum flat packs, integrated circuits, or cordwood modules.

15. Spraying Spraying is the most expeditious method for coating assemblies. With the proper combination of solvent reductions, spray pressures, nozzle pattern, and solids contents, excellent results are obtainable. Spray-coating is widely employed for protecting all types and sizes of assembly, and both hand and automatic equipment is used. Spraying is particularly adaptable for coating uneven surfaces. For spray application, coatings are used as supplied or thinned down to a spraying viscosity. If the viscosity is too high, difficulties are encountered in spraying a smooth coat. On the other hand, materials can be sprayed at extremely low viscosities if the solvent system is well formulated and the pressure and gun settings are adjusted properly.

Regardless of which technique is used, the coating assembly should be allowed to drain and air-dry. Turning the unit from time to time during the drying period prevents pooling of the coating and speeds up evaporation of solvents. If very sharp edges or asperities exist on one side of the board, especially where the leads protrude through the solder side of the board, it sometimes helps if, during the draining process, the board is held in a horizontal position with pins facing down. That allows "stalactites" to form on the tips of the sharp edges. It may be necessary to repeat the process two or three times in order to produce enough coating to cover the protrusions. Figure 12.1 further explains the need for multiple coats.

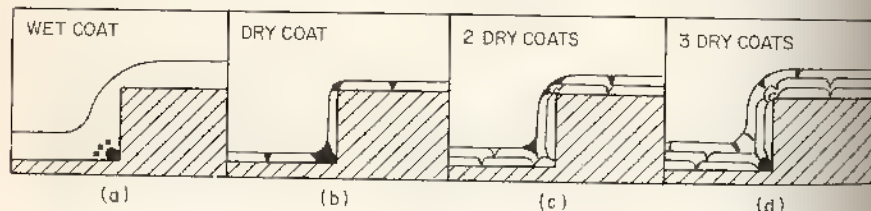


Fig. 12.1 An example of the need for multiple coats.

INSPECTION

After a printed circuit assembly has been coated, visual inspection is necessary. That becomes evident when one realizes that a void or bubble provides a path for moisture to penetrate to the substrate and defeat the original intent of the coating.

Inspection, usually under magnification, is simplified if the coating contains a fluorescent tracer. When a coated assembly is viewed under ultraviolet light, the areas not coated will produce a darkened or shadowed spot.

REPAIR OF COATINGS

An obvious prime consideration in the selection of a conformal coating is the degree of difficulty of removal if repairs should be necessary.

Acrylics. Their thermoplastic film-formers usually can be removed by most chlorinated solvents.

Urethanes. Most single- and two-components urethane materials may be removed by certain strippers. The strippers selectively dissolve the urethanes rather than destructively decompose them.

Epoxies. To date, chemical removal is not possible; obviously, anything that would dissolve the epoxy film would attack the board and components also.

Silicones. Some silicone coatings are soluble in a few select strippers.

In the use of any stripper, thorough rinsing is necessary to prevent any residual traces from becoming entrapped beneath the re-coated area.

Most coating materials when applied in thicknesses of up to 0.004-in may be soldered through. However, the degree of difficulty of soldering through a coating is dependent upon the specific resin used, the film thickness, and the aesthetic quality of the reworked area.

STORAGE OF COATING MATERIALS

The flammability of liquid coatings is determined by the specific solvents used in the formulation and the flash points those chemicals have. The flash point is defined as "the temperature at which the vapors of that solution will ignite and burn." To overcome any potential hazard, we should eliminate all ignition sources. Liquid coatings should always be stored in their originally sealed container away from fire or open flame. The shelf life of a coating is the time a coating may be stored in the original unopened container at room temperature without noticeable deterioration of properties. Polyurethane coatings are sensitive to moisture. Whenever possible, they should be stored in the original container. If the container is partially filled, it should be purged with dry nitrogen.

SAFETY PRECAUTIONS

Whenever conformal coatings are applied, the following precautions should be observed:

1. Do not inhale vapors from the coating. Use a ventilated hood.
2. Provide adequate ventilation and exhausting.
3. Avoid contact, especially of lips and eyes, with liquid coating.
4. Use protective clothing, respirators, and goggles if exposure to vapors is constant.
5. Request a material safety data sheet to establish safety vapor threshold limits and handling procedures.

Chapter 13

Assembly Repair

D. P. SCHNORR

RCA Missile and Surface Radar, Moorestown, New Jersey

Introduction	13-2
Preparation for Rework	13-2
1. Conformal Coating Removal	13-2
Conductors	13-3
2. Damaged, Defective, or Missing Conductors	13-3
3. Recommended Procedures for Repairing Damaged Conductors	13-3
4. Lifted Conductors	13-5
5. Recommended Procedures for Repairing Lifted Conductors	13-5
6. Shorts and Spurs	13-6
7. Removal of Conductors	13-7
Terminals	13-7
8. Lifted, Damaged, or Missing Terminal Areas	13-7
9. Damaged or Defective Contact Tabs	13-10
10. Missing Holes of Terminals	13-10
Component Replacement	13-11
11. Defective Components	13-11
12. Axial, Radial, and Multiple-Lead Components	13-11
13. Solder Joint Removal Methods	13-13
Plated-through Holes	13-15
14. Damaged Plated-through Hole	13-15
Warped Boards	13-15
15. Warped Boards without Components	13-15
16. Warped Boards with Components	13-16
Laminate Defects	13-16
17. Mismatching and Weave Exposure	13-16
18. Laminate Voids	13-16
19. Cracking	13-17
Multilayer Boards	13-17
20. Internal Shorts	13-18
References	13-18

INTRODUCTION

Printed boards, characterized by a serial processing sequence, arrive at their finished state only after the expenditure of a considerable amount of time and money. It is understandable, then, why it is easy to justify repair and/or correction of defects which may occur during later stages of manufacture or assembly. In most cases it is cost-effective to perform the rework; and when many expensive components are used, it is essential.

The rationale behind rework and repair of the board must be that the operation will not degrade the mechanical, electrical, or physical properties of the printed wiring assembly. In all cases the repaired article must meet the original, or design, requirements of the finished product. Techniques for repairing the conditions illustrated in this chapter have proved, through test or functionality, to be acceptable for normal applications. Finished product requirements should, however, dictate reliability and quality standards for the repaired article.

The design of a printed board should allow for change if a need for repair is anticipated. In all instances, technique and function criteria for the operation should be subject to engineering approval. In many cases it may be necessary to guarantee the reliability of the repair step by the use of a rudimentary performance test. The justification for repair should be flexible enough to consider the alternate solution; replacement of the part. Indeed, the economics of time, labor, material, and repair limitations, as well as customer preference, must be weighed against outright replacement.

It is essential that users of these guidelines pay particular attention to the following when considering a repair procedure.

1. No cumulative conditions warranting repairs are presented here; the conditions are indicated as singular variances. Note that they are intentionally depicted as isolated.
2. Repairs promoted should be made only by personnel specifically trained and, if possible, certified in rework and repair techniques.
3. The manufacturer and user must be in accord concerning repair criteria and limitations.
4. Repair can never compensate for good quality standards, process control, or manufacturing techniques.
5. Publication of these repair procedures does not signify approval per se; the procedures are only suggested as possible approaches. The approval of the responsible project engineer of the contracting organization should be obtained before repair is initiated. Acceptance of a repaired printed wiring assembly can be granted only after successful completion of functional tests under prescribed environmental conditions.

PREPARATION FOR REWORK

In addition to the assembly of suitable tools for repair and the designation of a definite repair work station, the printed board must be prepared for the rework operations which are planned for it. One of the most important operations is the removal of conformal coating from the assembly. If the boards are unassembled, that is, of course, no problem, but for the majority of military contracts and for federal agency and commercial work, some type of moisture- and fungus-proof (MFP) coating is specified on the assemblies following the assembly soldering operation.² The need to remove all the conformal coating from the connections or area to be reworked exists because the coating creates a heat barrier which makes it difficult to melt solder joints and because the resoldered joint may become contaminated by any residual conformal coating. In most cases only spot removal of the MFP from the work area is necessary, but in some instances it may be more efficient to remove all of the coating, by solvent means, on a production basis.

1. Conformal Coating Removal There are three recommended coating removal methods: (1) solvent, (2) thermal parting, and (3) abrasion. The specific removal method to be employed is based on the generic type of coating used, the specific condition of the coating, the nature of the parts, and the nature of the board. These methods are covered briefly below.

a. Solvent Method. Mild solvents, such as xylene and trichloroethane or the equivalent, or harsh solvents, such as methylene chloride, may be used to remove specific soluble coatings on a spot basis by brushing or swabbing the local area a number of times with

fresh solvent until the area is free of coating. When solvents are used, rapid solvent dry-off through forced evaporation, as well as neutralization if required, should be utilized.

CAUTION: If chloride-based or other harsh solvents are utilized, extreme care must be exercised to prevent damage to the component part and base laminates (e.g., measing). Solvents of the chloride type should be used on a short-term basis only.

The procedure is as follows:

1. Saturate the end of a cotton-tipped applicator with the stripping solution and apply a small amount of the material to the coating around the soldered connections of the part or the area to be repaired. Repeat until the coating is softened.
2. Rub the treated surface carefully with a bristle brush or the wood end of a wedge-shaped applicator tip until material is dislodged.
3. Neutralize the stripped area and dry.

4. Alkade the exposed solder connections, if present, and brush away the residue.

b. Thermal Parting Method. Thermal parting uses a controlled, low-temperature, localized heating method for removing thick coatings by an overcuring or softening means. Although MIL-approved coatings are now in the range from 0.001 to 0.003 in, the method is also useful for removing urethane resin strapping and adhesive component hold-downs.

CAUTION: Do not use soldering irons for coating removal, since their high temperatures may char and delaminate the base substrate.

The procedure is as follows:

1. Select an appropriate thermal parting tip to suit the workpiece configuration. Set the nominal tip temperature by following the manufacturer's recommended procedure.
2. With a light pressure, apply the thermal parting tip to the coating. The coating material will either soften (epoxy) or granulate (polymethane).
3. Gradually reduce the coating thickness around the area to be repaired without contacting the board surface. Low-pressure air or a bristle brush should be used to remove waste material.

4. Remove the remaining coating material by a combination of the thermal parting method and the abrasive method, below.

c. Abrasive Method. This method is often used in conjunction with the solvent and thermal parting methods after the coating has been softened. Rubberized abrasives of the proper grade and grit are ideally suited for removing thin hard coatings from flat surfaces but not softer thick coatings, which would cause the abrasive to load with coating and become ineffective. A wide variety of tools may be employed, including twist drills, ball mills, and rotary brushes, to suit the various coating types and configurations.

The procedure is as follows:

1. Apply an appropriate rotating abrasive tool to the coating with various degrees of pressure to test the rates of coating removal. Use coarse abrasive first and change to a finer abrasive or bristle brush to clean up the area.

2. When all coating has been removed from the desired surface, the area should be cleaned with an appropriate solvent to remove any remaining contaminants.

NOTE: This method is primarily suited to the circuit, or solder, side of the board, which is readily accessible. On the component side of the board use either the thermal parting or the solvent method.

CONDUCTORS¹

2. Damaged, Defective, or Missing Conductors As indicated in Fig. 13.1, a fault in this category may consist of (1) a complete break in the conductor, (2) scratches, (3) nicks, or (4) pinholes, all of which may reduce the cross-sectional area of the conductor below that required by the specification.

NOTE: The limitations are that rework should be limited to two repairs per conductor and not more than six repairs per board or as determined by the specific requirements of the customer. Of the repairs, no more than three per board should be made by procedure 3 below. Conductor widths and spacings must not be reduced below the allowable tolerances.

3. Recommended Procedures for Repairing Damaged Conductors

a. Soldered Bus Wire Method. (See Fig. 13.2.)

1. Clean area to be repaired of conformal coating if necessary. Clean both sides of the

break in the conductor at least $\frac{1}{4}$ in on each side with a rubber eraser and then with isopropyl alcohol.

2. Cut a piece of solid tinned copper wire of applicable gauge and a minimum of $\frac{1}{4}$ in longer than the break.

3. Hold wire on centerline of conductor across the break; apply flux; and solder in place.

4. Clean flux residue from joint with isopropyl alcohol.

5. (Optional) Flow a small amount of epoxy cement over the entire area and allow for proper cure.

b. Jumper Wire Method. (See Fig. 13.3.)

1. Select component leads, unused through holes, or terminals if available, and clean area to remove conformal coating if present. Clean with isopropyl alcohol. With soldering iron, clean excess solder off lead. Either side of board may be used.

2. Cut a piece of solid tinned copper wire of applicable gauge and wrap each end around terminals or in the hole to be connected.

3. Flux connections and solder in place.

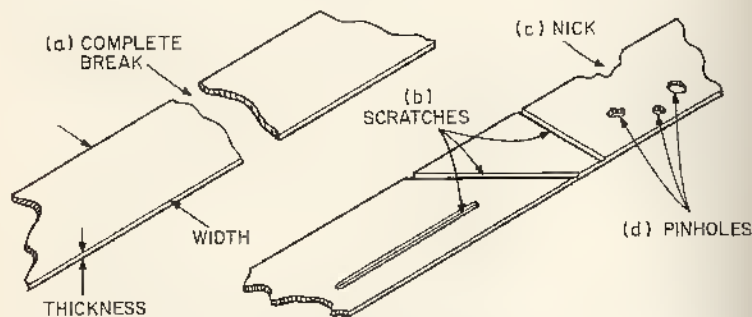


Fig. 13.1 Types of conductor damage.¹

4. Clean flux residue from the joint with isopropyl alcohol.

5. Whenever possible, connect wires to a point where soldering an adjacent component will not cause the wire to become unsoldered.

6. Jumpers over 2 in long should be covered with Teflon* sleeving and be firmly secured to the board.

c. Copper Foil Method

1. Peel off any damaged circuitry by using a surgical knife; remove conformal coating if necessary; and roughen the board surface under the removed circuitry by using a suitable abrader.

2. Clean board surface with isopropyl alcohol and a lint-free industrial wiper.

3. Cut a piece of adhesive film (epoxy bonding material) to the size and shape of the missing circuitry.

4. Prepare a piece of copper foil of suitable gauge to conform to the size and shape of the missing circuitry. Allowing for $\frac{1}{8}$ in overlap onto the existing circuitry. The repair conductor should be centered over the original conductor and be formed to the board unless the conductor break is small. In the latter case, delete the adhesive application step.

5. Prepare copper surfaces for soldering by abrading the overlapping ends.

6. Clean a piece of replacement circuitry with isopropyl alcohol and put a piece of cut film adhesive in place on the roughened board surface.

7. Put the piece of cut replacement circuitry over the film adhesive with the oxide side of the copper against the film adhesive. Tack the circuitry in place with a soldering iron.

8. After tacking, fasten foil in place by slowly drawing the flat side of the soldering

*Registered trademark E. I. du Pont de Nemours & Company.

iron across the repaired circuitry from the center to the ends while exerting a slight pressure.

9. Flux and solder-reflow lapped ends to the original circuitry.

10. Clean flux residue from joints with isopropyl alcohol and reapply suitable conformal coating if required.

d. Gap Welding Method.⁴ Prior to using this method, certain precautions should be taken:

1. Welding equipment should have electrodes cleaned, aligned, and set for proper board thickness.

2. Perform test runs on samples having conductor paths representative of the conductor size, finish, contour, etc. to be repaired. Observe weld quality of strip and laminate for appearance, alignment, discoloration, and fusion.

3. Perform tensile tests and microsectioning as required, and readjust weld schedule if necessary. Repeat the sequence until acceptable results have been achieved.

The procedure is as follows:

1. Cut ribbon wire to size. Material may be either nickel (gold-plated) or Kovar. The current-carrying capacity of the ribbon wire is to be equal to or greater than that of the severed copper conductor.

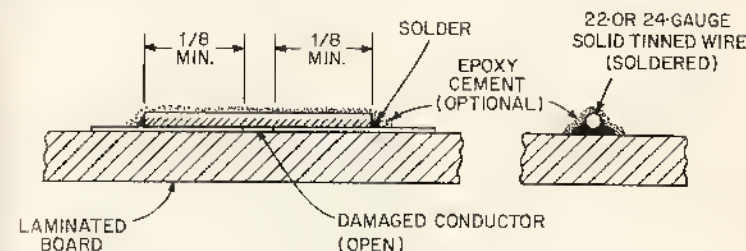


Fig. 13.2 Conductor repair, soldered bus wire method.¹

2. Thoroughly clean the ribbon wire, conductor line, and base laminate around the repair area with an abrader and isopropyl alcohol.

3. Place ribbon over the centerline of the conductor and perform weld per schedule based on accepted test sample.

4. Clean with isopropyl alcohol and inspect the weld quality and alignment.

5. Coat with epoxy or conformal coating if required.

4. **Lifted Conductors** This condition, as depicted in Figs. 13.4 and 13.5, occurs when a portion of the conductor separates from the substrate but remains unbroken.

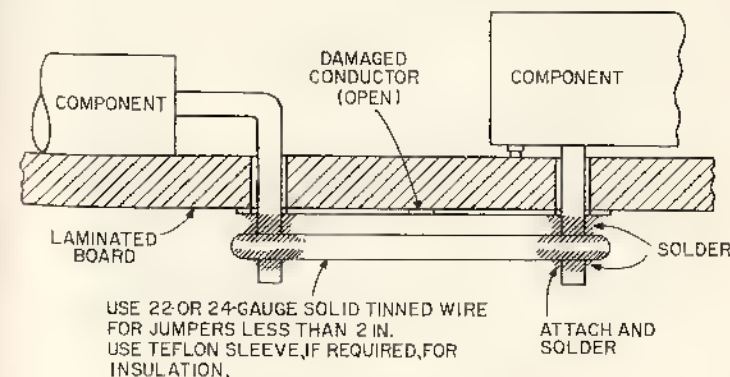


Fig. 13.3 Conductor repair, jumper wire method.¹

NOTE: The following limitations apply: repaired length of the lifted conductor should not exceed one-half the length of the conductor between two terminal areas. The maximum number of repairs per assembly should be subject to individual requirements and customer approval. Industry practice never exceeds 5 percent of the number of conductors.

5. Recommended Procedures for Repairing Lifted Conductors

a. Epoxy-under-conductor Method

1. Clean the underside of the lifted conductor and surrounding area with isopropyl alcohol after the conformal coating has been removed.
2. Remove, with suitable tools, anything which may prevent the lifted conductor from making intimate contact with the surface of the substrate.
3. Force epoxy under the entire length of the lifted conductor.
4. Press conductor into contact with the substrate and hold until set. Cure in air or bake in accordance with the material supplier's instructions. Units which have been repaired should not be handled until the epoxy has cured.

b. Epoxy-over-conductor Method. This technique is similar to the preceding one; it is used when component density does not permit manipulation of the conductor to place epoxy under the trace. Also, it is very important to remove any conformal coating from the board before starting the operation.

1. The cleaning method is identical with step 1 of the epoxy-under-conductor method.
2. Apply epoxy to the surface of the lifted conductor and to a distance of at least $\frac{1}{8}$ in. in all directions from the damaged area.
3. Cure in air or bake in accordance with the material supplier's instructions.
4. Repaired units should not be handled until the epoxy has cured.

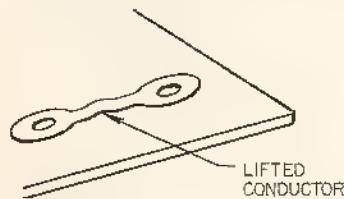


Fig. 13.4 Lifted conductor, between terminals.

6. Shorts and Spurs (Excess conductive material; see Fig. 13.6.) One of the most basic of all rework procedures consists of removing foil material. Most often it is carried out to preserve spacing between conductors, which in extreme cases may have resulted in a short. Rework of this type is especially important on internal layers of multilayer boards, where the presence of a short can have disastrous results after the board has been laminated and the processing has been completed.

The procedure is as follows:

1. By using a suitable point on a razor knife, carefully cut the foil and separate the excess material from the body of the conductor. Take care not to cut into the base laminate.
2. By using a chisel-point knife, skive up the conductor from the base material.
3. Scrape the area under the removed conductor and inspect it to make certain that there are no conductor traces. Apply epoxy material or conformal coating if specified. (Not in the case of internal layers.)

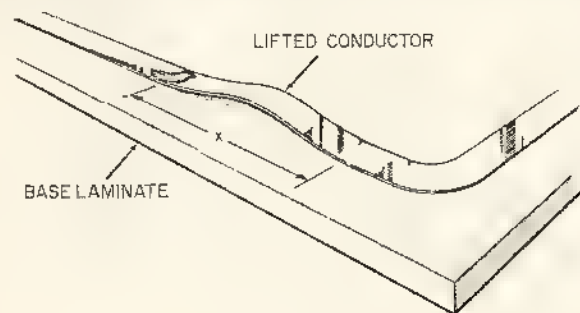


Fig. 13.5 Lifted conductor.¹

7. Removal of Conductors In some cases, design changes in an assembled board may necessitate rerouting of conductive paths. If so, the conductor must be removed at designated points.

NOTE: Not more than five conductor portions may be removed per board. (Shorts and spurs do not apply.)

The procedure is identical with that for shorts and spurs, above.

TERMINALS

Terminals are important because they are the points of component attachment or interconnection to the printed wiring pattern. For that reason, contact tabs are included in this category as input-output points. They share in functional importance with other board terminals.

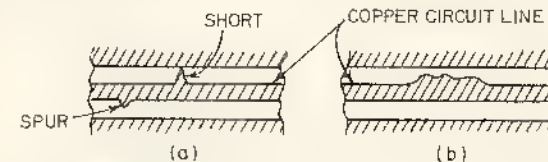


Fig. 13.6 Excess conductive materials. (a) Spur and short, and (b) under-etched area.

8. Lifted, Damaged, or Missing Terminal Areas¹ (See Fig. 13.7.) Defects in this category consist of terminal areas which have become separated, loosened, or lifted or which for some reason have become unbonded from the base material. Included are terminal areas damaged by some means in excess of established limits (Figs. 13.8 and 13.9) and those that are missing or were inadvertently removed during the manufacturing process.

NOTE: The limitations are as follows:

- a. No repairs on bare (or unassembled) boards should be made without customer approval.
- b. The spacing between conductors must not be reduced to less than the minimum specified in the original or governing specifications.
- c. Repairs are made if the lifting or separation exceeds the established limits during inspection. The unshaded areas in Figs. 13.8 and 13.9 represent the terminal areas and/or land areas to be inspected. In those areas (other than noted), the 0.0015-in. feeler gauge can penetrate a distance equal or no more than one-half the distance from the interface of the hole to the nearest edge of the terminal area (annular ring) for not more than 180° of the periphery. When feeler gauge penetration does not exceed the limits described above, the terminal area is acceptable and is not considered as lifted. If, however, the feeler

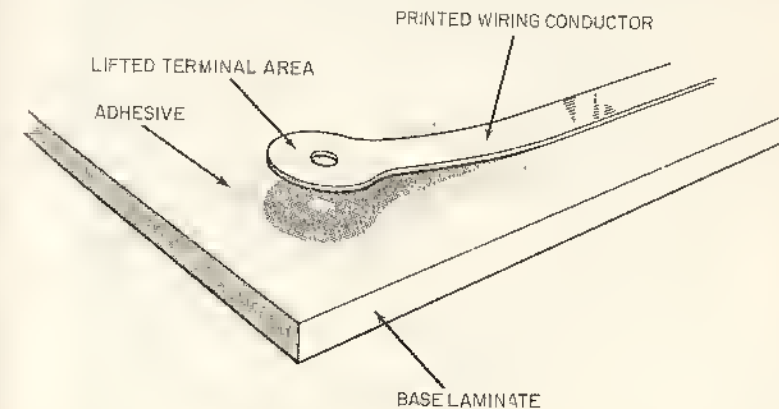


Fig. 13.7 Lifted terminal area.¹

gauge penetration is greater than described above or the defect can be determined visually, repair may be made in accordance with procedures and conditions listed below.

The repair procedure for *partially lifted terminal areas* is as follows:

1. Free immediate area of solder by using a vacuum-type solder remover.
2. Gently but firmly clean the areas to be repaired by brushing with clean isopropyl alcohol. Air-dry for 5 min; then oven-dry for 30 min at 60°C.
3. Prepare adhesive. One recommended material is 10.0 g of Epon 828 mixed thoroughly with 15.0 g of Versamid 140. Deaerate by centrifuging for 2 min.

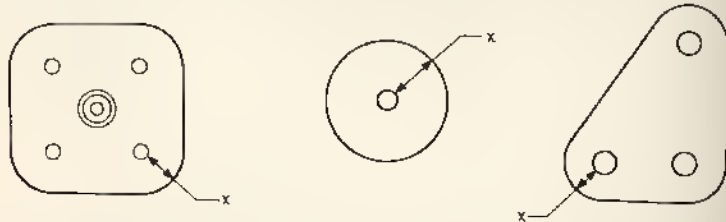


Fig. 13.8 Terminal area configurations; maximum feeler gauge penetration of $\frac{1}{32}$ applies.

4. Apply adhesive immediately after removing the board from the oven; carefully flow it under and around the lifted pad with a camel's hair brush, syringe, or other suitable applicator. Press the terminal down with the applicator or clamp until adhesive is set.

CAUTION: Solder side of terminal area must be free of contamination and/or epoxy.

5. Examine the board under ultraviolet light. The adhesive glows and should be seen to surround the reworked pad or trace.

6. Cure in 60°C oven for a minimum of 3 h. The surface buildup of adhesive should be smooth and neat in appearance.

The repair procedure for *lifted terminal areas around bare holes* is as follows:

1. Free immediate area of solder by using a vacuum-type solder remover.
2. Lift up the separated area without bending and clean all dirt, flux, residue, and foreign matter from under and around the terminal area by brushing with isopropyl alcohol. Air-dry for 5 min; then oven-dry for 30 min at 60°C.
3. Apply adhesive per steps 3 to 6 for partially lifted terminal area.
 - a. If the surface under the lifted terminal area is smooth and free of pits, adhesive film (Permacel P-18) may be used instead of liquid adhesive. Cut adhesive film to the exact circuit dimensions and place in position.
 - b. Adhere the loose area by applying the flat side of a clean soldering iron for approximately 5 s.
4. Clean the repaired areas with isopropyl alcohol or an approved solvent.
5. Install a fused or funnel eyelet of sufficient size to receive the component lead. Eyelet OD should be within 0.010 in. of hole ID. The ID of the eyelet should not be a tight

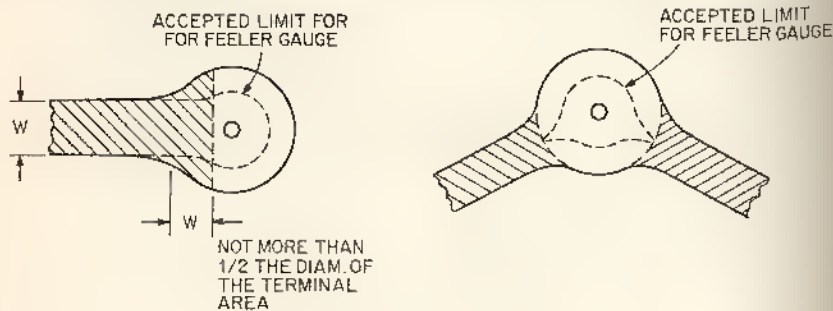


Fig. 13.9 Limits for terminal area repair

fit. For a lifted terminal area which has a flat side to facilitate circuitry spacing, the eyelet may be clipped by using a diagonal cutter to conform to the shape of the original pad. In no case shall the eyelet be clipped within 0.005 in. of the OD of the eyelet. All other procedures outlined in this section apply here.

The repair procedure for *lifted terminal areas which have been separated or otherwise broken away from the associated circuitry* (Fig. 13.10) is as follows:

1. Remove defective terminal area, if any; clean board surface with an applicator which has been saturated in isopropyl alcohol or other approved solvent; and allow the surface to dry.

2. Remove a similarly shaped section of good circuit from a scrap board or use a tinned copper foil of equal or greater thickness to replace the damaged or broken terminal area or pad. A minimum overlap with the connecting circuit conductor of 0.125 in. excess should be allowed for the solder lap joint. The overlapping area of terminal area segment and board circuitry should be completely cleaned of contamination by using an electric erasing machine or an ink eraser. Clean the area, after buffing it, with isopropyl alcohol or other approved solvent.

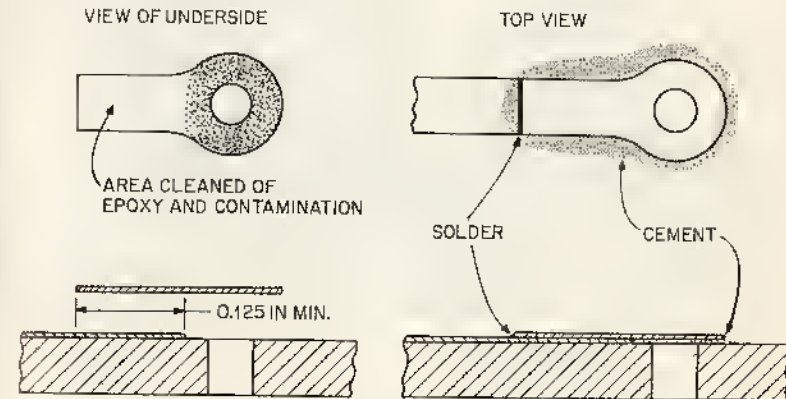


Fig. 13.10 Lifted and separated terminal area.

3. Solder the newly prepared terminal area segment to the existing circuit. Take care to match the terminal area with the hole.

4. Cement the replaced terminal area segment to the board per steps 2 to 4 for repairing lifted terminal areas around bare holes.

5. Install a fused or funnel eyelet (see step 5 for repairing lifted terminal areas around bare holes).

The repair procedure for a *lifted terminal area* is as follows:

1. Free the immediate area of solder by using a vacuum-type solder remover.
2. Lift separated area without bending and clean all dirt, flux residue, and foreign matter from under and around the pad with isopropyl alcohol or other approved solvent.
3. Insert Epon 828 or an approved epoxy resin under the copper with a camel's hair brush, syringe, or other suitable applicator. Make sure that the resin does not contaminate the solder surface of the terminal area.

NOTE: adhesive film method per step 3 for repairing lifted terminal areas around bare holes may be used.

4. After adhering and curing, check repaired areas for adequate adhesion.
5. Clean the repaired area with alcohol or other approved solvent.
6. Install a fused or funnel eyelet (see step 5 for repairing lifted terminal areas around bare holes).

The repair procedure for *lifted terminal areas which are not connected to circuitry (via holes)* is as follows:

1. Remove defective terminal area, if any, and clean board surface around the hole with isopropyl alcohol or other approved solvent.

2. Leave hole and board bare unless a mechanical solder joint is required.
3. If a mechanical solder joint is required, install a flat flange eyelet in the hole. (See step 5 for repairing terminal areas around bare holes.)

9. Damaged or Defective Contact Tabs Although this type of defect commonly involves pitted, peeled, or delaminated gold plating, only badly worn, missing, or damaged contact tabs will be discussed. (See Fig 13.11.) Gold plating is normally repaired on unassembled boards by completely replating the board and in assembled boards by a selective, or brush-plating, procedure. Information on the latter method is best obtained from the manufacturer of the equipment.

NOTE: The following limitations apply. The number of repairs of this type per board should be determined by the specific requirements of the procuring agency and/or customer. Industry practice never exceeds 5 percent of the contacts per board. Repair should not be performed if there is serious damage to the underlying base laminate.



Fig. 13.11 Damaged (worn) contact tabs.¹

The procedure is as follows:¹

1. Peel off the defective contact tab by using a razor knife and clean and roughen the board surface by using a suitable abrasive.
2. Brush with 50 percent isopropyl alcohol in naphtha and dry with a lint-free tissue.
3. Select a similar contact tab from a scrap board; remove it; and cut it to a length at least $\frac{1}{16}$ in longer than the defective removed tab.
4. Prepare the new tab by abrading its back surface with a glass-fiber eraser or equivalent and tinning the end.
5. Tin the end of the etched conductor on the board.
6. Flux ends and reflow lap solder in place with a minimum overlap of $\frac{1}{16}$ in.
7. Remove flux residue and clean areas to be adhesive-bonded with 50 percent isopropyl alcohol in naphtha.
8. Use masking tape to protect the neighboring contact tabs as well as the tabs on the opposite side of the board.
9. Coat the back of the tab with a thin film of epoxy adhesive and position the tab. Clamp the tab firmly in place until the epoxy is cured; use a slight, uniform, and constant pressure.

10. Missing Holes of Terminals Holes may be drilled in boards which have missing mounting holes or terminals or to interrupt internal layers or route added wires.

NOTE: The following limitations apply. Added terminals or holes drilled for any reason should not exceed eight per board. With the exception of drilling out holes in multilayer boards to break connections, no internal conductive layers should be pierced.

The procedure is as follows:

1. Drill the proper hole in the board according to the detail drawing. In the case of a missing terminal, install a terminal.

2. For interrupting internal conductive circuits, or for routing wires through holes, an oversized hole should be drilled and then filled with epoxy as follows:

- a. Clean area to be filled with a stiff bristle brush dampened in perchloroethylene and then rinse with isopropyl alcohol.
- b. Mask and/or plug adjacent holes to prevent their filling, and take care to keep the rework area localized.
- c. Select the proper bonding materials (resin and catalyst) and mix as recommended by the material supplier. Mix in Fiberglas No. 128 and stir until all the fibers are wetted.
- d. Fill hole and cure as recommended. Abrade flush. Small plates coated with release agent may be clamped on the filled area to minimize the amount of abrading required to flush.
- e. Refill the hole, if necessary, per the applicable drawing.

COMPONENT REPLACEMENT

Printed boards are basically used as interconnection media for increasingly more sophisticated and expensive electronic assemblies. Microelectronic devices are used; they are contained in a wide variety of package styles including flat packs, dual in-line packages, and a number of can- or TO- type configurations. When a device on such an assembly fails, it is most often cost-effective to remove the device and replace it with another and thereby salvage the cost of the assembly. When a device does fail and is replaced, it is of primary importance not to damage the board and adjacent devices and of secondary importance to remove the device in good condition. Valuable information regarding the manufacture and application of the device may be gained by examining the failure modes and mechanisms of an essentially intact "failed" device. It is therefore necessary to have procedures for removing devices in an expedient manner and without mechanical or electrical damage to the printed board, the adjacent devices, or, if possible, the removed device itself.

11. Defective Components In this category falls the removal of components which need to be replaced, including all axial-lead types, as well as device body styles incorporating multiple conductors with fixed leads.

NOTE: The number of repairs of this type per board is determined by the specific requirements of the procuring agency and/or customer. Extreme care is required to avoid damage to the circuitry, base laminate, or other components. Reasons for damage could be excessive heat application, improper or sloppy use of tools, inadequate training of the operator, or rough handling of the printed board.² Heat damage may occur when an iron is held on a terminal too long. A guideline for soldering iron application is this: No more than THREE SECONDS for a terminal connected to circuit traces, no more than SIX SECONDS for a terminal connected to a ground plane.³ The use of heater blocks, such as those illustrated in Fig. 13.12 for multiple-lead components in plated-through hole joints, is not recommended. It has not proved feasible to heat multiple leads uniformly; joints are either over- or underheated, and the result is either plated-through-hole damage or pad delamination and/or blistering.⁷

12. Axial, Radial, and Multiple-Lead Components

a. Through-Hole Procedure¹

1. Identify component to be removed and note polarity.
2. Remove strapping material, if necessary, as follows:
 - a. Use a thermal prying tool to remove large accumulations of adhesive. Take care not to cut into the board or discolor the board with heat.
 - b. Thin applications of remaining material may be carefully removed by methods outlined previously under conformal coating removal methods.
3. Remove conformal coating, if applied, as outlined previously.
4. Component can be removed by applying one of the following techniques:
 - a. Apply heat and remove excess solder by one of the methods outlined below (solder joint removal methods).

b. If leads are clinched, apply heat, remove solder as above, and straighten and remove lead.

c. Cut component leads, apply heat, and remove solder as outlined below. Remove the lead.

5. After a component has been removed and the terminal area has cooled, melt all

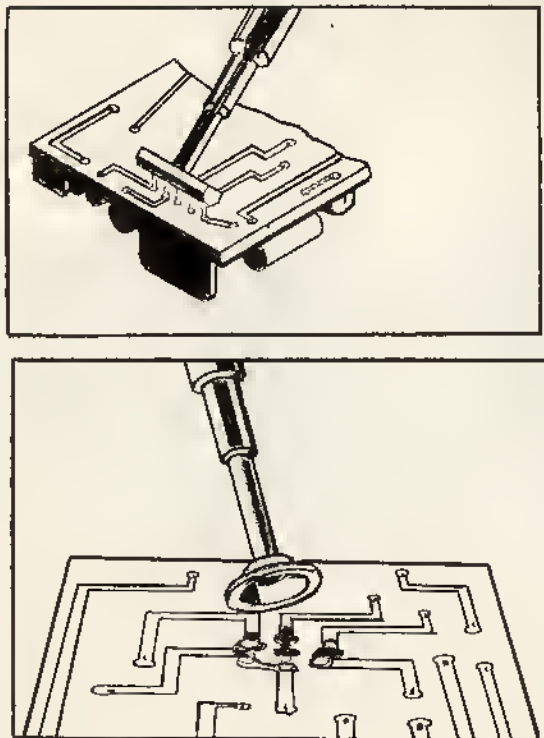


Fig. 13.12 Not recommended use of heater blocks for multiple-lead⁷ components in plated-through hole joints.

excess solder and remove it with an extractor as outlined below under solder joint removal methods.

6. Inspect hole thoroughly for damage before attempting to replace any components.

7. Form the leads of the replacement component and trim them to size.

8. Mount the component on the board. Note the polarity.

9. Solder leads in such a fashion as to avoid any undesirable heat buildup, especially on multiple-lead components. Allow cooling time when necessary.

10. Clean flux residue from joint with cotton-tipped applicator and isopropyl alcohol when necessary.

11. Perform the electrical and/or functional tests.

12. Restrap the component by using a tie-down, an approved adhesive, or both if required.

13. Conformal coat if necessary.

b. Surface-Soldered Components Procedure

1. Identify component to be removed and note polarity.

2. Remove conformal coating, if applied, as noted previously.

3. Cut each lead of a faulty component so that the remaining lead is perpendicular to

the board surface and slightly above the solder connection. If the component is to be salvaged, reflow each joint individually and lift the lead with a hook or tweezer.

4. Surface solder connections may be removed by one of the methods outlined below. If the board is double-sided, remove the solder joint on each side of the board as if the sides were two separate joints.

5. When all the leads are free, it may be necessary to debond the component body from the board by means of a razor knife or the thermal parting method already described.

6. Remove the solder remaining on the lead areas by means of the wicking or the vacuum extraction method (below).

7. Now resolder replacement components in place by using standard reflow soldering techniques.

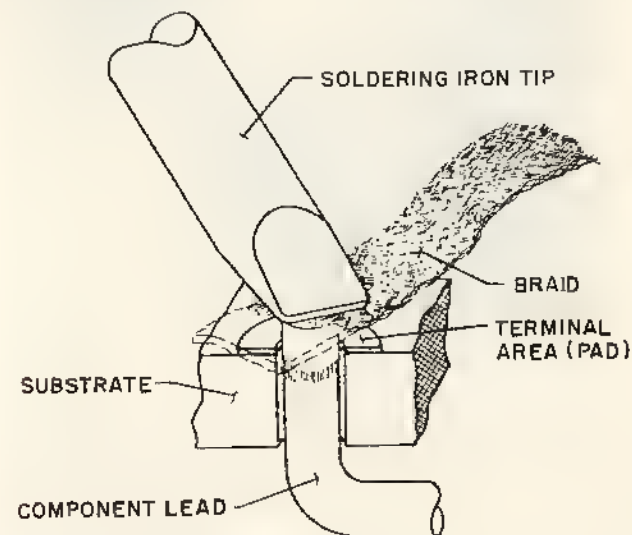


Fig. 13.13 Solder removal by the wicking technique.⁶

13. Solder Joint Removal Methods¹⁰

a. **Wicking.** This method incorporates stranded copper wire or braiding saturated with flux and then applied between the solder and a heated soldering iron tip. The combination of heat, molten solder, and air spaces in the wick creates a capillary action which causes the solder to be drawn into the wick.

The wicking method should be used only to remove surface joints such as those found on single- and double-sided boards (not with plated-through holes) or to remove excessive solder from any flat surface. Figure 13.13 illustrates satisfactory application of the wicking technique. Always be sure to saturate the braiding with liquid rosin flux and use a new end of fluxed braid each time. The previously used portion into which solder has already been wicked should be cut off.

b. **Solder Sucker.** The solder sucker usually employs a single-pulse vacuum-generating device utilizing either a squeeze bulb or a plunger mechanism with a Teflon orifice through which the vacuum is applied. The technique is to melt the solder joint with a soldering iron and insert the solder sucker tip with collapsed bulb or cocked plunger into the molten solder and then release the bulb or plunger (Fig. 13.14). The action creates a pulse of limited vacuum (level and flow) which removes the molten solder. Although the method offers a positive vacuum rather than the low capillary force of the wicking method, it has inherent limitations in its general application and is specifically used with plated-through-hole terminations.

Because of the resweat problem in plated-through-hole joint configurations, the solder sucker should be limited to use on single-sided or surface solder joints to which both the

iron and the sucker tip can be applied simultaneously. The method is not generally recommended for plated-through-hole solder joints.

c. Solder Extraction. The latest and most effective method for solder joint removal is solder extraction. It comprises controlled combinations of heat, pressure, vacuum, airflow, and manipulative qualities for the removal of any solder joint configuration. The solder

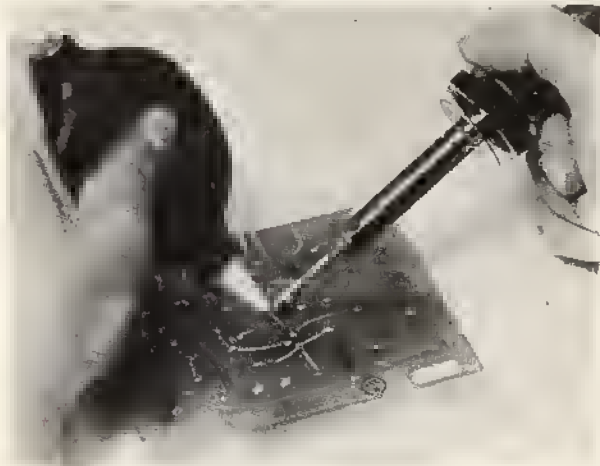


Fig. 13.14 Solder removal by solder sucker technique.¹⁰

extraction device is a coaxial, in-line instrument with the general configuration of a small soldering iron. It consists of a hollow tip, heating element, transfer tube, and chamber located within the handle in which the waste solder and clipped leads are collected. The unit is operated by a power source that provides controlled vacuum or pressure and electrical supply (Fig. 13.15).

The advantage of the solder extraction device is that it is easily manipulated and fully controllable and provides two basic modes of operation: (1) heat and vacuum and (2) heat and pressure (Fig. 13.16). The power source provides variable control over pressure and vacuum levels and flow rates, as well as temperature control over the heated tubular tip.

Power combinations and controls are available as required.

In the vacuum mode the heated tip is applied to the solder joint; and when a melt is noted, the vacuum is activated. That causes the solder to be withdrawn from the joint and deposited in the chamber. To overcome the critical problem of the lead resweating to the sidewalls of a plated-through-hole type joint, the lead is oscillated with the tip while the vacuum is being applied. That permits cool air to flow into and around the lead and the hole sidewalls which causes the lead and sidewalls to cool down and prevent resweat. Solder extraction with continuous vacuum is the only consistent method for overcoming the resweat problem for either dual- or multiple-lead devices terminating in through-hole solder joints. Therefore, it is the method recommended for removing through-hole solder joints (Fig. 13-16a).

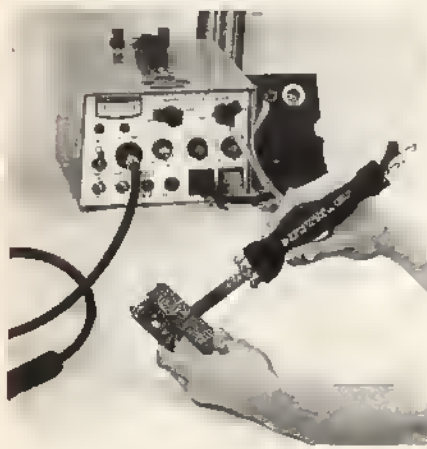


Fig. 13.15 Solder removal by solder extraction.¹⁰

In the pressure mode the tip can apply heat to a pin for melting a back-side sweat joint without contacting the delicate pad, and the air pressure is forced down the hole to break up the sweat joint (Fig. 13-16b).

PLATED-THROUGH HOLES

Plated-through holes, by the nature of their construction, are subject to damage or destruction for a number of normal-use, as well as normal-misuse, reasons. If the plated "barrel" of copper is damaged, so that the electrical continuity it is designed to provide is

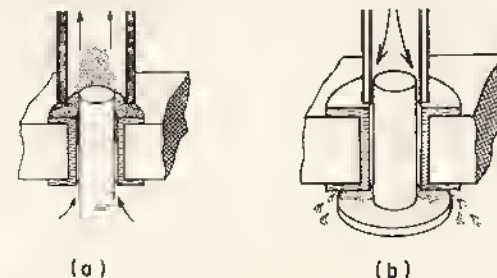


Fig. 13.16 Solder extraction methods; (a) Pressure mode, and (b) vacuum mode.

interrupted, either it must be repaired or the assembly must be replaced. The repair of plated-through holes is restricted to double-sided interfacial connections only; to date there is no reliable method for repairing multiple-layer plated-through holes.

14. **Damaged Plated-through Hole** NOTE: The following limitations apply.

- Restricted to interfacial connections only; not for component-mounting holes.
- For assembled boards only.
- Customer and/or MRB approval must be obtained prior to making repair.
- Number of repairs per assembly to be determined by agreement between supplier and purchaser.

The procedure¹ is as follows:

- Remove components and conformal coating as outlined previously if that is necessary to obtain access to the defective plated-through hole.
- Remove solder by using a vacuum-type solder remover.
- Cut a length of tinned copper wire and insert it into the defective plated-through hole. Clinch both sides so the lay of clinch is approximately $\frac{1}{16}$ in along the conductor.
- Solder clinched leads on both sides by using resin-core solder.
- Replace components if removal was required for access to damaged lead. Solder.
- Remove flux residue with cotton-tipped applicator moistened with solvent.
- Test electrically if applicable.
- Conformal coat if required.

WARPED BOARDS

Warped boards, before and after final assembly and soldering, are a very common and annoying defect that usually is caused by the unequal distribution of stresses on the surfaces of the printed board.³ The defect is particularly troublesome when the boards are mounted on close-tolerance centers or when the board is required to slide into guides or plug into a one-part (board-edge) connector.

15. **Warped Boards without Components** The following limitations apply:

- Specifically for NEMA C-10 and FR-4 or MIL-P-13949 GE and GF.
- This technique has limited application, since the boards may rewarpage.

The procedure¹ is as follows:

- Place the board between two steel plates or in a suitable straightening fixture. Clamp with sufficient force to hold the board flat between the plates.

2. Oven-bake 20 min at 240 to 300°F (115.6 to 148.9°C).
3. Remove from oven and cool to room temperature.
4. Remove from the straightening device and measure warp.
5. Repeat steps 1 to 4 if necessary.

NOTE: The number of times the procedure should be repeated is determined by results obtained from each cycle.

16. Warped Boards with Components

The following limitations apply.

a. Same as *a* and *b* in preceding section.

b. Components mounted on the board should be able to withstand the desired oven temperature without reduction of the service life of the components.

The procedure is as follows:

1. Place selected areas of the board between steel plates or in a suitable straightening fixture (usually edges).

2. Clamp with sufficient force to hold board flat between the plates. Arrange plates in such a way that only open areas of the board free of components are in contact with the clamping pieces.

3. Place in oven for 10 to 20 min at 240°F (115.6°C).

CAUTION: See limitations at beginning of this section.

4. Remove from oven and cool to room temperature.

5. Remove the straightening device and measure the warp.

NOTE: The number of times the procedure should be repeated is determined by results obtained after each cycle.

LAMINATE DEFECTS

Epoxy-glass laminate material is subject to a number of defects inherent in the nature of the material itself. Composed as it is of epoxy-impregnated glass cloth, it is prone to a large number of aesthetic and/or functional anomalies which may be inherent in the raw material or be aggravated by either the manufacture or repair of the printed board.² Unfortunately, the defects are not usually evident until the final steps of manufacturing the board (i.e., outer-layer etch). Even worse, they may be introduced during some type of repair or component replacement step. An overriding consideration is the value of the board or assembly, which usually justifies the cost of the salvage operation.

17. Measling and Weave Exposure Measling is defined as a condition existing in the base laminate in the form of discrete white spots or crosses below the surface of the base laminate reflecting a separation of fibers in the glass cloth at the weave intersection. Weave exposure is a surface condition in which the unbroken woven glass cloth is not uniformly covered by resin. Either defect is subject to agreement between board vendor and user as to acceptance. The defect can result in a functional failure if the condition abridges terminals on the printed board.

NOTE: The following limitation applies. Subject to agreement between manufacturer and user, but not usually exceeding 5 percent of the board area.

The procedure is as follows:

1. Area to be repaired must be thoroughly cleaned by brushing with isopropyl alcohol.
2. Bake board at 100°C for at least 45 min.
3. Brush authorized conformal coating material on the affected areas. The coating must extend a minimum of 0.062 in beyond the defect in all directions.
4. Cure conformal coating per manufacturer's recommendations.

18. Laminate Voids Included in this category are blistering and, in the case of multilayer boards, surface-layer delamination. Frequently found after dip or wave soldering, the defect is usually the result of moisture entrapped in the board in combination with a basic material shortcircuiting. Cross cases of widespread internal delamination are not treated here; they should be cause for rejection of the multilayer assembly.

NOTE: The following limitations apply.

a. Number of repairs (or area repaired) per board should be determined by the specific requirements of the procuring agency or customer.

b. Cost of the board or assembly should justify the rework.

c. Voids caused by blisters should be accessible by the methods used herein.

d. Only electrically functional boards should be repaired.

The procedure¹ is as follows:

1. Clean board thoroughly and remove all surface contamination and loose foreign matter.
2. Puncture each blister with a dental pick in at least two small areas, opposite each other, around the perimeter of the blister.
3. Place the board in the oven and use the time and temperature recommended to cure epoxy.
4. Remove from oven and add the epoxy material over one of the openings around the perimeter of each blister. Epoxy may be applied by using a hypodermic needle. The heat of the board will draw the epoxy into the void area and fill it completely.
5. (Optional) If it is determined that evacuation is necessary to remove possible included air, it should be accomplished at this point.
6. Cure the epoxy by using the recommended time and temperature.
7. Perform electrical tests of all interconnects in and around reworked area.

19. Cracking Cracking is defined as a condition existing in the base laminate in the form of connected white spots or "crosses" on or below the surface of the base laminate. It reflects the separation of fibers in the glass cloth and connecting weave intersections. It is serious because it can provide an entrance and/or dwelling place for moisture and, if terminals are abridged, a path for electrolysis and subsequent insulation resistance breakdown.

NOTE: The following limitations apply.

a. The number of repairs per board should be determined by the specific requirements of the customer.

b. The cost of the board or assembly should justify the rework.

c. Only electrically functional boards should be repaired.

d. Cracking in external layers with maximum length of 0.100 in.

The procedure is as follows:

1. Remove the affected laminate by means of the miniature machining abrasion method using a ball mill, or an S. S. White (or equivalent) air abrader with appropriate nozzles and abrasive material.
2. Clean areas to be filled by scrubbing with Freon TF.* Use a stiff nonmetallic brush, and take care not to scuff circuit lines, components, etc.
3. Preheat board at 65°C for a minimum of 30 min.
4. While the board is hot, fill the holes by using a syringe which has been filled with an appropriate epoxy replacement material (such as Epon 828). The board must be hot, and it must be reheated if it cools to room temperature.
5. After filling the void with epoxy, allow the board to sit in a dirt-free atmosphere for 15 to 20 min; then cure it in a controlled oven for four hours at 65°C.

MULTILAYER BOARDS

In many ways the multilayer board is similar to single- and double-sided boards when rework is involved. Nearly identical procedures are used for outer-layer rework, component replacement, repair of laminate defects, and conductor repair. Internal layers of multilayer boards are treated as single-sided boards prior to lamination. An added precaution when working on multilayer boards concerns susceptibility to damage from heat. Many of the repair techniques mentioned earlier in this chapter are applied in combination with some type of heat application (e.g., component replacement and conductor repair), and it is extremely important to minimize the amount of heat added to the multilayer assembly during any repair step. Among the problems which may be caused are damage to the plated-through hole, scorching of the board with subsequent possible loss of insulation properties, and hole contamination by flow of epoxy hinder.

The only repair technique reserved exclusively for multilayer boards involves the defects which may be found in internal layers after assembly; in this case they are internal shorts. Other repairs for this type of board may be found in detailed areas previously covered.

*Registered trademark E. I. du Pont de Nemours & Company.

20. Internal Shorts! This defect consists of shorts between internal layers and/or the ground.

NOTE: The following limitations apply.

a. The number of repairs per board should be determined by the specific requirements of the customer.

b. Shorts in the connector area are to be repaired in unassembled board only.

The procedure is as follows:

1. Make a series of resistance readings and/or continuity checks and record the data. Determine, if possible, the exact location of short or shorted holes before attempting any drilling.

2. Drill out the barrel of the suspected hole or holes slightly larger than the barrel OD.

3. Examine the hole carefully to be sure that the whole barrel has been completely removed.

4. Repeat resistance and/or continuity checks to verify removal of the short.

5. Clean hole or holes with isopropyl alcohol and mask area around the hole.

6. Fill the hole with epoxy and cure.

7. Abrade flush to surface of board and clean with isopropyl alcohol.

8. Hard-wire, externally, all circuits removed by drilling and solder with rosin-core solder.

NOTE: Care should be taken in routing external strappings to maintain wiring rules and circumvent crosstalk.

9. Remove all flux residue with cotton-tipped applicator and isopropyl alcohol.

10. Apply a conformal coat over external wires.

11. Cement all external wiring to the board surface if required or if board is not conformal-coated.

NOTE: A good rule of thumb is to tack external wiring down with cement in increments of approximately one inch. Each tack should be $\frac{1}{4}$ in in length.

REFERENCES

1. Rx for Repair and Modification of Printed Wiring Boards, UPC-R-700A, September, 1971, Institute of Printed Circuits, Evanston, Ill.
2. C. A. Harper (ed.), "Handbook of Electronic Packaging," McGraw-Hill Company, New York, 1969.
3. D. P. Schnorr, Printed-Wiring Processing Materials and Techniques, *Electronic Packaging and Production*, July, 1974.
4. Richard Martz, Welded Repair of Severed Conductors on Printed Wiring Assemblies, NEP/CON CENTRAL '70, April, 1970.
5. Solder and Soldering, *Department of the Army Technical Bulletin, TB-Sig. 222*, Change No. 1, unclassified.
6. Trouble Shooting and Repairing and Printed Circuits, Ungar Electric Tools, Inc. 16-page booklet, date unknown.
7. "Maintenance Procedures for Parts Mounted on Printed Wiring Boards," NAVSHIPS 94324, March, 1965, Sec. 10.
8. J. R. Matzinger, Repair of Damaged or Defective Printed Wiring Board Conductors, NEP/CON CENTRAL '70, April, 1970.
9. J. R. Matzinger, Repairability of Printed Wiring Boards Containing Microelectronic Devices, *Solid State Technology*, September, 1968.
10. William J. Siegel, "Support of Modern Electronic Equipment: Fact or Fantasy," Pace Incorporated, Silver Springs, Mo., March, 1969.

Section 4

Soldering

Designing and Presoldering Considerations for Soldering

HUGH COLE

Cobar Resources, Dallas, Texas

Introduction	14-4
Design Considerations	14-4
1. Wire-to-Hole Ratio	14-4
2. Size and Shape of the Land Area	14-4
3. Number and Direction of Extended Parallel Lines	14-4
4. Population Distribution	14-4
Material Systems	14-4
5. Common Metallic Surfaces	14-5
Wetting and Solderability	14-5
Solderability Testing	14-6
6. Testing Procedures	14-7
Plating Coatings for Preserving Solderability	14-8
7. Fusible Coatings	14-8
8. Soluble Coatings	14-8
9. Nonfusible and Nonsoluble Coatings	14-9
10. Organic Coatings	14-9
Tin-Lead Fusing	14-10
11. Thick Fused Coatings	14-10
12. Thin Fused Coatings	14-11
13. Problems in Reflowing Plated Coatings	14-11
Solder Resists	14-12
Solderability and the Plating Operation	14-12
14. Effect of Organic Plating Additives	14-12
15. Effect of the Plating Anode	14-13
Solderability Considerations for Additive Circuitry	14-13
Use of Pre-cleaners to Restore Solderability	14-14
16. Causes of Poor Solderability	14-14
17. Cleaning Tin-Lead Surfaces	14-15
Standard Solderability Tests	14-15
18. Edge Dip Test	14-15
19. Test for Wave-Soldered Boards	14-17

INTRODUCTION

This chapter deals with the specification of the materials system for the printed circuit and the design parameters which must be considered before a final circuit is laid out. The soldering operation must be considered from the inception of the board layout in order to ensure satisfactory performance. The rules are simple and straightforward; if they are followed, the operation should run smoothly and efficiently. If they are disregarded, however, the result is invariably a large number of touch-up operators handling recurring problems with bridges, icicles, and imperfectly formed fillets.

DESIGN CONSIDERATIONS

During the layout of the board, several soldering parameters should be carefully considered. They are (1) the wire-to-hole ratio, (2) the size and shape of the terminal areas, (3) the number and direction of extended parallel circuit runs, (4) the population density of the solder joints.

1. Wire-to-Hole Ratio The wire-to-hole ratio represents a compromise between the ideal situation for assembly (large hole and small-diameter lead) and the ideal situation for soldering (smaller lead-to-wire ratio). The minimum hole size can be established by the rule of thumb that it should be no less than the lead diameter plus 0.004 in. The maximum hole diameter should be no more than 2.5 times the lead diameter. Of course, if the board is a plated-through-hole or a multilayer circuit board, the hole-to-wire ratio should be lower than 2.5 to encourage the capillary action of the flux and solder during the soldering operation.

2. Size and Shape of the Land Area The pad area around the solder joint is normally either circular to slightly elongated (teardrop). It should not be more than 3 times the diameter of the hole in the board. There is sometimes a tendency, particularly on low-density boards, to leave large irregular land areas around the holes. That should be avoided! Excessively large land areas expose too much copper to the solder pot, cause excessive quantities of solder to be used in joint formation, and promote bridging and webbing.

If the leads are to be clinched during assembly, the land should be so oriented that the clinched lead will be in the center of the elongated pad. Both the pad and the component should be so oriented that the clinched lead is parallel to the direction of solder flow in the solder wave and not perpendicular to it.

3. Number and Direction of Extended Parallel Lines The use of automated printed board layout programs and the trend toward high-density circuit packaging have resulted in a tendency to group large numbers of circuit paths together and run them parallel to one another for long distances. If those paths are oriented perpendicularly to the direction of flow in the solder wave (i.e., at right angles to the direction of the conveyor), then they can contribute to bridging and webbing. Every effort should be made to maximize the spacing between lines which must be oriented perpendicular to the direction of the conveyor.

4. Population Distribution An excessive number of joints in one area promotes bridging, icicleing, and webbing. It may also cause a heat-sinking effect and interfere with the formation of a good solder joint.

MATERIAL SYSTEMS

In the soldering process there are two surfaces which must be considered before a solder flux is selected. They are the lead surface and the pad surface. The average printed circuit assembler has little control over the material systems used in component leads, since the selection is usually made for him by the component manufacturer. Furthermore, most components are mass-produced and supplied on large reels. It is not economically justifiable to treat each lead according to an individual assembly shop's specific requirement. Therefore, in selecting the components, care should be taken that the leads are solderable, and an incoming inspection should be established to ensure lead solderability.

The board itself, however, is a different story. Since each board is custom-manufactured, the assembly or soldering engineer can exercise a great deal of control over the material systems used on the board. Again, it is important, in order to keep defects to a minimum, that the board be made of a solderable material and that the solderability of the board be checked as a part of incoming material inspection. The next section will deal with some typical material systems encountered during the soldering process.

5. Common Metallic Surfaces

a. Bare Copper. Because of its low cost and ease of processing, one of the most common metallic surfaces encountered is bare copper. Chemically clean copper is the easiest material to solder; it can be soldered with even the mildest fluxes. But unless it is protected with a rosin-based protective coating, its solderability will rapidly degenerate because of oxides and tarnishes. As we shall see later in this chapter, however, the solderability of tarnished copper surfaces is easily restored with surface conditioners. If boards with bare copper surfaces are used, care should be taken to maintain solderability during handling and storage (storage time should be minimal), and the boards should not be stored in the presence of sulfur-containing material such as paper, cardborax, or newsprint. Sulfur produces a tenaciously adhering tarnish on copper which seriously impairs the solderability.

b. Gold. Gold is encountered most commonly on component leads and plug-in finger surfaces. It is a highly solderable material, but it is extremely expensive and it rapidly dissolves in the molten solder. Because it affects the properties of the solder joint, causing the joint to become dull and grainy, it is usually avoided or eliminated by pretinning the lead before soldering. Various studies have shown that all gold on a gold-plated lead can be dissolved in a solder pot of eutectic tin-lead solder within 2 s (plate thickness of about 50 μ m). Therefore, pretinning is economical as well as easy.

c. Kovar. Many dual in-line packs and related integrated circuitry are supplied with Kovar leads. Kovar is a very difficult metal to solder because it doesn't wet well. For that reason component manufacturers and/or assembly shops prefer to pretin Kovar. The pretinning is normally accomplished fully with organic acid fluxes and certain proprietary acid cleaners.

d. Silver. Although silver was once very popular in the electronics industry, it is not used on terminal areas or component leads. The reason is the problem of silver migration, a phenomenon discovered in the late 1950s and extensively researched during the early 1960s. Silver should be avoided. If it must be used, it is an easily soldered material and should be treated similarly to the bare copper surface (i.e., avoid sulfur-bearing materials and minimize storage and handling).

e. Immersion Tin. Immersion tin coatings are electrolessly deposited coatings of tin metal on bare copper surfaces. When tin is initially deposited, the coating is extremely solderable. It does, however, deteriorate rapidly, and it becomes more difficult to solder than bare copper. Originally, immersion tin coatings were used to protect the solderability of bare copper surfaces and thereby extend the shelf life of the board. Experience has shown, however, that fused tin-lead plate is far superior for the purpose.

f. Tin-Lead. Tin-lead coatings are put on printed boards and component leads to preserve the solderability of the material. They can be applied by electroplating, hot dipping, or roller coating. The mechanics of the processes are discussed elsewhere in this chapter. A properly prepared tin-lead surface should exhibit excellent solderability and long shelf life—about nine months to one year. Tin-lead coatings can be soldered with most rosin-based fluxes, even the nonactivated types. However, optimum results are obtained with the activated rosin fluxes.

WETTING AND SOLDERABILITY

Soldering is defined as a metallurgical joining technique involving a molten filler metal which wets the surface of both metals to be joined and, upon solidification, forms the bond. From the definition it is apparent that the materials to be soldered do not become molten and therefore the bonding occurs at the interface of the two metals and is strongly dependent on the wettability or solderability of the base metal by the molten alloy. Although the base metal does not become molten, some alloying can take place if the base metal is soluble in the filler metal. The bond which is formed is strictly metallic in nature,

and no chemical reaction which covalently or ionically bonds the metal to the surface occurs.

To understand the basic mechanism of soldering, it is necessary to understand the thermodynamics of wetting. Fortunately, however, in order to understand wetting it is not necessary to understand thermodynamics. Wettability or solderability of two materials is a measure of how well one material "likes" the other. The property can be easily visualized by using a water drop resting on the surface, such as the one shown in Fig. 14.1. When the water drop doesn't like the surface on which it rests, it pulls up into a ball and touches the surface, in the idealized case, at only one point. The angle between the drop and the surface at the point of contact is called the dihedral angle. If the drop likes the surface, it spreads out all over the surface and comes in intimate contact with it. Various degrees of

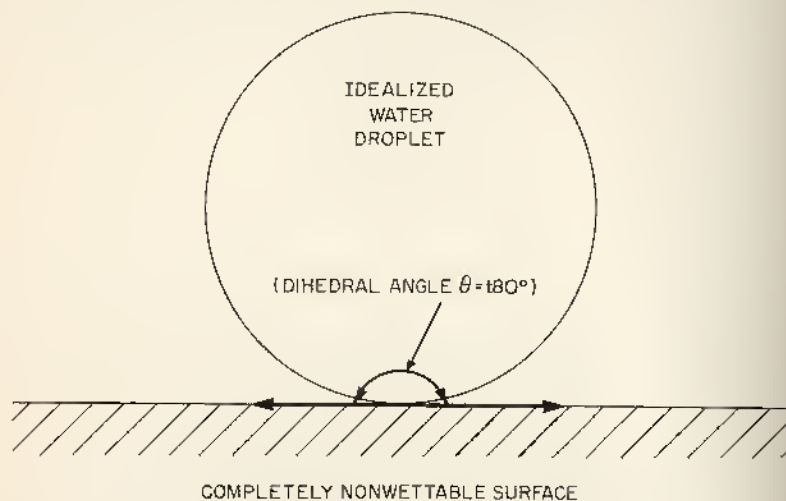


Fig. 14.1 Complete nonwetting of an idealized surface.

wettability are therefore related to the ability of the drop to spread out or wet the surface. Figure 14.2 shows the relation between the dihedral angle and the various wetting states. Wettability or solderability is related to the surface energy of the material. Wetting is substantially improved if the surface is clean and active (i.e., if all dirt and grease are removed and no oxide layer exists on the metal surface).

Therefore, to form a solder bond efficiently, we must start with a material system which can be wet by the molten solder, and the cleanliness of the parts must be maintained.

SOLDERABILITY TESTING

Solderability testing is an important quality control procedure in the electronics industry. It is a simple procedure but problems can occur if the fundamental test principles are not understood thoroughly. "Solderability" is a measure of the ease (or difficulty) with which molten solder will wet the surfaces of the metals being joined. When molten solder leaves a continuous permanent film on the metal surface, it is said to wet the surface. Wetting is a surface phenomenon which depends on cleanliness. Fluxing facilitates wetting by cleaning the surface, and the degree of surface cleanliness depends on the activity of the flux.

However, there is often a limitation on the activity of the flux in the electronics industry. Most electronics soldering operations require relatively weak rosin-based fluxes to avoid the possibility of current leakage caused by flux residues remaining on the part. To enhance the solderability of some surfaces and eliminate the need for active fluxes, electroplating often is employed to deposit a solderable coating over a base metal that tarnishes easily or is difficult to solder.

6. Testing Procedures Testing for solderability can be a simple procedure of inspecting production parts or of dipping an appropriately fluxed lead or portion of a printed board in a solder pot and observing the results. Good and bad wetting are then identified visually. The problem is to recognize borderline cases which simulate effective solderability but quickly deteriorate. In order to alleviate borderline solderability, the mildest possible flux should be used at the lowest soldering time-temperature relations which will give adequate results.

An effective solderability test involves the use of water-white rosin flux and a solder pot. The surfaces to be checked are fluxed and immersed for 3 to 4 s in the solder pot, which is maintained at approximately 500°F. The solder is then permitted to solidify, and the components are cleaned of flux residues prior to visual examination. The inspection is usually performed with either no or low magnification (5× to 10×). Most solderability tests will permit up to a 5 percent imperfection of the total surface, provided the entire imperfection is not concentrated in one area.

More elaborate solderability tests are described in governmental and industrial specifications. Testing of component leads is covered by Electronic Industries Association (EIA) Test Method RS17814, which is similar to the solderability method described in Military Specification 202, Method 208. The test incorporates a dip fixture which provides identical dip ratio and immersion times (Fig. 14.3).

a. Dip Test. For printed boards, the edge dip test, described in the EIA specification RS319 and by the Institute of Printed Circuits (IPC)* in standard S801, often is employed. The edge of a printed board is dipped first in a mild flux and then in a solder pot for a predetermined time and temperature. After the flux residues are removed, the board is inspected visually for the quality of wetting. A similar test is employed to determine the solderability of solid wire leads, terminals, and conductive accessories of component parts normally joined by soft solder. Applicable test standards are EIA

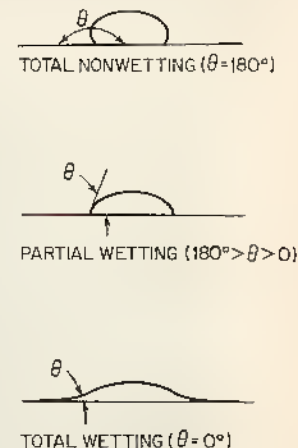


Fig. 14.2 Relation between dihedral angle θ and the degree of wetting.

*The Institute of Printed Circuits is now called the Institute for Interconnecting and Packaging Electronic Circuits, but is still referred to as IPC.

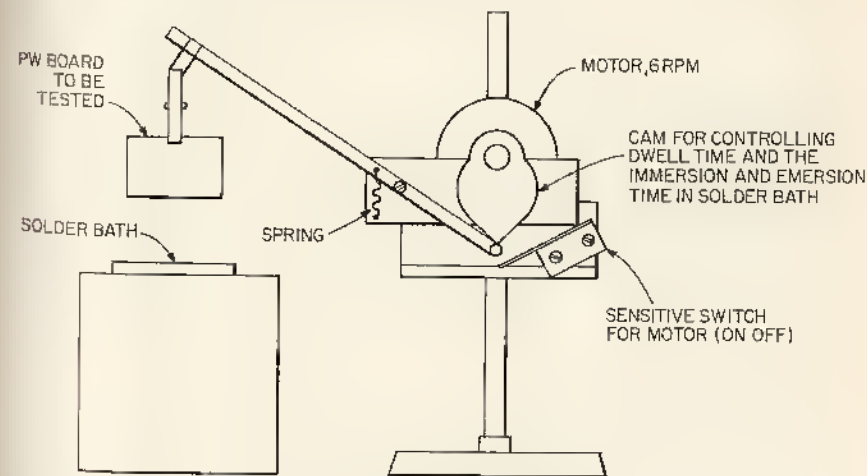


Fig. 14.3 Solderability dip tester.

Standard RS178A, Solderability Test Standard, and MIL-STD-202C, Method 208 A, Solderability.

To perform the dip test, the operator places the item to be tested into a holding arm. The arm lowers the sample section into the solder pot. After the preset dwell time has elapsed, the arm automatically raises the sample. Then a visual determination of solderability is made. The dip test can also be performed as a manual operation, but that leaves too many variables to the discretion of the operator.

Since interpretation of results is based on a subjective judgment by the operator, it is essential that the operator be provided with examples of good, marginal, and poor solderability. It is imperative, too, that pot temperatures, cleaning and fluxing procedures, dwell times, and solder purity be controlled carefully in order to obtain meaningful results.

b. Globule Test. This test is one that is prevalent throughout Europe and is mandatory for European suppliers whenever specifications dictate. It is described in the International Electrochemical Commission Publication 68-2 Test T, Solderability. The globule test provides a numerical designation for the solderability of wires and component leads. It measures the ability of the solder to wet the lead.

A lead wire, coated with a nonactivated rosin flux, is placed in a holding fixture. It is gripped and straightened and then lowered into a globule of molten solder. The volume and temperature of the globule are controlled. As the lead wire bisects the globule, the timer actuates and measures the time between the moment the wire contacts the solder and the moment the solder flows around and covers the lead. At this second point, the timer stops, and the elapsed time is registered on a read-out. Elapsed time is indicative of the solderability of the lead: the shorter the time, the greater the solderability.

This globule test method is completely automatic and is designed for continuous operation. Time is measured to $\frac{1}{100}$ s. Wires that are 0.008 to 0.062 in in diameter can be tested, and special heat regulators can hold solder temperature to $\pm 2^\circ\text{F}$.

When wire plated with a soluble or fusible coating is tested, it is advisable to perform a dip test to supplement and verify the globule test findings. The reason is that, under certain conditions, the plated coating might be totally reflowed or dissolved during the soldering operation, which would give misleading results.

PLATED COATINGS FOR PRESERVING SOLDERABILITY

The three commonly used types of plated coatings are generally referred to as fusible, soluble, and nonfusible and/or nonsoluble. Fusible electrodeposited coatings provide corrosion protection to a surface that has been activated for soldering. Whether the solder bond of a soluble electrodeposited coating is to the base metal or to the deposit depends on soldering conditions and coating thickness. Nonfusible and/or nonsoluble electrodeposited coatings are used frequently as barrier layers in electronic applications to prevent diffusion of solder and base metal.

7. Fusible Coatings Tin and tin-lead electrodeposited coatings are commonly used in electronic applications to preserve solderability because they are fusible and do not contaminate the solder pot or fillet. Contamination can adversely affect tensile, creep, and shear strengths at a solder connection. Also, contamination of a part can reduce the flow and spread of the solder on the part.

If the coating operation is not closely controlled, an electrodeposited coating may be applied over a partially contaminated surface. If that happens, dewetting will occur in the contaminated areas because the electrodeposited coating is fused during soldering. Therefore, adequate cleaning prior to plating is essential to obtain good solderability. Figure 14.4 shows a surface which has dewet after a reflow operation.

Another point to remember is that plating thickness should be sufficient that porosity is virtually eliminated. Porosity and codeposited impurities will lower the protective value of the electrodeposited coating and will eventually cause poor solderability.

8. Soluble Coatings Soluble coatings commonly used in electronic applications include gold, silver, cadmium, and copper. During soldering, these metal coatings are either completely or partially dissolved. The amount of dissolution depends on solubility of the coating metal, thickness of the deposit, and the soldering conditions. Silver and copper

tend to tarnish; and if a mildly activated flux is called for, they should be protected with a thin rosin coating. Cadmium offers sacrificial corrosion protection which often necessitates the use of highly activated fluxes to promote effective soldering.

Soluble gold coatings provide excellent corrosion and chemical resistance. However, because of the high cost, fairly thin coatings are used. Moreover, care should be taken because gold coatings of under $50\text{-}\mu\text{m}$ thickness tend to be porous and, as a result, lower the protective value of the metal. Corrosion of the base metal or barrier plate via pores causes soldering problems because the gold usually dissolves completely during soldering and it is difficult to wet the corroded base metal. Also, solderability decreases with the

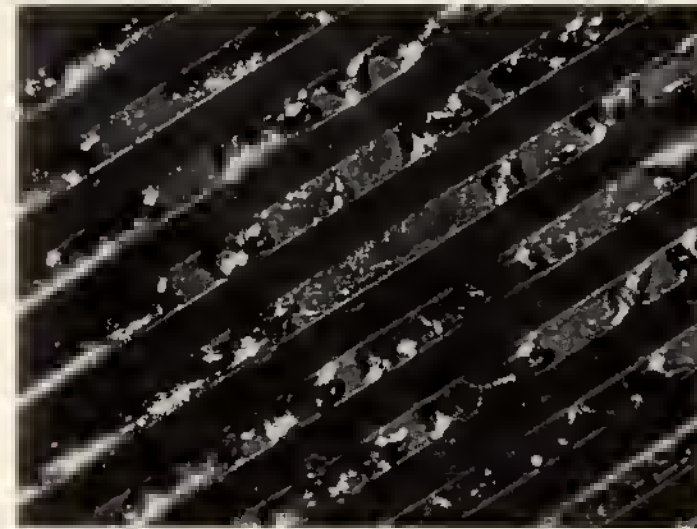


Fig. 14.4 Tin-lead-plated surface which exhibits dewetting after the reflow operation (Alpha Metals Inc.).

amounts of alloying elements that are used to increase hardness and are often codeposited with the gold. Thicker gold coatings, on the other hand, may cause brittleness in a solder connection because of formation of gold-tin intermetallic compounds.

9. Nonfusible and Nonsoluble Coatings Nickel and tin-nickel electrodeposited coatings are considered to be nonfusible and/or nonsoluble because they provide an effective barrier to prevent alloying of solder to the base metal in electronic applications. They allow for effective soldering to such materials as aluminum and silicon. However, problems can occur from passivity, which is caused by codeposited impurities, or from some addition agents that are used in nickel plating to increase hardness and reduce internal stress. In such cases, nickel and tin-nickel electrodeposited coatings should be protected with tin or tin-lead to improve the shelf life of the soldered part.

Furthermore, nickel and tin-nickel have limited solubility in solder, and a flash coating is not an effective barrier. When electroless nickel is used as a barrier coating on aluminum, the required thickness will depend on soldering conditions. A thickness of 50 to $100\text{ }\mu\text{m}$ is sufficient for most operations.

10. Organic Coatings Although the metallic coatings for preserving solderability are the most reliable and effective coatings, they are also the most expensive ones. For applications that do not require a long shelf life, considerable economics can be obtained by using an organic protective coating. There are several basic types of organic protective coatings—water-dip lacquers, rosin-based protective coating, and the benzotriazole-type coating. Organic protective coatings must be easily removable, and they must be compatible with the rosin-based fluxes normally used in the printed board industry.

Water-dip lacquers were once a very popular means of protecting the solderability of printed boards. However, they have fallen into disfavor because of their tendency to set up and polymerize with age. As they age, they become extremely difficult to remove. They also become insoluble in the flux solvent. If they are not properly removed after fluxing, they leave white residues which are not corrosive but are extremely unattractive.

Today rosin-based protective coatings are much more prevalent than water-dip lacquers. They are applied by dip, spray, or roller coating, and, depending on the thickness, they will provide solderability protection for six weeks to four months. They are composed of the rosin solids material, and as such they are quite compatible with the rosin thinners and the normal cleaning method used to clean rosin fluxes. When those materials are used, it is important to ensure that adequate fluxing occurs and that the preheating time and temperature are sufficient to allow the protective coat to melt and be displaced

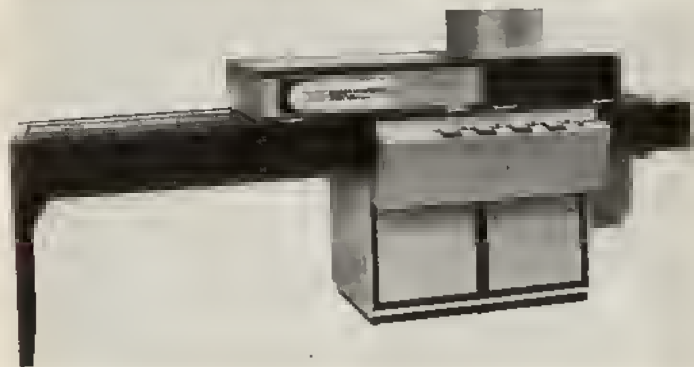


Fig. 14.5 Conveyorized infrared fusing machine (Research Incorporated).

by the flux. If the coat is not displaced, soldering will occur with a rosin nonactivated flux, no matter how active the solder flux really is.

A third alternative which some people have explored is the benzotriazole-type coating. Benzotriazole is an organic compound which is applied to the board surface during the final rinse operation of the plating line. It forms a thin nonporous film on the copper surface and prevents oxygen from reacting with the surface copper molecules. Benzotriazole films are very fragile, and they cannot be subjected to handling or scraping abuse.

TIN-LEAD FUSING

Fused coatings ordinarily are electrodeposited, low-melting metals or alloys that have been heated sufficiently above their melting points to become completely molten. In the molten state, alloying between the liquid and the basis metals is accelerated; on solidification, the deposit usually is dense and nonporous. The procedure is commonly known as "reflowing," and it usually connotes tin or tin-lead electrodeposits. However, it can also apply to the remelting of hot-dipped coatings.

Fused coatings are employed to guarantee that the cleaning procedure prior to plating is adequate, to produce a slightly denser deposit with less porosity, and to improve the appearance of the coating. Reflowing leaves a bright deposit which has a definite sales appeal. A typical conveyorized infrared reflow oven is shown in Fig. 14.5.

11. Thick Fused Coatings Tin-lead electrodeposits that are reflowed are usually between 300 and 500 μin in thickness. That thickness provides adequate protection with a minimum of reflow problems. When printed boards are reflowed, the tin-lead deposit forms a meniscus on the conductor pad. For that reason the edges of the pad have a much thinner coating than the original plating, whereas the center is thicker. The average thickness is the same, but it is distributed differently. As the plating thickness increases

over 500 μin , the surface forces are not always sufficient to hold the solder in the meniscus—especially on wide pads; the solder shifts upon solidification and the deposits appear uneven. On inspection, that may be mistaken for a dewetting condition, but it actually is a shifting of the molten solder before solidification.

When tin-lead with a thickness over 500 μin is reflowed, the boards must be in a horizontal position and be withdrawn in a smooth manner. Otherwise, shifting of the molten solder is inevitable. Because of that problem, there is a practical limit on the coating thickness. In recent years, some military specifications have been calling for 0.001 to 0.0015 in (1000 to 1500 μin) in reflowed tin-lead plating. The main objective is to obtain



Fig. 14.6 Hot-air solder-leveling machine. (Courtesy Electrovert, Inc.)

maximum corrosion protection. Deposits of that thickness have been reflowed, but not without processing difficulties.

Improvements in infrared reflow equipment have increased the use of thick fused coatings in recent years. Reflowed tin-lead electrodeposits offer several advantages. One advantage is that they provide a 100 percent quality control check immediately after etching. By examining the boards after reflow, problems in hole drilling, cleaning procedures, plating, and etching can be detected and corrective action can be taken immediately. The procedure has had a dramatic effect in improving printed board reliability.

Another benefit is that solder slivers are eliminated. During etching there is undercutting where the tin-lead coating overhangs the conductor pad. Under certain vibration conditions, the overhang can fall off and cause a short circuit. Fusing eliminates the condition and adds to board reliability.

12. Thin Fused Coatings Thin fused coatings level the solder during fusing when they are applied by procedures such as roller coating, spin coating, and hydro-squeezing. The leveling is accomplished by means of a hot liquid ejected from spray nozzles. The techniques usually result in coatings under 50 μin , hardly enough for adequate corrosion protection. Also, thin coatings can mask poor solderability when tin-lead is plated over unsolderable copper. For those reasons, use of leveling techniques has diminished in recent years. Figure 14.6 shows one system in use.

13. Problems in Reflowing Plated Coatings Codeposited impurities, especially copper in tin-lead, can cause dewetting in a reflowed deposit. Variations in alloy composition raise

the melting point and cause reflow problems. That is particularly true when organic contamination results in poor solution throwing power and consequent high lead deposit in the plated-through holes. Heavy oxidation or tarnish films that result from chemical attack by etching solutions must be removed prior to reflowing. The films can act as insulating barriers and interfere with reflowing.

SOLDER RESISTS

Solder resists are polymer materials which are silk-screened or laminated onto the board to protect the circuitry selectively while the pads are being soldered. The solder resists minimize bridging and webbing and restrict the amount of solder dragged from the pot during the soldering operation. A solder resist also minimizes the amount of board area in contact with the solder pot and thereby restricts the metallic contamination buildup in the solder pot.

There are two basic types of solder resists, temporary and permanent. Permanent solder masks are meant to remain on the board, so they are usually made of durable scratch-resistant material such as epoxy. Permanent solder masks can be applied over copper or over solder plate. Resists which are applied over solder plate must be capable of containing the molten solder when it flows during the soldering operation.

Temporary solder masks are applied as protection during the soldering operation. For instance, a temporary mask would be applied to the gold contact fingers during soldering to prevent the fingers from being soldered and thereby ruined.

SOLDERABILITY AND THE PLATING OPERATION

Blowholes in plated-through holes on printed boards normally are caused by solution entrapped in the hole, but they also can be caused by an excess of organics occluded in the electrodeposits. The heat involved in soldering causes moisture and entrapped chemicals in the laminate to build up in pressure and escape through voids or cracks in the plating. That results in blowholes. The problem can usually be alleviated by prebaking at 180 to 200°F.

In some cases blowholes can be caused by an excessive amount of occluded organics in the electrodeposit. The problem is more prevalent with bright plating deposits such as tin and tin-lead. When blowholes are caused by excessive organics occluded in the deposit, prebaking will not remedy the problem. The occluded material can be released only when the metal liquefies.

Figure 14.7 shows a cross section of a fused tin-lead surface that had an excessive amount of organic material. Note that the organic material tends to create voids in the plate which rise to the surface. The result is a grainy, pitted appearance after solidification.

14. Effect of Organic Plating Additives In order to solder successfully to electrodeposits that employ addition agents, it is extremely important that the additives be carefully controlled. Additives usually are essential to produce sound deposits of tin and tin-lead alloys. In electroplating, chemicals are added to the basic formulation of a plating solution to enhance the properties of the deposit. Some properties that can be improved by additives are throwing power, smoothness, hardness, leveling, brightness, and speed of deposition.

Normally, when foreign metals or organic materials are present in a plating solution, the properties of the deposit are adversely affected. In some rare cases, beneficial effects can be produced by codepositing small amounts of other metallic ions or occluding organic material. When that occurs, the materials are classified not as contaminants but as addition agents. When they produce a bright deposit, they are called brighteners. Strictly speaking, they are controlled impurities.

When organic additives are employed, some forms of the compounds are absorbed at the cathode surface during electrolysis. Often, the compound absorbed is a decomposition product of the original material. The amount that is absorbed is proportional to the nature and concentration of the compound and the time of electrolysis. Frequently, the organic decomposition products develop over a period of time and may affect the deposit

adversely. Six months to a year may pass before critical concentrations are reached. If the breakdown products can be removed by an activated carbon treatment, the problem can be controlled. It always is a worthwhile practice to remove organic addition agents and their breakdown products by a carbon treatment at least two or three times a year. That assures continuous operation of the plating solution without unscheduled purification treatments during peak production.

15. Effect of the Plating Anode When inorganic contaminants build up in a plating solution, increased concentrations of the additives are normally required to produce the desired effects. Poor-quality anodes constitute the major source of inorganic contaminants. In tin and tin-lead plating, at least a 99.9 percent purity anode is required. High-purity chemicals and efficient rinsing before plating are also essential to maintain a high-purity plating solution. Operation of tin-plating solutions at temperatures below 65°F can reduce the amount of organics occluded in the deposit.

When excess organics are occluded in a tin or tin-lead electrodeposit, bubbling is often observed during soldering. That often can be the cause of blowholes in soldering a



Fig. 14.7 Organic material from plating bath occluded in reflowed tin-lead deposit. (Alpha Metals, Inc.)

printed board. However, it should be noted that the majority of blowholes are caused by entrapped plating or cleaning solutions. When postbaking does not alleviate the problem, occluded organics in the tin or tin-lead electrodeposits are a likely cause.

SOLDERABILITY CONSIDERATIONS FOR ADDITIVE CIRCUITRY

Additive printed circuitry is a process whereby conductive patterns are formed on a plastic laminate by means of electroless plating. In the process, the copper conductor pads are usually of extremely high purity and, as deposited, are usually very solderable. However, there can be a problem in preserving solderability. Although additive circuitry is not subjected to an etching solution that activates the copper and renders it susceptible to reoxidation, it will oxidize in time and thus can produce solderability problems. The key to soldering additive circuitry is protection of the copper during processing.

In subtractive circuitry, in which the board is laminated with copper and additional copper is deposited on both by electroplating, solderability is usually preserved by the deposition of tin-lead on the copper. The tin-lead serves the dual function of being an etch-resist metal and preserving solderability. The copper is rendered solderable in the cleaning process prior to the deposition of the tin-lead, and the coating usually provides the necessary corrosion protection for preserving solderability during storage. That usually permits soldering with mildly activated fluxes. To obtain higher reliability, it is often specified that the tin-lead be reflowed. Reflowing is a process in which the coating is heated over the melting point and allowed to solidify in place. If nonsolderable conditions exist, they can be readily observed, because dewetting will occur or the tin-lead will not fuse. Proper fining will result in a smooth, shiny surface. The process affords a 100 percent inspection immediately after etching and has had a marked effect on improving reliability of solder joints.

In additive circuitry, tin-lead cannot be plated because there is usually isolated circuitry

that cannot be electroplated. Other means must therefore be employed to preserve solderability. One is to use rosin- or resin-type protective coats. The process is designed to preserve solderability of copper during in-plant processing, but it was never intended for long-term storage. If tarnishing of the copper occurs during storage, soldering can be a problem because, although the coating is a mild flux, it dilutes the activator content of the soldering flux. If soldering problems arise when this type of coating is present on a printed board, it is usually suggested that the coating be removed and precleaning be performed on the copper surface prior to the soldering process. In wave-soldering it is also important that adequate preheating be employed so that the activator in the soldering flux can be dispensed into the protecto-coat prior to soldering.

Another means of preserving solderability is to employ immersion tin, a chemical process by which a tin coating is applied over copper. The problem is that it is not a true electroless process and there are limitations on the thickness of the coating. Normally, 30 to 50 μin is the practical thickness that can be applied. The solution that deposits the coating acts as a cleaner and renders the copper solderable. However, a tin coating of under 50 μin is normally not sufficient to provide corrosion protection during long-term storage. Also, the tin alloys with the copper, and it can create an unsolderable condition during long-term storage.

There is often a misunderstanding of the electroless process; the process can be employed provided its limitations are thoroughly understood. If the coating could be applied with a thickness of 200 to 300 μin , the process would be an excellent one, but to date the technology has not developed sufficiently to get coatings that thick on a production basis.

A major limitation of the additive process is in preserving solderability. The future of additive circuitry is dependent to a large extent on the development of a process that can deposit a sufficient thickness of tin or tin-lead to provide the corrosion protection necessary for long-term storage. Another approach would be to improve organic protective coatings so they can provide long-term solderability shelf life.

USE OF PRECLEANERS TO RESTORE SOLDERABILITY

In the electronics industry, a restriction is often placed on the activity of the flux that can be employed in soldering, because an assumption has been made that postsoldering cleaning may not always be 100 percent effective. If ionic residues are left on a printed board after cleaning, there is a possibility that voltage leaks could develop under high-humidity conditions. Because of that restriction, a situation in which effective soldering cannot be accomplished with the specified flux can arise. In that event, solderability must be restored or the parts must be scrapped.

16. Causes of Poor Solderability It is important that the cause of poor solderability be understood. In some cases, oil, grease, or organic films may be responsible. A simple solvent or alkaline cleaning can remedy that situation. The most common cause of poor solderability, however, is heavy tarnish or oxidation on the surface of the metal being soldered. Precleaning in an acid cleaner usually will restore solderability. After acid cleaning, it is essential that the acid residues be thoroughly rinsed off. In critical applications, a neutralization step, followed by another rinse, is employed to ensure that all acid residues are removed. A quick, thorough drying is required after rinsing to prevent reoxidation.

It is also possible that the solderability problem may be a combination of the two situations. That would necessitate a solvent or alkaline cleaning to remove organic films and an acid cleaning to remove tarnish and oxidation. If the dual cleaning operation is impractical, then cleaning with an organic solvent containing acid should be considered. With that type of solution, effective cleaning can be performed in one operation followed by rinsing and thorough drying. That type of cleaning, which requires minimum space and equipment, is ideal when organic films on the surface are not extensive but do prevent 100 percent removal of oxides and tarnish.

A solvent containing acid cleaner is ideal for copper and brass, since it dissolves most organic films that could be on the surface and assures complete removal of tarnish and oxidation. Straight acid cleaners such as hydrochloric, sulfuric, and fluoroboric acids and sodium acid sulfate (sodium bisulfate) are completely effective only when organic films

are removed in prior operations. In severe cases of copper oxidation, etching type cleaners such as ammonium persulfate sometimes are used. Although the solutions are very effective, it must be noted that they leave the metal in an active state so that it can easily reoxidize. Hence it is a good practice to follow this procedure with a mild acid dip and thorough rinsing and drying.

17. Cleaning Tin-Lead Surfaces Leads and printed boards are often coated with tin or tin-lead to preserve solderability. When that type of coating is applied by electroplating, it is extremely important that the deposit be applied on a solderable surface. Adequate cleaning prior to electroplating is essential. Tarnishing of the tin or tin-lead coating during etching or storage can detract from solderability. Acid cleaners for removing tarnish from tin or tin-lead usually contain thiourea, fluoboric acid, wetting, and complexing agents. If spray equipment having titanium heating coils or rollers is used, then an acid cleaner containing fluoboric acid cannot be used. However, there are available equivalent cleaners that do not contain fluoboric acid that will clean tin or tin-lead effectively in a spray operation. It should be noted that if the tin or tin-lead is plated over an unsolderable surface, then the only recourse is to strip the tin or tin-lead and replating. Solutions containing glacial acetic acid and hydrogen peroxide often are employed for the purpose. Stripping and replating can be a costly operation, and economic considerations may rule out the procedure in some cases.

STANDARD SOLDERABILITY TESTS

Although many solderability tests are used throughout industry, we have selected two that are agreed upon by the IPC as effective. If a printed board has a special surface treatment, a special test may be required. In general, however, these will give acceptable and repeatable results.

18. IPC Edge Dip Test*

a. Purpose

1. This standard is intended to provide a test method which may be used by both vendor and user to determine solderability of printed boards with or without surface coatings.

2. The solderability determination is made to verify that the printed circuit manufacturing processes and storage have had no adverse effect on the solderability of the printed board. That is determined by evaluating the ability of the portions of the printed board normally soldered to be wetted by a new coat of solder.

3. The standard does not specifically relate to the solderability of internal plating in holes.

4. The standard should not be construed as a production procedure for preparing and soldering printed boards.

b. Definitions

1. *Solderability*. The ability of a metal surface to be wetted by solder.

2. *Wetting*. The formation of a uniform, smooth, unbroken, and adherent film of solder to a base metal.

3. *Dewetting*. A condition which results when the molten solder has coated the surface tested and then receded, leaving irregularly shaped mounds of solder separated by areas covered with a thin solder film. Base metal is not exposed.

4. *Nonwetting*. A condition which results when the molten solder has coated the surface tested and then receded, leaving irregularly shaped mounds of solder separated by areas covered with a thin solder film and by some areas of exposed base metal.

c. *Requirements*. The surface to be tested is immersed in molten solder under defined conditions (see Test Procedure, 3). It is considered solderable if, after removal from the solder, cooling, and cleaning, it meets the requirements as outlined under Evaluation.

d. Equipment

1. *Solder pot*. A thermostatically controlled solder pot of adequate dimensions to accommodate the samples and containing no less than 5 lb of solder.

*Courtesy of IPC, Evanston, Ill.

2. *Dipping device.* A dipping device, as shown in Fig. 14.3, is used. A similar device may be used provided the following requirements are met:

a. Rate of immersion, dwell time, and rate of withdrawal are within the test limits (Test Procedure, 3).

b. Perpendicularity of board and solder are maintained.

c. Wobble, vibration, and other extraneous movements are eliminated.

3. *Optical equipment.* Either a direct or projection lens system with a maximum of 10 diameters magnification is suitable.

e. *Material*

1. *Solder.* The solder should be composition Sn 60 or Sn 63 conforming to the latest revision of Federal Specification QQ-S-571, Solder; Lead Alloy, Tin-Lead Alloy, and Tin Alloy; Flux Cored Ribbon and Wire, and Solid Form.

2. *Flux.* A nonactivated rosin flux having a nominal composition of 25% by weight of water-white gum rosin in a solvent of isopropyl alcohol 99% should be used. The specific gravity of the flux should be 0.843 ± 0.005 at 77°F. The flux must be free of additional activators.

3. *Flux remover.* The flux remover should be either isopropyl alcohol, a chlorinated solvent such as trichlorethylene, or other suitable solvent.

WARNING: Do not use chlorinated solvents on silicone-base board materials, because delamination and finish damage may occur.

4. *Test sample.* The test sample required for this test consists of a metallic surface of the board which is so placed that good drainage of the molten solder is possible as the sample is withdrawn from the solder pot.

The sample can be a planned scrap edge, a finished product, or a specially prepared specimen processed with a group of production boards as mutually determined between vendor and user.

f. *Test Procedure*

1. The printed boards should be checked in an "as received" condition from the vendor, and care should be exercised to prevent contamination by grease, perspirants, etc. of the surfaces to be tested.

2. *Application of flux.* The test samples are to be dipped into flux described under Material, 2, and allowed to drain for 60 s before proceeding with the solder dip.

3. *Solder dip.* The molten solder (per Material, 1) should be stirred with a clean stainless-steel paddle to assure that the solder is of a uniform composition and at a temperature of $450 \pm 10^\circ\text{F}$. Dross and burned flux should be skimmed from the surface of the molten solder immediately prior to dipping. After fluxing and draining, the printed board should be immersed into the molten solder edgewise. The insertion and withdrawal rates should be 1 ± 0.25 in/s, and dwell time should be 4 ± 0.5 s. Upon withdrawal, the solder should be allowed to solidify by air cooling while the board is in the vertical position. No quenching or other means of accelerating solderification should be used. A mechanical device meeting the requirements of Equipment, 2, is recommended for dipping the specimens; however, by determination between the user and vendor, a manual operation simulating the above conditions may be used.

The solder bath should be analyzed periodically for conformance to Material, 1.

4. *Cleaning.* After the solder has solidified, residual flux should be removed from the specimens to facilitate inspection. Flux-removing material should be as specified under Material, 3.

g. *Evaluation*

1. After the parts have been solder-dipped and thoroughly cleaned of flux, they should be examined by using equipment of the type specified under Equipment, 3.

2. *Evaluation criteria.* The criteria for acceptable solderability is a new uniform adhering coating of solder which covers a minimum of 95 percent of the surface being tested. The balance of the surface may contain only small pinholes, dewetted areas, and rough spots, provided such defects are not concentrated in one area. For less critical applications, a smaller percent of coverage may be determined by vendor and user.

3. An area of $\frac{1}{8}$ in width approximately from the edge of each test specimen should not be evaluated.

4. *Aids to evaluation.* As an aid to evaluation of the test results, see Fig. 14.8. The aid is used primarily to illustrate types of defect rather than percent of area covered.

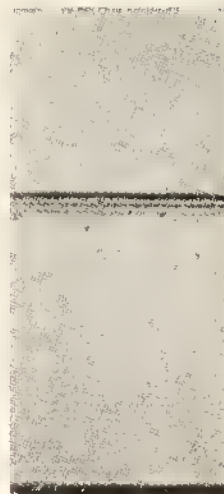
19. IPC Test for Wave-Soldered Printed Wiring Boards*

a. *Purpose*

1. This standard is intended to provide a test method which may be used by both vendor and user to determine solderability of completely processed printed boards, with or without surface coatings, which will be soldered by wave soldering or other machine methods.

*Courtesy of IPC, Evanston, Ill.

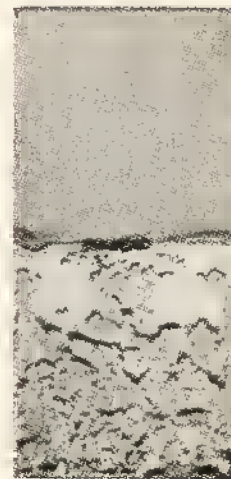
SOLDERABILITY TEST SAMPLES



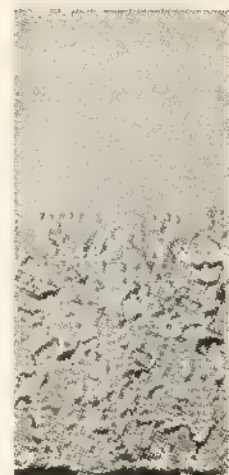
PREFERRED WETTING



SMALL AMOUNT OF DEWETTING



COMPLETE DEWETTING



NONWETTING

Fig. 14.8 Aid to evaluation of circuit wetting, IPC-S-803, June, 1975.

2. The solderability determination is made to verify that the printed wiring fabrication processes and storage have had no adverse effect on the solderability of the printed board. That is determined by evaluating the ability of the portions of the printed board normally soldered to be wetted by a new coat of solder. Determination is judged visibly by nondestructive methods.

3. The standard does not relate to the solderability of the walls of plated-through holes.

4. The standard should not be construed as a production procedure for preparing and soldering printed boards.

b. Definitions

1. *Solderability*. The ability of a metal surface to be wetted by solder.

2. *Wetting*. The formation of a relatively uniform, smooth, unbroken, and adherent film of solder to a base metal.

3. *Dewetting*. A condition which results when the molten solder has coated the surface tested and then receded, leaving irregularly shaped mounds of solder separated by areas covered with a thin solder film. Base metal is not exposed.

4. *Nonwetting*. A condition which results when the molten solder has coated the surface tested and then receded, leaving irregularly shaped mounds of solder separated by areas covered with a thin solder film and by some areas of exposed base metal.

c. *Requirements*. The surface to be tested should be wave-soldered under defined conditions (see Test Procedure, 3). It should be considered solderable if, after removal from the solder, cooling, and cleaning, it meets the requirements outlined under Evaluation.

d. Equipment

1. *Soldering machine*. Machine, such as a wave-soldering unit or system. Machine description may be the machine manufacturer's specification sheet with or without noted exceptions as agreed to between vendor and user.

2. *Optical equipment*. Inspection is generally by the unaided eye (corrected-vision glasses permitted) but on occasion either a direct or projection lens system with a maximum of 10 diameters magnification may be used.

e. Material

1. *Solder*. The solder should be composition Sn 60 or Sn 63 conforming to the latest revision of Federal Specification QQ-S-571, Solder; Tin-Lead Alloy, Solid Form. Other alloys may be used upon agreement between supplier and user.

2. *Flux*. A nonactivated rosin flux having a nominal composition of 25 percent by weight of water-white gum rosin in a solvent of isopropyl alcohol (99% should be used). The specific gravity of the flux should be 0.843 ± 0.005 at 77°F. Other fluxes as agreed to between supplier and user may be used.

3. *Flux remover*. The flux remover should be either isopropyl alcohol or other suitable solvent.

WARNING: Do not use chlorinated solvents on silicone-base materials, because delamination and damage to finish may occur.

4. *Test sample*. The sample required for this test consists of a metallic surface of the board which is so placed that good drainage of the molten solder is possible as the sample passes over the solder wave.

The sample can be planned scrap edge, a finished product, or a specially prepared specimen processed with a group of production boards, as mutually determined between vendor and user. Generally, components will not be mounted on the sample, although the sample may contain drilled and/or plated-through holes.

Specimens should be fixtured, as much as is practical, so as to be representative of the production setup.

f. Test Procedure

1. The printed boards should be checked in an "as received" condition from the vendor, and care should be executed to prevent contamination by grease, perspirants, etc. of the surface to be tested.

2. *Application of the flux*. The test samples are to be flux-coated (Material, 2) and preheated before proceeding with the solder application in accordance with the process

agreed to by vendor and user. Although flux may be applied manually, it will generally be applied as part of the machine system.

3. Application of solder

a. *Wave solder, machine solder*. Process agreed to between vendor and user. Usually it is the manufacturing process generated by the user for wave soldering, and it consists of a board holder, conveyor, flux applicator, preheater, solder unit, and operating instructions which include machine settings, machine process controls on conveyor speed, conveyor incline, preheat temperature, solder temperature [not less than 470°F], fixture description, etc.

b. The solder composition should be analyzed periodically for conformance to Material, 1.

4. *Cleaning*. After the solder has solidified, residual flux should be removed from the specimens to facilitate inspection. Flux-removing material should be as specified under Material, 3. Flux may be removed manually or by machine.

5. Sample size should be as determined between vendor and user.

6. Sample description should include finish (i.e., plated, refused, etc.), board thickness, and dimensional changes allowed (i.e., loss of spacing, tolerance, warpage, etc.).

g. Evaluation

1. After the sample parts have been wave-soldered and thoroughly cleaned of flux, they should be examined by using the type of equipment specified under Equipment, 2.

2. *Evaluation criteria*. The criterion for acceptable solderability is a relatively uniform adhering coating of solder which covers a minimum of 95 percent of the surface being tested. The balance of the surface may contain only small pinholes, dewetted areas, and rough spots, provided such defects are not concentrated in one area. For less critical applications, a smaller percent of coverage may be determined by vendor and user. Solder buildup or drainage on circuit areas where circuit leaves the solder wave is acceptable. A plated-through-hole wall portion should be partly wetted with solder, but it need not be filled with solder.

3. *Aids to evaluation*. As an aid to evaluation of the test results, see Fig. 14.8. The aid is used primarily to illustrate types of defect rather than percent of area covered.

Chapter 15

Solder Material and Process

HUGH COLE

Cobar Resources, Dallas, Texas

Introduction	15-2
The Solder Alloys	15-2
Purity of Solder Alloys	15-3
1. Metallic Impurities	15-3
Solder Flux	15-6
2. Basic Categories of Flux	15-7
3. Rosin Flux	15-7
4. Water-Soluble Fluxes	15-8
5. Application of Fluxes	15-8
6. Fluxer Maintenance	15-9
Preheaters	15-9
7. Types of Preheaters	15-10
Solder Pot	15-10
Dross Formation and Control	15-11
Solder Wave Dynamics	15-12
8. Types of Solder Waves	15-13
9. Surface Tension of Molten Solder after Bidirectional Waves	15-14
Soldering with Oil	15-16
Tin Drift	15-16
Conveyor	15-17
Automated Soldering Systems	15-17
Hand Soldering	15-18
10. Soldering Irons	15-18
11. Tip Selection	15-19
Solder Preforms and Solder Creams	15-19
12. Stamped Preforms	15-20
13. Solder Spheres	15-21
14. Handling of Solder Preforms	15-21
15. Solder Creams	15-22
16. Heating Techniques	15-24

INTRODUCTION

The primary purpose of this chapter is to discuss the materials and machinery used in the soldering process. The emphasis is placed on automated soldering techniques, but most of the information is germane to other techniques. Because of the sequential nature of the soldering process and the need for a high degree of control of the processing parameters, soldering lends itself well to automated techniques. The large throughput required for most modern high-volume production installations makes automated soldering techniques mandatory.

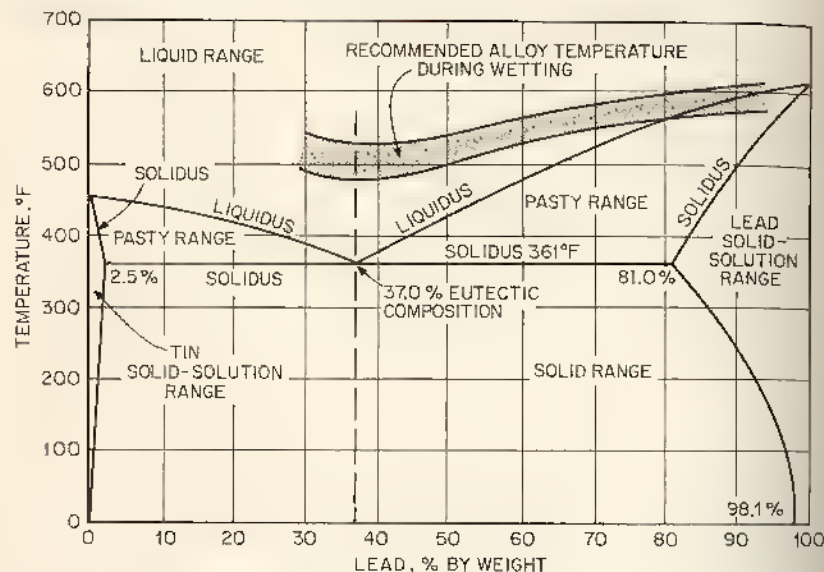


Fig. 15.1 Binary-phase diagram, tin-lead alloys. Note that the eutectic point occurs at 63 percent tin and 37 percent lead.

THE SOLDER ALLOYS

Solder alloys are normally composed of tin and lead in ratios close to the eutectic point. The eutectic composition is a distinct alloy composition that has a single melting point, rather than a discrete melting range. The melting point of the eutectic is always less than that of parent metals. In the tin-lead phase diagram shown in Fig. 15.1, the eutectic point is marked as 63 percent tin and 37 percent lead. As seen in the figure, compositions different than the eutectic do not solidify immediately on cooling; instead, they have a temperature range in which they are partly liquid and partly solid. That region, called the pasty range, becomes more pronounced the farther one departs from the eutectic. Large pasty ranges result in internal stresses and cold solder joints. Therefore, compositions at or near the eutectic are the most desirable for automated soldering.

Alloys other than tin-lead are available with melting temperatures covering the range from 200 to 600°F, but they are not often used in automated soldering processes. Table 15.1, from Federal Specification QQ-S-571, gives a rough idea of the number and types of alloys available. Solder alloy selection is made on the basis of soldering temperature (not melting temperature), mechanical strength, electrical characteristics, mechanical properties, and economy. Because tin is much higher in cost than lead, solder alloy cost is basically a function of the amount of tin in the alloy. The higher the tin content, the higher the cost. The highest tin content used is the eutectic composition, 63 percent Sn and 37 percent Pb. It provides the optimum soldering properties.

PURITY OF SOLDER ALLOYS

The purity of solder, as it is received from the manufacturer, is subject to much confusion, which volumes of advertising literature, claims, and counterclaims of solder manufacturers have done little to dispel. The confusion exists basically because of a tangle of antiquated solder specifications and a lack of appreciation for the complexity of solder manufacture. Federal Specification QQ-S-571 and ASTM B-32, the most often quoted specifications, were written during World War II. Since tin supplies were greatly restricted and the refining and recycling techniques we have today were unavailable, solder impurity levels greater than those currently acceptable were tolerated for expe-



Fig. 15.2 Solder specimens after outgassing test. Pockmarked appearance is due to outgassing of nonmetallic impurities (*Alpha Metals*).

diency's sake. Also at that time, modern high-speed production techniques were not available, and hand-soldered assemblies were most common. Contoured solder joints were rejected because they "didn't have enough solder on them to make a good joint," and the correlation between reliability and inspectibility wasn't fully recognized. In that environment, the type of difficulties caused by solder impurities simply were not recognized as problems.

Two basic types of impurities are of concern in tin-lead solder alloys; they are the metallic impurities, such as copper, gold, aluminum, and cadmium, and the nonmetallic impurities, such as the sulfides and oxides formed by the reaction of the solder alloy (tin-lead and the metallic impurities) with sulfur and oxygen. Impurities of both types are deleterious to the properties of the solder joint. Figure 15.2 shows two solder alloys. One was made in such a way as to minimize the metallic impurities; the other was further processed to remove the nonmetallics. Both specimens were then melted under vacuum. The gas pockets and blowholes in the specimen on the left result from the outgassing of the occluded nonmetallic impurities. Because the nonmetallic impurities affect the flow characteristics of the molten solder, in addition to the metallurgical properties of the joint, more joints can be made faster, and with less solder, if the nonmetallics are removed. Care should be taken, however, during the operation of the soldering process to minimize dross formation. Otherwise, the resultant buildup of impurities will negate the effects of initial impurity removal.

1. Metallic Impurities Metallic impurities also affect the performance of solder. Common contaminants and their effects are discussed in the following subsections.

TABLE 15.1 Solder-alloy Compositions

Composition	Tin, %	Lead, %	Antimony, %	Bismuth, max. %	Silver, %	Copper, max. %	Iron, max. %	Zinc, max. %	Aluminum, max. %	Arsenic, max. %	Cadmium, max. %	Total of all others, max. %		Approximate melting range, °C*
												Solidus	Liquidus	
Sn96	Remainder	0.10, max	—	—	3.6 to 4.4	0.20	—	0.005	—	0.05	0.005	—	—	221 221
Sn70	69.5 to 71.5	Remainder	0.20 to 0.50	0.25	—	0.08	0.02	0.005	0.005	0.03	—	0.08	—	183 183
Sn63	62.5 to 63.5	Remainder	0.20 to 0.50	0.25	—	0.08	0.02	0.005	0.005	0.03	—	0.08	—	183 183
Sn62	61.5 to 62.5	Remainder	0.20 to 0.50	0.25	1.75 to 2.25	0.08	0.02	0.005	0.005	0.03	—	0.08	—	179 179
Sn60	59.5 to 61.5	Remainder	0.20 to 0.50	0.25	—	0.08	0.02	0.005	0.005	0.03	—	0.08	—	183 191
Sn20	49.5 to 51.5	Remainder	0.20 to 0.50	0.25	—	0.08	0.02	0.005	0.005	0.025	—	0.08	—	183 216
Sn40	39.5 to 41.5	Remainder	0.20 to 0.50	0.25	—	0.08	0.02	0.005	0.005	0.02	—	0.08	—	183 238
Sn35	34.5 to 36.5	Remainder	1.6 to 2.0	0.25	—	0.08	0.02	0.005	0.005	0.02	—	0.08	—	185 243
Sn30	29.5 to 31.5	Remainder	1.4 to 1.8	0.25	—	0.08	0.02	0.005	0.005	0.02	—	0.08	—	185 250
Sn20	19.5 to 21.5	Remainder	0.60 to 1.2	0.25	—	0.08	0.02	0.005	0.005	0.02	—	0.08	—	184 250
Sn10	9.0 to 11.0	Remainder	0.20, max	0.03	1.7 to 2.1	0.08	—	0.005	0.005	0.02	—	0.10	—	268 280
Sn5	4.5 to 5.5	Remainder	0.50, max	0.25	—	0.08	0.02	0.005	0.005	0.02	—	0.08	—	308 312
Sn5	94.0 min	0.20, max	4.0 to 6.0	—	—	0.08	0.08	0.03	0.03	0.05	0.03	0.03	—	235 240
Pb80	Remainder	78.5 to 80.5	0.20 to 0.50	0.25	—	0.08	0.02	0.005	0.005	0.02	—	0.08	—	183 217
Pb70	Remainder	68.5 to 70.5	0.20 to 0.50	0.25	—	0.08	0.02	0.005	0.005	0.02	—	0.08	—	183 254
Pb65	Remainder	63.5 to 65.5	0.20 to 0.50	0.25	—	0.08	0.02	0.005	0.005	0.02	—	0.08	—	183 246
Ag1.5	0.75 to 1.25	Remainder	0.40, max	0.25	1.3 to 1.7	0.30	0.02	0.005	0.005	0.02	—	0.08	—	309 309
Ag2.5	0.25, max	Remainder	0.40, max	0.25	2.3 to 2.7	0.30	0.02	0.005	0.005	0.02	—	0.03	—	304 304
Ag5.5	0.25, max	Remainder	0.40, max	0.25	5.0 to 6.0	0.30	0.02	0.005	0.005	0.02	—	0.03	—	304 380

* For information only.

a. **Copper.** Copper forms two intermetallic compounds with tin, Cu_3Sn and Cu_6Sn_5 . The compounds weaken the solder joint and cause the solder to become sluggish and gritty. Since copper is a very common material in the electronics industry, care must be taken to minimize contact of the copper with the molten solder.

Because the copper-tin intermetallics freeze at about 10 to 20°F above the melting point of the solder alloy, they can be removed by dropping the temperature to that level and scooping out the resultant copper-tin crystals with a special tool. The procedure will not remove all the copper, and it necessitates the readjustment of the alloy composition with tin-rich solder. It should be used only as a stopgap if the pot cannot be changed immediately.

b. **Gold.** Gold is readily soluble in molten solder. Although gold was once considered a highly solderable surface, it is now recognized that it rapidly embrittles the solder joint and forms a dull grainy fillet. Gold and copper can act synergistically to deteriorate the performance of a solder pot rapidly. In addition, the value of gold makes it well worthwhile to monitor the level of accumulation in the solder pot.

c. **Iron.** Iron impurities also can be detrimental. Iron forms two intermetallic compounds with tin: $FeSn$ and $FeSn_2$. The formation of these compounds is more rapid at high temperatures such as 800°F. Iron surfaces in contact with the solder pot should be lined or heavily oxidized to prevent the dissolution of iron in the solder.

d. **Zinc.** Zinc is one of the most detrimental of solder contaminants. As little as 0.005 percent zinc will cause brittleness, lack of adhesion, and eventual failure of the joint. Other materials considered detrimental to the solder alloy are aluminum, magnesium, and cadmium. Table 15.2 lists the common contaminants and the levels at which they begin to affect the soldering operation.

TABLE 15.2 Solder Contamination

Contaminant	†QQ-S-571-E, %	‡New solder, %	§Contamination limits %
Aluminum	0.005	0.003	0.006
Antimony	0.2-0.5	0.3	—
Arsenic	0.03	0.02	0.03
Bismuth	0.25	0.006	0.25
Cadmium	—	0.001	0.005
*Copper	0.08	0.010	0.25
*Gold	—	0.001	0.08
Iron	0.02	0.001	0.02
Silver	—	0.002	0.01
Zinc	0.005	0.001	0.005
Others	0.08	0.01	0.08

*Copper and gold combined not more than .300

†Limits established by Federal Specification QQ-S-571-E for acceptable contaminant levels for various metals.

‡Levels of contaminants usually found in new solder delivered from the factory and before any use.

§Contamination levels which indicate the solder must be replaced for any use.

Both types of impurities, metallic and nonmetallic, can be found in raw materials from primary sources such as tin mines and reclaimed sources such as solder scrap. With the chemical refining techniques available today, there is virtually no difference in purity between primary or virgin grade material and properly refined reclaimed material. Therefore, both constitute acceptable raw material sources for solder alloys. No matter which source is used, however, the solder alloy should be processed to remove both metallic and nonmetallic impurities.

SOLDER FLUX

The first step in most automated soldering operations is the fluxer or flux pot; but before the mechanics of the fluxing operation are described, it is essential to understand the requirements of the solder flux and the way in which various types of fluxes fill those requirements.

To form a reliable solder joint, both the board surface and the component lead must be

free of oxidation and must remain so even at the elevated temperatures used in soldering. Also, the molten solder alloy must wet the surfaces of the metals to be joined. Without the solder flux, neither condition can be met reliably or economically. The requirements for a solder flux can be summarized as follows: the flux must react with and remove metal oxides from the surfaces to be joined and prevent the reoxidation of the cleaned surface even at elevated temperatures; it must promote wetting of surfaces to be joined by the molten solder alloy; it must act as a heat-transfer medium to ensure that the parts to be joined reach a temperature high enough to form a metallurgical bond; and finally, it must be either noncorrosive or easily removable.

2. **Basic Categories of Flux** Solder fluxes can be divided into two basic categories: rosin-based and water-soluble. Rosin-based flux has three components: the solvent or vehicle, the rosin, and the activator. (For very mild fluxes, the activator is sometimes omitted.) The function of the solvent is simply to act as a vehicle for the flux. To be effective, the rosin and activator must be applied to the board in the liquid state. That is accomplished by dissolving them in the flux solvent. Solvent vehicles can be simple one-component systems such as isopropyl alcohol, or they can be complex, multicomponent systems such as those used in foaming fluxes. When the flux vehicle is a multicomponent system, maintaining it properly becomes difficult. Because different components evaporate from the flux at different rates, they must be replaced with flux thinners of carefully controlled composition. Most manufacturers supply thinners blended to maintain a constant composition of the flux vehicle. Maintenance of the flux and thinner ratio will be discussed in another section of this chapter.

3. **Rosin Flux** Rosin is a naturally occurring substance extracted from pine trees. It is a mixture of many organic acids, and its purity and composition are difficult to control. The principal constituents, however, are abietic acid and pimaric acid, the structures of which are shown in Fig. 15.3. After rosin is extracted from the pine trees, it is given a superficial processing which removes undesirable impurities and neutralizes the acid residues remaining from the extraction operation. The purified material, called water-white rosin, is then used in the manufacture of rosin-based flux. Some manufacturers, in the hope of overcoming the difficulties associated with obtaining and processing natural rosin, have chemically synthesized substitute materials. Such materials are called resins.

Rosin by itself is a very weak flux; its ability to remove surface oxides is so limited that only exceptionally clean surfaces of high solderability can be joined with it alone. When surfaces require a more active flux, chemical compounds called activators are added to the rosin. Activators are thermally reactive compounds, such as the amine hydrochlorides, which break down at elevated temperatures and release hydrochloric acid to dissolve the surface oxides and tarnishes. The precise nature of the activators used in a given manufacturer's flux is normally a carefully guarded trade secret, but review of the literature on flux activators usually provides more than sufficient familiarity with the compounds available.

a. **Design of a Flux.** The design of a rosin-based flux is a careful balance: of the ratio of rosin to activator. If there is too much activator, the flux will be too corrosive and difficult to remove; if there is too little, the fluxing action will be impaired. The amount of the rosin and activator mixture in the solvent vehicle, called the solids content or density, is tailored for the specific application. The higher the solids content of the flux, the more rosin-activator mixture is deposited on the board and the more difficult the board will be to clean. To obtain adequate coverage, high-solids-content fluxes are used for boards with dense circuitry and small plated-through holes. Single-sided boards with less-dense circuitry can use fluxes with lower solids content. That is an important point: the rosin-to-activator ratio controls the activity of the flux, not the solids content. Solids content affects only the coverage. Thus a high-solids-content flux with a low ratio of activator to rosin may

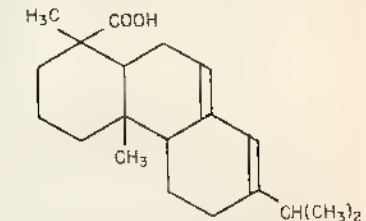


Fig. 15.3 Typical organic acid (abietic acid) found in natural rosin, which is a mixture of many such acids in different concentrations.

he the wrong choice for a surface with poor solderability. A more appropriate selection may be a lower-solids-content flux with a much higher activator-to-solids ratio.

b. Activity Levels. The activity levels of rosin-based fluxes have been divided into three categories: 10sin (R), 10sin mildly activated (RMA), and 10sin-activated (RA). R-type fluxes contain no activators. RMA fluxes contain small quantities of activators that for all practical purposes are completely inert and noncorrosive after soldering. For that reason, most government contracts and some commercial supply contracts require the use of RMA fluxes. RA fluxes contain the greatest quantities of activator and hence are the most active.

4. Water-Soluble Fluxes Water-soluble fluxes also are used to solder printed circuit assemblies. Water-soluble fluxes are much more active than rosin-based fluxes, and they will therefore rapidly and efficiently solder material of even very poor solderability. However, because they are so active, they are extremely corrosive and must be thoroughly and rapidly removed. Water-soluble fluxes are mixtures of compounds like glutamic acid hydrochloride and aniline hydrochloride in a solvent vehicle. In some cases, activators quite similar to those in rosin-based fluxes are used, but in much higher concentrations.

A common misconception concerning water-soluble fluxes is that the solvent vehicle itself is water. Although water can be used as the vehicle, it results in a flux which spatters and is quite difficult to use. More commonly, the solvent vehicle is an alcohol system similar to the systems used for rosin-based fluxes. Small quantities of water are sometimes added to multiple-component systems to aid in solubilizing the activator. The term "water-soluble flux" refers to the fact that the residues are generally water-soluble. Although that is true for very long rinse times and elevated rinse temperatures, it is better to neutralize the residues with ammonia water or a similar neutralizing agent. Neutralizing the residues greatly increases their solubility in water. That will be discussed in detail in a later chapter.

5. Application of Fluxes Solder fluxes must be applied to the underside of the board in the most effective and economical manner possible. The resultant flux coat must be uniform and must thoroughly cover the areas to be joined. An excessive amount of flux not only presents a fire hazard if it drips on the hot preheater but also is wasteful and ineffective and greatly increases the work required to clean the printed circuit assembly. No matter how the flux is applied, an air knife or adjustable squeegee should follow the flux pot to remove excess solder flux.

For all practical purposes, most automated solder machines in commercial use apply the flux by using one of these techniques: wave, foam, brush, spray, or dip. Of those techniques, wave and foam fluxing are the most popular, and they will therefore be discussed in more detail.

a. Foam Fluxer. A foam fluxer consists of a sump and a long high nozzle which contains a porous stone. A typical arrangement is shown in Fig. 15.4. The nozzle, called a chimney, shapes and constrains the foam head. The pump of the fluxer is filled with flux until the porous stone is about two inches below the surface. Air is pumped into the stone, and the foam head rises out of the chimney to meet the underside of the assembly. To prevent contamination of the foam flux with oils and water from the air lines, the inlet lines must be fitted with filters and traps. The pressure of the air line should be regulated to control the pressure and volume of air being passed through the stone. A nominal pressure of 3 to 5 lb is normally ample. The air line should also have a needle valve to adjust the air volume precisely and thereby control the quality of the foam head. The use of a foam fluxer allows a thin uniform layer of flux to be placed on the board surface. The foam fluxer is particularly effective when the assembly to be soldered has a lot of plated-through holes. Apparently the action of the bubbles bursting within the through hole is one of the most effective ways to distribute flux on such structures.

b. Wave Fluxer. A wave fluxer consists of a sump and a nozzle similar to the one in the foam fluxer. In a wave fluxer, however, the flux is pumped through the nozzle to form a standing wave of flux. The parts to be fluxed are passed over the top of the wave. This method provides less control over the amount of flux deposited on a printed board, but it is quite adequate for high-speed board production.

c. Spray Fluxing. Spray fluxing is another technique used for depositing flux on a board surface. It has the advantage of very precise control over the quantity, uniformity, and location of solder flux, but it is a very messy technique and is subject to frequent

maintenance. Fluxes used in this operation must have a very volatile solvent system. That makes them expensive and very difficult to control. Unless the precise control over flux deposition parameters afforded by the method is absolutely necessary, the foam fluxer may offer a more attractive alternative.

d. Brush and Dip Fluxing. Brush- and dip-fluxing techniques are self-explanatory. They are fairly gross techniques and are not readily adaptable to high-speed or high-volume work.

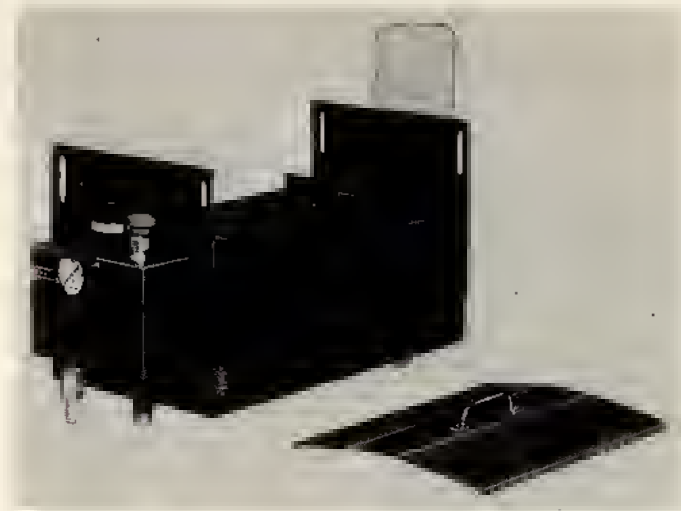


Fig. 15.4 Typical foam flux applicator (Electrovert, Inc.).

6. Fluxer Maintenance The key to proper flux application and performance is regular maintenance. As mentioned previously, fluxes that incorporate volatile vehicles such as alcohol must be periodically thinned to make up for solvent evaporation. Thinning must be done several times during a normal production shift. The most convenient way of doing it is to monitor the specific gravity. The procedure for replenishing the flux pot is as follows:

1. Bring the flux in the pot back to the proper specific gravity by diluting it with the appropriate thinner. Specific gravity information is normally found on the manufacturer's data sheet.
2. Add new flux to the system to bring it back up to the proper level.

It is also important to completely empty the flux pot at periodic intervals and fill it with a fresh change of flux. That minimizes the effect of drag in contaminants and is fairly inexpensive preventive maintenance. Depending on production conditions, it should be done once every 30 to 40 h of operation. All parts used in the construction of a fluxer should be acid-resistant material. Stainless-steel or polypropylene is often used for the purpose.

PREHEATERS

Before the solder flux can become effective, chemical reactions must take place within the flux to release the acidic activators. These chemical groups then react with the metal oxides to remove those oxides from the surface to be soldered. To initiate the reaction, the flux must be heated to what is called the activation temperature; for most rosin fluxes, it is about 190°F. If only the solder wave is relied upon to heat the flux to that temperature, the time in the wave is extended by the additional time required for the flux to clean the metal surfaces. A more palatable approach is to heat the flux to the activation temperature prior to the time it enters the wave.

Preheating of the assembly offers other benefits. In most fluxes a large portion of

volatile material will still be retained in the rosin when the assembly reaches the solder wave. Some organic acid fluxes contain water, which also is a relatively volatile component. If the boards are soldered in that state, the heat of the solder pot will rapidly volatilize the solvent and cause spattering and blowholes in the solder joint. By preheating the board, the excess volatiles are driven off, and that eliminates potential problems in the solder wave. If the flux contains water, the preheat temperature must be higher than it would have to be if it contained a more volatile solvent such as alcohol.

Preheating minimizes the thermal shock of the soldering operation by providing a more gradual increase in the temperature of the assembly. It thereby minimizes warpage of the board and moderates thermal stresses which could affect the mechanical integrity of the assembly. Preheating also minimizes the danger of damage to heat-sensitive components.

7. Types of Preheaters Two basic types of preheaters are in widespread commercial use; they are the radiant preheaters and the volumetric preheaters. Radiant preheaters can be



Fig. 15.5 Calrod radiant preheater (Hollis Engineering).

further divided into two subcategories: long electrically heated rods such as the Calrod type shown in Fig. 15.5 and flat-plate sources such as the one shown in Fig. 15.6.

a. Radiant Preheaters. Radiant preheaters transfer heat almost entirely by radiation; the temperature on the underside of the board is controlled by the heater temperature and the distance between the heater and the underside of the board. To increase their efficiency, heated-rod-type systems are provided with reflectors which focus the heat energy on the underside of the board. The reflectors are lined with aluminum foil, which can be changed regularly. Flux drippings, therefore, do not present a major maintenance problem. Plate-type heaters, on the other hand, are situated under the conveyor in close proximity to the board surface. That greatly enhances the efficiency of the heat transfer between

the plate and the board. Flux which drips onto plate-type preheaters must be regularly removed to eliminate the danger of fire and maintain the efficiency of the preheater.

b. Volumetric Preheaters. Volumetric preheaters pass a controlled volume of heated air across the underside of the board. Heat transfer is primarily by conduction and convection. The forced air currents aid in the rapid elimination of solvent vapors from the underside of the board. Volumetric preheaters are preferred when plated-through-hole boards are used, because the inside of the hole is often shielded from straight-line contact with radiant sources. That is particularly important in the case of multilayer boards. Many commercial applications use both techniques—a volumetric preheater to eliminate excess solvent followed by a flat-plate radiant heater to maintain the proper temperature.

SOLDER POT

The solder alloy is contained in a large reservoir called the solder pot, and the solder pot is heated electrically to maintain the solder in a molten state. The pot should be well insulated and should have sufficient heating capacity to maintain the alloy temperature adequately. Since soldering is a time- and temperature-dependent phenomenon, accurate regulation of the heat input is essential. To control the temperature effectively, many solder pots have a powerful bulk heater to melt the metal rapidly and a smaller regulated heat supply to control the temperature thermostatically. The smaller heater also prevents thermal loads, such as the printed board, from affecting the pot temperature seriously. Of course, another approach is to maintain a very large volume of solder at the appropriate temperature.

Heater placement in the solder pot also is important. Heaters must be so placed that the solder heats uniformly and there are no hot spots. The temptation to heat only the bottom of the pot and not the sides is great, but it should be resisted. Careful consideration of the initial melting pattern of the solder is necessary. If the solder at the bottom of the pot melts and the surface remains intact, large stresses are built up as the molten solder expands. If the stresses are not relieved by melting the metal in the surface, they can build to such a level that an area on the surface will rupture and molten metal will be propelled in a fountain several feet high. That is a very dangerous condition. To avoid it, the sides of the pot also should be heated. That applies equally to small dip pots and the larger pots used for automated soldering of assemblies, such as the one shown in Fig. 15.7.

When the solder pot is used in conjunction with a pump and a nozzle to make a solder wave, there are several other commonsense considerations. The pump should be pro-



Fig. 15.6 Flat-plate radiant preheater with forced-air convection unit (Electrovert, Inc.).

vided with a safeguard such that it can't be operated until the metal is molten. The volume of the pot should be great enough that, when the wave is operational, the solder level will not be affected substantially. The pot should be easy to drain and refill with a new solder charge, and all parts should be easily accessible for regular maintenance.

All parts of the solder pot which come in contact with the molten metal must be made of metals which are not readily wetted by the solder and which will not dissolve in the solder and contaminate it. The most common alloy for that purpose is stainless steel. All tools used to maintain the solder pot, such as cross skimmers, sampling scoops, and perforated paddles, should also be made of stainless steel. Cleaning the inside of the solder pot with a wire brush prior to adding a new charge of solder should be avoided. The wire-brushing operation imbeds contaminants in the wall of the solder pot and creates an iron-rich dust which is difficult to remove. Both the dust and the iron-rich particles imbedded in the wall will rapidly contaminate the new solder charge.

Wire brushing can cause damage to a cast-iron solder pot also. Cast-iron pots are normally heavily oxidized to prevent the iron from being dissolved by the tin in the molten solder. Wire brushing or similar abrasive cleaning and scraping can damage the integrity of the oxide layer and expose a potential source of iron contamination to the molten solder.

DROSS FORMATION AND CONTROL

"Dross" is the name given to the layer of metallic impurities on the surface of the solder pot. It is composed of the oxides of tin and lead formed either on initial melting or on contact of the molten metal with air. Dross formation is principally a function of solder pot

design, i.e., exposed surface area on the pot, turbulence of the wave, etc. There is, however, a tendency for dross to be self-propagating. The more dross in the solder pot, the more likely it is that dross will continue to form. Therefore, dross should be removed on a regular basis (at least once per shift), and new solder should be added to make up for the removed dross.

Dross is harmful to the soldering process and the solder wave. If the dross layer becomes thick enough to be pulled into the solder pump, it will rapidly abrade the



Fig. 15.7 Wave-Solitar pot with nozzle and automatic solder level control (Electrocraft, Inc.).

impeller and increase maintenance costs. Dross in the wave results in a bumpy uneven wave with excessive turbulence. If the dross particles are small, they may become occluded in the solder and produce a dull grainy joint. Larger particles of dross may adhere to the bottom of the board and contribute to webbing and bridging.

Dross formation can be minimized by using a blanket, such as rosin or oil, to reduce the amount of exposed solder surface area. When solder blankets are used, however, it is important to change them on a regular basis. Rosin-based blankets should be changed after 4 h of use; soldering oils can be used for 8 to 16 h before they need to be changed.

SOLDER WAVE DYNAMICS

One of the most critical but least understood variables in the automated soldering machine is the design of the solder wave. Historically, the solder wave was considered little more than a fountain of solder which supplied the molten alloy to the joint area. That was, of course, the logical extension of the drag-and-dip soldering operation. As the industry matured, it became apparent that the dynamics of the solder wave contributed greatly to the formation of smooth shiny joints that were free of webbing and iciling. To understand the importance of wave dynamics better, it is necessary to examine the constraints imposed by the use of automated soldering. Because of the high temperatures involved, the printed board should not be exposed to solder bath too long. However, to

ensure formation of a good metallurgical bond, the board must remain in contact with the solder long enough to bring the joint areas up to the proper temperature. Molten solder moving past the underside of the board induces a scrubbing motion in the joint area which greatly aids the fluxing action. To aid in the scrubbing action, the solder wave should have maximum contact time with the board; but to minimize dross formation, the wave must have minimal contact with the air. To minimize iciling, the board should be moving at zero velocity relative to the solder when it leaves the wave. All these apparently conflicting requirements must be satisfied if optimum performance of an automated soldering process is to be attained.

In addition, the solder wave must be even and smooth enough to contact the entire board surface as the board passes over the solder wave. An uneven wave could cause molten solder to pass over the board, rather than under it, and thereby ruin the assembly.

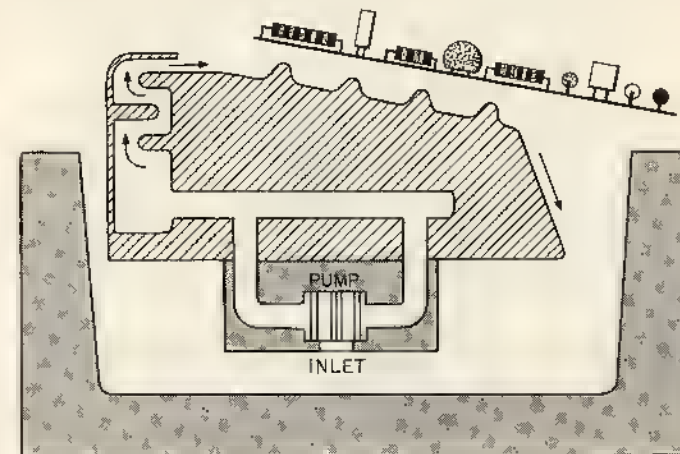


Fig. 15.8 Typical multiple-wave soldering scheme (From RCA Corporation, by permission).

Dross formation must be minimized because dross accelerates wear of the solder-pumping system and causes an uneven wave. Excessive dross formation also promotes tin depletion and consumes inordinately large quantities of solder. From those conditions it's easy to see that the solder wave is much more than a fountain of molten metal.

B. Types of Solder Waves Many solder wave systems, each purporting to optimize the above list of constraints, have been marketed. They can be divided into two classes: unidirectional (all solder flows in one direction) and bidirectional (solder flows in both directions). Some variations of the bidirectional system offer extenders to broaden and flatten the wave on the trailing side. We will discuss one such nozzle in detail later on.

a. Unidirectional. Variations of the unidirectional scheme have included a multiple-wave cascade such as the one shown in Fig. 15.8 and an arching stream of molten solder called a solder jet. Neither technique is seen on newer machines.

b. Bidirectional. The most commonly used system is the bidirectional wave. Because of the velocity profile on the surface of the wave, the bidirectional solder wave can be designed to minimize iciling. Since solder is flowing both forward and backward in the wave, it follows that there will be an area of zero velocity on the surface of the solder. The velocity profile of the flowing solder in the vicinity of the zero-velocity line is critical to the formation of icicle-free joints.

The bidirectional wave is formed by pumping the molten solder into a nozzle with a large plenum chamber. The resulting head of solder rises up through the lip of the nozzle to form the solder wave. The contour of the nozzle controls the shape of the solder wave and hence the wave dynamics (Fig. 15.9). Baffle screens can be placed in the nozzle to ensure laminar flow and a smooth wave. To minimize dross formation, the solder from the wave must be returned to the pot without excessive turbulence. Two schemes are normally used to accomplish the return. In the wave shown in Fig. 15.10 sluice gates are

placed at the outside of the nozzle. The sluice gates return the solder to the pot below the surface level, and so the surface remains undisturbed. If sluice gates are not used, the other alternative is to return the solder to the pot via gradually sloping ramps. That minimizes surface turbulence, but it requires slightly more surface area than the sluice gates.

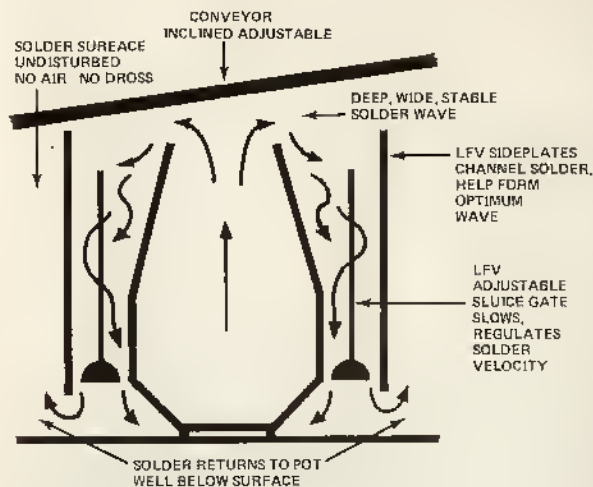


Fig. 15.9 Cross section of a nozzle on an automated solder wave. The large plenum on the nozzle is designed to provide smooth laminar flow through the nozzle (Hollis Engineering).

9. Surface Tension of Molten Solder after Bidirectional Waves To understand how a bidirectional nozzle minimizes iciling, we must first understand the phenomenon of surface tension and its relation to wetting. Surface tension forces make liquids such as molten solder and water form into beads or droplets when they are incapable of wetting a surface. They control how well a surface "likes" a given liquid; hence they control how well

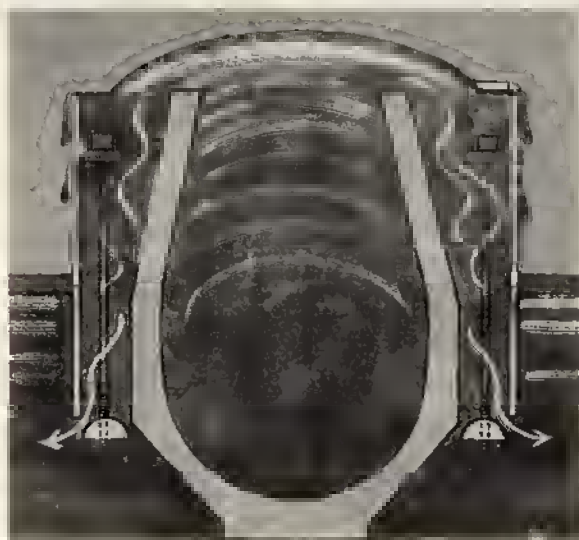


Fig. 15.10 Solder nozzle with sluice gates to minimize dross formation on the pot (Hollis Engineering).

solder wets the fluxed copper. In Fig. 15.11 we see a printed board passing over a bidirectional wave. The solder has wet the board surface and is being pulled from the solder wave in the form of a web. The size of the web is controlled by the surface tension of the solder, the velocity profile at the point where the wave contacts the web, and the weight of the molten solder in the web.

As the web becomes larger, it becomes more difficult for the surface tension forces in the molten wave to pull the excess solder back into the wave. At some critical web size, surface tension forces cause the web to separate; and if the excess solder has not been pulled back into the wave, an icicle is formed. From this very crude model we can see that our task is to minimize web formation by altering either the surface tension of the solder

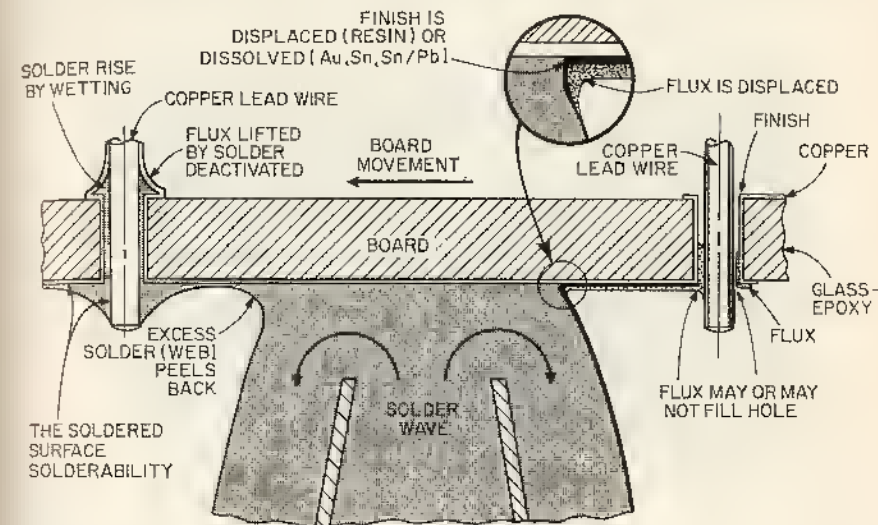


Fig. 15.11 Printed board passing over a typical solder wave. Note the tendency to pull a web of solder from the wave as the board passes over (Alpha Metals, Inc.).

or the velocity profile of the wave at the point where the web occurs. That can be done in several ways. The surface tension of the solder is affected by the temperature of the solder. Higher temperatures will result in a lowering of the surface tension forces. The result, however, may be damage to heat-sensitive components, and the heating will not significantly improve the surface tension of the solder. Surface tension can be reduced by injecting oil into the solder wave. That will be discussed in detail later on.

The size of the web can also be decreased by inclining the conveyor. The friction coefficient of the solder is about 3%, so inclining a conveyor at 4 to 9° will aid in causing the solder to peel back more rapidly. Another technique which has been used is to shape the wave in such a way that it becomes quite wide. The wide wave, when used in conjunction with an inclined conveyor, allows the board to exit from the solder wave at or near the zero relative velocity point. That allows sufficient time for the surface tension forces to draw the web completely back into the solder wave. Figure 15.12 shows a nozzle which was designed by using that approach. Note also in this figure that the board is initially forced to contact the wave at a point of high velocity; thus the scrubbing action of the molten solder also is optimized. Several other features of this type of nozzle design are noteworthy. Since the shape, and hence the velocity profile, of the wave is controlled by positioning the baffles on the front of the nozzle, a region of zero velocity which covers a substantial portion of the front of the nozzle is created. Hence the board can be made to emit the wave at a zero relative velocity point for a wide range of inclination angles. After the board exits from the wave, its close proximity to the hot solder causes a postcating effect which aids in minimizing icicle formation. By using this type of nozzle, icicle-free soldering has been successfully accomplished at conveyor speeds in excess of 20 ft/min.

SOLDERING WITH OIL

Oil has commonly been used in soldering systems to reduce the amount of dross formed on the molten solder. The oil forms an insulating blanket which prevents contact of the molten solder in the pot with the air. Automated soldering systems which inject the oil into the solder wave at the pump impeller also have been perfected. The oil is dispersed uniformly through the wave and greatly reduces the surface tension of the molten solder. It also reacts with the tin and lead oxides which make up solder dross to form tin and lead soaps. That reduces the abrasive action of solder dross on the pump impeller and minimizes dross in the solder wave itself.

From the preceding discussion it should be apparent that the real reason for mixing oil with molten solder is to reduce the surface tension between the printed board and the molten solder at the point where the board departs from the wave. This effectively minimizes the solder web and eliminates icicles. Proponents of oil intermix systems have pointed out several other "processing type" benefits that can be obtained. The solder joint

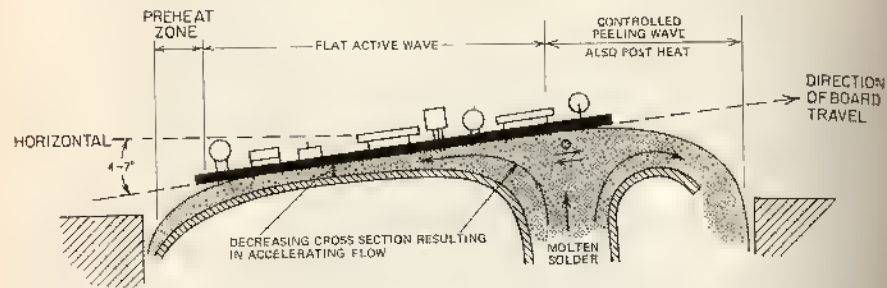


Fig. 15.12 Lambda Wave; the system is designed to ensure exit conditions of zero velocity from the solder wave (Electrovert, Inc.).

is coated with oil when it leaves the wave. The coating prevents oxidation and results in brighter and shinier joints which are easy to inspect. Since the tendency for icicling has been minimized, conveyor speeds can be accelerated and process throughput is increased. Lowered surface tension increases the ability of solder to wet copper pads on the circuit board, and the temperature of the solder wave can therefore be reduced substantially (20°F) without affecting joint formation. When oil is applied in proper amounts, it can assist in solubilizing rosin residues and thereby aid in the cleanup of the finished assembly.

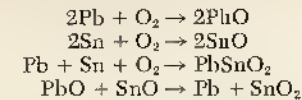
According to a different, and somewhat more conservative, school of thought, soldering with oil results in inclusions of soldering oil in the solder joint. The oil in the inclusions could affect the mechanical and electrical properties of the joint and could be a source of corrosive acidic residues. The critics also point out that soldering with oil results in a messy, oil-coated assembly which is difficult to clean effectively. When soldering is with oil, cleaning is a necessity.

The key to effective use of soldering oil is process maintenance. The soldering oil must be changed on a regular schedule to prevent its degradation and carbonization. Carbonized oil in the solder wave negates most of the advantages that can be obtained by using oil. The soldering machine and the machine area must be kept spotless. Oil spills and the results of similar small handling accidents should be cleaned up before the cumulative effects turn the area into a pigpen. Finished assemblies should be cleaned as soon as they are processed, and the assemblies should be inspected to make sure all oil is removed.

TIN DRIFT

One other subject which is of interest to companies using automated soldering is tin drift: the slow depletion of tin in a solder pot as the pot is used. Tin drift can be very

troublesome when compositions at or around the eutectic must be maintained. It occurs because tin is easier to oxidize than lead and dross formation is therefore biased toward the formation of tin oxide rather than lead oxide. Basically, the reaction is as follows:



When soldering oils are used, the reaction is somewhat different but the results are generally similar. Tin drift can easily be corrected by replenishing the solder pot with solder that is slightly tin-rich. Another, and probably more palatable alternative, is to use tin ingots or tin bits to bring the solder pot back to specification.

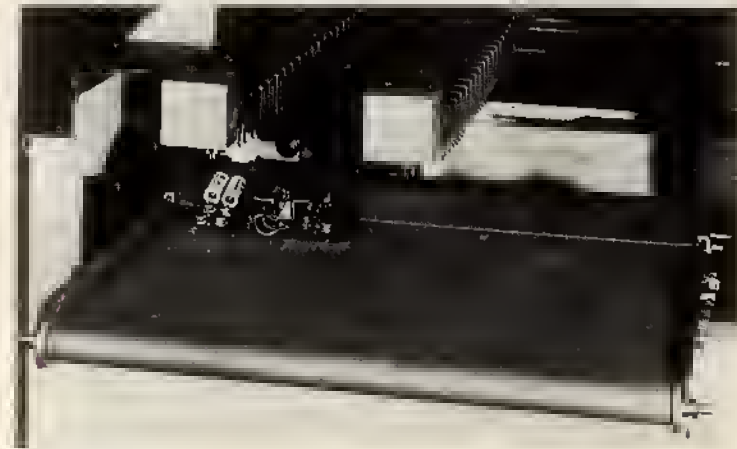


Fig. 15.13 Finger conveyor for holding printed boards in an automated soldering system without the need for pallets. Boards are held in place by the spring tension of conveyor fingers (Electrovert, Inc.).

CONVEYOR

Another parameter which should be considered when an automated soldering system is designed is the conveyor. It controls the speed at which the boards are passed through the process; hence it controls preheat time and temperature and dwell time in the solder wave. Conveyors are available in two basic types: the pallet type in which boards are held in carriers or holders, and the finger type in which boards are held in spring fingers. Finger-type conveyors can be adjusted to accommodate the width of the board; they can automatically discharge boards at the end of the process; and they are easier to load. Figure 15.13 shows a typical finger-type conveyor. Both pallet- and finger-type conveyors are available in either horizontal or inclinable models.

AUTOMATED SOLDERING SYSTEMS

So far we have discussed each segment of the system in great detail; only postsoldering cleaning has been omitted because it is covered in another chapter. Most manufacturers sell complete systems rather than just component parts. Whenever possible, it is advisable to purchase a complete system and assure yourself that it is properly engineered and integrated into your entire process. Most automated soldering systems also contain provisions for adequate ventilation and proper employee protection. In today's legal and ecological environment, purchasing a complete system from a single manufacturer is an extremely effective way to shift a major portion of the burden of complying with federal and state regulations onto the machine manufacturer. Figure 15.14 shows two samples of typical automated systems which are commercially available.

HAND SOLDERING

10. Soldering Irons Although the great majority of solder joints are formed during the automated soldering operation, there are still many instances in which manual soldering techniques are essential. Individual joints may be faulty and therefore require repair and touch-up at a manual-soldering station. Heat-sensitive and solvent-sensitive components may need to be hand-inserted, or defective components may need to be replaced.

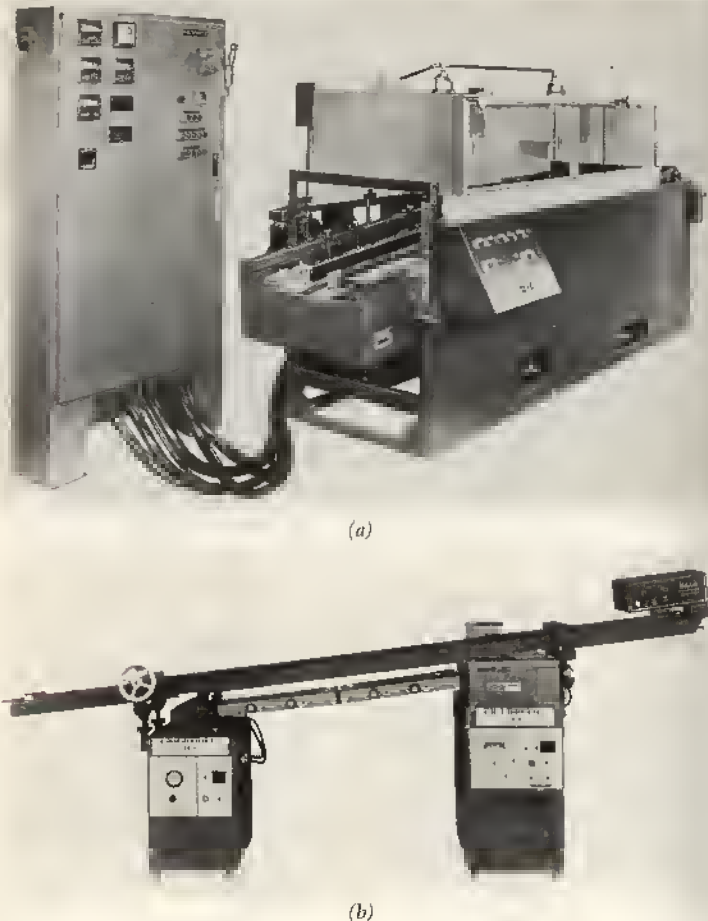


Fig. 15.14 Typical automated solder systems. (Courtesy (a) Electrovert, Inc. and (b) Hollis Engineering.)

Therefore, a knowledge of manual-soldering techniques is essential for the printed circuit processing engineer. Manual soldering is accomplished with a soldering iron and depending on the application, a length of wire or core solder or a solder preform. The function of the soldering iron is to transfer heat to the joint area rapidly and efficiently. Because of the wide variety of joint configurations and joining problems, there are many different sizes and types of soldering iron and tips. To help you evaluate your particular job and requirements, some of the important considerations in selecting a soldering iron are outlined below:

a. Wattage. This refers to the power consumption of the soldering iron. It is a rather coarse measure of the total amount of energy available for heating the soldering iron tip.

All other things being equal, an iron with a higher wattage will have a higher soldering temperature.

b. Maximum Tip Temperature. This is the steady-state or equilibrium temperature of the soldering iron tip. A properly designed iron should maintain a constant tip temperature when operated at its rated voltage.

c. Heat Content of the Iron. Heat content is a measure of the thermal bulk of the soldering iron, the amount of thermal energy that can be stored by the metallic components of the tip. The heat content must be carefully matched to the job. If the iron is too small, the heat drain caused by making a single joint will lower its temperature below that necessary for making the next joint. Valuable production time will then be lost in waiting for the iron to reheat. If the heat content is too high, excessive heating of the components may cause damage or degradation of some of the electrical components.

d. Recovery Rate. The recovery rate is a measure of the recycle time of the soldering iron, it is the time necessary to reach the equilibrium tip temperature after a single joint has been made. Obviously, the recovery time is a function of wattage, heat content, and tip geometry.

e. Selection of Irons. In practice it is not feasible to maintain a diverse inventory of soldering irons and tips to match each production situation, nor is it anything but naive to think that a single iron and tip will cover all requirements. Therefore, compromises must be made in the interest of economy and efficiency. The following general guidelines should be considered when soldering irons are selected for individual applications.

1. The caloric requirements of the work. This parameter depends on several other variables: the solder alloy and flux used, the thermal properties of the base metals, and the geometric configuration of the joint.

2. The production environment in which the tool is to be used. The number of joints to be produced per unit of time, the human factors involved in handling the tool, and the limitations imposed by the assembly to be soldered are important considerations.

11. Tip Selection Careful tip selection is just as important as proper iron selection. As a matter of practice, the tip should be evaluated in conjunction with the soldering iron to be used. Soldering irons with equivalent wattage ratings but different efficiencies will yield different tip temperatures when used in conjunction with the same tip. There, it is essential that the soldering iron and tip be evaluated as a system. The following guidelines will aid you in the selection of your soldering iron tip.

1. Select a configuration that affords maximum contact between the tinned areas on the work and the soldering iron tip. That maximizes the heat transfer between the iron and the work.

2. Select a tip that will facilitate the manual work of gaining access to the solder joint. For example, in some cases the use of a right-angle tip greatly simplifies the manual-handling problems.

3. The taper of the tip should be as short as possible to achieve optimal heat transfer to the tip.

4. Select the shortest reach available to minimize wobble and thermal loss due to convection heat transfer.

5. The diameter of the shank determines the heat transfer between the elements of the iron and the tip. The larger the diameter, the better the coupling between the iron and the tip.

6. The life of bare copper tips is short, and the cost of maintaining a proper tip's shape is relatively high. Iron-clad tips, on the other hand, have much longer lives, but they cannot be filed or shaped.

SOLDER PREFORMS AND SOLDER CREAMS

We have shown that solder joints can be made easily, reliably, and rapidly through the use of automated soldering equipment. However, there are applications and designs which simply are not compatible with wave- or dip-soldering techniques. They may be encountered when the circuit assembly is nonplanar or when the proximity of sensitive components makes the use of a hot soldering wave impossible. If, in addition, we broaden the definition of printed board to include the ceramic substrates used in microcircuit assembly, then the automated techniques previously described simply become inapplicable.

For such applications, techniques based on either solder preforms or solder creams are often the most useful. In this section we will discuss both the nature and application of such materials. However, because of the inherent versatility and flexibility, we can provide only a few insights into the properties of the materials and a small cross section of typical applications. Your imagination and your own possible applications will soon provide you with an estimate of the design flexibility and cost-saving advantages of solder preforms and creams.

12. Stamped Preforms Solder preforms are precision-manufactured solder parts made to the exact shape desired for an application. Typically, they are punched from a strip of solder alloy by using either a standard-shaped die such as a disk or washer or a die which has been custom-manufactured for some specific application. Obviously, the cost of each

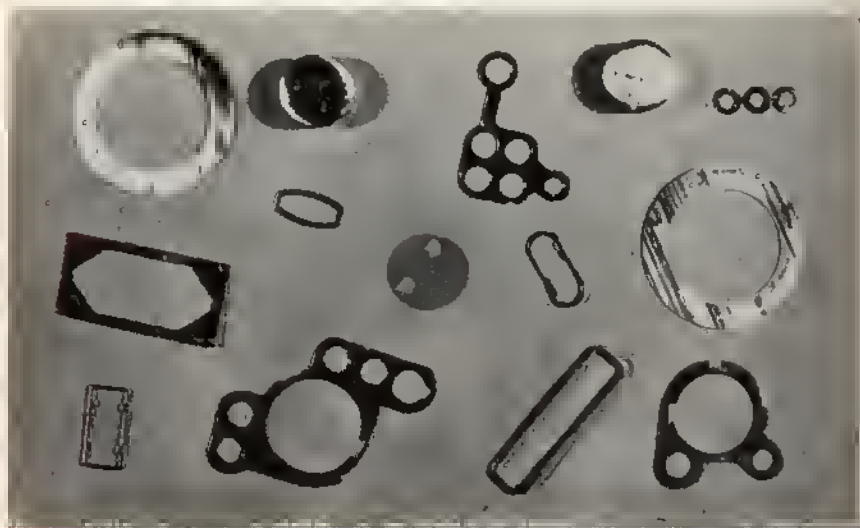


Fig. 15.15 Solder preforms can be stamped in a variety of shapes and sizes. Preforms simplify complex handling problems (Alpha Metals, Inc.).

part is significantly higher if a custom-made die is necessary. Some typical stamped preforms are shown in Fig. 15.15.

Some solder manufacturers will also supply preforms stamped from laminated metal systems such as copper-solder or solder-copper solder. Continuous strips of those materials systems are made by soldering the molten solder alloy onto one side of a copper strip and then skiving the solder alloy to the desired thickness. Preforms made from such material systems offer some major advantages which are not so apparent on a first examination. For joining problems involving difficult surfaces such as heavily tarnished copper or nickel, the copper segment of the preform will provide a solderable surface in close proximity to the joint. Thus the solder will be retained in the joint area until the flux can activate the surfaces to be joined. Without the solderable surface to retain it, the molten solder might have dewetted and moved to another area of the assembly. In some instances the parts to be joined may have excessive tolerances that will produce a joint area that cannot be filled by the molten solder. In that case the copper core can act like a bridge or filler to maintain the integrity of the joint.

Preforms can be supplied with or without flux. The flux can be an integral part of the preform—that is, the preform can be flux-filled, or it can be coated on the outside of the preform in a subsequent manufacturing operation. In the case of the flux-filled preform, the strip from which the preforms are punched is made from a flattened piece of core solder. This results in a sandwich structure that has flux in the center and solder alloy on the outside. Preforms made from this type of material offer the advantage of very precise

control over the final dimension of the part. The other side of the coin is that the flux is not released until after the metal is molten. In some applications that could be a disadvantage. When precise dimensional control is not essential, it is a simple matter to coat the preform with flux. The flux coating results in a dull powdery appearing surface.

Solder preforms offer a number of advantages. They not only provide solder in a carefully controlled shape to aid in joint formation but also provide a controlled volume of solder alloy. The use of a solder preform also allows complete control over the placement of the solder alloy, which ensures that the joint is formed in areas specified by the designer and nowhere else. Considerations such as these lessen dependence on operator skill and judgment in the assembly of electronic equipment and put control in the hands of the designer.

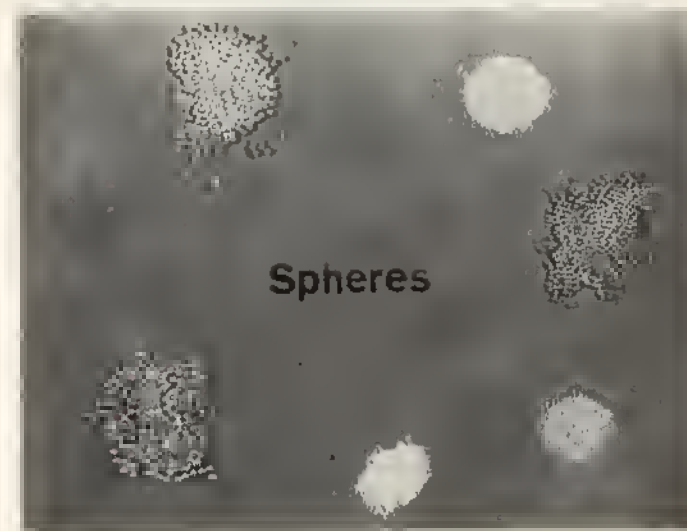


Fig. 15.16 Typical solder spheres. The spheres contain precisely controlled volumes of solder and are easily handled (Alpha Metals, Inc.).

13. Solder Spheres Another type of solder preform which should prove quite useful is the solder sphere. Solder spheres are made by using one of two basic techniques. For large spheres, a wire of the desired alloy is chopped into small segments that contain the proper volume of solder. The segments are spherized by passing them through a bath of hot oil maintained at a temperature above the melting point of the solder. While the segments are passing through the heated section of the column, they melt and surface tension forces cause them to become spherical; and when they reach the cooler sections of the bath, they solidify as spheres. For smaller spheres, the molten solder is sprayed into an oil bath from a nozzle with a very small orifice. The technique can yield extremely small spheres, but the spheres must be sorted carefully in order to have them uniform in size. Some typical solder spheres are shown in Fig. 15.16.

14. Handling of Solder Preforms Solder preforms can be placed in position by using a variety of methods. For high-speed or high-volume applications, they are normally positioned on special tools or fixture which are loaded by using a linear or rotary vibrator feeder. In this type of equipment, the preforms are loaded into vibrating bins or hoppers. The bins are structured in such a way as to accept the preform at the loading point only when it has vibrated to the proper altitude for dispensing. Preforms with a flux core or no flux at all are more suitable to these applications than are flux-coated preforms. The latter will lose some of their coating in the hopper, and the resultant dust will gum up the fixtures and adjacent machinery. Figure 15.17 shows some typical vibratory feed apparatus.

For applications in which the symmetry and spacing can be controlled appropriately, preforms can be manufactured in "daisy chains" that is, connected to one another in a symmetrical pattern. The operator can then simply take a precut length of daisy chain and position it on the assembly. When the preforms are heated, the connections will melt and wet into the appropriate joint. Examples of where such a technique would be applicable are pins from a plug-in connector or the wire wrap pins common to many forms of electrical assemblies.

Many types of specialized jigs and fixtures have been designed for manual placement of preforms. A common technique which is often used with disks and washers is the "shuffle

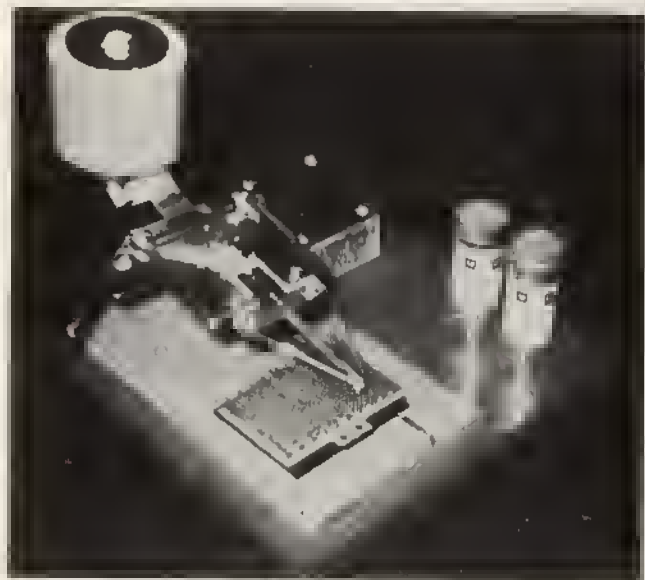


Fig. 15.17 Vibratory feeder for loading solder preforms (Alpha Metals, Inc.)

board." With the shuffle board, a large quantity of preforms is placed at one end of the fixture. The preforms are then pushed or brushed across the fixture plate in such a way that one preform drops into each hole in the plate. A retaining plate is then secured on the fixture and the loaded jig is moved to the assembly area.

A variation of the shuffle board technique is often used for loading solder spheres. In the vernacular, it is called the seesaw, and the author has also heard it called a pinball machine. Quite simply, it consists of two wells, one on either side of the fixture plate. The fixture has a pivot in the center so it can be tipped back and forth like a seesaw. The solder spheres are rolled across the fixture plate each time the fixture is tipped until the holes in the fixture are loaded.

Manual placement of the solder preform can be accomplished by using tweezers, hand tools, and small vacuum pickups. The methods normally require some operator skill and judgment, to say nothing of a steady hand.

15. Solder Creams Solder creams are mixtures of finely powdered solder in a viscous vehicle. The vehicle contains the fluxing agent, and it controls the flow properties of the solder cream. Solder creams can be manufactured either with rosin-based or water-soluble fluxes, so they can be used in any application which requires a solder joint. Because of the many unique ways in which it has been used, solder cream is considered to be the most versatile method for soldering unusual assembly configurations. Solder cream, in fact, can be used as a fluid preform capable of being shaped and molded to fit the application. Solder creams can be cured in place and stored for several days before the joint is flowed to make the bond. They can be used as a kind of viscous putty to hold components in place until the reflow operation. Because they can be applied equally well

to planar and vertical surfaces, they can be used to overcome some of the fixturing problems inherent in preforms.

The manufacture of a solder cream is a fairly sophisticated operation because of the tremendous number of variables which affect properties and performance. The viscosity of the final solder cream is a function not only of the viscosity of the base or vehicle but also of the particle size distribution, the amount of metal in the cream, and the shape and surface roughness of the individual powder particles. Variations in the properties of the cream caused by variations in the parameters may not affect some of the grosser applications, but they are critical to the reliability and reproducibility of many of the microelectric joining applications. Because an understanding of the importance of those parameters is critical to the proper use of solder creams, we will discuss each parameter in turn and try to show the interrelations among the factors.

a. Alloy Powder. Solder powder is made by blowing molten solder alloy through a small nozzle into a cooling liquid in much the same way solder spheres are made. Because of the manufacturing method used, a large variation in particle size distribution is unavoidable. The size variation results in even greater variations in the actual surface area of solder alloy for a given standard volume of solder powder. In particular, the presence of a large number of very fine particles, called fines, greatly increases the actual surface area of the given standard volume.

To separate the powder, or, more appropriately, to grade it, two methods are normally used. In the first method, called an air separator, the particles are introduced into a stream of air and are carried distances dependent on their masses. Bins placed along the stream collect particles of various sizes. The method is tedious and expensive, but it is highly effective. The second method is probably more commonly used. It involves the use of various grades of sieves to sort the particles. When the system is used, the powder is classified by the smallest screen mesh through which it has passed.

b. Flux and Vehicle. The properties of a solder cream are also to a very great extent dependent on the properties of the vehicle and flux mixture; it is those properties that make the solder cream so versatile. Depending on the application, solder creams that cure in place can be obtained; i.e., they dry to a nontacky finish and can be handled after deposition. To aid in deposition, the solder cream can be formulated to be thixotropic, i.e., shear thinning. That means they will run freely when a force is applied to them during deposition; but when the force is removed, they remain in place as thick viscous amalgams of solder and flux. Thus solder creams can be easily and rapidly deposited on the parts to be joined. Then, if desired, they can be cured and handled during subsequent operations and finally reflowed to form the final coating.

The viscosity of the solder cream is affected by the amount of metal in the cream. The more metal added, the more viscous the cream will be. However, as can be seen in Fig. 15.18, the relation between metal content and viscosity is nonlinear. A typical commercial solder cream will contain about 80 percent by weight of metal powder and have a viscosity in the 300,000- to 400,000-cP range.

c. Application of Solder Creams. One of the most popular methods for applying solder cream is by screening. In the technique, shown in Fig. 15.19, a stainless-steel screen which conforms to the desired pattern of solder cream is made. The part is placed under the screen, and the solder cream is forced through the screen by using a squeegee. The method is similar to silk screening. Screening can be used as a fairly gross technique, or it can be an accurate, precise way of depositing solder creams.

Solder cream can also be applied by using pneumatically operated dispensers which meter out precisely predetermined amounts of solder cream. A typical dispenser is shown in Fig. 15.20. Pneumatically controlled dispensers are capable of placing a 0.015-in dot of solder cream on a representative part, an attribute which has made them most useful in the fabrication of microcircuitry. For applications which do not require the precise metering control of a pneumatic dispenser, a manually operated syringe or even a plastic ketchup bottle can be used.

An interesting application for solder creams takes advantage of the fact that the solder cream can be applied either by screening or by dispensing in a gross sort of way. Because the solder cream becomes molten solder when it is heated to the proper temperature, it will wet only surfaces which are properly fluxed and are wettable in the fluxed state. Therefore, a blob of solder cream can be applied to the general vicinity of a joint or a pad. When it is melted, the solder will retract onto the metal surface and form a bond. If a chip

or large ceramic integrated circuit is placed on the solder cream before melting the cream, it will be pulled into perfect alignment with the metal pads as the solder cream is melted.

16. Heating Techniques Heating techniques are germane to the discussion of both solder creams and preforms. They can, therefore, be discussed without considering the form of

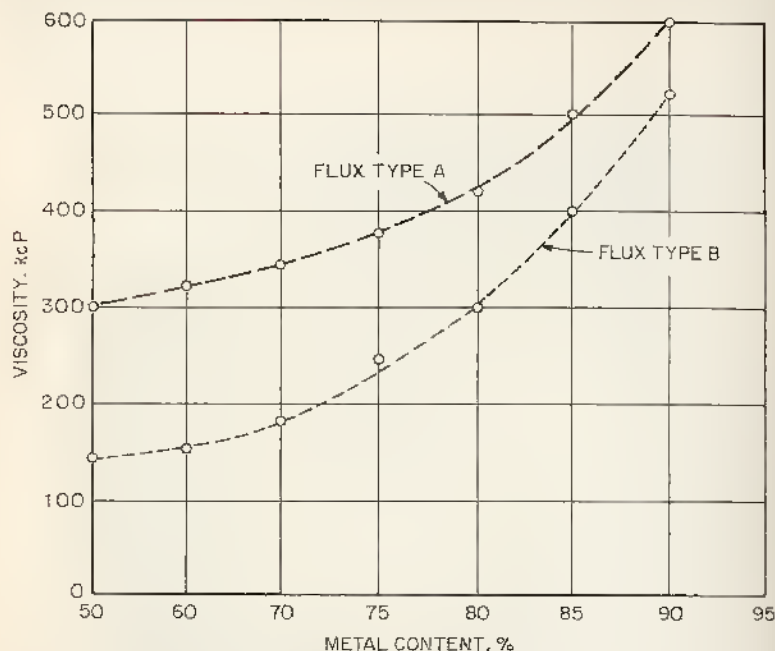


Fig. 15.18 Viscosity as a function of metal content in a solder cream.

the solder on the part. Just as solder creams and preforms challenge the imagination of the designer, so the techniques available for meeting the heating requirements of the assembly call for a great deal of versatility on the part of the assembly engineer. Some applications may simply require the touch of a hot soldering iron, but others will require the complex and intricate techniques inherent in induction and radiation heating. Don't be afraid to let your imagination wander a bit to take advantage of the flexibility of the available technologies.

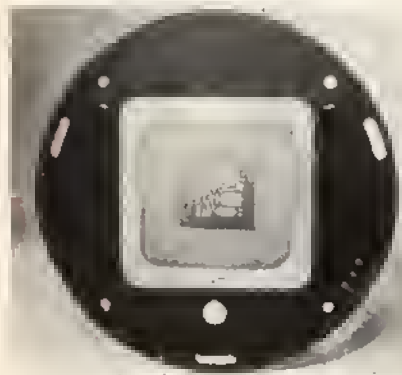


Fig. 15.19 Stainless-steel screen for automated application of solder cream (Alpha Metals, Inc.).

a. Soldering Iron. The use of a soldering iron or a hot plate is a fairly straightforward technique. It is discussed elsewhere, so there is no need to dwell on it here.

b. Hot-Air Gun. A surprisingly versatile heating technique which is often overlooked is the electrically heated hot-air gun. Air from such a gun can be hot enough to flow even some of the high-temperature solders. The gun can be used in a gross sort of way to heat the entire assembly, or, through the use of nozzles, a stream of hot air can be directed into minute spaces of approximately the dimensions of a soldering iron tip. Hot-air guns can be manually

or automatically operated. However, because of the critical relations between distance and temperature inherent in the hot-air technique, an automated setup generally enhances the reproducibility of the operation.

c. Infrared Heating. Infrared heating is a very common and very attractive technique. It is a form of radiation heating; therefore, like all radiation, it can be focused onto a small



Fig. 15.20 Pneumatic solder cream dispenser (Otto Engineering, Inc.).

area. It can induce rapid temperature changes on metallic surfaces (surfaces which absorb infrared radiation) and, at the same time, cause only minor changes in the temperature of the epoxies generally used for printed board construction. Because infrared heating is done principally by radiation, some of the design parameters you must consider are the geometry of the parts to be joined, the emissivity and thermal conductivity of both the parts and the surrounding region, and the color of the parts and the surrounding area.

If it is desirable, shields and masks can be made to allow the parts to be heated in one region and not in another. A typical application is shown in Fig. 15.21.

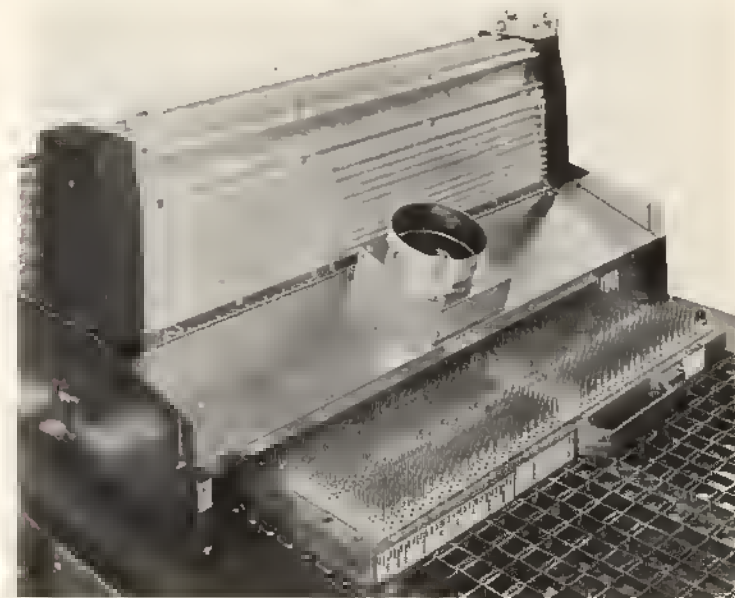


Fig. 15.21 Stainless-steel screen used to prevent excessive heating of the area surrounding a solder joint during infrared soldering (Research Inc.).

Infrared heating adapts itself well to automated processing and high-speed production. Most printed boards which are protected by reflowed tin-lead coatings are reflowed by using infrared techniques. Infrared heating is also a common technique for handling silicon devices to substrates, bonding substrates to printed boards, and soldering and splicing flexible cables.

d. Induction Heating. Induction heating is a technique for heating electrically conductive materials to high temperatures. It is accomplished with high-frequency eddy currents induced in the parts by an external induction coil. Because of the current and voltage characteristics of the eddy currents, it is not a good idea to use the technique if sensitive active devices are present. However, the author has seen induction heating used quite successfully for soldering connectors on both flat and stranded interconnection cable. The large-scale use of induction heating is generally limited by the high power requirements and the bulky electrical equipment needed to generate the high frequencies that are required.

e. Hot Oil. Hot oil or hot glycol reflowing techniques are a common heating method. They are, however, quite messy, and they are somewhat difficult to control. They are not generally used except for reflowing solder plate onto printed boards or component leads. Do not confuse the hot oil technique with the hydro squeegee. Reflowing with hot oil requires the parts to be immersed in a bath of hot oil or glycol; the hydro squeegee technique impinges jets of hot oil onto the surface of a printed board to reflow the plate under temperature and pressure. Hydro-squeegeed boards have been shown to have minimal thicknesses of reflowed tin-lead on their surfaces after processing, and they are therefore quite susceptible to solderability degradation during storage. Boards reflowed with hot glycol, on the other hand, retain all solder plate and have excellent shelf life if properly cleaned.

Chapter 16

Cleaning of Soldered Boards

HUGH COLE

Cobar Resources, Dallas, Texas

Introduction	16-1
Types of Contamination	16-2
Measurement of Cleanliness	16-2
Selecting among Process Alternatives	16-4
Use of Solvent-based Cleaning Processes	16-4
1. Redistillation	16-5
2. Health Requirements	16-5
3. Flash Point	16-5
4. Threshold Limit Value	16-5
Water-based Cleaning Processes	16-6
5. Cleaning Water-Soluble Fluxes	16-7
Cleaning Equipment—Cold Cleaning	16-8
6. Wave Cleaning	16-8
7. Brush Cleaning	16-9
Cleaning Equipment—Vapor Cleaning	16-9
8. Vapor Degreasers	16-10
9. Selection of Construction Materials	16-10
10. Boiling and Ultrasonic Agitation	16-11
11. Nonezeotropic Vapor Cleaning	16-11
Effectiveness of a Process Alternative	16-12
Defects Related to Cleaning—White Residues	16-12
Defects Related to Cleaning—Meeling	16-14
Circuit Cleanliness	16-14
12. MIL-P-28809 Excerpts on Cleanliness	16-15
13. Water Extract Conductivity Method for Determining Ionic Contamination	16-15
14. Lieberman-Storch and Helphen-Hicks Tests for Rosin	16-17
15. Chloride Determination with Silver Chromate Paper	16-18
16. Determination of Ionizable Surface Contamination	16-18

INTRODUCTION

Cleaning has become one of the most important steps in printed board fabrication; it affects the ultimate reliability of the board. Ionic residues on a printed board lead to electrical leakage and dielectric breakdown, two phenomena which seriously impair the electrical performance of the product. Flux residues left on the board may become corrosive and thereby affect the mechanical and electrical properties of the board. The economics of processing the board are also significantly affected by the choice of a cleaning procedure. Particularly in today's competitive environment, it is important to understand the nature of the trade-offs between reliability and cost of various postcleaning systems.

Types of Contamination

Dirt or contamination on a printed board can be divided into two categories: polar and nonpolar. Polar, or ionic, contamination ("polar" and "ionic" are used interchangeably in the electronics industry, a practice that is somewhat disconcerting to a theoretically oriented chemist) consists of chemical compounds that will dissociate in solution into positively and negatively charged species. It is undesirable both because it can carry a current in the presence of moisture and because it can enter into chemical reactions



Fig. 16.1 The Alpha Ionograph, an instrument used to determine the amount of ionic contaminant on a circuit assembly. (Alpha Metals.)

associated with cerrosium. Ionic contamination typically comes from plating residues, finger salts, and flux activators. Nonpolar (or nonionic) contamination, although not as detrimental as polar contamination, will still interfere with circuit performance. Nonpolar dirt typically consists of resin from solder fluxes and oils and greases associated with handling operations. Oils, greases, and rosin can form insulating films on plug-in contact surfaces, and the films can result in intermittent open circuits which are difficult and expensive to troubleshoot. Nonpolar soils, however, are readily removed by most chlorinated or fluorinated solvents. They do not usually present a problem unless the mechanics of the cleaning process are improperly specified. For example, if pallet design causes entrapment of rosin residues or if dwell time in the cleaning solution is too short, problems are likely to occur.

MEASUREMENT OF CLEANLINESS

Before a level of desirable cleanliness is specified, it is necessary to establish a quantitative measure of the degree of dirtiness of the part. Cleanliness and dirtiness are very general terms, and, understandably, many tests have been devised to determine objectively the amount of dirt on a printed circuit assembly. As enforceable quality control standards, however, they all have one common disadvantage: they don't work! Most notable among these types are ultraviolet inspection and the Lieberman-Sterch test. Both methods are based on the detection of rosin residues and the implicit assumption that,

when all the nonpolar rosin residues are gone, all polar residues also will be gone. That simply isn't true. In addition, the Lieberman-Sterch test is a carefully controlled laboratory procedure; it is tedious, cumbersome, and difficult to adapt to high-volume production.

To quantify the level of cleanliness, it is necessary to measure the amount of ionic soil

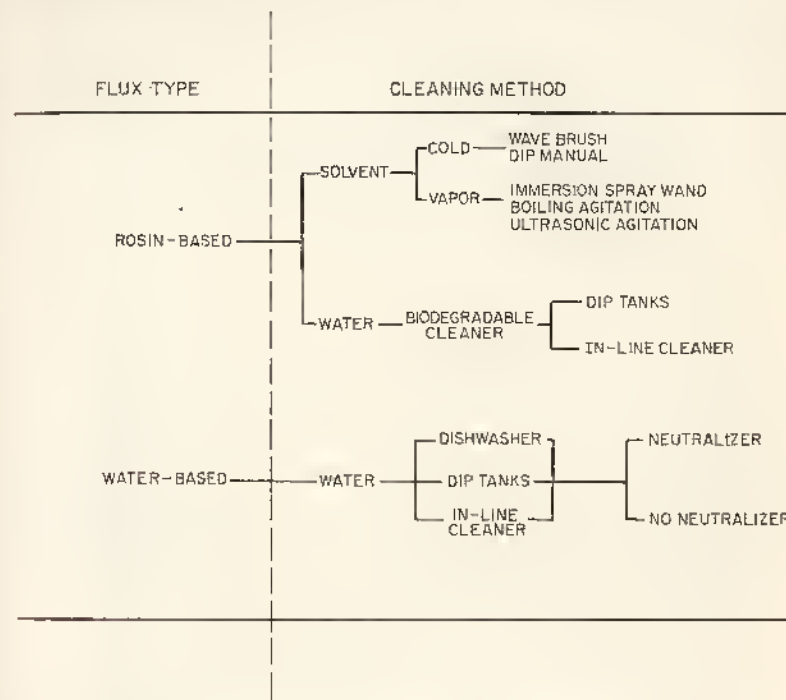


Fig. 16.2 Process alternatives: flux can be removed in a variety of ways. The proper sequence depends on the requirements of the printed board.

on the board surface. Since ionic soil is the most dangerous, a control on the level of ionic dirt promises to be most effective. Several methods of control have been devised. Most notable among them are the solvent extract resistivity tests and the Ionograph. In both methods, the ionic soil on the electronic assembly is dissolved in a water and alcohol mixture and the resistivity or conductivity of the solution is monitored. Variations use high-purity water as a solvent, and that, as we shall see in a subsequent part of this chapter, will result in error because water will not dissolve residual rosin. Therefore, ionics trapped in or under the rosin will not be measured.

When the water-alcohol solutions are monitored, most people prefer to use conductivity rather than resistivity. Conductivity is the reciprocal of resistivity. At the concentrations used to measure cleanliness of electronic assemblies, it is a linear function of the concentration of dissolved ionics; hence it is much easier to interpret. Because of the linear relation, the Ionograph can quantitatively measure the amount of ionic material present on the board surface.

The Ionograph, shown in Fig. 16.1, continuously recirculates the water-alcohol mixture through an ion-exchange column, and the result is a solution of very high purity. By monitoring the change of conductivity as the ionics are dissolved in the water-alcohol mixture, the Ionograph rapidly determines the amount of ionic material on the part. The use of a calibration solution of known ionic content permits quantitative determination of ionic levels on the electronic assembly. Thus the Ionograph can be used either for relative

comparison of cleaned parts or for sensitive analytical determination of the ionic levels. Unlike the solvent resistivity test, which measures only the conductivity of a water-alcohol mixture used to wash the finished assembly, the ionograph will determine the actual ionic level on the finished part.

SELECTING AMONG PROCESS ALTERNATIVES

When a printed circuit assembly operation is selected and installed, the process engineer is faced with three basic alternatives. Two of them involve a rosin-based flux, and the third involves a water-soluble flux. Figure 16.2 shows the alternatives and the implications of each one.

If the most common path is chosen, the engineer will use a rosin-based flux and clean with an organic solvent. The mechanics of the cleaning process depend on the constraints

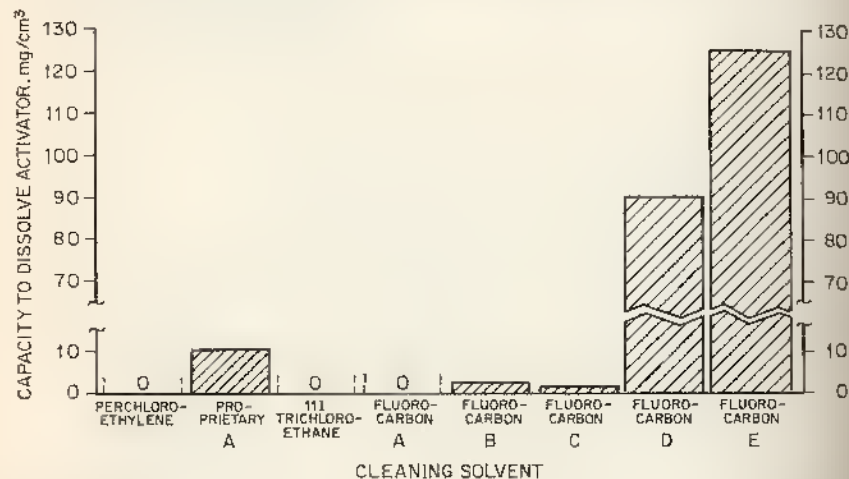


Fig. 16.3 Solubility of a typical ionic flux activator in commonly used flux solvents. Note that only a polar solvent will dissolve a polar salt. Single-component systems: perchloroethylene, 1,1,1-trichloroethane; fluorocarbon A. Designed blends: proprietary A, and fluorocarbons B, C, D, and E.

of the particular application. The second alternative, and one which is growing in popularity, is to remove the rosin-based flux with a biodegradable (i.e., water-based) cleaner. A water-based cleaning system can offer some distinct advantages over solvent cleaning for certain types of assemblies. The third alternative is a water-based cleaning process for water-soluble fluxes. Details of the three cleaning alternatives will be covered in depth in this chapter.

USE OF SOLVENT-BASED CLEANING PROCESSES

Because soil on a printed board can be either ionic or nonionic, solvents used in the final cleaning process must be capable of dissolving both kinds of soil. A solvent capable of dissolving only nonpolar soils will remove oil and greases and rosins, but it will not remove plating residues and flux activators. Figure 16.3 shows the final result of an experiment which illustrates this point. In the experiment, a typical amine hydrochloride flux activator was dissolved in several solvents commonly used to clean electronic assemblies. The more activator dissolved in a given solvent, the greater its propensity to remove polar activators and residual plating salts. Nonpolar solvents such as perchloroethylene, 1,1,1-trichloroethane, and trichlorotrifluoroethane are incapable of dissolving any of the flux activators. The other solvents dissolved various amounts of the activator. The more polar the blend, the greater the amount of activator dissolved. So in choosing a

solvent for postcleaning of electronic assemblies, it is important to choose one that has the capacity to remove both ionic and nonionic soil.

Another consideration is the compatibility of the solvent with the assembly to be cleaned. The solvent must be capable of rapidly removing rosin and handling soils and ionic residues without damaging marking inks or the materials used in the construction of the assembly. That property is related to strength as an organic solvent. If the solvent is too strong, it will dissolve more than the rosin. If, on the other hand, it is too weak, an excessively long time may be required for it to dissolve the rosin residues. The strength of an organic solvent is related to chemical structure and the effective temperature of the cleaning process. The role of temperature is straightforward; the higher the temperature the more active the solvent. The role of structure is more complex. We can generally relate the strength to the number of chlorine atoms on the molecule. When fluorine atoms are added to the solvent molecule, the molecule becomes less active. So, as a general rule of thumb, chlorocarbon solvents are stronger than fluorocarbon solvents. Chlorocarbon solvents have higher boiling temperatures and lower volatilities, and they are therefore adaptable to cold cleaning processes. Fluorocarbon solvents have lower boiling temperatures and higher volatilities, and they are much more adaptable than chlorocarbon solvents to vapor degreasing processes.

1. Redistillation To be used economically in a cleaning process, the solvent must be easily redistilled. The individual components of any given solvent blend may boil over a very broad range, which will make recovery by distillation almost impossible. Unless solvents that boil within a very narrow range can be selected, redistillation will be economically infeasible. Solvent blends which boil at only one temperature are the most preferable alternatives. Those solvent compositions, called azeotropes, are also ideal for vapor-cleaning operations. It is important to realize that azeotropes exist at one and only one temperature, the boiling point. An azeotropic mixture used at temperatures below boiling will rapidly change composition because of the unequal volatilization of the solvents. The ability to redistill the solvents also means that stabilizers and acid acceptors added to the original solvent must be present after distillation. They should come over during distillation in sufficient amounts to insure the stability of the redistilled solvent.

2. Health Requirements Another area of concern is safety and health requirements. In that respect the two most often quoted, and least understood, parameters are flash point and threshold limit value (TLV).

3. Flash Point The flash point of a solvent can be measured in several ways: tag open cup test, tag closed cup test, Cleveland open cup test, and Cleveland closed cup test. For our purposes it is unimportant to know the mechanics of each test. It is enough to realize that the flash point of a given solvent measured by each of the tests could vary within about a 30°F range. Obviously, most manufacturers quote the most optimistic figure. So when you compare two solvent blends on the basis of flash point, make sure you are comparing apples and apples, not apples and oranges. OSHA requirements are standardized on the tag open cup test. Be sure you have that number before approving a solvent for use.

The flash point doesn't really measure flammability; it measures the temperature at which the solvent vapors will momentarily sustain a flame. In many cases, the solvent will flash and then extinguish itself. In general, most chlorocarbon solvents do not have flash points and are nonflammable materials; fluorocarbon solvents such as FC 113 and FC 112 also are nonflammable. Usually, flammable mixtures result from the addition of polar solvents such as the alcohols to increase the solvency power of ionic materials. In that case, it's strictly a question of the ratios. The more flammable solvent in the vapors, the greater the propensity to flash. So when a solvent blend is designed, a delicate balance is normally struck between the ability to remove ionic soil (the principal objective of the solvent) and flammability.

4. Threshold Limit Value The other parameter of concern is toxicity or TLV. It refers to the tolerance level of an "average" person for the solvent vapors. It is given as an index, running from 1 to 1000, that is derived from the maximum amount of solvent (given in parts per million) that a person can be exposed to during an 8-h working day. The lower

the index, the more toxic the solvent. In general, chlorocarbons have lower TLVs than fluorocarbons. The index is somewhat subjective but is nonetheless an excellent tool for ranking the toxicity of solvents. However, TLV doesn't tell the whole story; the other half is evaporation rate. The more a solvent evaporates, the higher the concentration of solvent vapors in the ambient. Therefore, a solvent with a high TLV and a high evaporation rate may be just as toxic as, if not more toxic than, a solvent with a low TLV and a low evaporation rate. TLV is a factor inherent in the solvent itself; process modifications will not change it. Evaporation rate, on the other hand, can easily be minimized by proper equipment design and ventilation. Along those lines, don't forget the human element. The adverse effects of organic solvents have been given widespread publicity. Unless job assignments are made with a view toward the psychological implications, don't be surprised if delays, complaints, and union grievances result when sensitive individuals begin working around organic solvents.

WATER-BASED CLEANING PROCESS

Water is one of the most effective solvents for rosin activators; but because it is a polar liquid, it will not dissolve such nonpolar materials as rosin. One way to overcome that

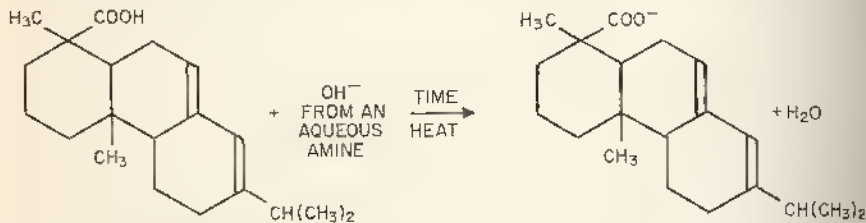


Fig. 16.4 Saponification of rosin. When a biodegradable rosin cleaner is used, the rosin is changed chemically so that it is water-soluble.

difficulty is to react the rosin with other chemicals to form a water-soluble material called a rosin soap. The reaction, called saponification, is shown in Fig. 16.4. Like most chemical reactions, the saponification of rosin is highly dependent on time and temperature. If the temperature is too low or the saponification agents are not in contact with the rosin for a long enough time, the reaction will not go to completion. If that happens, the result is a very dirty board. Because saponification agents react with the rosin, they will be depleted according to the amount of rosin on the boards; therefore, they must be replenished on a carefully controlled basis. Unlike solvents, the exhausted material cannot be reclaimed and must be discarded. Once the rosin has been converted to a rosin soap, the soap must be completely rinsed from the printed board. That is normally done by spraying large quantities of water on the board surface. Several rinse steps are normally used to ensure that all the rosin has been completely cleaned off the board. The important point to remember here is that the parts must be thoroughly rinsed in water of as high a quality as possible.

If the circuit assembly has very high reliability requirements, a final rinse in deionized water is essential. Water can be one of the most deleterious contaminants a board will encounter. If the water is high in ionic salts, improper rinsing may leave the board more contaminated than if it had not been cleaned. If a board has capillary spaces, such as open relay coils, or if it has tight component spacing, such as the space between dual-in-line packs and the board surface, then it may be impossible to rinse it thoroughly.

Once the parts have been thoroughly rinsed, they must be dried. Drying is accomplished by using air knives to blow off excess moisture and then baking the board at elevated temperatures to remove residual moisture. Since water has a lower volatility than fluorinated organic solvents, the boards must be baked for an extended period of time to remove the moisture. Alternatively, fluorocarbon drying systems which will replace the excess moisture with a higher-volatility organic solvent are available. The use of this type of procedure, however, negates the original objective of using the biodegradable rosin cleaner, namely, to avoid the use of organic solvents.

The process just described adapts itself quite well to automation, and several manufacturers have introduced conveyorized equipment to that end. An automated system such as the one shown in Fig. 16.5 can be readily adapted to a specific application. However, one fact to consider when such a conveyorized system is designed and installed is drying time. Because of the time and temperature involved in assuring complete drying of parts, a conveyorized system moving at any reasonable speed would be ludicrously long. Most manufacturers (and rightly so) skimp on the length to allow the machine to operate in a reasonable manner. Be prepared, therefore, for either a supplemental baking operation or a reasonable dwell time at the end of process to allow all the moisture to evaporate.



Fig. 16.5 Typical conveyorized water cleaner (Hollis Engineering).

5. Cleaning Water-soluble Fluxes Water is also used as the principal cleaning solvent for water-soluble fluxes, the residues from which are far more corrosive than those from an activated rosin flux. They must be thoroughly rinsed off to insure trouble-free operation of the printed board. The cleaning of water-soluble flux residues, however, is quite a complex subject and one which seems to be very controversial. Some of the residues obtained after soldering with a water-soluble flux are "sparingly soluble" in water. That means that, in order to remove them with water alone, very high temperatures and very large volumes of water must be used. Even then, we are not assured of their complete removal. To hasten the removal of the residues and convert them into salts that are more soluble, many people use a neutralizing agent such as ammonia water, alkaline detergents, dilute ammonium hydroxide, or similar proprietary chemicals. Most of the agents have the disadvantage that they are probably more corrosive and more deleterious to board performance than the flux residue itself. Although they are for the most part essential to the effective cleaning of residues from water-soluble fluxes, the neutralizing agents must also be thoroughly rinsed with large volumes of clean deionized water to ensure their complete removal. Entrapment of any of those residues, or the flux residues themselves, in a capillary or beneath a component, virtually assures a failure of the printed board. Extreme care must be taken in the design of an assembly which is to be processed by these kind of techniques to avoid capillaries and potential sites of residue entrapment.

Also, to minimize the dangers of using a neutralizing agent, materials based on subliming ammonia salts should be selected. If they are trapped, they can be dissipated either in the heating stage of the cleaner or during storage at factory ambient.

The equipment for cleaning water-soluble flux residues can be as simple as a common dishwasher or as complex as the conveyorized systems previously described. For applications in which reliability of the finished product is important, however, it is highly

recommended that the in-line conveyorized type of cleaner be used. It assures that parts are cleaned immediately, which provides minimal time for the formation of difficult-to-remove corrosion products. Also, the more automated the system is, the more uniform and controlled the cleaning and drying process will be.

Admittedly, the preceding discussion espouses a somewhat conservative viewpoint concerning water-based cleaning systems. That is because of the myriad of field problems encountered by companies unaccustomed to some of the nuances of water cleaning. The secret of an effective water cleaning system is design not just of the process, but of the assemblies to be cleaned. If either is incorrect, the disadvantages of water cleaning rapidly show up.

Unquestionably, water is a better solvent for flux activators than any organic solvent currently available, and it will obviate all of the legal and environmental requirements

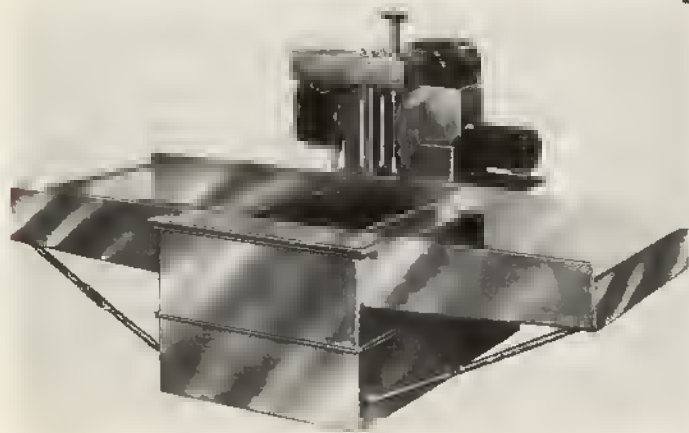


Fig. 16.6 Typical solvent wave cleaner with optimal brush installed. Solvent is pumped in a broad flat wave, and the boards are passed over the wave. (Electrovert, Inc.)

inherent in solvent cleaning. However, it is not a panacea. Not all assemblies are compatible with water cleaning; the probability of failure of an improperly water-cleaned unit is much higher than that of an improperly solvent-cleaned unit; and the cost advantage may be more imaginary than real. Weigh your alternatives carefully; the rewards for success are quite attractive, but the penalty for failure is unduly severe.

CLEANING EQUIPMENT—COLD CLEANING

The type of equipment chosen for a cleaning process will depend on the solvent used, the temperature at which the solvent is to be used, the level of cleanliness desired, and the process throughput. For applications requiring moderate cleanliness and a rapid in-line cleaning process at a minimum cost, an in-line wave or brush cleaner is the ideal choice. Cleaners of those types are generally used with chlorocarbon solvents, although some of the lower-volatility fluorocarbons have also been successfully used.

6. Wave Cleaning A wave cleaner such as the one shown in Fig. 16.6 works in a similar manner to the solder wave discussed in the preceding chapter. The solvent is pumped through a nozzle to form a large standing wave of solvent. The lip of the nozzle is contoured to broaden and flatten the wave. That maximizes the contact time of the wave with the board to be cleaned. Because the board is still warm from the soldering operation, the solubility of the flux residues is greatly enhanced. The system should be connected directly to a still so that cleaning solvent can be continuously introduced into the system at the same time dirty solvent is being removed for reclamation. Otherwise, flux contamination will rapidly build up in the solvent and result in a redeposition of the

flux residues on the printed board to be cleaned. Obviously, that will defeat the purpose of the cleaning operation.

7. Brush Cleaning Brush cleaning is a variation of wave cleaning. In brush cleaning, a large rotating brush is added to the system in the center of the wave. The soft nylon bristles of the brush rotate in a direction counter to the forward motion of the printed board and add a scrubbing action to aid in cleaning. The height of the brush is adjustable so that the brush can remain just below the wave surface and still contact the board surface effectively. The brush must be changed periodically because flux residues have a tendency to make the bristles gummy and stiff. A typical brush cleaner is shown in Fig. 16.7.

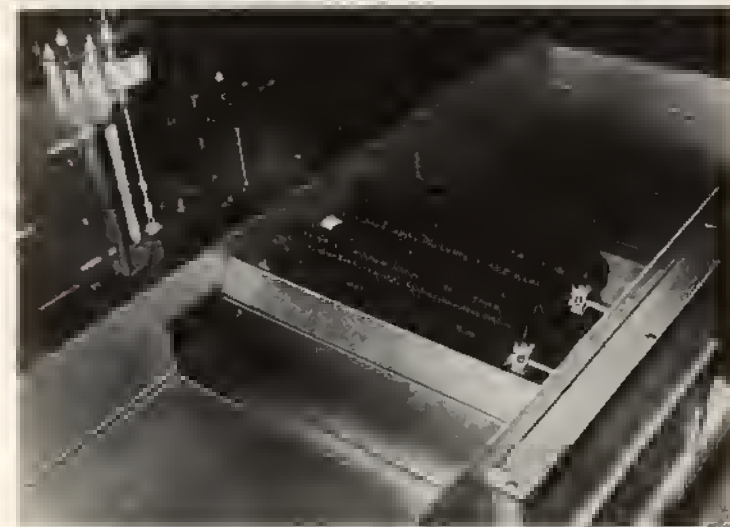


Fig. 16.7 Typical brush cleaner. The brush can be mounted either in the cleaner or above it. (Electrovert, Inc.)

Off line, the same type of cleaning operation can be done with a brush and a pan of solvent, but that poses many problems. If the operation is limited to hand dipping of parts, several immersions must be made, each in successively cleaner solvent, in order to minimize redeposition of the flux residues. The first tank must be changed often to minimize contamination buildup. After the contents of the first tank are dumped into an appropriate holding tank, the second tank becomes the first, etc., and the purest solvent is used in the last tank. If the procedure sounds tedious in theory, it is even more so in practice. Most often it degenerates into a single tank changed once a shift, if that often, and, under the guise of solvent economy, the dirty parts are sent to the next operation.

A variation of the procedure is brush cleaning after touch-up and repair. In that operation, the solvent is contained in individual dispensers at the work stations. Using a small brush, the operator dabs solvent on a localized glob of repair flux. Scrubbing vigorously, he or she redistributes the localized glob over the entire board; and when the solvent evaporates, he or she passes the uniformly contaminated board on to the next process step. Of course, not all the flux residue remains on the board, quite a bit remains on the brush to assist in contaminating the next part.

CLEANING EQUIPMENT—VAPOR CLEANING

For solvent-cleaning applications in which cleanliness is paramount, vapor degreasing is the most popular choice. Depending on the application, vapor cleaning can be done in a small off-line batch-cleaning unit or in an incline machine capable of high process throughput. Vapor degreasing is a technique long popular in the plating and metal-

finishing industries, and it is ideally suited to cleaning of printed boards. It is based on the fact that the hot vapors of a boiling solvent will condense on any surface that is at a temperature lower than the boiling temperature of the solvent. Since the vapors are high-purity solvent, the part will be rinsed with solvent of very high purity until the temperature of the part equilibrates with the boiling temperature of the solvent.

8. Vapor Degreasers A typical vapor degreasing unit is shown in Fig. 16.8. It consists of two sumps, one of which will continuously boil and contain the bulk of the flux residues. The other sump will contain continuously redistilled solvent held at any desired temperature. Any contaminant in the second sump will be continuously diluted as clean solvent drips into the tank and dirty solvent cascades into the boiling sump. The vapors from the

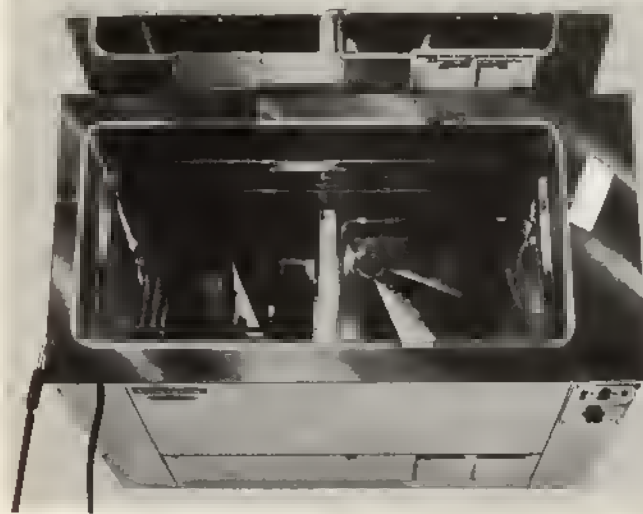


Fig. 16.8 Batch vapor cleaner. (Electrovert, Inc.)

boiling sump are condensed on coils maintained at low temperatures. The coils, situated above both sumps, prevent vapor from escaping from the degreaser and continuously supply the clean sump with redistilled solvent. Some degreasers also are equipped with a spray nozzle that draws solvent from either the cold sump or the distillate and allows it to be sprayed on the part to be cleaned. As shown in Fig. 16.9, large vapor degreasers can be designed with conveyor belts to adapt them for in-line use. The conveyor belts can be made to move through just about any desired sequence of spray, dip, and vapor cleaning.

9. Selection of Construction Materials Stainless steel is the best material for the construction of a vapor degreaser. For economic reasons, some of the plumbing in the degreaser can be made from copper, but that does not include any plumbing which is likely to be carrying a water and organic solvent mixture. Active metals such as aluminum and zinc must be strictly avoided because they catalyze the breakdown of organic solvents and result in the formation of acidic groups that attack both the degreaser and the parts being cleaned. Materials containing iron, which could rust in the presence of moisture, should be avoided for the same reason. Most organic solvents used in vapor degreasing are fortified with stabilizers and acid acceptors to guard against acid breakdown. However, the use of active metals in the degreasing fixtures or in the material used in the construction of the degreaser rapidly negates the effectiveness of the stabilizers. The discussion has been confined to vapor cleaning, but similar considerations apply to the use of active metals in any solvent-cleaning operation.

10. Boiling and Ultrasonic Agitation One other topic germane to our discussion of vapor cleaning operations is the role of boiling and/or ultrasonic agitation. There is a widespread meanness in the electronics industry concerning the use of ultrasonic agitation to remove soil. The meanness stems from the theoretical consideration that beam leads and fine wires used in the assembly of active microelectronic components might hit resonant frequencies and be destroyed. The possibility seems to delight many theoreticians who thrive on computer modeling of abstract concepts, but in practice the notion is farfetched.



Fig. 16.9 In-line conveyorized vapor cleaner. In-line cleaning lends itself well to high-volume production.

Properly designed and implemented ultrasonic agitation can provide an exceptionally effective scrubbing action to aid in the removal of some types of soil.

If ultrasonic agitation is too costly, or if the fears of the theorist are not easily quelled, then the other alternative is boiling agitation. Immersing the part to be cleaned in a sump of clean boiling solvent is an extremely effective procedure. Of course, all components of the assembly must be compatible with the boiling solvent and must remain intact after exposure to the boiling temperature of the solvent.

11. Nonazeotropic Vapor Cleaning Some systems that have been marketed recently utilize nonazeotropic mixtures for vapor degreasing. A typical machine is shown in Fig. 16.10. The systems rely on the fractionation of mixtures of fluorocarbon 113 and alcohol during the distillation process. Each rinse sump will thus have a successively lower content of the lower-volatility alcohol. The vapor composition of these systems is fairly high in alcohol and must be monitored carefully to ensure that the ratio of alcohol to fluorocarbon does not exceed the flammability point. The procedure is extremely effective for cleaning rosin-based fluxes, but it must be run on specially constructed equipment and it must be carefully maintained to ensure that the alcohol content does not reach the region of flammability.

EFFECTIVENESS OF A PROCESS ALTERNATIVE

It is always difficult to draw a comparison of one process with another. The effectiveness of a process depends on many factors which are quite independent from one specific application to the next. However, without offering a lot of weasel words and a long list of exceptions, we can examine some laboratory data obtained on boards of similar configurations with similar soldering procedures. The data were presented at the April 1974 meeting of the Institute of Printed Circuits by the author, and it should suffice, at least, to make generalized rules of thumb and broad generalizations.

The data shown in Fig. 16.11 were obtained by using the ionograph to measure the cleanliness of boards fluxed with an activated rosin flux and cleaned with various solvents. The processes selected for evaluation were cold dip (or immersion) and two vapor-degreasing cycles. The length of the bar in the graphs represents the amount of ionic soil

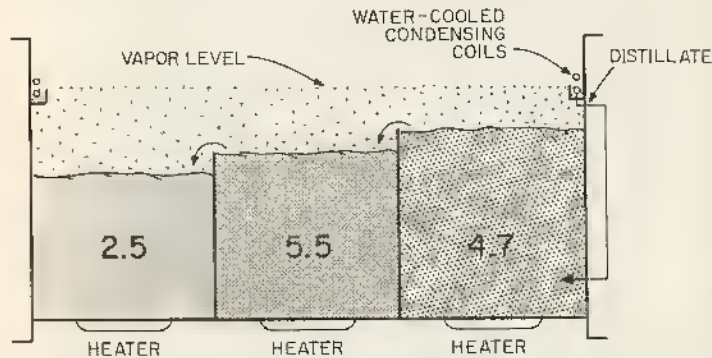


Fig. 16.10 Nonazeotropic vapor-cleaning processes offer the advantage of higher alcohol contents than azeotropic systems. Boldface numbers indicate alcohol concentration. (*Imperial Chemical Industries, Inc.*)

remaining after the cleaning operation. The results correlate well with the results obtained in Fig. 16.3 relating to the ability of the solvents to dissolve flux activators.

As can be seen from the data, vapor degreasing with a solvent designed to remove ionic material is the most effective process. Fluorocarbon solvents adapt best to the vapor-cleaning processes. Cold cleaning with nonpolar chlorinated solvents is essentially a futile attempt to clean the boards, whereas while cold cleaning with a properly designed chlorocarbon cleaner can match vapor cleaning in effectiveness. When these data were presented, no data on water-based cleaning processes had been obtained. Since then, however, a wealth of data has been produced by several people. The data tend to show that a properly designed water-based cleaning process will be almost as effective as vapor cleaning with an effective fluorinated blend. However, the number of boards on which the cleaning was incomplete is alarming. Almost the same thing holds true for water-soluble flux-water-cleaning processes. Use of a properly designed process with an appropriate neutralizer and sufficient quantities of deionized rinse water results in an end product that is cleaner than any solvent-cleaned board measured. Again, however, the same qualification holds true. The number of highly contaminated boards seen in actual field measurement is exceptionally high.

DEFECTS RELATED TO CLEANING—WHITE RESIDUES

Two of the most common defects related to improper cleaning of the printed circuit assembly are white residues and mealing. A board exhibiting white residues is shown in Fig. 16.12. White residues have a number of causes, and not all problems identified in the field as white residues are the same thing. White residues can be caused by improper cure of the base laminate or by solvent attack on improperly cured solder resists. Those defects, however, are relatively rare. Most commonly, white residues are caused by the polymeri-

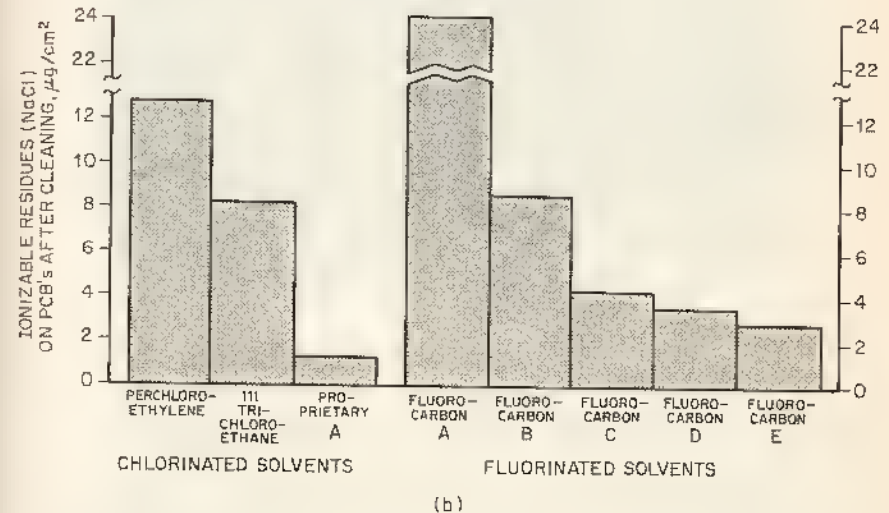
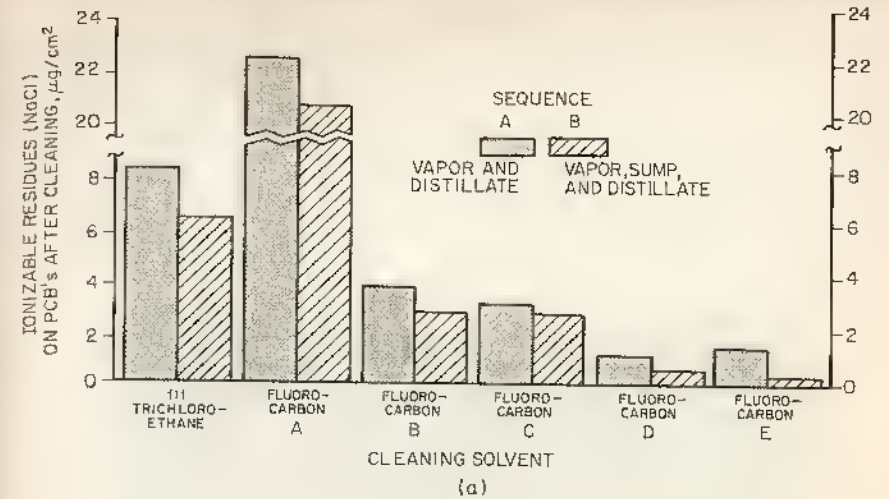


Fig. 16.11 Relative effectiveness of various cleaning solvents in (a) vapor cleaning and (b) cold cleaning. A solvent which is a blend of polar and nonpolar solvents should always be selected. (*Courtesy Alpha Metals.*)

zation of rosin fluxes and/or water-dip lacquers during the soldering operation. When the rosin in those materials polymerizes, some of the rosin becomes a very long chain molecule which cannot be dissolved in the commonly used solvents. Therefore, the flux-cleaning solvents dissolve only the short-chain rosin segments and the original rosin and leave behind a tenaciously adhering white powder which is polymerized rosin. Once the polymerized rosin has formed, not even the best flux solvents will dissolve it. There is, however, a way to remove it. Very simply, like materials can dissolve one and another; so if the board is recoated with a rosin-based flux, the rosin in the flux dissolves the polymerized rosin. As long as there is sufficient rosin to maintain the polymerized rosin in solution, the entire board can be readily recleaned by using normal cleaning methods (i.e., vapor cleaning).

DEFECTS RELATED TO CLEANING—MEALING

The other cosmetic defect directly related to improper cleaning procedures is mealing: white granular-appearing spots which form between the printed board and a conformal coating. The board shown in Fig. 16.13 exhibits mealing. Don't confuse mealing of the conformal coat with mealing of the base laminate. Although the defects appear to be similar, mealing is related to selective delamination of multilayer printed boards. It is caused by resin-starved regions in the laminate, and it shows up long before the soldering operation.

There is a school of thought in the printed circuits industry that feels that mealing is not detrimental to the printed board. That is simply not true. Mealing is symptomatic of ionic contamination on a printed board. As such, it indicates inadequate cleaning of the assembly, and it casts serious doubt on the long-term reliability of the circuit.



Fig. 16.12 White residues on circuit board.

Since conformal coatings are essentially polymer films, they will behave in a manner similar to that of thin polymer membranes. They are not impervious to water vapor; instead, they will experience a distinct osmotic pressure and, depending on the coating used, will be fairly porous to water vapor. When a void occurs in the conformal coat, either in the coating itself or between the board and the coating, the transmission of water vapor will essentially become localized to the void area. Hence the void induces a kind of localized pumping action for the water vapor. When ionic materials are near the void area, they dissolve in the water vapor and are carried to the void. Since the membrane transmits the moisture and not the salt, the salt concentrates in the area of the void to form a meal. Obviously, the effect is more than just cosmetic.

Mealing is sometimes cured by switching to a different conformal coat, one with a much lower propensity to transmit water vapor. That camouflages the problem, but it doesn't solve it. If mealing occurs, the problem is in the cleaning of the board, not the conformal coating. Cleaning procedures should be reexamined and modified to ensure that all ionic contaminants are removed.

CIRCUIT CLEANLINESS

In normal handling of circuit boards and final assemblies, a certain amount of contamination is inevitable. A cleaning process can be added, but it also adds expense and time to the total cycle. However, the complication of ionic contamination is the electrical effect of leakage paths at unknown resistance. To see if this is a potential problem, check the circuit for the resistance levels required and then test the board for contamination by means of one or more of the following tests. If unacceptable contamination is detected,

both preventive measures in the basic process and a cleaning step should be installed to facilitate reaching the acceptance level.

12. MIL-P-28809 Excerpts on Cleanliness

3.7.1 Resistivity of solvent extract (see . . . 6.7). When uncoated printed-wiring assemblies are tested as specified in 4.8.3, the resistivity shall not be less than 2,000,000 ohm-centimeters. Equivalent test methods may be used in lieu of 4.8.3 only when specifically approved by the government procuring activity. Such approval will be determined on the basis that the alternate method is demonstrated to have equal or better sensitivity, and employs both polar and nonpolar solvents with the ability to dissolve flux residue as does the alcohol-water solution specified in 4.8.3.

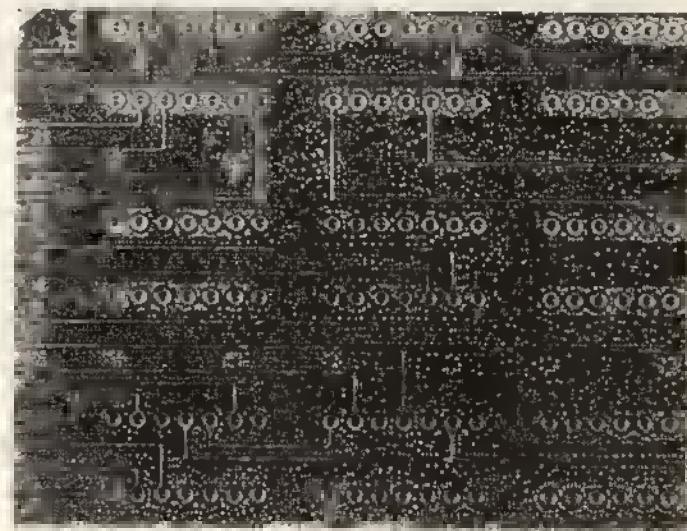


Fig. 16.13 Circuit board with mealing.

4.8.3 Cleanliness and resistivity of solvent extract (see . . . 3.7.1). A convenient sized funnel shall be positioned over an electrolytic beaker. The printed-wiring assembly shall be suspended within the funnel. A wash solution of 75 percent by volume of ACS reagent grade isopropyl alcohol and 25 percent by volume of distilled water shall be prepared. This wash solution must have a resistivity equal to or greater than 6×10^6 ohm-centimeters (see 6.7). The wash solution shall be directed in a fine stream from a wash bottle onto both sides of the assembly, until 100 milliliters of the wash solution is collected for each 10 square inches of board surface (including both sides of the board, but not counting the surface area of the parts mounted thereon). The time required for the wash activity shall be a minimum of one minute. It is imperative that the initial washings be included in the sample to be measured for resistivity. The resistivity of the collected wash solution shall be measured with a conductivity bridge or other instrument of equivalent range and accuracy. Note: All laboratory ware must be scrupulously clean (see 6.7).

6.7 Resistivity of solvent extract (see 4.8.3). This test procedure, including solution preparation and a laboratory ware cleaning procedure, is documented in Materials Research Report No. 3-72, "Printed-wiring assemblies; detection of ionic contaminants on." Application for copies of this report should be addressed to the Commander, Naval Avionics Facility, Indianapolis, Indiana 46218.

13. Water Extract Conductivity Method for Determining Ionic Contamination

a. Outline of Method. The plated parts are placed in water of known conductivity and agitated for a specific time. The conductivity of the water extract is measured, and the increase in conductivity due to residual plating salts and other conducting impurities is calculated.

b. **Applicability.** This method is applicable to the detection of residual plating salts and other soluble, ionizable contaminants on parts that are completely plated.

c. **Reagents and Apparatus**

1 **Conductivity monitor.** A conductivity monitor with a primary scale of 0 to 2 $\mu\text{S}/\text{cm}$ and additional scales for higher readings. An instrument such as the Balsbaugh Laboratories Model 910M-2C Monitor, obtainable from Balsbaugh Laboratories Incorporated, 25 Industrial Park Road, South Hingham, MA 02043, or equivalent, may be used.

2 **Working conductivity cell.** A conductivity cell with an integral thermostat for automatic temperature compensation with a constant of 0.1 cm^{-1} . A conductivity cell such as the Balsbaugh Laboratories No. 910-JTD Dip Conductivity Cell, or equivalent, may be used.

3 **Standard conductivity cell.** A duplicate of the working conductivity cell to be used only for checking the calibration of the working conductivity cell.

4 **Cell selector switch.** A cell selector switch for connecting either the working or standard conductivity cell to the conductivity monitor. A switch such as the Balsbaugh Laboratories No. 910SS-2 Selector Switch, or equivalent, may be used.

5 **Patch cord.** A shielded patch cord for cell selector switch to conductivity monitor connection. A patch cord such as the Balsbaugh Laboratories No. 910PC-10 Patch Cord, or equivalent, may be used.

6 **Jogger.** A jogger (agitator), 3600 cycles/min (60-H current), with adjustable amplitude. A jogger such as the Single Action Model DL-1A, obtainable from Syntro, a division of FMC Corporation, Humer City, PA 15748, or equivalent, may be used. Joggers with larger tables are also available.

7 **Water bottle.** A polyethylene bottle of about 1.9 L (1 gal) capacity with a polyethylene or polypropylene screw cap.

8 **Deionized water.** Deionized water with a conductivity of 0.5 $\mu\text{S}/\text{cm}$ or less (resistivity 2 M Ω or more).

d. **Preparation of Equilibrated Water and Calibration of Working Conductivity Cell.** Put the working and standard conductivity cells into separate containers of deionized water. Before removing them to make a conductivity measurement, *always* measure the water in which they are stored to be certain it is less than 1 $\mu\text{S}/\text{cm}$. That ensures the cells are clean.

CAUTION: The standard conductivity cell is used only for checking the working conductivity cell in equilibrated water. It must not be put in any other liquid.

Rinse the polyethylene water bottle thoroughly with deionized water; then half fill the bottle with deionized water. Shake the bottle vigorously for 2 min to equilibrate the water with the carbon dioxide in the air. (See Note.)

NOTE: Carbon dioxide is a normal and necessary component in air. It is soluble in water and forms carbonic acid, which ionizes and is at equilibrium at 0.8 $\mu\text{S}/\text{cm}$. In a closed polyethylene bottle, the equilibrated water will remain in the range of 0.8 to 1.0 $\mu\text{S}/\text{cm}$ for at least a week.

Transfer some of the equilibrated water to a Pyrex or other borosilicate glass beaker of sufficient size to hold both the working and standard conductivity cells. Insert both cells and measure the conductivity with the standard cell first. Check to see that the level of equilibrated water is sufficient to cover the side holes in the cells. That is necessary to obtain a correct reading. The conductivity reading should be between 0.8 and 1.0 $\mu\text{S}/\text{cm}$; if not, see the following note.

NOTE: If the conductivity is higher, either the polyethylene bottle or the beaker was contaminated. Clean both by rinsing several times with deionized water and prepare new equilibrated water. If the conductivity of the equilibrated water is lower, continue shaking until water in the desired range is obtained.

Next, measure the conductivity with the working conductivity cell. Both conductivity cell readings should agree within 0.1 $\mu\text{S}/\text{cm}$; if they do not, see the following note.

NOTE: If the conductivity cells do not agree within 0.1 $\mu\text{S}/\text{cm}$, clean the cell surface by washing the working conductivity cell in reagent grade acetone, followed by rinses with equilibrated water.

Store the working and standard conductivity cells in equilibrated water in separate Pyrex or other borosilicate glass beakers.

e. **Procedure for Determining Conductivity of Extract of Plated Parts**

1 **Test facilities.** This test procedure should not be performed in a plating shop or in the presence of acid or alkaline mist or in a laboratory with noticeable acid fumes.

2 **Sample.** Calculate the surface area of the plated parts to be extracted. A sample of the plated parts having a total surface area of 32.2 cm^2 (5 in^2) should ordinarily be used and extracted in 100 cm^3 of equilibrated water. For other sample sizes, the volume of equilibrated water should be adjusted to keep the ratio of surface area to volume of water fixed at 0.32 cm^2 (0.05 in^2) to 1 mL of water.

3 **Extraction.** Measure the correct volume (see sample, 2 above) of equilibrated water into a clean, dry Pyrex or other borosilicate glass beaker. Measure and record the conductivity of the equilibrated water (C_W) with the working cell. The conductivity should be in the 0.8- to 1.0 $\mu\text{S}/\text{cm}$ range. If the conductivity is greater than 1.0 $\mu\text{S}/\text{cm}$, see the following note.

NOTE: Either the equilibrated water or the beaker or both are contaminated. Check the conductivity of the equilibrated water and wash the beaker to correct the condition.

Add the plated parts to the beaker containing the equilibrated water and place the beaker on the jogger. Do not touch the parts with bare fingers, and be certain that the parts are completely covered by the water.

NOTE: To prevent the lateral movement of the beaker while the sample is being agitated, cover the top of the jogger with an approximately 1.27-cm (0.5-in.) thick rubber pad with holes of suitable size to accommodate the beakers.

Turn on the jogger and adjust the amplitude of the jogger until a standing wave exists on the surface of the liquid.

CAUTION: Do not increase the amplitude of the jogger to a point at which droplets of liquid will be expelled from the beaker.

After 15 min of operation, turn off the jogger and remove the beaker.

Measure the conductivity of the water in which the working cell has been stored. If it is greater than 1.0 $\mu\text{S}/\text{cm}$, replace with equilibrated water.

Measure and record the conductivity of the sample extract (C_E) in the beaker with the working cell.

Calculate the increase in conductivity of the sample extract due to residual plating salts and other conducting impurities as follows:

$$\Delta C = C_E - C_W$$

where ΔC = increase in conductivity, $\mu\text{S}/\text{cm}$

C_E = conductivity of sample extract, $\mu\text{S}/\text{cm}$

C_W = conductivity of water, $\mu\text{S}/\text{cm}$

NOTE: This is hitherto unpublished material. It will also be published in *Plating*.

14. **Lieberman-Storch and Halphen-Hicks Tests for Rosin**

a. **Scope.** This method covers procedures for the qualitative detection of rosin in varnishes by the Lieberman-Storch and Halphen-Hicks tests. The rosin may be present as either free rosin (abietic acid), esterified rosin, or metal salts.

b. **Apparatus**

1. Test tubes, 2.5 \times 15 cm

2. Filter funnel, 75 mm in diameter.

3. Porcelain spot plate.

c. **Reagents**

1. Acetic anhydride.

2. Sulfuric acid reagent. Add 35.7 mL of concentrated sulfuric acid (H_2SO_4 , sp. gr. 1.84) slowly to 34.7 mL of water, and cool to room temperature. Store in a glass-stoppered bottle.

3. Phenol reagent. Dissolve one part by volume of phenol in carbon tetrachloride.

4. Bromine reagent. Dissolve one part by volume of bromine in four parts by volume of carbon tetrachloride.

d. **Procedure**

1 **Lieberman-Storch test.** Place 0.1 to 0.2 g of the sample in a test tube and add 15 mL of acetic anhydride. Heat gently until the sample is dissolved or dispersed. Cool and filter, with an ashless rapid filter paper, into a clean test tube. Place a few drops of the clear solution in a depression of the spot plate and add one drop of sulfuric acid reagent, so that the acid will mix slowly with the filtrate. If rosin is present, a fugitive violet color develops

immediately. A pink or brown coloration should be ignored. A control sample containing rosin should be run simultaneously.

2 Halphen-Hicks test. Dissolve a small quantity of the sample in 1 to 2 mL of the phenol reagent. Pour the solution into a cavity of the spot plate until it just fills the depression. A portion of the solution will spread out on the flat part of the plate a short distance beyond the rim of the cavity unless too much of the carbon tetrachloride has been lost by evaporation, when a drop or two more should be added to produce the spreading effect referred to. Immediately in an adjacent cavity place 1 mL or more of the bromine reagent, so that the bromine vapors evolved will contact the surface of the solution in the other cavity. Sometimes it is necessary to blow a gentle current of air in the proper direction to accomplish this satisfactorily, or both cavities may be covered by a watch glass of suitable size. The development of a fugitive violet coloration, best observed upon a flat portion of the test plate, indicates the presence of rosin.

15. Chloride Determination with Silver Chromate Paper

a. Application. This test method is used to determine qualitatively if chlorides or bromides are present on solder joints or other materials or surfaces. The chloride or bromide must be on the surface or otherwise soluble in water.

b. Test Material

1 Source. Silver chromate reagent paper may be obtained from A. Eichhorn Co., 644 Salem Ave., Elizabeth, NJ 07209, (201) 351-6975.

2 Characteristics. Silver chromate test paper is extremely sensitive. The paper should not be touched with the hands or allowed to come in contact with materials that have been contaminated by handling with bare hands. It should be handled with clean forceps; contact with clean scissors, glass rod, etc., is permissible. The use of disposable plastic gloves is advisable.

The paper is also light-sensitive, and its original brick-red color will fade if exposed to direct daylight for a prolonged time. It should always be kept in clean containers and shielded from daylight when not in use.

c. Procedure

1. Moisten a piece of silver chromate reagent paper of suitable size with distilled water. Use forceps or other suitable means to handle the paper. Do not permit the paper to come in contact with the hands.

Drain excess water by touching the edge of wet reagent paper with blotting paper. Place moist reagent paper in intimate contact with the surface area to be tested.

Use a clean glass rod or forceps to press the reagent paper down. Allow the paper to remain in position for 1 min; then remove it and examine it for color change. An off-white color indicates the presence of soluble chlorides or bromides.

2. As an alternate procedure, pipette a small quantity of distilled water into a capillary tube and transfer it to the area to be tested. One drop should be sufficient. Allow the water to remain in contact with the test area for 1 min. Remove water with the capillary tube and transfer it to the test paper. Observe for the color change to off-white or yellow-white, which indicates the presence of soluble chlorides or bromides.

NOTE: Certain acidic solutions may react with the reagent paper to provide a color change similar to that obtained with chlorides. When a color change is observed, it is advisable to check the acidity of the affected area by means of pH indicating paper. If pH values of 3 or less are obtained, the presence of chlorides should be verified by other analytical means.

16. Determination of Ionizable Surface Contamination

a. Scope

1 Purpose. This test method establishes a procedure for determining the amount of surface ionic soil on a circuit board. The soil must be soluble in water, alcohol, or some mixture of the two. The determination can be made on either a quantitative or a qualitative basis.

2 Restrictions. The Ionograph does not differentiate between specific ionic species. It determines their presence and ranks them according to their ionic mobilities. Salts with higher ionic mobilities are weighted heavier than salts with lower ionic mobilities.

b. Uses. This method has application as a quality control tool and also as a method for developing and evaluating cleaning process parameters. As a quality control tool, it can be used to inspect parts to determine if they conform to predetermined levels of cleanliness. In process development the procedure can be used to evaluate solvent and process efficiency and also to set levels of acceptable cleanliness.

c. Theory

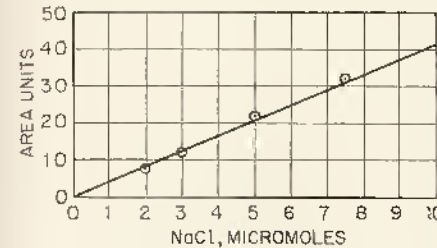
1 Description of measurement technique. The Ionograph utilizes a dynamic measurement of ionic conductivity of a rinse solution which extracts the ions from the surfaces being measured. The solvent is pumped through a recirculating loop which includes a plastic tank, conductivity cell, and ion-exchange column to remove all traces of ions from the solvent before entering the tank. The conductivity cell used is temperature-compensated to avoid reading variations caused by temperature changes. A metering pump pumps the solution through the loop at a constant rate.

Without a sample in the tank, a condition will be established in which the conductivity of the solvent, as measured by the conductivity cell, will attain a constant low value. With

the introduction of a contaminated sample into the tank, the conductivity reading measured at the cell will rise rapidly. A recorder is used to follow the change of conductivity with time. The sample remains immersed in the solvent until the conductivity of the solvent returns to its original equilibrium level. At that point, no further ionic material can be removed from the sample.

2 Theory of measurement technique.

Fig. 16.14 Ionograph calibration: area measured under recorded curve vs. μ moles of NaCl.



The entire amount of ionic material removed from the sample can be related to the integral of the conductivity readings over the period of time required to dissolve the material and pump it through the system as follows: At any instant t the number of moles n_t of ionic material within the conductivity cell is $n_t = V_c \times C_t$, where C_t is the concentration of ions and V_c is the cell volume, which is constant. Over an infinite amount of time, the total number N of moles of ions passing through the cell N will be

$$N = \int_0^{\infty} n_t dt = V_c \int_0^{\infty} C_t dt$$

Since we are dealing with very low concentrations ($10^{-4} N$), we can assume complete ionization; therefore,

$$\text{Conductivity} = L = kC$$

(assuming one salt to be present. Of course, different ionic salts with different ionic mobilities will give different conductivities for a given concentration).

$$N = kV_c \int_0^{\infty} L_t dt$$

If the monitor and recorder responses are linear with respect to L , then, according to this last equation, the area under the conductivity-time curve which is charted on the recorder is a linear function of N , the total amount of ions removed from the sample.

3 Solvent systems. The Ionograph can be used with either pure water or water-alcohol mixtures. Water is used when only water-soluble salts such as plating salts are to be measured. The use of pure water in the Ionograph results in a measuring fluid with an initial conductivity of about $0.1 \mu S$ ($10 M\Omega$).

Water-alcohol systems are used when nonpolar soils might encapsulate or otherwise mask the water-soluble ionic soils. Various alcohols have been used successfully. The preferred systems use either *n*-propanol or isopropanol as the alcohol solvent. Because of the high dielectric constant of the alcohols, excessive alcohol in the mixture will generally degrade the sensitivity of the measurement. To obtain maximum sensitivity and to ensure sufficient alcohol to readily remove all nonpolar residue, the recommended mixtures are 40% (by volume) *n*-propanol and 60 percent water or 50 percent (by volume) isopropanol

and 50 percent water. Mixtures with as high as 75 percent by volume isopropanol have been successfully used.

d. Test Method

1 Calibration. Once the fluid in the system has established a stable level of conductivity for the pump rate, solvent mixture, and tank size selected for the test, a precision hypodermic needle is used to inject a known volume of a solution containing $1\mu\text{g}/\mu\text{L}$ of sodium chloride. A $30\text{-}\mu\text{L}$ injection is normally an adequate initial one. The output of the ionograph chart recorder is then integrated to determine the total area under the curve corresponding to the initial injection of salt. Integration can be accomplished by using an electronic integrator or a planimeter or simply by weighing the cut-out area under the curve on a sensitive analytical balance.

Additional volumes of the calibration solution are then injected until a curve such as the one shown in Fig. 16.14 is obtained for the chosen set of test conditions.

2 Test procedure. Once the system has been calibrated in accordance with 1, above, the test sample is immersed in the sample tank. Care must be taken not to handle the sample or any of the appliances used to insert the sample into the tank. Finger dirt contains highly mobile ionic soils and may give spurious readings on the ionograph.

During the course of the measurement, the conductivity of the solution will depart from the baseline of conductivity and then gradually return. When it has returned to the baseline, no additional soil can be removed and the measurement is complete.

The curve for the sample is then integrated as per 1, above, and the calibration curve is used to determine the amount of contamination on the part.

3 Treatment of test data. The number obtained from 2, above, will be the ionic contamination on the surface of the board in terms of equivalent micrograms of sodium chloride. (If the calibrating solution contained a salt other than sodium chloride, the number will be in terms of equivalent micrograms of that salt.) It is common practice to divide that figure by the total area on both sides of the board and present the data in terms of equivalent micrograms of salt per unit of area.

By using a scale of measurement based on only one salt, i.e., sodium chloride, the ionic contaminants are being measured in terms of their ionic mobility. The more mobile or active an ion is, the more likely it is to cause trouble on a circuit assembly. Thus while the ionograph method will not differentiate between specific ions, it is an effective way of quantifying the presence of many ions.

e. Miscellaneous

5.1 Other Uses of the Ionograph. Because the ionograph measures the ionic activity of any part or solution which contains ionic material, it has been used for various other purposes. A partial list appears below:

a. Incoming inspection of reflowed tin-lead boards to determine if residues have been completely removed.

b. Measurement of purity of incoming and redistilled solvents.

c. Measurement of amount of activated rosin flux dissolved in the boiling sump of a vapor degreaser.

d. Measurement of activity level of activated rosin fluxes.

Chapter 17

Quality Control in Soldering

HUGH COLE

Cobar Resources, Dallas, Texas

Introduction	17-1
Process Control	17-2
1. Solderability	17-2
2. Fluxes	17-3
3. Preheaters	17-3
4. Solder Pot	17-3
5. Cleaning Systems	17-3
Causes for Defective Joints	17-3
Defects Arising from Poor Solderability	17-3
6. Nonwetting	17-4
7. Dewetting	17-5
Process-related Defects	17-5
8. Excess Solder	17-6
9. Bridging	17-6
10. Icicle Formation	17-6
11. Blowholes	17-8
12. Cold or Fractured Joints	17-8
13. Dull or Grainy Solder Joints	17-9
Design-related Defects	17-9

INTRODUCTION

From the preceding chapters it should be apparent that an automated soldering operation can be a smooth-running, efficient way to assemble printed boards. When the processing parameters remain within their specified boundaries, the operation requires minimal maintenance and supervision. However, when one or two of the critical parameters drift away from their expected values, the result is mounting backlogs of quality control rejects and an inordinate amount of high-level supervisory and engineering time spent in

hunting down the cause. That is why maintenance and quality control of the soldering process are so important. With a well-designed, well-documented quality control program, the loss of many hours of valuable time and much unwanted visibility can easily be avoided. This chapter will cover the quality control programs required for an automated soldering operation, and it will touch on various common defects and their causes.

PROCESS CONTROL

Once a suitable process has been designed (i.e., the proper flux, the proper method of application, the proper alloy, etc.), a few simple maintenance operations can easily be set up to ensure its smooth operation. This discussion covers a "typical soldering operation" and should serve as a skeleton on which to build your own program.

1. Solderability Both boards and component leads are subject to a deterioration of the solderability of their surfaces as a result of sloppy manufacture or excessive storage environments. To prevent solderability problems from disrupting the smooth flow of work through a soldering process, solderability checks should be made on all incoming materials and on all materials entering the soldering process from storage. Boards and components with marginal and inferior solderability should either be rejected or have their solderability restored with an appropriate surface conditioner (see Chap. 14). Components placed in bins for hand insertion operations should also be checked periodically, since there is a great tendency to unknowingly subject some of those components to the factory environment for extended periods of time because of the last-in first-out scheme inherent in most bins.

The solderability of a surface is related to the ability of the molten solder to wet or spread out on the surface to be soldered. Several parameters can be examined to gain a good understanding of the solderability of a surface. However, the most common and the simplest indicators are based on the observations of either the total area covered by a controlled volume of solder or the angle which the solder makes with the surface to be tested. Since solderability is an empirical concept, rather than one which can be precisely formulated, solderability measurements are often surrounded with a mystical, semiquantitative set of procedures and numerical scales designed to give the tester a sense of making a precise analytical determination. That is not really the case, because the solderability of a surface is greatly dependent on your requirements and quite independent of any definition I might formulate for solderability. In the final analysis, the solderability test should be an objective attempt to answer the question, "Will it work in my process?" rather than a subjective attempt to quantify the dihedral angle of a solder blob.

Solderability tests should be performed away from the production area, and they should be performed by using the flux and alloy to be used in the final assembly. They can be as simple as a hand-dipping operation on a statistically significant number of parts or as complex as a 100 percent inspection of all parts using the meniscograph. (For more information on solderability testing, see Chap. 14 and/or contact the Institute of Printed Circuits, 1717 Howard Street, Evanston, IL 60202). The type and extent of the solderability testing done for each process is a function of the activity of the flux (recall that RA and water-soluble organic acid fluxes are much more forgiving of solderability defects than RMA or R-type fluxes) and the final requirements of the assembly.

Do not forget that the solderability check is in reality a check both of your vendor's ability to produce solderable parts and your ability to store the parts in a suitable environment. Along those lines, work with your vendor to ensure that he knows your requirements and rapidly feed back to him any difficulties you encounter. Also, maintain a clean storage area with minimal handling of the parts and minimal dwell time.

2. Fluxes The flux density should be checked at least twice during an 8-h shift. The simplest way is to use a hydrometer. The flux should be thinned with the appropriate thinner and makeup flux added.

Depending on the process throughput, the flux should be dumped after 30 to 40 h of operation and new flux added to the appropriate level. If a foam fluxer is being used, the stone should be cleaned periodically by soaking it in a tank of thinner and bubbling air through it for 10 to 15 min to assist in removing any flux residues which may be trapped.

Since the foaming properties of most fluxes are affected by moisture and oil, air lines feeding the stone must be fitted with traps to remove oil and moisture from the lines. The traps must be maintained periodically to ensure trouble-free operation. If the flux pot has brushes or squeegees to spread the flux on the board, they should be cleaned regularly (once a week) to prevent flux buildup.

3. Preheaters The secret to the successful and safe use of preheaters lies in the regular and thorough cleaning of flux drippings from the preheater surface and/or the reflectors. Preheaters should be cleaned about once each shift to ensure proper operation. The preheater temperature should also be checked to ensure that temperature settings are accurate and all heating elements are operating properly. If forced-air preheaters are used, the air-intake filter and/or grating must be periodically checked to ensure unrestricted passage of air across the heating elements.

4. Solder Pot The most important solder pot parameters to be controlled are temperature and composition. The temperature should be verified with an independent thermometer at least once per shift to ensure reliability of the solder process. The alloy composition should be tested at least once per month to guard against rapid buildup of metallic impurities. Also, dross should be removed periodically and replaced with new solder alloy to maintain the proper solder level. Depending on the process throughput, it is also a good idea to set up a schedule of regular additions of fixed amounts of solder chunks to the pumping system. If oil is being used, a regular cleaning and replacement schedule is essential. No soldering oil should be used for more than 16 h. Most oils should be changed every 8 h to be effective.

5. Cleaning Systems If a solvent-based cleaning system is used, the level of acid acceptors in the solvent should be checked periodically to guard against acid degradation. Most manufacturers will supply procedures or simple test kits to perform the test. If the solvent is recovered by batch distillation, the test should be performed after each distillation. If vapor degreasing is used, it is usually sufficient to test the level of acid acceptance once every three or four weeks.

If a water-based cleaning system is used, the conductivity or resistivity of the rinse water should be continuously monitored to ensure adequate cleaning. In any case—whether solvent or water cleaning, a check of some fraction of the boards with a device such as the Ionograph should be made. That ensures that adequate levels of cleanliness are being maintained.

CAUSES FOR DEFECTIVE JOINTS

Under ideal soldering conditions, perfect wetting of the component leads and the printed board terminal area results in solder joints with smooth, bright, concave fillets that exhibit very low contact angles with both the leads and the component leads. Such a solder joint is shown in Fig. 17.1.

If ideal conditions prove somewhat elusive, then the solder joint will be imperfect. The material in this section is designed to identify the types of defects that are indications of serious problems somewhere in the soldering process and that call for immediate corrective action.

Imperfect solder joints may be traced to one or more of the three following general sources:

1. Poor solderability of the printed board terminal area and/or the component lead
2. The soldering process itself (i.e., incorrect adjustment or control of the soldering conditions)
3. Incorrect design of the assembly to be soldered

DEFECTS ARISING FROM POOR SOLDERABILITY

When the basis material has marginal or poor solderability, incomplete wetting by the molten solder will result in a discontinuous film of solder on the termination or board pad. This will result in two basic types of defects: dewetting and nonwetting, which are sometimes difficult to separate. In the case of nonwetting, the color of the basis metal will

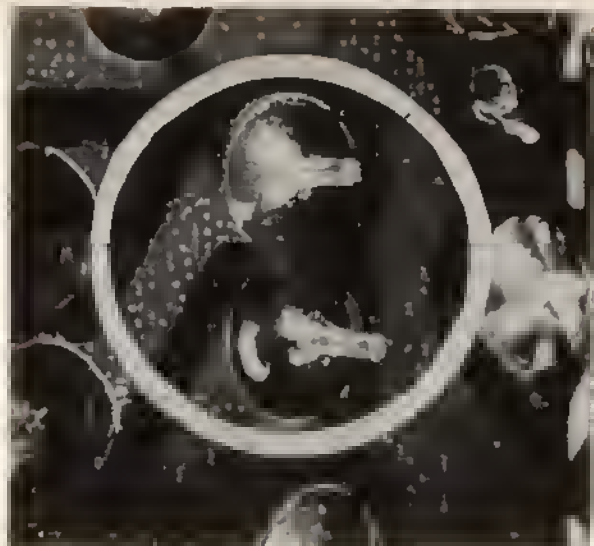
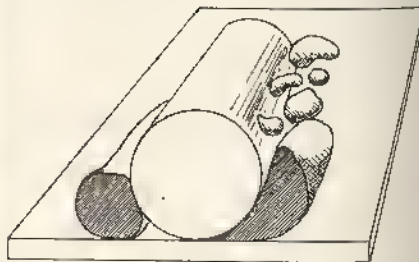


Fig. 17.1 An acceptable solder joint. Note that the contour of the lead is clearly visible.

be readily visible through the discontinuous film of solder. However, if the basis metal is similar in color to the solder, as would be true with a roller-tinned or palladium-coated surface, the nonwet surface may be extremely difficult to discern from one which has simply dewet. A typical dewetted surface is shown in Fig. 17.2. By contrast one which has not wet the terminal areas is shown in Fig. 17.3. Both dewetting and nonwetting are serious problems and indicate inadequate bonding of the solder to the circuit metal.



(a)



(b)

Fig. 17.2 A dewetted surface. The solder has initially wet the terminal areas (a) and then drawn back into discontinuous blobs (b).

6. Nonwetting Nonwetting can occur for a number of reasons, some of the most typical of which are the following: (1) inadequate solderability of the base metal in relation to the activity of the flux used, (2) oil or grease on the surfaces, which prevents the flux from coming in contact with the surface; (3) improperly controlled time-temperature cycle during the soldering process.

Figure 17.4 shows a solder joint which

exhibits good wetting of the wired paths but nonwetting of component lead. Note that partial wetting of the base of the lead results in a cup-shaped depression around the lead. Although the outline of the lead is quite visible, it is not because the solder has wet the lead and feathered away from it as it drained; it is because the complete lack of adherence of the solder to the lead has resulted in the lead itself being visible. Nonwetting can also be masked by excessive solder on the joint, as shown in Fig. 17.5. If there is too much solder on the joint, the lead becomes encapsulated in the solder and any visual indication of its condition is destroyed.

7. Dewetting In contrast to nonwetting, a typical dewet condition results when the solder first wets the basis metal and then retracts, leaving a thin but continuous coating of solder over the basis metal and discrete globules of solder spaced discontinuously across the surface. The large globules of solder have very large contact angles where they meet the basis metal. The large contact angles are indicative of a poor wetting condition.

Dewetting of the basis metals can be traced to sources similar to the nonwet condition.

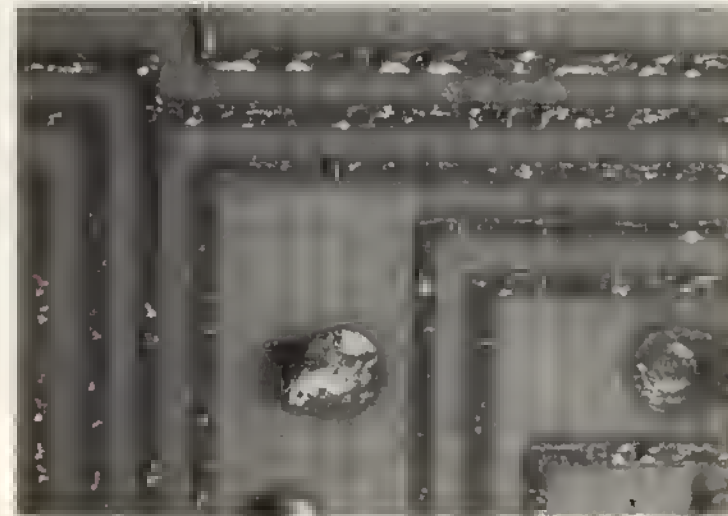


Fig. 17.3 A nonwetted surface. The solder has never wet the terminal areas. The metallic surface below is visible.



Fig. 17.4 Nonwetted component leads on a highly solderable terminal area. Note that the solder has completely pulled away from the lead.

In addition, however, dewetting can be caused by certain types of contaminants on the surface of the base metal; embedded cleaning abrasives are an example. Metallic impurities in the solder bath can, in sufficient concentrations, also result in dewetting.

In extreme cases of poor solderability due to gross contamination of a surface, both nonwetting and dewetting may be seen to occur simultaneously on the same surface. In such cases, solder fillet formation will be poor or nonexistent. On the other hand, a very slight tendency for nonwetting or dewetting on a surface may merely show as pinholes in the solder coating on, for example, printed board conductors; the pinholes are, however, unlikely to have a significant adverse effect on joint formation.

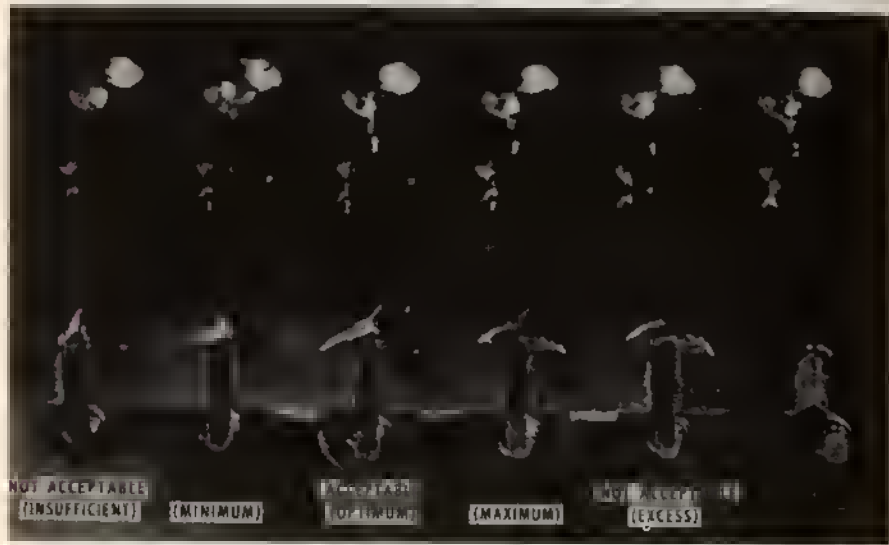


Fig. 17.5 Variation in the amount of solder required to form the joint. Note that excess solder results in a rounded configuration completely masking the lead contour.

PROCESS-RELATED DEFECTS

8. Excess Solder The presence of an excessive amount of solder in a joint, enough to cause the fillet to have a convex surface, may mask the tendency for nonwetting on either the land or the component termination. If no termination is visible, it is not possible to decide whether the component termination wire was too short or whether, in fact, no component has been mounted on the board. For those reasons, excessive solder is undesirable. It may occur because a dip- or wave-soldering bath is operated at too low a temperature, but it more generally occurs in hand-soldering operations.

9. Bridging Bridging can take the form of a web or film between adjacent vertical terminations, as seen in Fig. 17.6, or if it is more extensive in nature, it can appear as a web of solder joining the legs of adjacent conductors (Fig. 17.7).

The cause of bridging may be the use of such incorrect soldering conditions as low temperature, insufficient flux, or the presence of a tenacious oxide film on the surface of the molten solder. The oxide film may be due to the presence of an impurity such as zinc or aluminum. Soldering machine design factors, such as the form of the wave and the angle at which the printed board approaches and leaves the molten metal during wave soldering, may have a strong effect on the tendency for bridging to occur, and the design of the printed circuit assembly is also an important factor.

10. Icicle Formation Stalactites or icicles of solder such as those shown in Fig. 17.8 arise mostly in wave-or-drag soldering operations when the draining of the molten solder from the soldered board is restricted. Icicle formation may often be alleviated by causing the printed circuit assembly to leave the solder bath surface at a small angle, for example, 7°. Solder temperature, conveyor speed, and activity of the flux also may influence the



Fig. 17.6 Solder bridging: an unwanted short between two conductors.

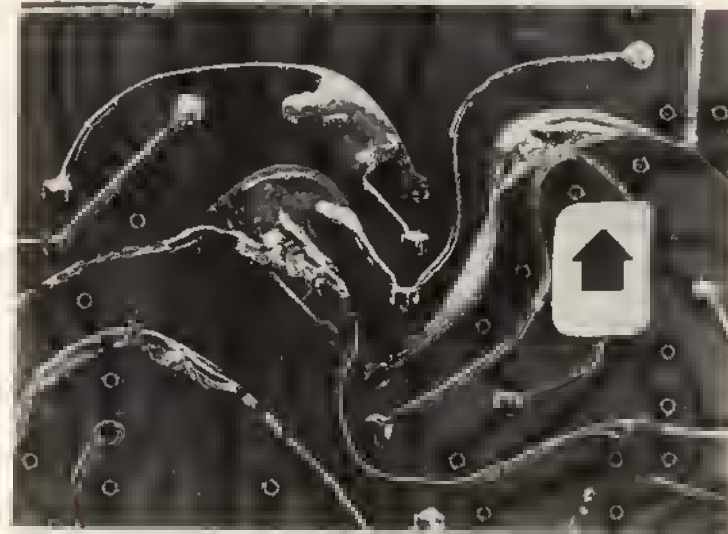


Fig. 17.7 Solder webbing. Note that, in webbing, nonmetallic surfaces can be involved and many conductors can be shorted together.



Fig. 17.8 Icicleing, a common soldering defect.

tendency for icicle formation. As in the case of bridging, icicle formation may be exaggerated by the presence of tenacious oxide films due to high levels of an impurity such as zinc or aluminum in the bath.

11. Blowholes Blowholes or small spherical cavities in the surface of a solder fillet may arise from solidification of the solder around entrapped bubbles of air or flux vapor. A typical blowhole is shown in Fig. 17.9. Usually the interior of the cavity can be seen to be bright and smooth. Blowholes may be quite small, but they should not be confused with the normal surface indentations due to solder contraction upon solidification.

Excessive flux or insufficient evaporation of the flux solvent before soldering can give rise to blowholes and internal porosity in the joint. Blowholes may also result when moisture or organic plating residues become trapped in plated-through holes.

12. Cold or Fractured Joints Cold joints are caused when the component lead moves as the solder joint is freezing. The movement of the lead results in uneven freezing of the solder joint, and ultimately the stress set up can cause a crack in the joint. Any indication of a cold or fractured joint casts serious doubt on the integrity of the joint, and the joint should be



Fig. 17.9 Blowholes, a defect caused by the trapping of liquid or vapor inside the joint as the joint is forming.



Fig. 17.10 A cold joint. Cold joints are caused by excessive movement of the joints during formation.

reworked. If cold joints are a continuing problem, the difficulty can easily be corrected by providing a smooth transfer of the printed board from the solder pot to the next station. A typical cold joint is shown in Fig. 17.10.

13. Dull or Grainy Solder Joints As outlined in one of the preceding chapters, the principal cause of dull solder joints is the accumulation of metallic impurities in the solder pot. Metallic impurities, particularly gold and copper, will rapidly cause the solder joints to take on a dull gray appearance. The problem can easily be corrected by replacing the solder in the pot. Several stopgap measures, outlined in Chap. 15, will allow continued operation until a new solder charge can be obtained. To prevent significant buildup of metallic impurities, it is important to have the solder pot analyzed on a regular basis.

A dull appearance can also result from a solder alloy which is low in tin. The bright shiny appearance of the solder joint is due to the tin, not the lead. Therefore, low-tin alloys such as 50/50 will be substantially duller than high-tin alloys such as 63/37. If the tin becomes depleted from the solder alloy, then the result will also be duller solder joints.

Dull joints will also result when the residues of certain highly active fluxes are left on the solder joint for an extended period. In that case, the dull appearance is the result of chemical attack on the solder joints and the solder process should be modified to ensure rapid cleaning of the boards.

Dross particles or carbonized oil in the solder wave often gives rise to a solder joint with a grainy, lumpy appearance. Both dross and carbonized oil result from improper maintenance of the solder pot. If dross has been sucked into the solder wave, the pot level is too low and additional solder must be added as makeup.

Once dross has entered the solder pump, it is advisable to remove the pump and clean it, since dross is a highly abrasive material and will not purge itself through the pump. Carbonized oil results from using inferior soldering oils or from running a good soldering oil at too high a temperature for too long a time. When that occurs, the oil should be changed immediately and new oil should be passed through the system to purge the carbonized material. In extreme cases, the solder pot must be dumped and recharged.

DESIGN-RELATED DEFECTS

An important feature of the design of printed circuit assemblies is the selection of the optimum component termination size for a given board hole diameter. Even with perfect solderability and soldering conditions, excessive clearance between the terminations and the hole in a terminal area, in the case of unplated holes, often leads to incomplete solder fillet formation. A marginally suitable clearance may become unsuitable when the termination is displaced from the axis of the hole, a condition which is extremely difficult to avoid in the real world of production line assembly.

Insufficient spacing between terminations can lead to bridging, and incorrect printed circuit conductor spacing, size, and layout may also result in bridging or webbing. A large area of copper, or a high density of terminal areas and terminations, tends to act as a heat sink which may, again, cause bridging or, in extreme cases, localized nonwetting.

Section **5**

Quality Assurance

Chapter **18**

**Quality Assurance
Procedures and Methods**

GEORGE A. SMITH
Department of Defense

BERNARD W. ZIMMERMAN
Department of Defense

Introduction	18-4
1. Quality Conformance Test Circuitry	18-4
2. Sampling and Testing	18-4
3. Preproduction Testing	18-4
4. Production Testing	18-4
5. Final Testing	18-5
6. Costs for Testing	18-6
7. High-reliability Concerns	18-6
Testing	18-7
8. Production and Inspection Survey Checklist	18-7
9. Testing Conditions	18-7
10. Calibration and Inspection Interval	18-7
Quality Control Program	18-8
11. Controls	18-8
12. Sampling Plans	18-8
13. Sequential Testing Grouping	18-9
Defective Characteristics and Causes	18-10
14. Major Defects	18-12
15. Minor Defects	18-12
Traceability	18-12
Reliability Testing	18-13
Appendix A	18-13
Appendix B	18-14

INTRODUCTION

This chapter covers test methods and reliability considerations applicable to rigid plated-through-hole printed boards. The boards may consist of two or more layers of conductor patterns on insulating material and be interconnected by a continuous plated-through-hole metallic connection. For purposes of this chapter, the term "boards" will be used hereafter when referring to finished printed wiring boards.

1. Quality Conformance Test Circuitry A series of test coupons, which we refer to as Quality Conformance Test Circuitry, was designed for MIL-P-55640 to provide a series of test areas to permit testing without destroying boards (Fig. 18.1). The quality conformance test circuitry also provides the feature of a perpetual preproduction sample that is representative of each multilayer board made throughout the project. The design allows for either placing all coupons in a strip configuration or separating and placing them at various locations on the panel. Strategically locating the individual coupons on the panels can be a definite advantage to the manufacturer, because he can then observe in-process steps more accurately. For example, he may (1) observe the drilling done by each drill bit on panels being processed by multiple drilling, (2) examine the plating thickness at various time intervals of the plating process, (3) compare the plating thickness at many locations on the panel, and (4) compare the plating thickness at any location of all panels in the plating tanks. If a process should go out of control, the manufacturer can immediately shift to 100 percent testing until the process is corrected without needlessly destroying end-product boards during the screening operation. Obviously, accurate identification, storage, and retrieval of coupons is essential. In-process testing early in production is a self-governing evaluation system.

2. Sampling and Testing After the capability of testing representative coupons has been incorporated, the next step is to develop, with an eye toward minimizing costs, a realistic sampling technique for testing. Certainly any testing is expensive, but not testing at all is ultimately more expensive. Sampling need not be risky, especially if we keep in mind that all tests can be performed at a later time simply by retrieving the desired coupons for lot screening or for diagnostic work. Preproduction samples must pass all requirements of the specification in order for the manufacturer to prove his ability to fabricate boards of acceptable quality. Caution must be used in evaluating preproduction boards, because they are often made in a laboratory hand-line operation. It is very difficult and unusual for a manufacturer to arrange all the special provisions necessary to process preproduction samples in the true production mode. Examples of special provisions are (1) preparation of numerical control tape for drilling, (2) fabrication of racks and other fixtures that would normally be used in production, (3) rearranging plating anodes, board orientation, agitation, etc., and (4) interrupting other production work to process a few sample boards.

3. Preproduction Testing Preproduction testing includes visual and dimensional, microsectioning, plating adhesion, terminal pull, warp and twist, water absorption, solderability, plated-through-hole structure, thermal shock, thermal stress, interconnection resistance, moisture and insulation resistance, current breakdown, and dielectric strength. In special applications, fungus resistance, mechanical shock, vibration, and outgassing characteristics also are tested for.

4. Production Testing During the day-to-day production phase, testing is held to a minimum provided the quality is maintained sufficiently high. Under normal conditions, only visual and dimensional examination, microsectioning one plane, plating adhesion, moisture and insulation resistance tests on the coupon, and circuitry tests on the boards are required. In an effective sampling procedure, destructive testing is performed on the coupons after every four weeks or every 5000th board is produced, whichever occurs first. The testing includes visual and dimensional examination, microsectioning in three principal planes, terminal pull, water absorption, solderability, plated-through-hole structure, thermal shock, thermal stress, interconnection resistance, current breakdown, current carrying capacity, and dielectric strength. If failures occur during those tests, retesting of associated coupons is performed. Repeating the failure causes temporary shutdown, recall of test coupons, and a fact-finding study, although not necessarily in that order. Normal

production is resumed only after the cause is detected, corrective action is taken, and a complete screening is accomplished. However, tightened inspection sampling should be in effect for a prescribed period of time to assure that corrective action was indeed effective and permanent.

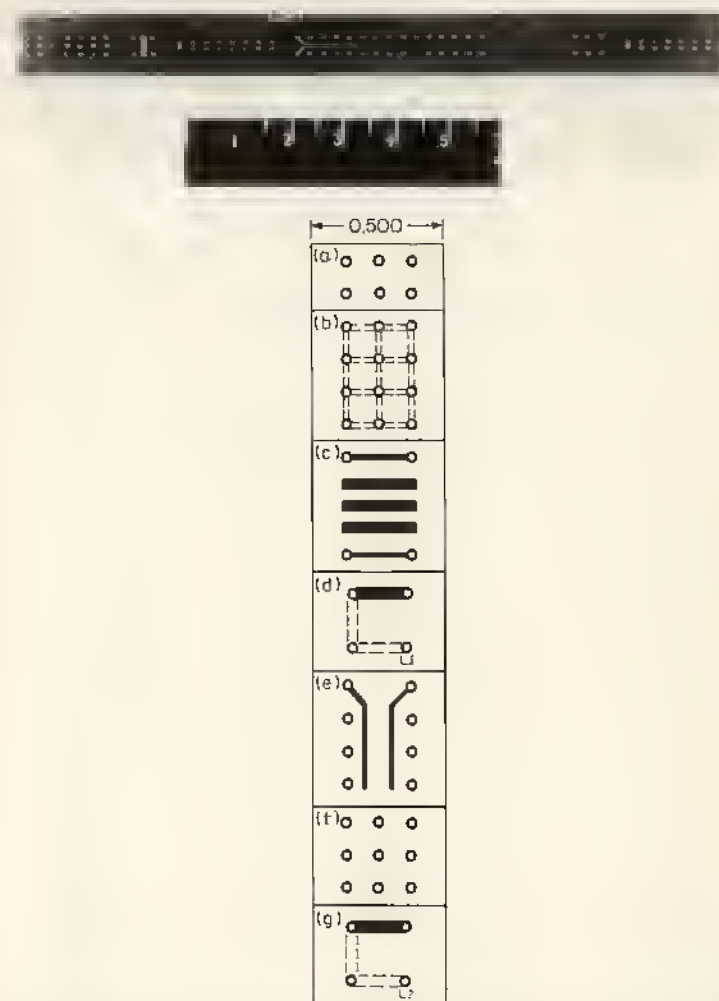


Fig. 18.1 Quality conformance test circuitry. (A) (B) Notes: (1) For economy and ease of artwork layout, the location of each coupon of the quality conformance test circuitry is optional. All test coupons illustrated must appear on each panel. (2) The number of layers must be identical with the number of layers in the boards derived from the panel. Layers shown are for illustration purposes only. (3) Etched letters on coupons are for identification purposes only. (4) Length of coupons D, E and G is dependent upon the number of layers in the panel. (5) All pads should be 0.070 ± 0.005 in diam or 0.070 ± 0.005 in square. Holes in pads should be the diameter of the smallest hole in the associated board. (6) All conductor lines should be 0.020 ± 0.003 in wide unless otherwise specified. (7) All tolerances must meet the requirements of the imposed specification.

5. Final Testing For multilayer boards, only two most critical requirements must be tested 100 percent; (1) microsectioning of one test coupon from each production panel and (2) electrical circuitry testing of the finished boards. It is not feasible to wait for the systems test final phase of production equipment before testing for electrical performance. To

have hundreds of dollars worth of components on defective boards and not find electrical failures until later is extremely costly. Since the function of the multilayer board is to provide reliable electrical interconnection, mechanical integrity of the plated-through hole is of paramount importance. Heavy emphasis is therefore placed on reliable plated-through holes. Hence, periodic microsectioning of the x , y , and z planes and periodic removal of all glass cloth and epoxy-resin for viewing the outer structure of plated-through holes (plated-through-hole structure test) was developed. Appropriate test methods are discussed later in this chapter. During production, the periodic test coupons mentioned above are sent to the customer for complete analysis. Contractors are often required to submit test coupons to the customer regardless of whether boards were made in-house or by a subcontractor.

6. Costs for Testing The testing cost for low-priced radio and TV boards is practically nil because there are very few requirements, but the cost of high-reliability multilayer boards can be approximately 3 percent of the board cost. The characteristics which may be required on various levels of quality boards are shown in Table 18.1. The letters LR designate commercial boards have limited requirements. The letters A, B, and C designate die Institute of Printed Circuit's three levels of quality. MIL designates military specifications, and hi-rel indicates high-reliability requirements (outer-space requirements).

TABLE 18.1 Characteristics Which May Be Required on Various Levels of Quality

Requirement	LR	A	B	C	MIL	Hi-rel
Visual and dimensional	x	x	x	x	x	x
Etchback						x
Plating adhesion		x	x	x	x	x
Microsectioning					x	x
Terminal pull		x	x	x	x	x
Copper strike						x
Warp and Twist		x	x		x	x
Traceability						x
Water absorption						x
Copper pyrophosphate						x
Solderability	x	x				x
PTH structure					x	x
Fungus resistance						x
Mechanical shock						x
Vibration						x
Thermal shock					x	x
Thermal stress					x	x
Outgassing						x
Interconnection resistance		x			x	x
Insulation resistance		x	x	x	x	x
Dielectric strength		x	x	x	x	x
Hot-oil resistance		x	x	x	x	
Moisture resistance		x			x	x
Current-carrying capacity	x	x	x			x
Internal shorts	x	x	x	x		x
Circuitry electrical test (100%)					x	x
Flammability		x	x		x	x

7. High-reliability Concerns One of the most serious problems can be the administrative delay of obtaining test coupons, especially when subcontractors are involved. The delay results in large volumes of boards being manufactured in the interim. The intent is to test coupons promptly to help in the assessment of the quality of production. Another problem is that, if a manufacturer incorrectly drills holes, plates a quantity of the boards, and then stores the boards until the assembly people order the particular part number, he does not know he is producing a bad product up to the plating stage. Because of these problems, it is advisable, in high-reliability programs, to expedite things by having a coupon from each board or panel sent directly from the production floor by bypassing all engineering, production, and administrative channels. Appropriate follow-up can come later. One

finished board should be required periodically for float solder testing, because the coupons have very little surface area for that test and delamination and outgassing are important properties.

TESTING

8. Production and Inspection Survey Checklist To assure checking all process, production, inspection, and test stations, an MLB production and inspection survey checklist such as that in Appendix A is suggested. It assures that, during visits to the plants, none of the many steps are inadvertently overlooked. Perhaps even more importantly, the checklist immediately shows up any significant materials or process changes.

9. Testing Conditions Three main elements must be considered in connection with obtaining data required at room conditions: (1) the test setup for supporting and operating the equipment, (2) the instrumentation for measuring the data from the tests, and (3) the instrumentation for measuring the room conditions.

Considerable emphasis is placed on thorough consideration of the factors involved in obtaining the data, which serve as standard, comparative, environmental data when a test report is presented properly. In most cases, the step consists of a simple recording of the barometric pressure, relative humidity, and temperature. Since room ambient conditions are relative and not absolute, the temperature, pressure, humidity, vibration, etc., both of the room and of the area immediately adjacent to the test setup, should be considered. The conditions to which the specimen is to be subjected and the type of test being performed will vary, but temperature, pressure, and humidity, at least, are recorded in nearly all cases.

10. Calibration and Inspection Interval Consideration must be given to the instrumentation necessary for measuring the required data. Attention must be given to (1) the required accuracy, (2) the effects of the means of measurement on the quantity being measured, and (3) the method of calibrating the instrumentation. Judgment must be employed in correlating the accuracy requirements of the final test data with the individual accuracies obtainable from the instruments employed.

If a choice is permitted by the setup, the most direct means of measurement is usually most reliable. For example, the manometer is preferable to the face-reading pressure gauge. Of course, the choice of instrument depends not only on the accuracy requirements but also on the specific conditions under which the measurement must be made.

TABLE 18.2 Calibration Schedule for Production Tooling and/or Equipment

Item	Required checks	Interval
Degreaser	Temperature and Baumé	Weekly
Automatic plating system	Temperature, Baumé, mechanical operation	Weekly
Cleaning tanks	Temperature, Baumé	Weekly
Oven	Temperature	Daily
Screening machine	Alignment	Weekly
Ether	Temperature	Daily
Numbering machine	Alignment	Daily
Double-tape machine	For clean punch and holes	Weekly
Countout press and die	For clean sharp edges	Weekly
Heat scale	Temperature	Weekly

Depending on the accuracy requirements, the standards against which the instrument calibrations are made must be considered. The probable accuracy of the laboratory standards, the frequency with which the standards are checked against primary standards, the techniques of calibration, and the environmental conditions under which the calibration is performed must be assessed against the determined accuracy requirements of the test data.

Based on previous experiences, production tooling, inspection gauges, and equipment should be checked within the intervals listed in Tables 18.2 and 18.3.

QUALITY CONTROL PROGRAM

11. Controls

a. **Incoming Material Control.** Quality control samples all raw materials received for production and inspects materials for compliance with drawings, specifications, and all other required contractual criteria. Material testing which requires specialized equipment, chemical analysis and/or test which cannot be performed by the inspector should be forwarded to the laboratories for testing. No material should be used until the inspector certifies that it is in compliance with all contract requirements or is accepted on a waiver. A material history file should be maintained on all items received.

b. **In-process Control.** Random sampling of material in process is performed at established inspection stations located at critical points within the production flow by the inspector. This type of inspection will alert production personnel if a process goes out of control and requires remedial action. It will also reveal nonsalvageable rejects soon after they occur and thereby preclude further processing of such items.

TABLE 18.3 Inspection Gauges and Equipment Calibration Schedule

Item	Inspection interval
Go-no-go plug gauges	6 months
Surface plates	1 year
Vernier calipers	6 months
Micrometers	6 months
Profilometer	1 week
Microscope	1 year
Critical circuit area overlay	6 months
Ph meter	1 year
Electrophotometer	1 year

At the end of each shift of plating on either automatic plating line at least one board is selected at random for plating thickness determination. The determination is performed by the microscopic method. Adhesion test also is performed by the inspector at least three times during each shift of operation.

c. **Tooling, Equipment, and Solution Control.** Production and inspection tooling and equipment is inspected at specific intervals to determine wear, mechanical alignment, operation, and electrical calibration. Records are kept to indicate when replacement, repair, calibration, or realignment will be required.

d. **Electrical Inspection Control.** Each board receives an electrical test for shorts, current-carrying capacity, circuit continuity, and circuit verification. The electrical test equipment is checked at least once each shift to ensure it is operating according to requirements. The tests are performed by the inspector.

e. **Complete Product Control.** The inspector selects at least one board from each 300 completely packaged boards and inspects for completeness of packaging, identification, and overall product quality and performs a functional electrical test. A permanent record is maintained on all boards shipped. The record includes the serial numbers and quantity shipped and the name of person who originated shipment.

12. **Sampling Plans** The sampling plan to be used varies considerably with board type. In deciding which plan is best suited for a particular category, five factors are considered: (1) allowable in-process lot quantity delay time, (2) required frequency of examination per lot quantity, (3) the complexity of the operations to be controlled, (4) the lot size to be sampled, and (5) the defect occurrence level of the operations through which the lot quantity has passed. In the following example subsections, the sampling plan used in each control category and the method used to perform the quality control check are outlined.

a. **Incoming Material Sampling.** Acceptability of incoming material is determined by use of sampling plan K as outlined in MIL-STD-105D, GEN. INSP. LVL. II NORM. I% Acceptable Quality Level (AQL) in 2-3000 pieces.

As an example, the sampling plan chosen for incoming material requires that 125 pieces be drawn at random from 3000 for evaluation. If three or more are found to have major

defects, the lot is rejected and the tightened sampling plan as outlined in MIL-STD-105D is used on all subsequent lots until quality returns to the acceptable level.

b. **Sampling of the Product in Process.** The complexity of the manufacturing process increases the possibility of defect occurrence. Therefore, frequent lot sampling is done to reduce buildup of material at any point and reveal any out-of-control production process which should have corrective action.

One may find it necessary to deviate from MIL-STD-105D in establishing sample size per lot quantity because the complexity of manufacturing processes dictates tighter sampling procedure. A sample of each product lot quantity is checked to determine the presence of major and minor defects. The sample size to be checked at each station varies proportionally to the complexity of the operations the boards have passed through since their last quality control check. The defects sampled for at each station are those that conceivably could result from out-of-control operations that the boards have passed through since their last quality control check.

Sampling is performed at those designated stations. The check sheet shows

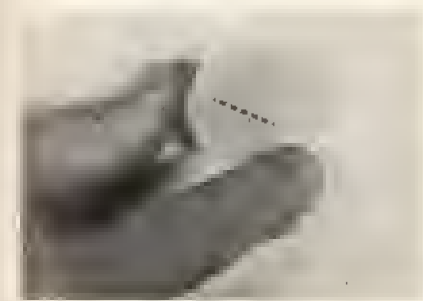


Fig. 18.2 Encapsulated cross section for microscopic evaluation.

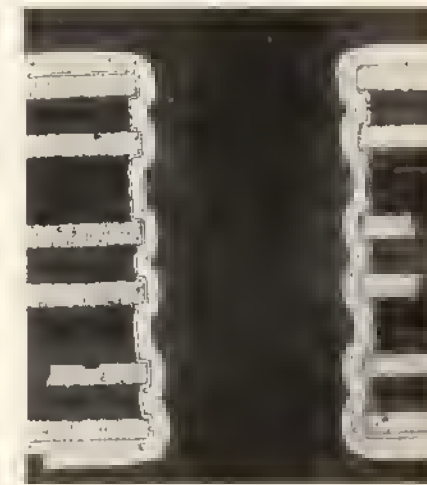


Fig. 18.3 Photomicrograph of a six-layer multi-layer board; original magnification 50X.

which defects will be inspected for and reported on at each quality control check station. When a series of like major or minor defects at any given sampling station has been determined, the quality control representative will determine the operation number(s) of the out-of-control process(es) and will notify the production supervisors immediately to preclude the occurrence of additional defects.

c. **Tooling, Equipment and Solution Control.** Inspection records are maintained on all production tooling, inspection gauges, and electrical test equipment. It is the responsibility of quality control to maintain such records. The records should indicate when replacement, repairs, calibration, or realignment will be required.

A plating solution control form is employed to record all data required to control the plating solutions. It is the responsibility of the plating room leader to maintain the records.

d. **100 Percent Electrical Check.** Proper functioning of the boards depends, of course, on the boards being electrically acceptable. To assure acceptability, each circuit on every board is checked for such critical defects as opens and shorts. As an additional precaution against a board with a critical defect inadvertently being shipped to a user, samples of each lot quantity and the test equipment itself are checked by quality control at least once each week.

The 100 percent electrical check is usually performed by using automatic electrical test equipment as a normal inspection function during production operation. The tests simulate field use of the boards but subject the boards to greater voltages than actual field use.

13. **Sequential Testing Grouping** A typical series of groups A, B, and C testing requirements are given as follows:

Group A. Visual and dimensional. The visual and dimensional tests are listed in Table 18.4. See also Figs. 18.2 and 18.3.

Group B. Group B inspection is performed on production boards or on the quality conformance test circuitry area that has passed group A inspection. See Table 18.5.

Group C. A sufficient quantity of boards should be selected from lots which have passed groups A and B inspections. The frequency should be on a 4-week basis or 5000 boards produced, whichever occurs first. In addition, the manufacturer must, on a 4-week basis, submit at least one sample of a quality conformance test circuitry area to the customer for testing. See Table 18.6.

TABLE 18.4

Examination or test	To be performed on	
	Production board	Quality conformance test circuitry area
Visual examination	100%*	100%
Dimensional examination	Sampling	100%†
Plating adhesion	No	Sampling
Warp and twist	Sampling	No

*The production boards will not require microsectioning.
 †Microsectioning shall be in one vertical plane only.

TABLE 18.5

Examination or Test	To be performed on	
	Production board	Quality conformance test circuitry area
Moisture and insulation resistance	No	Yes
Circuitry	100%	No

TABLE 18.6

Examination or test	To be performed on	
	Production boards	Quality conformance test circuitry area
Visual and dimensional examination (including microsections)	No	Yes, 3 planes
Terminal pull (Bond Strength)	No	Yes
Water absorption	No	Yes
Solderability	No	Yes
Plated through-hole structure	No	Yes
Mechanical shock	Yes	No
Vibration	Yes	No
Thermal shock	Yes	No
Thermal stress (float solder)	Yes	No
Interconnection resistance	No	Yes
Current breakdown	No	Yes

DEFECTIVE CHARACTERISTICS AND CAUSES

The following list of frequently encountered deficiencies, together with the most likely causes, will be useful to the in-process or final-product inspector or tester:

- | | |
|--|---|
| 1. Annular ring (failure to meet minimum requirements) | Unstable artwork |
| Design | Overetched |
| Drilling accuracy | Worn tooling plates and registration pins |
| Improperly registered layers | Reduction of artwork improper |
| Lay-up pinning | |

- | | | |
|------------------------|---|--|
| 2. Delamination | CAUSES:
Aged prepreg
Low temperature and pressure during lamination
Moisture
Inadequate baking
Uncontrolled lay-up area (temperature and humidity)
Silicone release agent | Improper heat transfer through stack
Handling during lay-up
Improperly cleaned layers
Contamination
Improperly vented ovens
Temperature control during assembly
Excessive antistaining agent on copper |
| 3. Dewetting | Contaminated surfaces
Contaminated reflow media
Improper fluxing
Oxides
Porous copper that may result in volatile migration
Improper angle of product into fusing oil
Inadequate volume of fusing media resulting in temperature variation
Improper heat during reflow | |
| 4. Inadequate etchback | Incorrect agitation
Improper lamination and curing
Nail heading
Hardened epoxy smear
Unbalanced and/or depleted bath | Improper bath temperature
Improper time exposure
Presence of Novalac in Epoxy resin
Incompatibility of materials |
| 5. Laminate voids | Improperly cured resin
Improper flow of resin
Border barriers
Too little prepreg to allow fill | Temperature and/or pressure improper
Poor material selection |
| 6. Mealing | Insufficient removal of fluxes
Improper cleaning | Moisture entrapment
Hygroscopic coating |
| 7. Measling | Moisture
Pressure
Internal stresses | Volan in lieu of silane glass treatment
Excessive heat
Circuit design geometry |
| 8. Misregistration | Dimensional stability of base materials
Registration of artwork
Care in punching layers and artwork | Out-of-parallel laminating plates
Inadequate pinning
Unstable artwork
Worn or deformed holes in artwork |
| 9. Plating adhesion | Improperly cleaned surface
Resist residue
Imbedded pumice on surface | Out-of-control electroless or under-plating process
Fingerprints
Silicones |
| 10. Plating, cracks | Brittle copper
Excessive brightness
Excessive levelers
Resin content contributing to z expansion | Contaminated copper bath
Improper current density
Design
Deletion of nonfunctional pads
Excessive etchback resulting in etching during plating |
| 11. Plating, peeling | Improperly cleaned surface
Pumice imbedded | Careless handling during processing
Plated at improper current density
High-stress plating |
| 12. Plating voids | High current density
Inadequacies in electroless copper process | Depleted electroless bath
Particles of photoresist not removed
Improper anode displacement
Improper agitation (cavitation) |

- | | |
|--|---------------------------------------|
| Starting boards with no copper strike following electroless deposition | Contamination in bath |
| 13. Resin starvation | Unbalanced plating solution |
| Improper mix of resin | Excessive pressure during lamination |
| Laminating plates out of parallel | Resin content |
| 14. Slivers | Weave count of fabric |
| Excessive overhang | Overetched |
| 15. Solderability | Brittle overplate |
| Oxidation | Porous copper |
| 16. Weave Exposure | Photoresist residue |
| Inadequate butter coat | Fabric size too large for end product |
| Excessive pressure | Thickness |
| Overexposure to electroless tin | Careless workmanship practices |

14. Major Defects The most significant major defects which relate to the boards are listed here as an aid in the evaluation of finished printed wiring assemblies:

- Solder bridging between adjacent circuits or excessive protrusions or peaks.
- Holes in the board which are not specified on the assembly drawing.
- Corrosion, discoloration, or both on the metal surfaces of boards.
- Softening, blistering, or discoloration of the board.
- Inadequate coverage of conformal coating on the printed wiring assembly.
- Conformal coating on areas which are specified to be free of it.
- Flux residues and contaminants on uncoated printed wiring assemblies.
- Pinholes or small spots of copper showing on conductors that exceed 5 percent of the conductor area.
- The presence of conformal coating on the adjustable portion of adjustable components.
- The presence of conformal coating on electrical and mechanical mating surfaces such as connector contact points, screw threads, and bearing surfaces.
- Incompatibility of the conformal coating with the other parts of the printed wiring assembly.
- Construction, masking, or protection of printed wiring assemblies inadequate to prevent degradation of the electrical properties by the conformal coating.
- Unauthorized repair. (This pertains to repairs but not to rework or touch-up.)
- Degradation of the mechanical, electrical, or physical properties of the printed wiring assembly caused by repair operations.
- Circuit boards that are charred, burned, blistered, chipped, gonged, separated, or otherwise damaged.
- Physical damage or change in rating of components resulting from the straightening, cutting, bending, insertion, or clinching of wire leads.
- Component parts not securely supported on the board.

15. Minor Defects

- Wrong flux
- Nonuniformity of appearance of the printed wiring assembly.
- Inadequate cleanliness of the printed wiring assembly as evidenced by the presence of dirt, foreign matter, oil, fingerprints, corrosion, salts, flux residues, and contaminants.
- Component polarity markings not visible.

TRACEABILITY

Because of the possibility of latent defects, it is essential that the fabricator establish and maintain a traceability program for all boards used in high-reliability programs. The traceability program must begin at the lamination panel level and continue to reflect the exact disposition of each board from that panel, including the serial number of the equipment in which the board is located. A record of rejected boards should be kept, and the reason for rejection should be identified. The program may be set up in such a manner that the data can be sorted and compiled by electronic data processing.

RELIABILITY TESTING

Although reliability testing is normally thought of as a means of testing discrete components on printed wiring assemblies, there is merit to evaluating the exposed boards for plated-through-hole quality.

Figures 18.4 to 18.6 depict three recommended reliability tests. The tests are recommended in sequence.

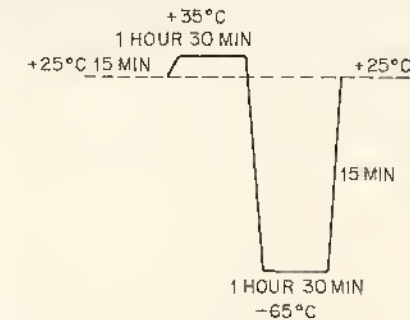
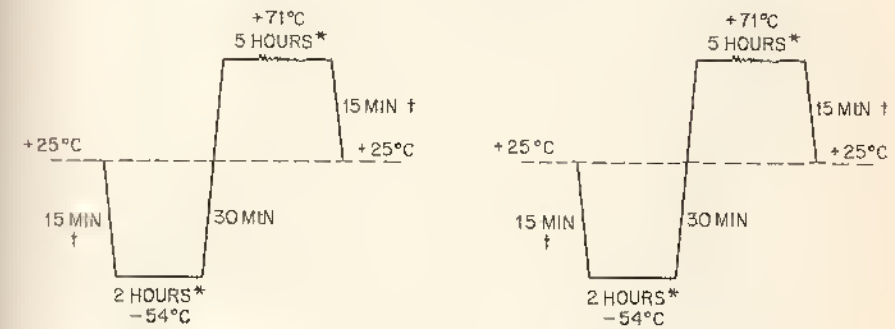


Fig. 18.4 Fast burn-in cycle.



* 10-MIN. VIBRATION EACH HOUR AT 2-2 g, 20-60 Hz (NONRESONANT)

† VOLTAGE CYCLING 90-110% DURING THESE TRANSITIONS.

* 10-MIN. VIBRATION EACH HOUR AT 2-2 g, 20-60 Hz (NONRESONANT)

† VOLTAGE CYCLING 90-110% DURING THESE TRANSITIONS.

Fig. 18.6 Reliability demonstration test; identical to regular burn-in, except that there are 150 cycles and tests are done on a sampling basis.

APPENDIXES

A. PRODUCTION, INSPECTION, AND TESTING CHECKLIST

1. Cut laminate material to size
2. Stamp copper thickness (two sides)
3. Pumice-scrub
4. Chemically clean surface
5. Degrease
6. Photoresist application
7. Print pattern (two sides)
8. Develop pattern (two sides)

- _____ 9. Touch up layers (two sides)
- _____ 10. Etch layers
- _____ 11. Touch up
- _____ 12. Pumice-scrub
- _____ 13. Chemically clean surface
- _____ 14. Inspect layers
- _____ 15. Store in lay-up room in plastic container
- _____ 16. Prebake material
- _____ 17. Cut prepreg material to size
- _____ 18. Punch prepreg material
- _____ 19. Prebake material
- _____ 20. Lay up board per lay-up
- _____ 21. Inspect lay-up (X-ray)
- _____ 22. Laminate boards (use appropriate fixture)
- _____ 23. Trim edges of excess resin
- _____ 24. Inspect laminated board (trim decal)
- _____ 25. Pumice-scrub
- _____ 26. Drill holes
- _____ 27. Etch-back holes
- _____ 28. Vapor-blast
- _____ 29. Inspect holes
- _____ 30. Pumice-scrub
- _____ 31. Electroless copper-plate
- _____ 32. Copper-plate
- _____ 33. Inspect plating (microsection)
- _____ 34. Sand board (both sides)
- _____ 35. Pumice-scrub
- _____ 36. Chemically clean board
- _____ 37. Degrease
- _____ 38. Apply resist
- _____ 39. Print image
- _____ 40. Develop image
- _____ 41. Touch up
- _____ 42. Electroplate
- _____ 43. Inspect plating
- _____ 44. Remove photoresist or ink
- _____ 45. Blacklight inspection (photoresist)
- _____ 46. Etch
- _____ 47. Touch up
- _____ 48. Reflow, if solder plated
- _____ 49. Blank, pierce, and notch (machine if desired)
- _____ 50. Test boards
- _____ 51. Inspection manufacturing
- _____ 52. Mark screen
- _____ 53. Serialize
- _____ 54. Inspection manufacturing
- _____ 55. Inspection quality control
- _____ 56. Pack

B. REFERENCE AND GOVERNMENT SPECIFICATIONS

- MIL-P-18177 Plastic Sheet, Laminated, Thermosetting, Glass Fiber Base, Epoxy-Resin
- MIL-P-13949 Plastic Sheet, Laminated, Metal Clad for Printed Wiring
- MIL-S-45743 (M1) Soldering, Manual Type
- MIL-P-46843 (M1) Printed Circuit Assemblies, Design and Production of
- MIL-S-46844 Solder Bath Soldering of Printed Wiring Assemblies, Automatic Machine Type
- MIL-P-55110 Printed Wiring Boards

- MIL-P-55424 Printed Wiring Boards, Multilayer
- MIL-P-55561 Foil, Copper, Cladding for Printed Wiring Boards
- MIL-P-55617 Plastic Sheet, Thin Laminate, Metal Clad
- MIL-P-55636 Glass Cloth, Resin Preimpregnated (B-Stage)
- MIL-P-55640 Printed Wiring Boards, Multilayer (Plated-Through-Hole)
- MIL-P-82585 (OS) Printed Wiring Boards
- MIL-I-46058 Insulation Compound, Electrical (for Coating Printed Circuit Assemblies)
- MIL-Y-1140 Yarn, Cord, Sleeving, Cloth, and Tape-Glass
- MIL-T-10727 Tin Plating, Electrodeposited or Hot-Dipped
- MIL-F-14256 Flux, Soldering, Liquid (Rosin Base)
- MIL-C-14550 Copper Plating, Electrodeposited
- MIL-C-45204 Gold Plating, Electrodeposited
- MIL-L-13808 Lead Plating, Electrodeposited
- MIL-R-46085 Rhodium Plating, Electrodeposited
- MIL-P-81728 Tin-Lead, Electrodeposited
- MIL-STD-130
- MIL-STD-202 Test Methods for Electronic and Electrical Components
- MIL-STD-275 Printed Wiring for Electronic Equipment
- MIL-STD-429 Printed Circuit Terms and Definitions
- MIL-STD-454 Standard General Requirements for Electronic Equipments
- MIL-STD-781 Reliability Tests: Exponential Distribution
- MIL-STD-810 Environmental Test Methods
- MIL-STD-883 Test Methods and Procedures for Microelectronics
- MIL-STD-1495
- QQ-S-571 Solder, Tin, Alloy, Lead-Tin Alloy, and Lead Alloy
- QQ-N-290 Nickel Plating, Electrodeposited
- L-T-90 Tape, Pressure, Sensitive
- FED-STD-151 Metals: Test Methods
- FED-STD-406 Plastics: Test Methods

Chapter 19

Acceptability of Fabricated Circuits

A. D. ANDRADE

Sandia Corporation, Livermore, California

Inspection Operations	19-2
Use of Test Patterns	19-2
How to Determine What is Acceptable or Rejectable	19-3
The Materials Review Board	19-3
Visual Inspection	19-3
1. Surface Defects	19-4
2. Base Material Defects	19-6
3. Epoxy Smear	19-8
4. Layer-to-Layer Registration, X-ray Method	19-8
5. Plated-through Holes: Roughness and Nodulation	19-8
6. Eyelets	19-8
7. Base Material Edge Roughness	19-10
8. Visual Inspection	19-10
Dimension Inspection	19-10
9. Annular Ring	19-11
10. Pattern-to-Hole Registration	19-12
11. Conductor Width	19-12
12. Conductor Spacing	19-13
13. Edge Definition	19-14
14. Hole Specifications	19-14
15. Warp and Twist	19-14
16. Conductor Pattern Integrity	19-15
17. Contour Dimensions	19-16
18. Plating Thickness	19-16
19. Undercut	19-20
20. Overhang	19-21
21. Etchback	19-21
22. Layer-to-Layer Registration	19-22
23. Flush Printed Circuits	19-23

24. Dimensional Inspection	19-23
Mechanical Inspection	19-23
25. Plating Adhesion	19-23
26. Solderability	19-24
27. Alloy Composition	19-25
28. Thermal Stress, Solder Floal Test	19-26
29. Peel Strength	19-26
30. Plated-through-Hole Structure	19-26
31. Terminal Pull	19-26
32. Mechanical Inspection	19-26
Electrical Inspection	19-27
33. Continuity	19-27
34. Insulation Resistance	19-27
35. Current Breakdown, Plated-through Holes	19-28
36. Current-Carrying Capacity	19-28
37. Dielectric Withstanding Voltage	19-28
38. Electrical Inspection	19-29
Environmental Inspection	19-29
39. Thermal Shock	19-29
40. Moisture Resistance	19-29
41. Outgassing	19-29
Summary	19-30
Test Specifications and Methods Related to Printed Boards	19-30
General Specifications Related to Printed Boards	19-31
References	19-32

INSPECTION OPERATIONS

Quality assurance operations can be performed by the purchaser or contracted out to an independent company. Wherever it is performed, however, there must be access to a facility equipped with the required types of mechanical gauges, as well as microsectioning, chemical analysis, dimensional measuring, and electrical and environmental test equipment called for in these procedures.

Quality assurance requirements are usually met by one of the following methods:

1. Inspection data submitted by the fabricator are reviewed for compliance to design requirements.
2. Inspection data submitted by the fabricator are reviewed and a sample lot inspection is performed.
3. A complete inspection is performed to all design requirements including destructive testing.

Where and how the inspections are performed is a question of economics and availability of experienced personnel. Management should review all aspects of the question, such as equipment cost and maintenance, work volume, time to and from quality assurance facility location, and personnel availability, prior to making a decision.

USE OF TEST PATTERNS

The use of test patterns located outside the finished circuit border area to monitor all processes and also be utilized for microsectioning is a controversial subject. Some persons state that test patterns located outside the border area usually have thicker plating thicknesses than the actual printed circuit pattern and, therefore, are not representative of the actual pattern. Others debate that the cost savings of not destroying actual circuits overshadow any slight differences in plating thicknesses. Test patterns are useful mechanisms for performing destructive tests such as peel strength, terminal pull strength, flammability, and solder shock as an indication of the integrity of the finished printed boards.

The use of test patterns for the acceptance of printed boards is therefore an individual

question and must be resolved for each design requirement and the economic advantage of various inspection operations.

HOW TO DETERMINE WHAT IS ACCEPTABLE OR REJECTABLE

The question of what is acceptable or rejectable must be answered for individual printed board design functions. "Functionality" should be the ultimate criterion for acceptance. A printed board which will see extreme environmental conditions should not be inspected to the same requirements as a printed board for an inexpensive radio. Some of the characteristics normally inspected come under the heading of workmanship. In the majority of designs those characteristics are cosmetic and pertain to how the printed board looks and not how it performs. In most cases, the characteristics are inspected to establish a confidence level for integrity. The inspection results should be used to establish a fabrication quality level and not to scrap parts that will meet the functional criteria. Scrapping functional parts has a considerable impact on unit cost. However, the same workmanship characteristics can have an effect on function of some designs, and the effects of the different characteristics will be discussed later in this chapter. The determination of printed boards that do not meet specified acceptance requirements but are functional should be made by a materials review board. The levels of acceptance have to be established by each company; they are dependent on the functional criteria to which the printed board will be subjected.

IPC has established recommended guidelines for acceptance levels categorized as preferred, acceptable, and reject. The acceptance guidelines in this chapter, however, utilize only two categories: preferred and nonpreferred. The illustrations in the nonpreferred category are typical examples of rejectable conditions. They will guide individuals in establishing minimum acceptance levels applicable to their functional requirements.

THE MATERIALS REVIEW BOARD

The materials review board (MRB) is usually comprised of one or more representatives from the departments of quality, production, and design. The purpose of the materials review board is to effect positive corrective action to eliminate the cause of recurring discrepancies and prevent occurrence of similar discrepancies. The board's responsibilities include:

1. Reviewing questionable printed boards or materials to determine compliance or noncompliance with quality and design requirements.
2. Reviewing discrepant boards for effects on design functionality.
3. Authorizing repair or rework of nonconforming materials, when appropriate.
4. Establishing responsibility and/or identifying causes for nonconformance.
5. Authorizing scrapping of excessive quantities of materials.

VISUAL INSPECTION

Visual inspection is the inspection of characteristics which can be seen in detail with the unaided eye. Magnification, approximately 10 \times , is advantageous for viewing questionable characteristics after their initial location with the unaided eye.

The viewing of printed board microsections for defects is the exception. Defects such as epoxy smear and plated-through-hole quality require magnifications of 50 \times to 350 \times , depending on the characteristic. Visual inspection criteria are difficult to define grammatically, because individuals interpret words differently.

An effective way to define visual inspection criteria is to use line illustrations or photographs. IPC utilized this method in the publication "Acceptability of Printed Wiring Boards Manual" to "visually standardize the many individual interpretations to specifications on printed wiring boards."

Of the different types of inspection, visual inspection costs the least. Visual inspection by unaided eye is usually performed on 100 percent of the printed boards or on a sample taken following an established sampling plan. MIL-STD-105, Sampling Procedures and Tables for Inspection by Attributes, is frequently used for this purpose. Inspection for visual defects, following a sampling plan, is done on the premise that the boards were 100 percent visually screened during the fabrication process.

Defects which usually can be detected by visual inspection may be divided into three groups: surface defects, base material defects, and other defects.

1. Surface Defects Surface defects include dents, pits, scratches, surface roughness, voids, pinholes, inclusions, and markings. Dents, pits, scratches, and surface roughness usually fall in the class of workmanship. The defects, when minor, are normally considered to be cosmetic, and they usually have little or no effect on functionality. However, they can be detrimental to function in the plated contact area (Fig. 19.1).

Voids in conductors, terminal areas, and plated-through holes can be detrimental to function depending on the degree of defect. Pinholes and inclusions are in the same category. Voids or pinholes either of which reduce the effective conductor width, reduce current-carrying capacity. Voids in the hole wall plating are a result in reduced conductivity, increased circuit resistance, and voids in plated-through hole solder fillets (Fig. 19.2).

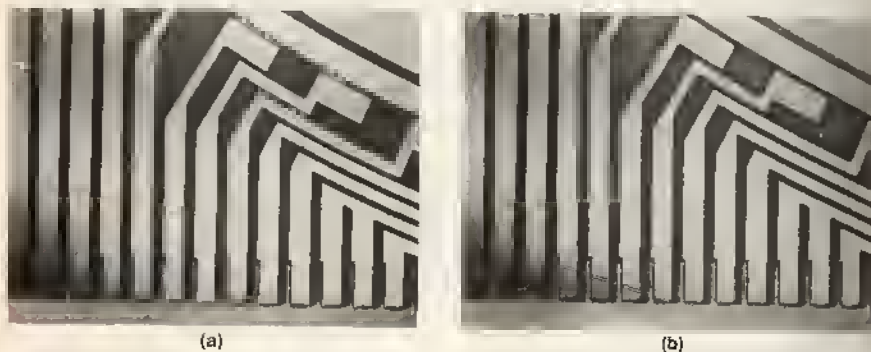


Fig. 19.1 Plated contact area. (a) Preferred: plated contact area free of delamination, pits, pinholes, dents, nodules, and scratches. (b) Nonpreferred: (1) scratch depth at the contact area exceeds the microinch surface roughness requirements; (2) delamination of one of the plated contacts (IPC).

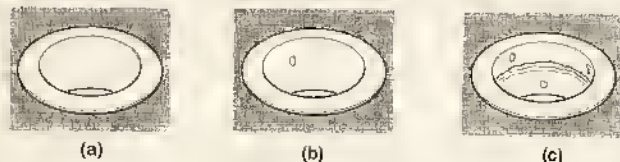


Fig. 19.2 Voids in hole. (a) Preferred: no voids in hole. (b) Acceptable: no more than three voids in the hole; total void area does not exceed 10 percent of the hole wall area. (c) Nonpreferred: voids exceed 10 percent of the hole wall plating area; circumferential void present (IPC).

Voids in terminal areas are also detrimental to solderability. Pinholes or voids can undermine the top metal plate (Figs. 19.3 and 19.4). The degree of undermining depends on when during the fabrication process the defect occurred.

The defects in this group are defined as follows:

- 1 **Dents.** Depressions which do not significantly decrease the surface thickness.
- 2 **Pits.** Small holes occurring as imperfections which do not penetrate entirely through the metal foil.
- 3 **Scratch.** Slight surface marks or cuts.
- 4 **Surface roughness.** Not smooth or level, having bumps, projections, etc.
- 5 **Voids.** Absence of a deposited substance in a localized area. Base material exposed.
- 6 **Pinholes.** Imperfections occurring as small holes which penetrate entirely through the metal foil.
- 7 **Inclusions.** Foreign particles in the conductive layer or plating and/or the base material. Inclusions in the conductive pattern, depending on degree and material, can affect plating adhesion. Inclusions in a conductor greater than 0.005 in (0.127 mm) in their greatest dimension are usually cause for rejection, but inclusions less than 0.0001 in (0.00254 mm) in their largest dimension are usually allowed. Metallic inclusions in the base material reduce the electrical insulation properties and are not normally acceptable.

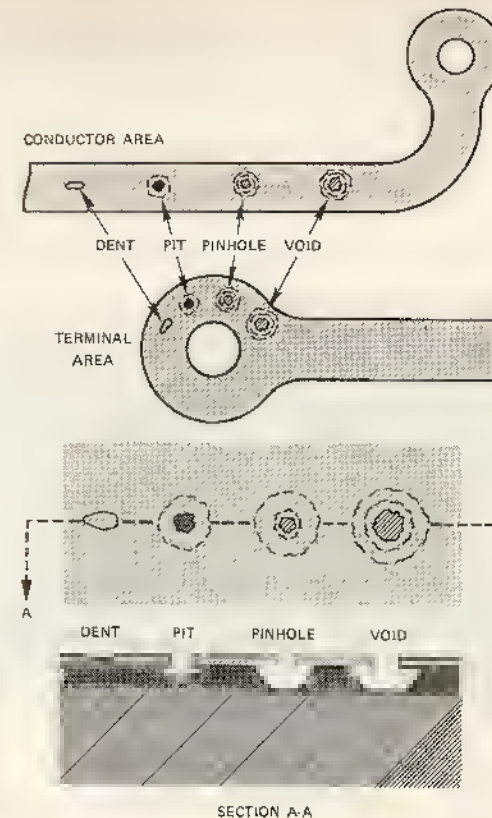


Fig. 19.3 Pits, dents, pinholes, and voids (Sandia Laboratories).

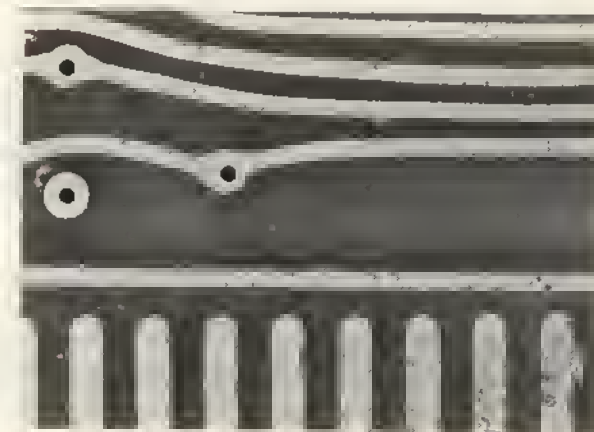


Fig. 19.4 Severe pitting and pinholing (IPC).

8 **Markings (legend).** Marking is a method of distinguishing printed boards with identifying information such as part number and revision letter. It can be the origin of another surface defect. The condition is usually considered minor; but if there are different revisions to the same part number, markings that are missing or partially obscured could have an effect on functionality (Fig. 19-5).

2. **Base Material Defects** Visual inspection is also used to detect the following base material defects: measling, crazing, blistering, delamination, weave texture, weave exposure, fiber exposure, and haloing. (Fig. 19-6). These defects have been a source of controversy as to what is good or what is bad. IPC formed a special committee in 1971 to consider base material defects and to better define them with illustrations and photographs. The conditions are defined and discussed here.

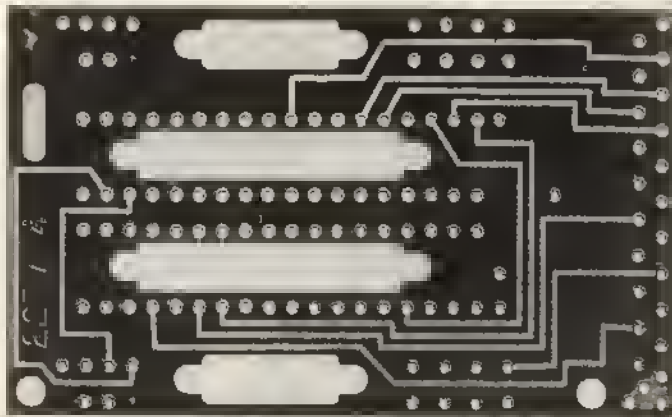


Fig. 19.5 Part numbers partially obscured and missing (Sandia Laboratories).

1 **Measling.** A condition existing in the base laminate in the form of discrete white spots or crosses below the surface of the base laminate. It reflects a separation of fibers in the glass cloth at the weave intersection.

A report compiled by IPC, titled "Measles in Printed Wiring Boards," was released in November 1973. The report stated that "measles may be objectionable cosmetically, but their effect on functional characteristics of finished products are, at worst, minimal and in most cases insignificant."

2 **Crazing.** A condition existing in the base laminate in the form of connected white spots or crosses on or below the surface of the base laminate. It reflects the separation of fibers in the glass cloth and connecting weave intersections.



Fig. 19.6 Base material defects: (a) measling, (b) blistering, (c) fiber exposure, and (d) crazing (IPC).

When two or more measles are interconnected, the defect is then termed "crazing." The condition can affect the printed board function if it bridges between two plated-through holes. Crazing is considered to be an internal separation within a fiber bundle. Moisture or other corrosive residues could be trapped in the crazed area and reduce electrical insulation.

3 **Blistering.** A localized swelling and separation between any of the layers of the base laminate and/or between the laminate and the metal cladding.

4 **Delamination.** A separation between any of the layers of the base laminate, and/or between the laminate and the metal cladding, originating from or extending to the edges of a hole or edge of the board.

Blistering and delamination are considered to be major defects. Whenever a separation of any part of the board occurs, a reduction in insulation properties occurs. The separation

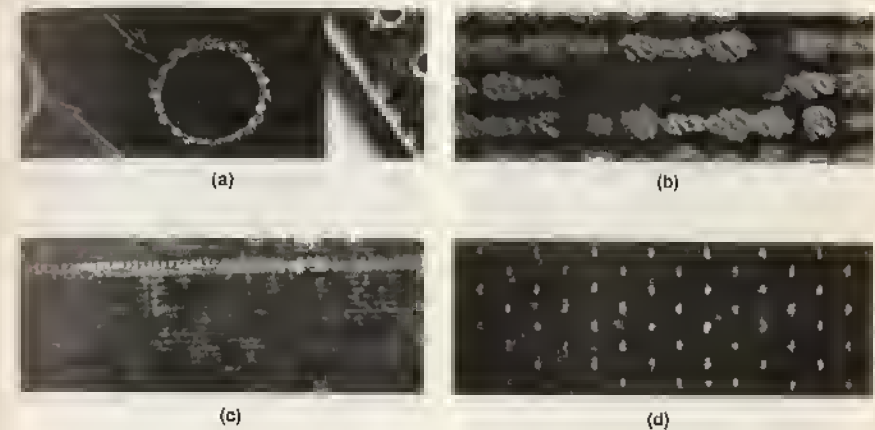


Fig. 19.7 Base material defects: (a) haloing, (b) weave exposure, (c) delamination, and (d) weave texture (IPC).

area could also house entrapped processing solutions that contribute to corrosion and other detrimental effects in certain environments (Fig. 19.6). There also is the possibility the delamination or blister area will increase to the point of complete board separation. Last, but not least, is the question of solderability in plated-through holes. Entrapped moisture, when subjected to soldering temperatures, has been known to create steam that blows holes into the sidewalls of the plated-through holes and creates large voids in the solder fillet.

5 **Weave texture.** A surface condition in which the unbroken fibers of the woven glass cloth are completely covered with resin but exhibit the definite weave pattern of the glass cloth.

6 **Weave exposure.** A surface condition in which the unbroken woven glass cloth is not uniformly covered by resin. Weave texture and weave exposure differ in the degree of defect (Fig. 19.7). A condition of weave texture after the board has been completely fabricated is considered a minor defect. However, if the condition materializes during processing, a judgment must be made concerning the possible attack of subsequent processing chemicals. Weave texture, usually caused by the lack of sufficient resin, can become weave exposure if processing chemicals do attack the thin resin layer.

Weave exposure is considered a major defect. The exposed glass fiber bundles allow wicking of moisture and entrapment of processing chemical residues.

7 **Fiber exposure.** A condition in which broken glass cloth fibers are exposed on machined or abraded areas. The effect on printed board operation is the same as that of weave exposure.

8 **Haloing.** A base laminate condition consisting of a light area around a hole or other machined area on or below the surface of the base laminate. Haloing is usually a condition of edge delamination either at the board edge or at the edges of non-plated-through holes. The open fissures are a potential source of electrical problems; they can entrap moisture or corrosive chemical salts. In either case a definite defect is present, but the depth of the

defect must be taken into consideration. Some board users will allow up to 0.050 in (1.27 mm) on a limited scale. The end use of the board must be taken into consideration before halving can be labeled a major or minor defect.

3. Epoxy Smear The condition known as epoxy smear is caused by excessive heat, generated during drilling, softening the epoxy in a hole and smearing it over exposed internal copper areas. The condition creates an insulator between the internal terminal areas and subsequent plated-through holes, and the result is "open circuits." The defect is removed by chemical cleaning. Inspection for epoxy smear is performed by viewing vertical and/or horizontal microsections of plated-through holes.

Chemical cleaning is the chemical process used in the manufacture of multilayer boards. Its purpose is to remove only epoxy from conductive surfaces exposed in the inside of the holes (Fig. 19.8). Sulfuric or chromic acid is used in the chemical cleaning process; subsequent inspection indicates whether it has fulfilled its purpose.

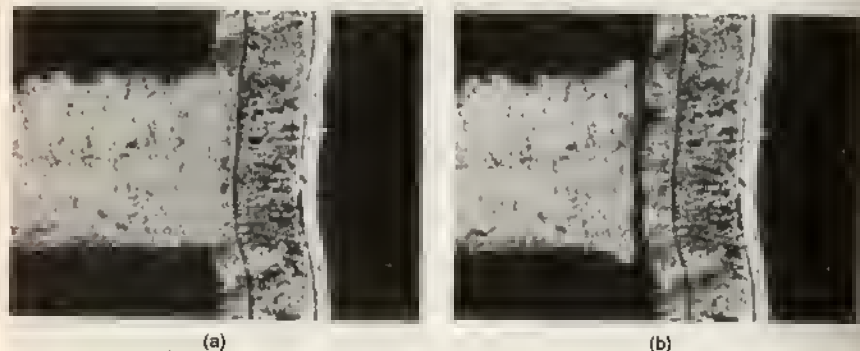


Fig. 19.8 (a) Preferred: no evidence of epoxy smear between layer and plating in the hole. (b) Non-preferred: evidence of epoxy residue or epoxy smear between internal layer and plating in the hole (IPC).

4. Layer-to-Layer Registration, X-ray Method The X-ray method provides a nondestructive way to inspect layer-to-layer registration of internal layers of multilayer boards. It utilizes an X-ray machine and, usually, Polaroid film. The multilayer board is X-rayed in a horizontal position. The X-ray photos are then examined for hole breakout of the internal terminal areas. The lack of an annular ring denotes severe misregistration (Fig. 19.9).

5. Plated-through Holes: Roughness and Nodulation Roughness is an irregularity in the sidewall of a hole; nodulation is a small knot or an irregular, rounded lump. Roughness and/or nodulation creates one or more of the following conditions:

1. Reduced hole diameter
2. Impaired lead insertion
3. Impaired solder flow through the hole
4. Voids in solder fillet
5. Possible entrapment of contaminants
6. Highly stressed areas in the plating

Although roughness and nodulation are not desirable, they are allowable in small amount. Specifications have a tendency to use simple, generalized statements such as "good uniform plating practice" when defining acceptability criteria. Such statements require that a judgment be made by the inspector as to what is acceptable or rejectable. The use of visual aids allows judgments to be made by different inspectors within a close degree of consistency. Figure 19.10 illustrates the acceptance criteria recommended by the Institute of Printed Circuits.

6. Eyelets Metallic tubes, the ends of which can be bent outward and over to fasten them in place, are called eyelets. Eyelets are used to provide electrical connections with mechanical strength on printed boards. Acceptability of eyeleted printed boards is based on eyelet installation. Eyeleted boards should be inspected for the following conditions:

1. Form of the flange and/or roll should be set in a uniform spread and be concentric to the hole.
2. Splits in the flange or roll should be permissible provided they do not enter the barrel and provided they allow proper wicking of solder through the eyelet and around the setting.
3. Eyelets should be set sufficiently tight that they cannot move.
4. Eyelets should be inspected for improper installation, deformations, etc.
5. A sample lot of eyeleted holes should be microsectioned to inspect for proper installation.

Institute of Printed Circuits acceptance criteria for roll and flange eyelets are shown in Figs. 19.11 and 19.12. (See pages 19-10 and 19-11, respectively.)

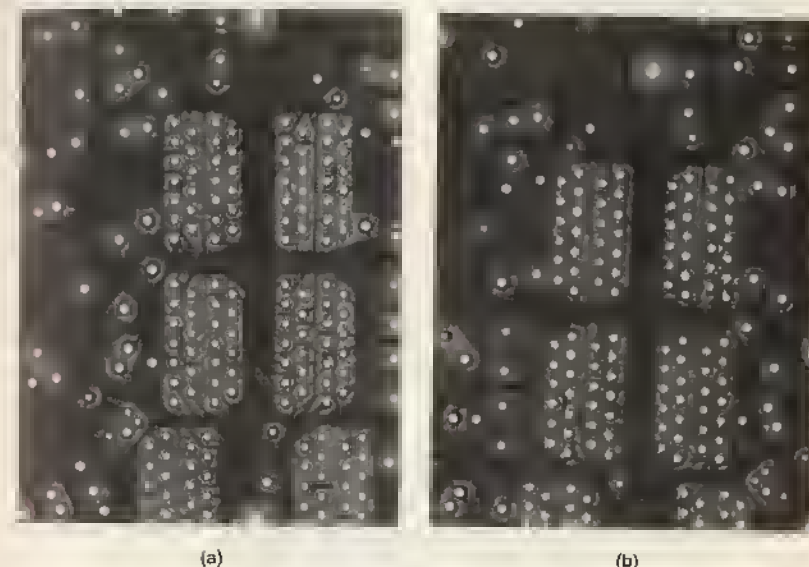


Fig. 19.9 Layer-to-layer registration, X-ray method. (a) Preferred: all layers accurately registered. (b) Non-preferred: extreme misregistration; insufficient measurable annular ring exists on a segment of circumference (IPC).

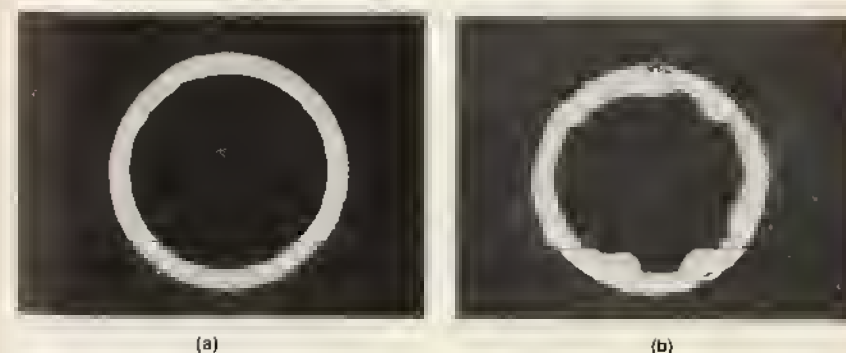


Fig. 19.10 Plated-through-hole roughness or nodulation. (a) Preferred: (1) plating is smooth and uniform throughout the hole; (2) no evidence of roughness or nodulation. (b) Non-preferred: (1) roughness or nodulation reduces plating thickness below minimum requirements; (2) roughness or nodulation reduces finished hole size below minimum requirements; (3) excessive roughness or nodulation permits outgassing of the hole when it is solder dipped (IPC).

7. Base Material Edge Roughness Base material edge roughness occurs on the printed board edge, cutouts, and non-plated-through-hole edges (Fig. 19-13). It is classified as a workmanship condition, and it is created by dull cutting tools causing a tearing action instead of a clean cutting action.

8. Visual Inspection The board attributes subjected to nondestructive and destructive visual inspection are listed in Table 19.1.

TABLE 19.1 Visual Inspection Chart

Subject	Nondestructive	Destructive
Dents	x	
Pits	x	
Scratches	x	
Voids	x	x
Pinholes	x	
Inclusions	x	
Surface roughness	x	
Markings	x	
Measling	x	
Crazing	x	
Blistering	x	
Delamination	x	
Weave texture	x	
Weave exposure	x	
Haloing	x	
Chemical cleaning		x
Layer-to-layer registration, X-ray method	x	
Plated-through-hole roughness and modulation	x	x
Eyelets	x	x
Base material edge roughness	x	

DIMENSIONAL INSPECTION

Dimensional inspection is the measurement of the printed board attributes such that dimensional values are necessary to determine compliance with functional requirements. The methods of inspection vary, but the basic inspection equipment consists of gages and measuring microscopes. More sophisticated equipment is available; it includes comparators, numerical control measuring equipment, coordinate measuring systems, micro-ohm meters, and beta backscatter gages.

Dimensional inspection is usually performed on a sampling plan basis. One such plan, termed acceptable quality level (AQL), is specified as the maximum percent of defects

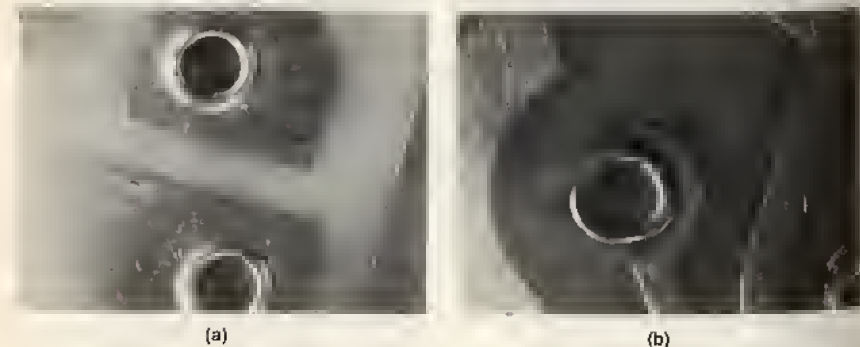


Fig. 19.11 Roll flange eyelets. (a) Preferred: (1) eyelet set uniformly and concentric to hole; (2) strain or stress marks caused by roll-over kept to a minimum. (b) Nonpreferred: (1) eyelet flange uneven or crushed; (2) splits entering the barrel (IPC).

which, for the purpose of sampling, can be considered statistically satisfactory for a given product. The following attributes fall in the category of dimensional inspection.

9. Annular Ring The circular strip of conductive material completely surrounding a hole is called the annular ring (Fig. 19.14). The primary purpose of the annular ring is that of a flange surrounding a hole; it provides an area for the attachment of electronic component leads or wires.

Annular ring width of 0.010 in (0.254 mm) is a standard requirement, but some

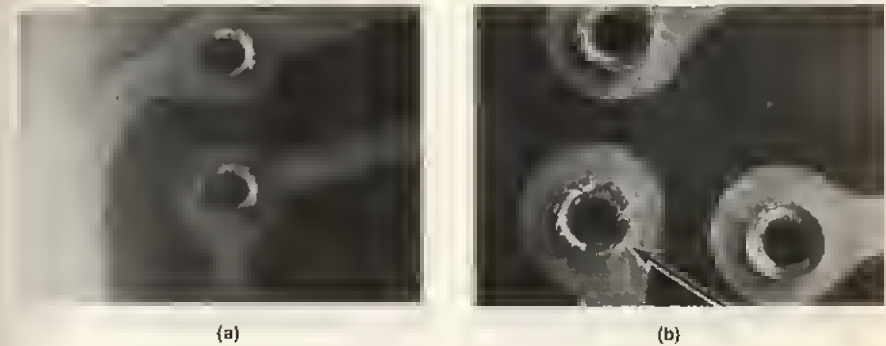


Fig. 19.12 Funnel flange eyelets. (a) Preferred: (1) funnellet set uniformly and concentric to hole; (2) strain or stress marks caused by setting kept to a minimum. (b) Nonpreferred: (1) funnellet periphery uneven or jagged; (2) splits enter into barrel (IPC).

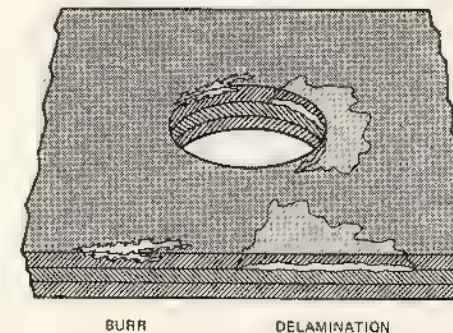


Fig. 19.13 Base material edge roughness (Sandia Laboratories).

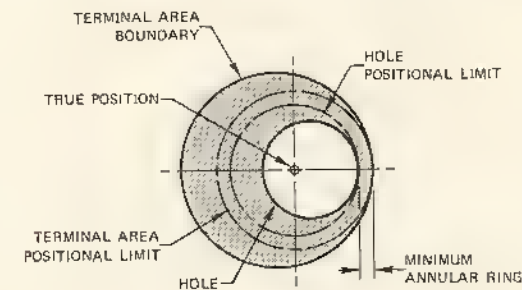


Fig. 19.14 Annular ring (Sandia Laboratories).

specifications have allowed rings as small as 0.005 in (0.127 mm). Figure 19.15 shows holes neatly centered in the terminal area, and shows holes on the extreme edge of the terminal area and holes breaking the edge. Holes that extend beyond the terminal area are generally not acceptable.

10. Pattern to Hole Registration The annular ring can also be used for determining registration between the pattern and the holes (Fig. 19.16). Some designers dimension a terminal area to each datum on the master drawing. By verifying the dimensions on the printed boards and then verifying that the minimum widths of the annular rings, on all

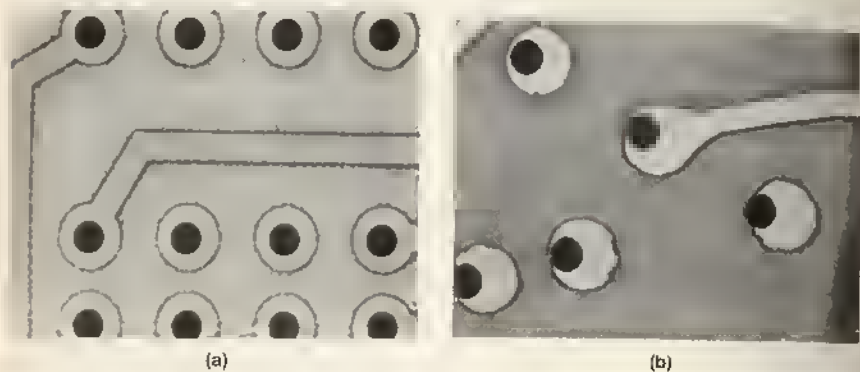


Fig. 19.15 Terminal area registration. (a) Preferred: holes neatly centered in the terminal area. (b) Nonpreferred: holes not centered in the terminal area (IPC).

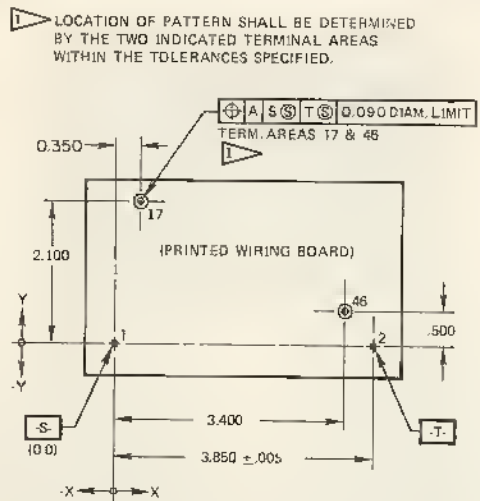


Fig. 19.16 Terminal area to datum registration (Sandia Laboratories).

other terminal areas, are within the drawing requirements, the pattern is considered to be in registration with the drilled hole location. Front-to-back registration is also inspected in this manner.

11. Conductor Width The conductor width affects the current-carrying capacity of the conductor. A decrease in conductor width decreases the current carrying capacity and increases electrical resistance. Most current carrying capacity graphs take the fabrication process into consideration and adjust the current carrying capacity to allow for a margin of safety.

Although the conductor width definition is very basic, there are two different interpretations as to where on the conductor the measurement is performed. (1) The minimum conductor width is measured at the minimum width of the conductor (Fig. 19-17) and (2) the conductor width is the observable width of the conductor at any point chosen at random on the printed board when ready for delivery and viewed vertically from above (Fig. 19.18). The minimum conductor width is always measured. Usually it can be measured only on a conductor cross section, and it is a destructive inspection. Plating growth, during pattern plating, can prevent the minimum width from being seen unless a cross section is performed.

The conductor width is always nondestructive and easily measured; see Fig. 19-19.

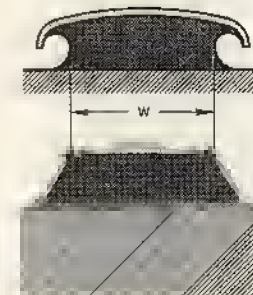


Fig. 19.17 Minimum conductor width (Sandia Laboratories).

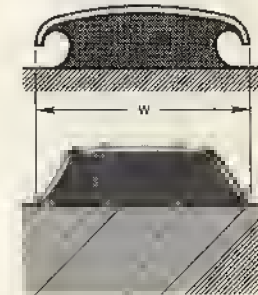


Fig. 19.18 Conductor width (Sandia Laboratories).

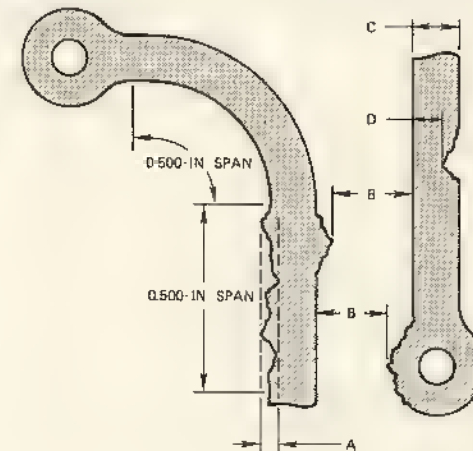


Fig. 19.19 Edge definition (Sandia Laboratories).

dimension C. The difference between the minimum width and the point which is measured vertically can have an effect on the current-carrying capacity and the impedance. Therefore, check the design drawing notes and associated specifications to determine which conductor width measurement method is required.

12. Conductor Spacing The distance between adjacent conductors on any single layer of a printed board, is defined as conductor spacing. The spacing between conductors and/or terminal areas is designed to allow sufficient insulation between circuits. A reduction in the spacing can cause electrical leakage or affect the capacitance. The cross-sectional width of conductors is usually nonuniform. Therefore, the spacing measurement is taken at the closest point between the conductors and/or terminal areas; see Fig. 19-19, dimension B.

13. Edge Definition The fidelity of reproduction of pattern edge relative to the original master pattern is called edge definition. It falls in the cosmetic effects category and does not normally affect functionality. It can, however, have an effect on high-voltage circuits: a corona discharge may be caused at the irregular conductor edges.

Measurement of edge definition is performed by measuring the distance from the crest

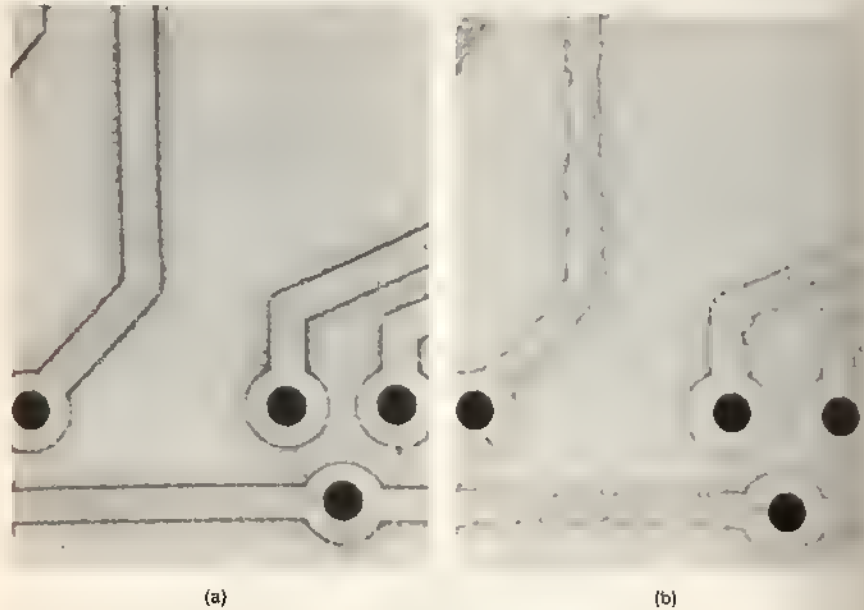


Fig. 19.20 Edge definition (a) Preferred: conductor edges are smooth and even within tolerance. (b) Nonpreferred: conductor edges are poorly defined and outside tolerance (IPC).

to the trough; see Fig. 19.19, dimension A. A popular specification is 0.005 in (0.127 mm) crest to trough. Isolated indentations which do not reduce the conductor width by more than 20 percent are usually allowed. See Fig. 19.19, dimension D. Also allowed are isolated projections which do not reduce the conductor spacing below specification requirements. As stated above, projections can produce corona discharge in high-voltage circuits. Figure 19.20 illustrates isolated indentation and projection and edge definition.

14. Hole Specifications Hole size is the diameter of the finished plated-through or unplated hole. A *plated-through hole* is an interfacial or interlayer connection formed by deposition of conductive material on the sidewall of a hole through the base. An *unplated hole* is a hole which has not been plated. The hole size measurement is performed to verify that the hole meets minimum and maximum drawing requirements. The size requirement is usually associated with a fit requirement of a component lead, mounting hardware, etc.

Two basic methods are used to measure hole size: (1) drill blank plug or suitable gauge (Fig. 19.21) and (2) optical. The latter method is utilized when soft coatings over the copper are used. The optical method prevents deformation of the soft coatings within the hole. When Kwik-Chek or drill blank plug gauges are used, the inspector should acquire a soft touch to prevent damage to the hole.

Plating nodules are sometimes present in the hole and restrict the penetration of the gauges. Forcing the gauges into the holes causes the nodules to be dislodged, and that results in voids in the plated-through-hole sidewall.

15. Warp and Twist Warp is a deviation from flatness of a board characterized by a roughly cylindrical or spherical curvature such that, if the board is rectangular, its corners are in

the same plane. *Twist* is a bending or curving distortion from a true, or plane, surface, in a direction parallel to a board diagonal. No more than three corners are in the same plane.

Warp and twist, on a printed board, are inspected when the conditions impair function. Warp and/or twist on a board is detrimental when the board must fit in card guides or in packaging configurations in which space is limited. Two methods are recommended for

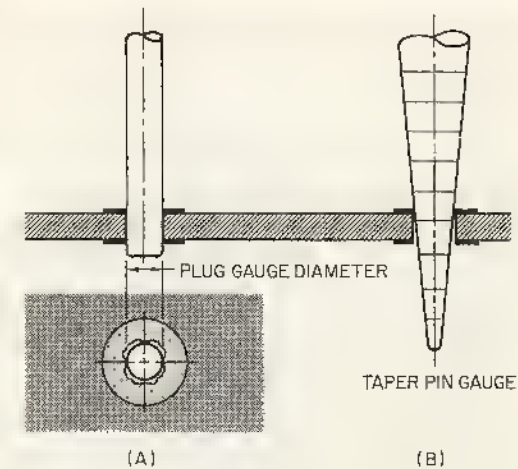


Fig. 19.21 Hole-measuring methods: (a) Drill blank plug method, and (b) Kwik-Chek or taper pin method of measuring the hole diameter (Sanilia Laboratories).

measuring the degree of warp and/or twist: the indicator height gauge method and the feeler gauge method.

The indicator height gauge method (Fig. 19.22, page 19-17) is as follows:

1. Place the board, unrestrained, on a flat horizontal surface (surface plate) with the convex surface of the board upward.

2. Take reading at edge of board. Contact flat surface R_2 .

3. Take reading at maximum vertical displacement R_1 .

4. Subtract R_2 from R_1 .

NOTE: Care must be taken to make both readings on circuitry or base material!

5. Multiply length of board by allowable inch per inch warp.

6. The difference of R_2 from R_1 should be equal to or less than the product in step 5.

The feeler gauge method as shown in Fig. 19.23, page 19-18 is as follows:

1. Place the board, unrestrained, on a flat horizontal surface (surface plate) with the convex surface of the board upward.

2. Multiply length or width of board by inch per inch allowance.

3. Select feeler gauge (gauge blocks, or gauge blanks), for the maximum allowance.

4. Attempt to slide the feeler gauge under the board at point of greatest deviation.

5. If the gauge does not enter under the board, board is within tolerance. If the gauge enters under the board, the board is out of tolerance.

Use either method described above to measure warp, but measure diagonally (corner to corner) for inch per inch multiplier.

a. IPC Tolerance Recommendations. The presentations in Tables 19.2 to 19.4 are for the tolerances to be multiplied by the maximum board dimension. The maximum board dimension for a rectangular board is the longest edge distance; for a circular board it is the diameter; and for an elliptical board it is the major axis.

A gross differential of conductor area and distribution may increase the tolerances given in Tables 19.2 to 19.4. Closer warp tolerance may limit selection of raw materials or make unusual manufacturing operations or shipping procedures necessary.

16. Conductor Pattern Integrity Several methods are used to determine conductor pattern integrity, i.e., the quality or state of being complete. Some methods include the use of

comparison equipment and overlays. The use of positive or negative overlays made from the master pattern is an inexpensive and effective method. By overlaying the film on the finished printed board, differences are easily detected. Overlays can also be used to determine if conductor widths are within tolerance and annular ring and contours are within drawing requirements.

TABLE 19.2 Tolerances—Pattern on One Side (IPC-D-300)

Base material thickness	Paper base	Glass base
Less than 0.0625 in (1.59 mm)	0.025 in/in (0.025 mm/mm)	0.15 in/in (0.015 mm/mm)
0.0625 to 0.0937 in (1.59 to 2.38 mm)	0.020 in/in (0.020 mm/mm)	0.010 in/in (0.010 mm/mm)
0.0938 to 0.0125 in (2.39 to 3.18 mm)	0.012 in/in (0.012 mm/mm)	0.008 in/in (0.008 mm/mm)
Over 0.0125 in (3.18 mm)	0.008 in/in (0.008 mm/mm)	0.006 in/in (0.006 mm/mm)

TABLE 19.3 Tolerances—Pattern on Two Sides (IPC-D-300)

Base material thickness	Paper base	Glass base
Less than 0.0312 in (0.79 mm)	0.020 in/in (0.020 mm/mm)	0.015 in/in (0.015 mm/mm)
0.0312 to 0.0625 in (0.79 to 1.59 mm)	0.015 in/in (0.015 mm/mm)	0.010 in/in (0.010 mm/mm)
0.0626 to 0.0937 in (1.60 to 2.38 mm)	0.010 in/in (0.010 mm/mm)	0.007 in/in (0.007 mm/mm)
0.0938 to 0.0125 in (2.39 to 3.18 mm)	0.007 in/in (0.007 mm/mm)	0.005 in/in (0.005 mm/mm)

TABLE 19.4 Tolerances; Multilayer Boards (IPC-ML-950A)

Thickness	Class A	Class B	Class C
All thicknesses	0.010 in/in (0.010 mm/mm)	0.030 in/in (0.030 mm/mm)	No requirement

Class A. Specified for stringent functional requirements.

Class B. Specified for normal functional requirements.

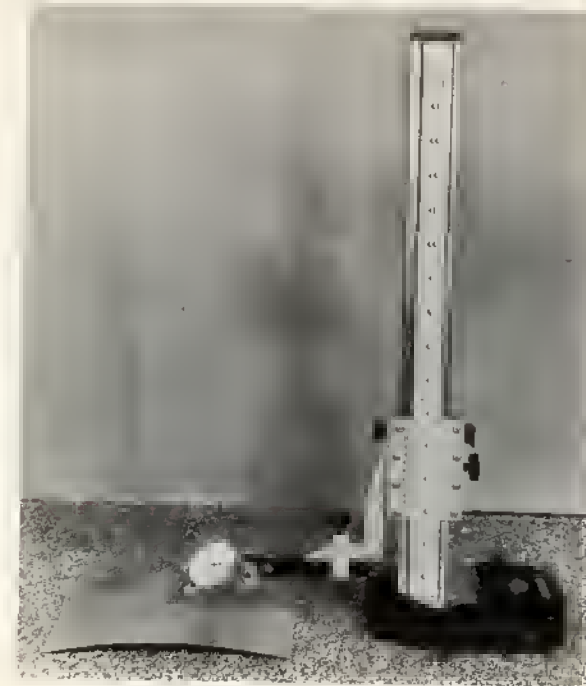
Class C. Specified for minimum functional requirements.

17. Contour Dimensions Contour dimensional inspection verifies that the outside border dimensions are within the drawing requirements. Contour dimension requirements can be considered a fit requirement. Both undersized and oversized printed boards can affect functionality, depending upon the degree of requirement violation. Measuring methods vary from the use of a ruler or calipers to sophisticated numerical control equipment. The sophistication of the method is naturally, dependent on the required dimensions and tolerances.

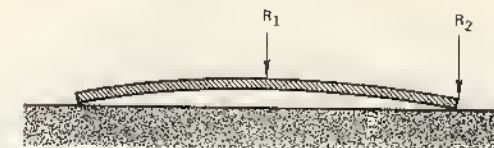
18. Plating Thickness

a. Nondestructive Methods. The plating process is used in the fabrication of numerous printed board designs to produce plated-through holes and conductive pattern circuitry. When the plating process is utilized, a plating thickness requirement is usually specified on the drawing or in the accompanying specifications. Verification of the plating thicknesses on the pattern and in the holes is usually required to assure circuit functionality. Two of the presently popular ways to measure plating thicknesses nondestructively are by the beta backscatter and micro-ohm methods.

1. Beta backscatter method. (Reference ASTM-B-567-72) This method utilizes an energy source (a radioisotope) and a detector (a Geiger Müller tube). It functions on the beta radiation backscatter principle. It allows thickness measurements to be made within a short period of time, and it can be utilized on metallic, nonmetallic, magnetic, and nonmagnetic materials.



(a)



(b)

Fig. 19.22 Indicator height gauge method (Sandia Laboratories).

The beta backscatter method can be used to measure plating thicknesses on the circuitry and in the plated-through holes during or after the board fabrication process and prior to or after the etching operation. It is useful only in measuring the top metal thickness, and it reads the thickness as an average.

2. Micro-ohm method. (Reference IPC-TM-650-2.2.13.1) The micro-ohm method functions as a four-wire resistance bridge circuit that imposes a constant current across the area under measure and measures the voltage drop across the same area. The answer is given in micro-ohms. In plated-through-hole measurements, it treats the hole as a cylindrical resistor. The method is effective only after the circuitry has been etched. When two different metals are used on top of one another, only the lowest-resistance metal thickness is detected. Like the beta backscatter method, the micro-ohm method reads the thickness only as an average.

b. Destructive, Microsection Method. The verification of plating thicknesses on the surface and in the holes, by the microsection method, usually requires three measurements at

three different locations on each plated hole sidewall (Fig. 19.24). The results are reported either individually or as an average. Care must be exercised to select locations free from voids; they are usually 0.010 in (0.254 mm) apart. Examples of plating thickness are shown in Fig. 19.25. Typical plating thickness requirements are as follows:

Copper	0.001 in (0.0254 mm) minimum
Nickel	0.0005 to 0.0010 in (0.0127 to 0.0254 mm)
Gold	0.000050 to 0.000100 in (0.00127 to 0.00254 mm)
Tin-lead	0.0003 in (0.00762 mm) minimum
Rhodium	0.000005 to 0.000020 in (0.000127 to 0.000508 mm)

Polishing vertical plated-through-hole cross sections to the mean of the hole is critical. If the hole is polished less than or greater than the mean of the hole, a plating thickness error is introduced. Horizontal plated-through-hole cross sections are recommended as



(b)

Fig. 19.23 Feeler gage method (IPC).

references (Fig. 19.26, page 19-21). Although horizontal plated-through-hole cross sections are usually more accurate for plating thickness measurements, they do not allow adequate inspection of other attributes such as voids, plating uniformity, adhesion, and nodules.

Surface plating thickness measurements are taken on vertical cross sections of conductor areas. Microsection mounts are usually etched with an appropriate etchant to show grain boundaries between the copper-clad foil and the copper plating. Copper plating surface measurements usually exclude the copper foil thickness unless otherwise specified.

c. Method of Preparing Microsections. (Institute of Printed Circuits, test method 2.1.1) Cut from the printed board or test coupon a specimen for vertical evaluation that contains at least three of the smallest holes adjacent to edge. Cut additional specimens from the board or test coupon for horizontal evaluation.

The test equipment and apparatus necessary for preparation of specimen is as follows:

Glass plate 5 × 7 in (127 × 177.8 mm)

Aluminum rings 1¼ in ID (31.75 mm)

- Silicone release agent
- Room-temperature-curing potting material
- Wooden spatulas
- Plastic cups (at least 6-oz cups)
- Saw or shear
- Engraver
- 240-grit abrasive
- Double-coated tape
- Metallographic polishing tables
- 240-, 320-, 400-, and 600-grit disks
- Number 2 liquid alumina polish
- Polishing cloths
- Chemical etchants

For metallographic evaluation the following equipment is necessary:

- Microscope and camera accessories
- Filar eyepieces or graduated reticle
- Engraver
- Photographic film or polaroid film
- Filter lens

The procedure for preparing the specimen is as follows:

1. Clean glass plate and aluminum rings and dry thoroughly.
2. Apply strip of double-coated tape to plate to support specimen. Apply thin film of release agent to glass plate and ring; then place ring on plate.
3. Sand the long edge of the perpendicular specimen until the edges of the conductor pads appear and specimen will stand on edge on a flat surface. Use 240-grit abrasive.

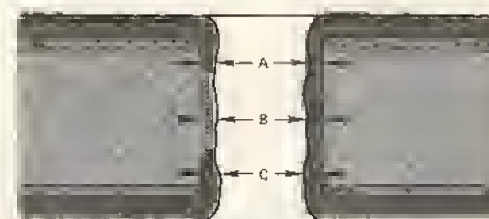


Fig. 19.24 Verification of plating thickness in holes by use of vertical cross sections (Sandia Laboratories).



(a)

(b)

Fig. 19.25 Examples of plating thickness. (a) Preferred: plating is uniform and meets thickness requirements. (b) Nonpreferred: plating thickness less than requirements (Sandia Laboratories).

4. When secondary plating thickness is being measured, overplate specimen with a harder electroplated metal. Specimens may be overplated as per ASTM method E3-58T.

5. Measure inside diameter of plated-through holes prior to encapsulation.

6. Stand specimen on edge on double-coated tape in aluminum ring with the plated-through hole edge down. For parallel specimens delate tape and lay specimen flat on glass plate inside ring.

7. Mix potting material and pour to one side of the specimen until it flows through the holes. Support the specimen in the vertical position if necessary. Continue pouring until the ring is full. Avoid entrapment of air.

8. Allow specimen to cure at laboratory temperature. Accelerated curing at a higher temperature following manufacturer's instructions is permissible, provided cracking or deformation does not occur.

9. Identify specimen promptly by engraving.

10. If more than one specimen is potted in one ring, the specimens should be spaced apart to facilitate filling the holes. Specimens should be identified with a strip of paper marked with traceability information and molded into the mounting.

The grinding and polishing procedure is as follows:

1. Rough-grind the face of the specimen to the approximate center of the plated-through holes by using 240-, 320-, 400-, and 600-grit disks in that order.

2. Flush away all residue by using tap water at room temperature. Wash hands repeatedly to avoid carrying over coarse grits.

3. Rotate specimen 360° about the axis of the wheel and opposite the direction of rotation of the wheel. Keep the face of the specimen flat on the wheel.

4. Micropolish by using a nylon disk and alumina polish number 2 until the specimen is smooth and free of sanding marks and a clear sharp image of plating lines is evident.

5. Rinse specimen thoroughly and, while it is still wet, chemically etch by using a cotton swab to highlight plating boundaries.

6. Lightly rub twice across the specimen and immediately rinse in distilled water. Repeat as necessary.

7. Rinse thoroughly after etching to eliminate carry-over of acids to the microscope.

8. Dry specimen prior to viewing through the microscope.

To make a microscopic examination, proceed as follows:

1. Place specimen on the microscope stage and adjust to get specimen centered with the eyepiece.

2. Focus and adjust lighting for best viewing; then scan specimen.

3. With filar micrometer or graduated reticle make three thickness measurements of each plating on both walls of the plated-through hole.

The following is the procedure for photomicrographing (composite photographs):

1. Locate specimen and mount camera to microscope tube. Insert film pack.

2. Set lighting and exposure time to prearranged settings.

3. Focus on one part of one wall and push camera plunger.

4. Immediately remove negative from the camera and develop.

5. Take sufficient photographs to illustrate the entire length of the wall.

6. Repeat steps 1 to 5 and photograph the opposite wall.

7. If defective junctions, voids, etc., appear, enlarged photographs may be made. Enlargements should be referenced to the photomicrographs.

19. Undercut The reduction of the cross section of a conductor, caused by the etchant removing metal from under the edge of the resist or plating, is called undercut. Measurement of the degree of undercut present is not usually required, because undercut is included in the minimum conductor width measurement. If, however, the undercut measurement is required separately, it is performed by first measuring the minimum conductor width and then subtracting that dimension from the master line width and dividing by 2 (Fig. 19.27). For example:

1. Master line width dimension, a
2. Minimum conductor width dimension, b
3. Undercut dimension, c

Then

$$\frac{a - b}{2} = c$$

20. Overhang The increase in conductor width, caused by plating buildup, is called overhang. The measurement of the degree of overhang is required in some specifications. Excessive overhang can eventually lead to "slivoring," a thin metallic piece that breaks off the conductive pattern.

Overhang length dimensions of one to one and one-half times, or less, the overhang thickness, at the point where the overhanging material extends beyond the main conductor configuration, is usually allowed. The degree of overhang is measured on a conductor cross section (Fig. 19.27). An example of excessive overhang is depicted in Fig. 19.28.

A method of determining if the overhang is prone to slivoring utilizes an ultrasonic cleaner; the total printed board is suspended in water in an ultrasonic cleaner for 1 to 2 min. If the board is prone to slivoring, metallic slivers will be evident along pattern edges.

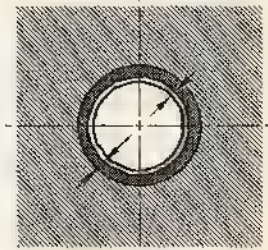


Fig. 19.26 Verification of plating thickness in holes by use of horizontal cross sections (Sandia Laboratories).

21. Etchback Etchback is the controlled removal of base material, by a chemical process, on the sidewalls of holes in multilayer boards. It is used to enhance internal layer-to-hole connections in multilayer boards. The degree of etchback is critical to function. Too much etchback creates excessively rough hole sidewalls and causes weak plated-through-hole structures.

Etchback requirements range from 0.0002 to 0.0030 in (0.00508 to 0.0762 mm). The requirement is usually specified as a minimum and a maximum. The degree of etchback is measured by utilizing vertical plated-through-hole cross sections of multilayer boards (Fig. 19.29).

Typical etchback acceptance criteria recommended by the Institute of Printed Circuits are shown in Fig. 19.30.

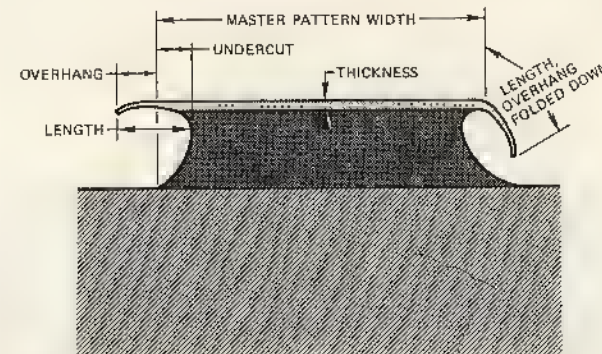


Fig. 19.27 Undercut and overhang (Sandia Laboratories).



Fig. 19.28 Overhang extension is approximately ten times the thickness (Sandia Laboratories).

22. Layer-to-Layer Registration Layer-to-layer registration has to do with the relative positions of printed circuitry patterns, or portions thereof, with respect to their desired locations on or within a multilayer printed board. It is required to ensure electrical connection between the plated-through holes and the internal layers. Misregistration of internal layers increases electrical resistance and decreases conductivity. Severe misregistration creates an open-circuit condition; complete loss of conductivity. Two popular methods of measuring layer-to-layer registration are the X-ray method described in Sec. 8, and the microsection method, which is similar to the plating thickness inspection method. The microsection method consists of measuring each internal terminal area on a vertical

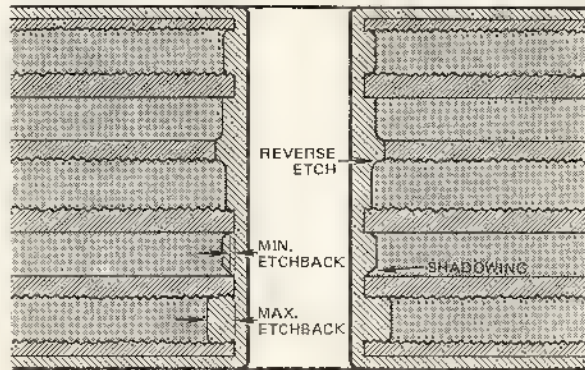


Fig. 19.29 Etchback configurations (IPC).

plated-through hole cross section and determining the centerline. The maximum variation between centerlines is the maximum misregistration (Fig. 19.31).

23. Flush Printed Circuits A printed circuit in which the outer surface of the reproduced pattern is in the same plane as the outer surface of the insulating base is defined as a flush printed circuit. Flush printed circuits are primarily used in rotating switches, commutators, and potentiometers. Common to those uses are brush or wiper and conductor pattern combinations. Making the conductors flush in the above applications reduces wiper vibrations, wear, and intermittent or noisy signals.

The height of the step allowed between pattern and base material is dependent on the

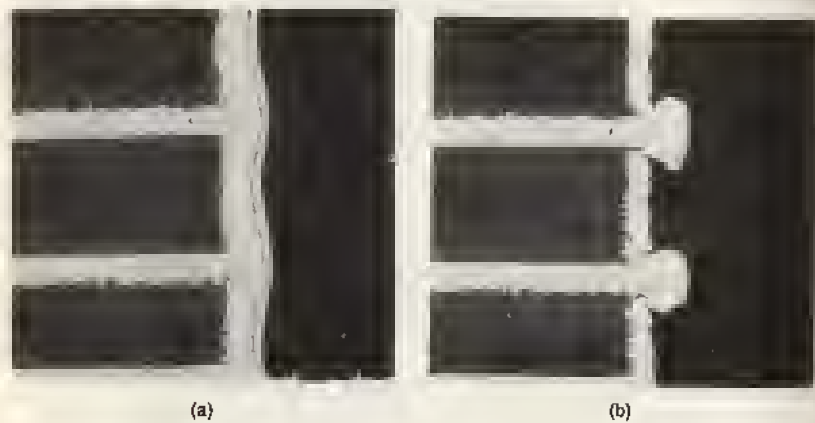


Fig. 19.30 Etchback. (a) Preferred: uniform etchback of base laminate; uniform plating in the plated-through hole (b) nonpreferred; nonuniform and excessive etchback of base laminate results in unacceptable nonuniform plating in the hole (IPC).

relative wiper speed, the materials used, and the degree of signal error and electrical noise tolerable in the circuit. A commonly accepted height allowance is as follows:

- Up to 50 rpm 80-200 μin (2.032×10^{-4} to 5.08×10^{-3} mm)
- 51 to 125 rpm Better than 50 μin (1.27×10^{-4} mm)
- 126 to 500 rpm Better than 30 μin (0.762×10^{-4} mm)

Inspection of the degree of flushness is performed with height gauges or measuring microscopes (Fig. 19.32).

24. Dimensional Inspection The dimensional board attributes subjected to nondestructive and destructive inspection are listed in Table 19.5.

TABLE 19.5 Dimensional Inspection Chart

Subject	Nondestructive	Destructive
Annular ring	x	
Pattern-to-hole registration	x	
Conductor width	x	x
Conductor spacing	x	
Edge definition	x	
Hole size	x	
Warp and twist	x	
Conductor pattern integrity	x	
Contour dimensions	x	
Plating thickness	x	x
Undercut		x
Overhang		x
Etchback		x
Layer-to-layer registration		x
Flush circuits	x	

MECHANICAL INSPECTION

Mechanical inspection is applied to characteristics which can be verified with qualitative physical tests. Mechanical inspection methods can be either nondestructive or destructive. The following methods are used to verify printed board fabrication integrity.

25. Plating Adhesion A common method of inspecting for plating adhesion is the tape test, which is described in detail in the IPC Test Method IPC-TM-650, 2.4.1 and military specification MIL-P-55110. The basic test method is shown in Fig. 19.33.

Another method of determining plating adhesion is applied during the preparation and viewing of the microsection mounts. If adhesion is poor, the layers of plating will separate during the microsection specimen preparation or will indicate lack of adhesion at plating boundaries during microsection mount viewing.

Figure 19.34 indicates lack of adhesion at the different plating boundaries, copper foil to copper plate and copper plate to tin-lead.

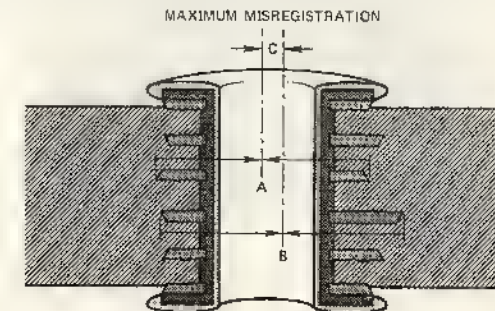


Fig. 19.31 Layer-to-layer registration—microsection method (IPC).

26. Solderability Solderability inspection measures the ability of the printed pattern to be wetted by solder for the joining of components to the board. It involves the use of three terms: (1) wetting, (2) dewetting, and (3) nonwetting. The terms are defined as follows:

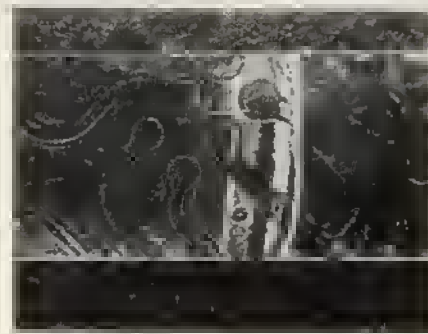
Wetting. The free flow and spreading of solder on conductors and terminal areas to form an adherent bond.



Fig. 19.32 Flush printed circuits. (a) Preferred: conductor is flush to the board surface. (b) Nonpreferred: conductor is not flush to the board surface and does not meet the specified tolerance (IPC).



(a) (b)



(c)

Fig. 19.33 The tape test. (a) Step 1. Place transparent cellophane tape across the circuits to be tested and press the tape onto the circuits. Eliminate all air bubbles with finger. (b) Step 2. Lift tape on one end enough to get a grip. Pull tape off the printed board at approximately 90° to the board. Use a rapid pull. (c) Step 3. A clear tape is the preferred test result (IPC).

Dewetting. The phenomenon of the solder first wetting the surface and then drawing back because of improper wetting, leaving behind a thin coat of solder over the base metal. Most of the solder balls up at random locations on the surface.

Nonwetting. A condition in which a surface has contacted molten solder but has had none of the solder adhere to it.

Many methods for inspecting printed boards for solderability both quantitatively and qualitatively have been established. However, the most meaningful method is that which will be used in the assembly soldering operation—hand soldering, wave soldering, drag soldering, etc. The Institute of Printed Circuits has adopted the criteria illustrated in Fig. 19.35 for printed board solderability acceptance.

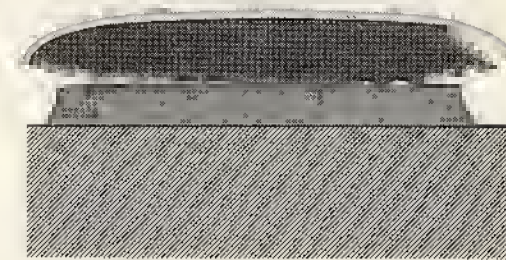
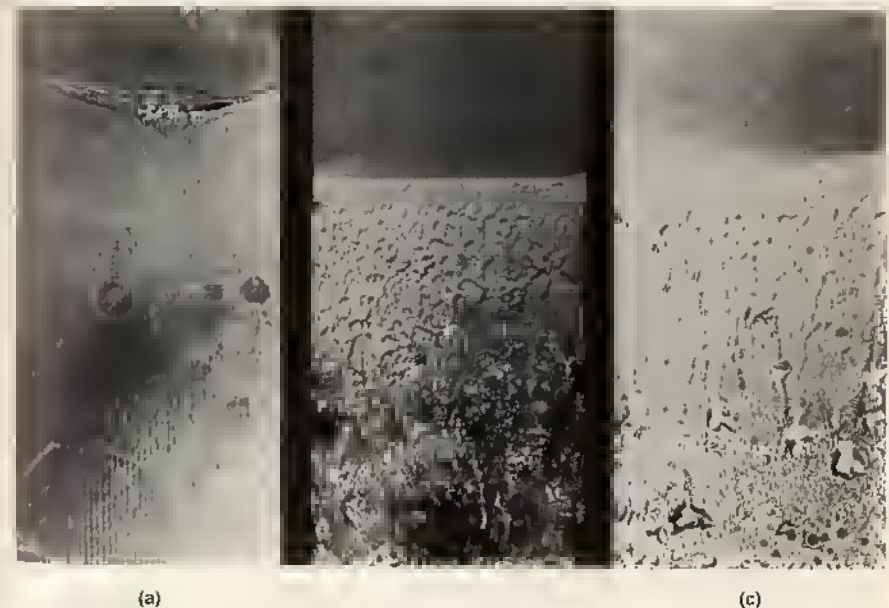


Fig. 19.34 Plated metal adhesion Sandia Laboratories.



(a) (b) (c)

Fig. 19.35 Solderability acceptance criteria (a) Preferred: good solderability; smooth, bright, good wetted copper; and good solder coverage. (b) Acceptable: some dewetting is visible at left surface. (c) Nonpreferred: surface shows severe dewetting and nonwetting (IPC).

27. Alloy Composition Two popular alloys used in printed board manufacture are tin-lead and tin-nickel. The composition of the tin-nickel bath is usually 65 percent tin and 35 percent nickel. The composition of the deposit, during electroplating, remains nearly constant despite fluctuations in bath composition and operating conditions. However, the composition of the tin-lead deposits can vary with bath composition and operating conditions, and that will influence the melting temperature of the alloy and affect

solderability. Specifications usually require the tin content to be between 50 and 70 percent.

Methods available for analyzing the alloy composition on the plated printed board include wet analysis, atomic absorption, and beta backscatter. The beta backscatter method is relatively new and is gaining in popularity because of the ease in obtaining the alloy composition nondestructively.

28. Thermal Stress, Solder Float Test Temperature-induced strain or straining force that is exerted upon the printed board and tends to stress or deform board shape can be a serious problem during the soldering operation. Thermal stress inspection is performed to predict the behavior of the printed boards after soldering. Plated-through-hole degradation, separation of platings or conductors, or laminate delamination is detectable by the thermal stress test. The test is usually performed by floating the printed board in a solder bath (tin between 58 and 68 percent) maintained at 550°F for 10 s. Visual inspection for defects is followed by cross-sectioning the plated-through holes and inspecting with magnification for integrity. Test patterns also are used for thermal stress inspection. (See Fig. 19.39 for a test pattern used in this and other connections.)

29. Peel Strength Peel strength is a measure, in pounds per inch width, of the force required to separate a strip of conductive material from the adjacent base by a pull applied to the edge of the conductive material. It is usually associated with the acceptance testing of copper-clad laminate material upon receipt, but it is sometimes utilized to test the adhesion of the conductors to the finished board.

Peel strength tests of conductors are usually performed after the specimens have been dip- or flow-soldered. Peel strength values in MIL-P-13949, after solder dip, also are used for printed board conductor peel strength tests. However, the values must be adjusted for the particular conductor width, because the peel strength values in MIL-P-13949 pertain to a 1.00-in (25.4-mm) conductor width. As an example:

Conductor width = 0.050 in (1.27 mm)
 Peel strength value in MIL-P-13949 = 10 lb
 Conductor width in MIL-P-13949 = 1.00 in (25.4 mm)

$$\frac{0.050 \times 10.0}{1.00} = 0.5 \text{ lb}$$

A calculation with width in millimeters will, of course, yield the same results.

Peel strength tests of conductors are a good method of assuring that the conductor-to-laminate adhesion is sufficient to withstand assembly soldering operations. Actual printed boards or test patterns can be utilized for the test. Figure 19.36 is an example of a typical peel strength test pattern.

30. Plated-through-hole Structure The inspection of plated-through-hole structures, including the metallic barrel and terminal areas, involves chemically etching the glass and epoxy away from the plated-through hole. The hole structure is then examined for voids and structural integrity. See Fig. 19.37 for a typical plated-through-hole structure test coupon.

31. Terminal Pull The terminal pull test is a test of the plating adhesion to the glass-epoxy in a hole.

Poor adhesion in the holes can occur from the drilling or electroless copper deposition operations being out of control or from the use of undercured laminate material. Whatever the cause, poor plating adhesion in the hole affects functionality.

Lack of plating adhesion to the glass-epoxy in the hole is detectable during microsection analysis also. Whenever poor adhesion is detected during microsection analysis, a terminal pull test should be performed to substantiate the condition (see Fig. 19.38).

32. Mechanical Inspection The mechanical board attributes subjected to nondestructive and destructive inspection are listed in Table 19.6.

TABLE 19.6 Mechanical Inspection Chart

Subject	Nondestructive	Destructive
Plating adhesion	×	×
Solderability		×
Alloy composition	×	×
Thermal stress (solder float test)		×
Peel strength, printed wiring conductor		×
Plated-through-hole structure		×
Terminal pull		×

ELECTRICAL INSPECTION

Electrical inspection is performed to verify electrical circuit integrity after processing and also to substantiate that the electrical characteristics of the processed board meet design intent.

Electrical inspection methods are both destructive and nondestructive. Nondestructive tests are usually performed on the actual printed boards. Destructive tests are performed on either printed circuit boards utilized for destructive acceptance testing or on test patterns fabricated outside the border of the board.

Two popular nondestructive electrical tests are insulation resistance and continuity. They are usually performed on 100 percent of complex printed boards, especially multilayer ones. Care should be taken to prevent arcing as the probes approach the printed

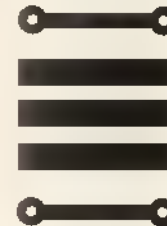


Fig. 19.36 Peel strength test



Fig. 19.37 Test coupon used for plated-through-hole structure testing (IPC).



Fig. 19.38 Typical terminal pull test coupon (IPC).

circuit pattern when the insulation resistance test is performed. An easily activated switch in series with one of the probes allows the probes to make contact with the pattern prior to current flow.

33. Continuity Continuity tests are performed basically on multilayer circuits to verify that the printed circuit pattern is continuous. They can be performed with an inexpensive multimeter or with more elaborate equipment such as a constant-current power supply and a voltmeter if currents greater than 50 mA are required.

In the printed circuits industry the continuity test is performed in one of two ways: (1) as a go/no-go test to verify that the pattern is continuous or (2) to verify that the pattern is continuous as well as to verify the integrity of the measured pattern. The latter result is reported in electrical resistance values (ohms). The preferred method is to perform the continuity test on all printed boards submitted for acceptance. This is especially recommended for multilayer circuitry, the internal patterns and interconnections of which cannot be inspected visually after fabrication.

Test coupons are sometimes used for the continuity test. See Fig. 19.39 for a pattern used on a typical coupon.

34. Insulation Resistance The purpose of this test, as stated in MIL-STD-202, is to measure the resistance offered by the insulating members of a printed board to an

impressed direct voltage that tends to produce a leakage of current through or on the surface of those members. Low insulation resistances can disturb the operation of circuits, intended to be isolated, by permitting the flow of large leakage currents and the formation of feedback loops. The test also reveals the presence of contaminants from processing residues.

In printed circuitry the test is performed either between conductors on the same layer or between two different layers. The test is also utilized before and after thermal shock and temperature cycling tests. Test voltages of 100 and 500 V dc and minimum insulation resistances of 500 M Ω are popular. The insulation resistance test is performed on either the actual printed board or a test coupon fabricated on the same panel as the board (Fig. 19.40).

When special conditions such as isolation, low atmospheric pressure, humidity, and immersion in water are required, they should be specified in the test method instructions.



Fig. 19.39 Continuity, thermal stress, current-carrying capacity, and moisture resistance test pattern (IPC).

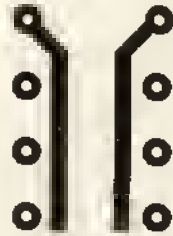


Fig. 19.40 Insulation resistance, dielectric withstanding voltage, and moisture resistance test pattern (IPC).



Fig. 19.41 Typical current breakdown test pattern (IPC).

35. Current Breakdown, Plated-through Holes The current breakdown test is used to determine if sufficient plating is present within the plated-through hole to withstand a relatively high current potential. The time and current selected determine if this test is destructive or nondestructive. IPC test method TM-650-2.5.3 recommends a current of 10 A for 30 s. The test is performed as follows:

1. Place a load resistor, of predetermined value, across the negative and positive terminals of a current-regulated power supply.
2. Adjust the supply for a current of 10 A or any other desired value.
3. Remove one end of the resistor from the positive supply terminal.
4. Connect the desired plated-through hole between the disconnected end of the resistor and the positive supply terminal.
5. Perform the test for the desired time.

The test is performed either on an actual printed board or on a test pattern (Fig. 19.41).

36. Current Carrying Capacity The current-carrying capacity test is performed to verify that the conductor will meet designated electrical functional requirements. It is performed by selecting a current from the current-carrying capacity chart, MIL-STD-275, for the cross-sectional area to be tested. The rest of the test is as follows:

1. Select a load resistor of predetermined value such that, when positive and negative terminals of a regulated power supply are shunted, the desired current will flow.
2. Place the circuitry to be tested in series with the resistor.
3. Connect a temperature-measuring device to the circuitry to be tested.
4. Compare results with stated values in the current-carrying capacity chart.

The current-carrying capacity test is nondestructive if the values in the MIL-STD-275 current-carrying capacity chart are used. Higher currents can result in destructive testing. The test can be performed on either a functional printed board or on a test pattern (Fig. 19.39).

37. Dielectric Withstanding Voltage This test is used to verify that the component part can operate safely at its rated voltage and withstand momentary overpotentials due to switching, surges, and similar phenomena. It also serves to determine whether insulating

materials and spacings in the component part are adequate. It is thoroughly defined in IPC-ML-950, Multilayer PWB's, and MIL-STD-202, Electronic Components Method 301.

One of three different test voltages—500, 1000, and 5000 V—is usually specified. The test is performed on either an actual board or a test coupon. Voltage is applied between mutually insulated portions of the specimen or between insulated portions and ground. The voltage is increased at a uniform rate until the specified value is reached. The voltage is held for 60 s at the specified value and then reduced at a uniform rate.

Visual examination of the part is performed during the test for evidence of flashover or breakdown between contacts. See Fig. 19.40 for a typical test pattern. The test can either be destructive or nondestructive depending on the degree of overpotential used.

38. Electrical Inspection The electrical board attributes subjected to nondestructive and destructive inspection are listed in Table 19.7.

TABLE 19.7 Electrical Inspection Chart

Subject	Nondestructive	Destructive
Continuity	×	
Insulation resistance	×	
Current breakdown, plated-through holes	×	×
Current-carrying capacity	×	
Dielectric withstanding voltage	×	×

ENVIRONMENTAL INSPECTION

Environmental inspection consists of performing specific tests to ensure the printed board will function under the influence of climatic and/or mechanical forces to which it will be subjected during use. Environmental tests are performed on preproduction printed boards to verify design adequacy. Specific tests are sometimes specified as part of the printed board acceptance procedure to expose a prospective failure situation. Specific environmental tests, as part of the acceptance procedure, are prevalent in high-reliability programs.

Environmental tests are performed on either actual printed boards or coupon test patterns. This section briefly reviews some of the popular environmental tests. Specific details on performing the tests are referenced at the end of this chapter in the section titled Related Test Methods.

39. Thermal Shock A thermal shock is induced on a printed board by exposure to severe, and rapid, differences in temperature. The thermal shock test is particularly efficient in identifying (1) printed board designs with areas of high mechanical stress and (2) the resistance of the printed board to exposure of high- and low-temperature extremes.

The test is usually performed by transferring the board from one temperature extreme (e.g., 125°C) to the other (e.g., 0°C) rapidly, usually within 5 min. Thermal shock effects on the board include cracking of plating, in the holes, and delamination. See Fig. 19.42 for typical thermal shock test pattern.

40. Moisture Resistance The moisture resistance test is an accelerated method of testing the printed board for deteriorative effects of high humidity and heat conditions typical of tropical environments. The test conditions are usually a relative humidity of 90 to 98 percent at a temperature of 25 to 65°C. After the required test cycles are completed, the board is subjected to electrical testing including current breakdown and interconnection resistance. Test specimens should exhibit no blistering, moisting, warp, or delamination after the moisture resistance test. See Fig. 19.39, 19.40, or 19.42 for typical test patterns.

41. Outgassing Outgassing is known to cause contamination of spacecraft equipment. A test method for outgassing has been designed to measure the weight loss, if any, through the degree of outgassing of low-molecular-weight materials such as trapped plating

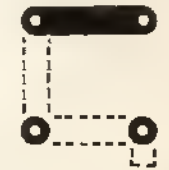


Fig. 19.42 Typical thermal shock and moisture resistance test pattern (IPC).

solutions and uncured adhesive layers. The test is performed by weighing the board, placing the board in a temperature-vacuum system, and heating to 125°C at a vacuum of at least 1×10^{-6} torr for 24 h. The board is then removed from the chamber and reweighed, and the weight loss is calculated. Board weight loss greater than 0.1 percent after 24 h exposure or an outgassing of products which condense on nearby surfaces is considered unsuitable for spacecraft application. Outgassing tests should be performed on specimens of actual printed board patterns.

SUMMARY

To select the printed board attributes to be inspected in an individual program, proceed as follows:

1. Review the environment in which the printed board will operate and the life expectancy of the board.
2. Review the electrical and mechanical parameters associated with functionality.
3. Consider the total assembly unit cost and the importance of the printed board before determining if a quality assurance program is economical.
4. Consider both functionality and economics in quality assurance program selection. Functionality should prevail, however.
5. Design the quality assurance program for at least a 90 percent confidence level by using sampling plans when they are appropriate.
6. Select for inspection the attributes which will satisfy requirements 1 and 2.
7. Select test methods which verify the printed board functionality and integrity.

Test methods may require modification or creation to satisfy the quality assurance requirements.

TEST SPECIFICATIONS AND METHODS RELATED TO PRINTED BOARDS

Attributes	Method
Annular ring	IPC-TM-650, 2.2.1
Base material edge roughness	IPC-TM-650, 2.1.5
Blistering	IPC-TM-650, 2.1.5
Chemical cleaning	IPC-TM-650, 2.2.5 & 2.1.1
Conductor spacing	IPC-TM-650, 2.2.2
Conductor width	IPC-TM-650, 2.2.2
Crazing	IPC-TM-650, 2.1.5
Current breakdown, plated-through hole	IPC-TM-650, 2.5.3
Current-carrying capacity	IPC-TM-650, 2.5.4
Dents	IPC-TM-650, 2.1.5
Dielectric withstanding voltage	IPC-TM-650, 2.5.7
Edge definition	MIL-STD-202, 301
Etchback	IPC-TM-650, 2.2.3
Eyelets	IPC-TM-650, 2.1.5
Haloing	IPC-TM-650, 2.1.5
Uole size	IPC-TM-650, 2.2.6
Inclusions	IPC-TM-650, 2.1.5
Insulation resistance	IPC-TM-650, 2.5.9, 2.5.10, 2.5.11-MIL-STD-202,302
Layer-to-layer registration, destructive	IPC-TM-650, 2.2.11
Markings	IPC-TM-650, 2.1.5
Measling	IPC-TM-650, 2.1.5
Microsections, methods of preparing	IPC-TM-650, 2.1.1
Moisture resistance	IPC-TM-650, 2.6.3
Outgassing	MIL-STD-202, 106C
Overhang	IPC-TM-650, 2.6.4
Pinholes	IPC-TM-650, 2.2.9
Pits	IPC-TM-650, 2.1.5
Plated-through-hole structure	IPC-TM-650, 2.1.3

Plating adhesion	IPC-TM-650, 2.4.10
Plating thickness, nondestructive	IPC-TM-650, 2.2.13.1
	ASTM-B-567-72
Plating thickness, destructive	IPC-TM-650, 2.2.13
	ASTM-A-219-58
Scratches	IPC-TM-650, 2.1.5
Solderability	IPC-TM-650, 2.4.12 and 2.4.14
Surface roughness	IPC-TM-650, 2.1.5
Terminal pull (board strength)	IPC-TM-650, 2.4.20 and 2.4.21
Thermal shock	IPC-TM-650, 2.6.7
	MIL-STD-202, 107C
Thermal stress	IPC-TM-650, 2.6.8
Voids	IPC-TM-650, 2.1.5
Warp and twist	IPC-TM-650, 2.4.22
Weave exposure	IPC-TM-650, 2.1.5
Weave texture	IPC-TM-650, 2.1.5

GENERAL SPECIFICATIONS RELATED TO PRINTED BOARDS

The following printed board specifications are some of those used throughout the industry. The specifications usually cover both processing and acceptance requirements.

IPC Specifications

IPC-FC-240B	Specifications for Flexible Printed Wiring
IPC-FC-250	Specifications for Double Sided Flexible Wiring with Interconnections
IPC-D-300	Printed Circuit Board Dimensions and Tolerances
IPC-D-320	Printed Wiring Board Rigid, Single and Two Sided End Product Specification
IPC-TC-500	Copper Plated Through Hole Two Sided Boards, Rigid
IPC-TC-510	Procedure for Installing and Inspecting Clinched Wire Type Interfacial Connections in Rigid Printed Circuit Boards
IPC-TC-550	Procedure for the Design, Assembly, and Testing of Fused-in-Place Interfacial Connections in Rigid Printed Circuit Boards
IPC-A-600	Acceptability of Printed Wiring Boards
IPC-TM-650	Test Methods Manual
IPC-ML-910	Design and End Product Specification for Rigid Multilayer Printed Wiring Boards
IPC-ML-950	Performance Specifications for Multilayer Printed Wiring Boards
IPC-ML-990	Performance Specification for Flexible Multilayer

Military Specifications

MIL-Q-9858	Quality Assurance
MIL-P-55110	Printed Wiring
MIL-P-55640	Multilayer Printed Wiring
MIL-P-50884	Flexible Printed Wiring
MIL-STD-105	Sampling Procedure and Tables for Inspection by Attributes
MIL-STD-202	Test Methods for Electronic and Electrical Component parts
MIL-STD-275	Printed Wiring for Electronic Equipment
MIL-STD-429	Printed Circuit Terms and Definitions
MIL-STD-810	Environmental Test Methods
MIL-STD-1495	Multilayer Printed Wiring Boards for Electronic Equipment

Other Publications

ASTM-B-567-72	Measurement of Coating Thickness by the Beta Backscatter Principle
ASTM-A-219-58	Standard Method of Test for Local Thickness of Electrodeposited Coatings
EIA-RS-319	Solderability of Printed Wiring Boards
IEC No. 326	Performance Specification for Single and Double Sided Printed Wiring Boards
NSA No. 68-8	Multilayer Printed Wiring
UL 796	Printed Wiring Boards

REFERENCES

1. C. R. Draper, "The Production of Printed Circuits and Electronics Assemblies," Robert Draper Ltd., London, 1969.
2. C. A. Harper, "Handbook of Electronic Packaging," McGraw-Hill Company, New York, 1969.
3. F. W. Keen, The Design and Manufacture of Printed Circuits, part 14, *PCM-PCE Magazine*, February, 1970.
4. IPC-A-600, Acceptability of Printed Wiring Boards, Institute of Printed Circuits, 1974.
5. IPC Design Guide, sec. 9, Quality Assurance, Institute of Printed Circuits.
6. IPC-TM-650, Test Methods Manual, Institute of Printed Circuits, 1973.
7. MIL-STD-202, Test Methods for Electronic and Electrical Component Parts.
8. J. A. Scarlett, "Printed Circuit Boards for Microelectronics," Van Nostrand Reinhold Company, New York, 1970.
9. H. R. Shemilt, Inspection of Printed Circuits, *Electronics Manufacturer Magazine*, vol. 15, 1971.

Section **6****Multilayer Circuitry**

Multilayer Design and Layout

DIETER W. BERGMAN

Aeroneutronics-Ford, Willow Grove, Pennsylvania

G. L. GINSBERG

General Atronics Corp., Philadelphia, Pennsylvania

General Considerations	20-4
1. The Need for Multilayers	20-4
2. Advantages and Disadvantages	20-5
3. Packaging Density	20-6
4. Relative Costs	20-8
Technology Description	20-6
5. Multilayer Types	20-8
6. Plated-Through Holes	20-9
7. Clearance Holes	20-9
8. Comparison of Plated-Through Hole and Clearance Hole	20-11
9. Drilling	20-12
Specifications and Standards	20-12
10. Institute of Printed Circuits Specifications	20-12
11. Department of Defense Specifications	20-12
General Design Considerations	20-12
12. Initial Design Development	20-13
13. Overall Design—Production Cycle	20-14
14. Tolerance Analysis	20-14
Mechanical Design Features	20-17
15. Thickness and Size	20-17
16. Holes and Terminal Areas	20-18
Electrical Design Factors	20-18
17. Conductor Spacing and Clearances	20-19
18. Capacitance	20-19
Layout	20-20
19. Early Decisions	20-22
20. Input-Output	20-23

21. Conductor and Terminal Area Layout	20-23
22. Layout Feasibility Concepts	20-24
23. Master Drawing Preparation	20-25
24. Digitizing	20-26
Artwork Generation	20-27
25. Registration	20-27
Tooling	20-28
26. Corner Marks	20-28
27. Reduction Targets	20-29
Test Coupon	20-29
Marking	20-29

GENERAL CONSIDERATIONS

Multilayer printed wiring emerged as a solution to the problem of interconnecting miniature electronic components into complex systems. The volume taken up by conventional wiring was disproportionately greater than the volume and weight of the components interconnected. Multilayer printed wiring made possible, by virtue of the multi-planar conductor structure, a reduction in weight and volume of the interconnections commensurate with the size and weight of the components it interconnected. The exact amount of space and weight savings is determined by the particular packaging design, and several packaging designs are described in this chapter.

The area of application which shows multilayer printed wiring to its greatest advantage is the interconnection of integrated circuits. Integrated circuits can be interconnected by other wiring techniques such as welded matrices, point-to-point wiring, or two-sided printed boards, but with these methods the packaging is done at the expense of interconnection density and with an increase in weight and volume. By a combination of integrated circuits and multilayers drastic reductions in the overall size and weight of a complete system can be achieved.

Another distinct application of multilayer boards is to accommodate the problems of heat distribution and heat removal in systems utilizing integrated circuits. With multilayers, all interconnections can be placed on internal layers, and a heat sink of thick, solid copper or other material can be placed on the outer surfaces. Components can then be mounted directly on the metallic surface.

A third feature of multilayer printed wiring is the possibility of incorporating ground or shield planes. Solid copper planes, with clearance holes etched out in areas where no contact is desired, can be placed anywhere within the structure of the multilayer board. The planes either serve as electrical decoupling to minimize the noise interference or cross-talk between various critical circuits or are used for shielding some circuits within the board, or the entire board, from any internal or external interference. A number of such planes can be incorporated within one board and also be used for power distribution. This unique feature of multilayer printed boards offers some interesting possibilities, and in many cases it provides the only solution to some interconnection problems. The ground plane feature of multilayer boards can be very important in high-frequency circuitry.

The expected growth of multilayer printed wiring has been realized, and it continues to expand with the advent of more sophisticated semiconductor devices (LSI) to the extent that system design engineers should become thoroughly familiar with its potentials and limitations. The new generation of electronic systems requires ever-increasing complexity and density of interconnections, and the definite trend toward higher operating frequencies demands extremely careful design of even a single conductor. The width, length, location, and relations of conductors, the spacing, the dielectric material, etc., are interrelated, and in many cases they exert critical influence on signal propagation, noise ratio, shape of the pulses, and other electrical parameters of the system. Therefore, the involvement of system design engineers as well as electrical design engineers in electronic packaging has become much more intensive.

1. The Need for Multilayers The conditions which will necessitate the use of multilayer printed wiring generally are the following:

1. When weight and volume savings in interconnections are of paramount importance, as in military, airborne, and missile and space applications or when integrated circuits are utilized.

2. When the complexity of interconnection in subsystems will require very complicated and costly wiring or harnessing.
3. When coupling or shielding of a large percent of interconnections is required.
4. When frequency requirements call for careful control and uniformity of conductor impedances with minimal distortion in signal propagation and when uniformity of those characteristics from board to board is of high importance.
5. When the spacing of terminal points does not allow a sufficient density of interconnection on two-sided circuit boards.

2. Advantages and Disadvantages In order to make an assessment as to whether to use multilayer boards, a comparison should be made of the different printed wiring forms that could be used in the end product equipment. A determination should be made as to how many double-sided boards of various sizes would be required to mount and interconnect all of the circuitry associated with a given equipment. A similar determination should be made of multilayer boards. The costs of the candidate technologies can be obtained and a cost-effective decision made. The comparative costs are often not limited to dollars and cents; they can include the cost to the user in volume required, amount of back-panel interconnection wiring, design-through-production time cycle, logistics support, etc. An equivalent analysis can be performed only after appropriate weighting factors for the intended application have been applied.

There are several factors which can be noted as favoring multilayer circuitry as compared with wiring utilizing conventional round hook-up cable with conductors individually insulated. One of the most obvious advantages is that, in practically all multilayer circuits up to about five layers, depending upon the thickness of the dielectric layers utilized between conductor layers, the presence of breaks, lesions, etc., in circuitry can generally be examined because of the translucent nature of the film or glass-reinforced materials. Another obvious advantage is in the tools of miniaturization which the multilayer circuits offer. In many cases lightweight adhesives can replace the heavy hardware and awkward nuts, bolts, clamps, and brackets used as fastening devices in conventional harness. An etched T splice is obviously a fraction of the cost and weight of such a junction in regular wiring.

The ability of the multilayer circuit to incorporate ground planes which can be (1) etched according to a predetermined pattern, (2) may consist of a wire mesh screen, (3) can be fabricated from a solid layer of foil, or (4) can even be grounded, by means of a tap-off connection to supporting metal frames or cases, permits the ready utilization of structural hardware for shielding purposes.

A further advantage of incorporating the ground plane into the three-dimensional circuit lies in high-speed applications, in which that capability makes possible the production of much lower cost constant-impedance conductors for strip-line application. The ability to combine multiplicity of circuits into a small capsule obviously reduces the chance for errors and the cost for testing of back panel wiring. To sum up the potential advantages and disadvantages of multilayer wiring, the following factors usually apply:

Advantages

1. Controlled assembly process through mechanization.
2. Space saving through use of thin films and high terminal density.
3. Wiring errors eliminated.
4. Uniform electrical impedance and coupling (especially with strip-line).
5. Possible cost savings with high quantity and proper tooling.
6. Assembly time reduction—by simplification.
7. Capable of high inherent reliability (dependent upon process control and type of interconnection).
8. Can combine structural and electrical functions.

Disadvantages

1. Realistic simulation during breadboard testing is difficult or requires sophisticated computer programs.
2. Repair requires complicated special tools and skill.
3. High cost of small quantities.
4. Extended design time.
5. Long lead time for fabrication.
6. Thermal sensitivity (warp).
7. Inspection of end product is difficult and usually requires microsectioning.
8. Changes in completed board are difficult and costly.

3. Packaging Density Multilayer boards are often used with integrated circuits when the component densities possible with double-sided boards must be exceeded. For instance, on two-sided boards (all of the interconnections of which must be distributed on two surfaces) a usual maximum is 2.0 TO cans per square inch. With multilayer boards that packaging density can be increased to over 3.0 if secondary board space allocations are taken into account. A similar condition exists for flat packs. Two-sided boards can interconnect a maximum of approximately 2.5 flat packs per square inch; multilayer boards are capable of interconnecting over 4.0, assuming a minimum of four layers. At least two of the four are for interconnection wiring and separate layers for ground and voltage distribution. In some designs it is possible to place flat packs on both sides of a multilayer board and thus double the component density without appreciably increasing the volume of interconnections.

Dual in-line integrated circuit package (DIP) densities per square inch of overall multilayer board surface are on the average up to 2.0, which is twice that normally achieved with two-sided boards.

4. Relative Costs In the design of any product an item of concern, after the technical problems have been resolved, is the cost. More often than not potential cost factors have changed the course of a particular project, equipment, or product. Since initial costing is usually estimated, it is important that all factors be considered in proper perspective in order to avoid canceling an approach that in total would have been the most cost-effective. Some of the key elements of multilayer board costs are (1) board layout, (2) choice of specification, (3) selection of a base and B-stage material, (4) selection of copper weight, (5) choice of multilayer board thickness and overall size, (6) choice of terminal area pad size and hole sizes, and (7) provision for maintainability, testing, and repair.

Figure 20.1 identifies most of the cost factors that should be considered when MLPV costs are estimated and indicates how the costs influence final pricing. The base costs used in the figure are for a three-layer board having an area of 24 in² and 25 plated-through holes of 0.050-in diameter. Adding layers, increasing the number of holes, increasing board area, and/or changing hole size increase costs by certain approximate percents. Base costs used in the charts for quantities of 500 multilayer boards are \$10.00 each for a 1/16-in-thick board, \$10.50 each for a 1/8-in-thick board, \$11.00 for a 3/32-in-thick board, and \$12.50 for a 1/2-in-thick board. Percent changes from those base costs are estimated by drawing a horizontal line from each factor to the percent line. The percentage total can then be applied to the appropriate base cost. If hole size is other than 0.050 in, an additional percent correction is involved. It must be emphasized that the costs are only a guide to the variables in pricing. Other factors can easily raise or lower the estimates, and the base costs are continually subject to change. Each board fabricator should develop his own version of these charts.

TECHNOLOGY DESCRIPTION

The processes for fabricating multilayer boards are basically extensions of the methods used for double-sided boards; therefore, they will be described in less detail. As identified by the board fabrication process, multilayer printed boards can be broken down into two basic types: laminated and built-up multilayer printed boards. Other than the difference between the laminated approach and the screened or deposited insulation used in the built-up approach, the various methods differ only in the way in which interlayer connections are made. As in single-side printed-wiring technology, there are countless variations of the basic multilayer processes. Only the methods identified with significant process departures will be discussed here. Multilayer concepts classified as laminated printed boards are the clearance-hole process and the plated-hole process.

Multilayer circuits normally consist of two or more layers of separate and different circuit patterns which have been laminated together under heat and pressure to produce a strong unit. Figure 20.2 illustrates the lay-up details of two multilayer boards, one composed of single-sided boards (a) and the other of double-sided boards (b).

Frequently there will be a conformal coating on both sides of the multilayer circuit, though it is not always essential. A multilayer circuit will consist of layers of circuitry

separated by an insulating dielectric and usually interconnected between layers by one of several methods which will be dealt with later. The multilayer circuit may also incorporate certain shielding and ground plane layers as well as discrete circuit patterns. As compared with a conventional harness, the multilayer bus bar distribution cable offers predetermined and accurately reproducible layer-to-layer capacitance.

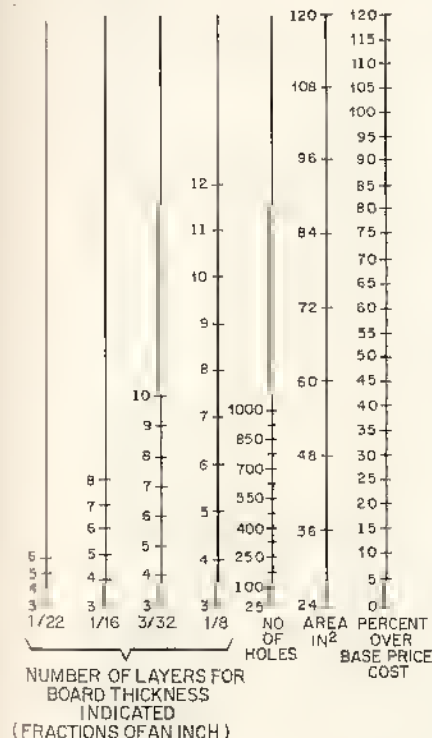


Fig. 20.1 Cost comparisons.

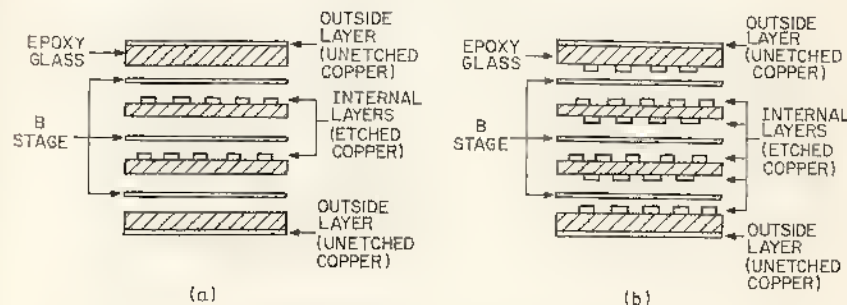


Fig. 20.2 Lay-up details. (a) Four-layer board. (b) Eight-layer board.

In general the process steps in producing multilayer boards follow in the sequence listed here:

a. Artwork

1. Check registration of layers and inner layers.

2. Photosensitize layer (or screen-on resist if tolerances permit, in which case skip steps 3 and 4).
3. Expose photosensitized layer.
4. Develop exposed layer.
5. Etch and remove resist.

b. Multilayer

6. Laminate layers
 - a. Perform lay-up (stack up layers with a sheet of B-stage material between layers).
 - b. Perform lamination (under heat and pressure to cure B-stage material and form a monolithic multilayer structure).
7. Drill and clean holes.
8. Deposit copper to plate through holes.
9. Photosensitize outer surfaces of multilayer board (or screen-on resist if tolerances permit, in which case, skip steps 10 and 11).
10. Expose photosensitized surfaces.
11. Develop exposed surfaces.
12. Plate copper to required thickness on pads and holes.
13. Plate tin-lead or noble metal as a resist.
14. Strip resist and etch.

5. Multilayer Types Multilayer printed boards using plated-through holes can be divided into several types:

a. Internally Connected

1. Boards containing inner layers each composed of a single sheet of copper. The layers are commonly called planes, and they may be either ground or voltage planes. Clearance holes are employed to eliminate unwanted connections at the plated-through holes. Multilayer boards containing voltage and ground planes are particularly useful for mounting integrated circuits where those planes provide the power connections. Ground and voltage planes of identical configuration can often be employed in several boards having different external circuitry patterned to suit the operational requirement of the module. The use of the same plane in several different boards will, of course, result in economies.

2. Boards containing inner layers composed of circuit patterns with numerous conductors. Boards of this type are particularly advantageous for highly complicated circuits in which space limitations are an important consideration. Another frequent application of this type of board is as a mother board for connecting other modules together. In that application the multilayer board serves as a very efficient cable. It is light in weight, and it occupies little space while providing mechanical support for the modules.

3. Boards containing both circuit patterns and planes as inner layers. Such boards, of course, provide the basic advantages of both types. They are being found increasingly useful in high-density packaging that employs numerous integrated circuits.

b. Externally Connected. In addition to the basic purpose of a multilayer printed board—that of being an electronic interconnecting device—other interesting and useful characteristics of multilayer construction are available to the designer. Since inner layers of the plane type have considerable surface area and planes may be spaced close together in the multilayer laminate, a relatively large electrical capacitance value exists between the plane surfaces. The capacitance value can be utilized, or used in conjunction with the distributed inductance in the multilayer, to obtain a certain fixed impedance value. An increasingly common component produced by printed board multilayer techniques, although considerably simpler in design, is the bus bar. This device makes use of many features of multilayer construction in the distribution of relatively large amounts of power.

Another method of connecting the layers in a multilayer board is sometimes used. It does not employ plated-through holes, and no connections between conductors within the laminate are made. Instead, all conductor terminations are brought to the exterior of the board and all connections are made externally by some suitable method such as wire wrap. This particular type of board finds its most prevalent and useful application as a mother board.

6. Plated-Through Holes The plating method of making interconnections through holes in the dielectric substrate of printed circuits has been well established and proved out in two-sided circuit boards, and the use of the technique on circuit boards of more than two layers is an extension of the art (Fig. 20.3). This would appear to be the most widely accepted technique, and in practice it must account for 80 to 90 percent of the multilayer circuitry used in the United States.

The cross-sectional sketch of a typical multilayer circuit shows surface conductors or pads on both sides of the sandwich, encapsulated circuits and lands, and through-hole connection holes with electroplated sides which provide for interlayer connection.

a. Advantages. The principal advantages of the plated-through-hole method include greater miniaturization than is obtainable by other methods, lower tooling cost for

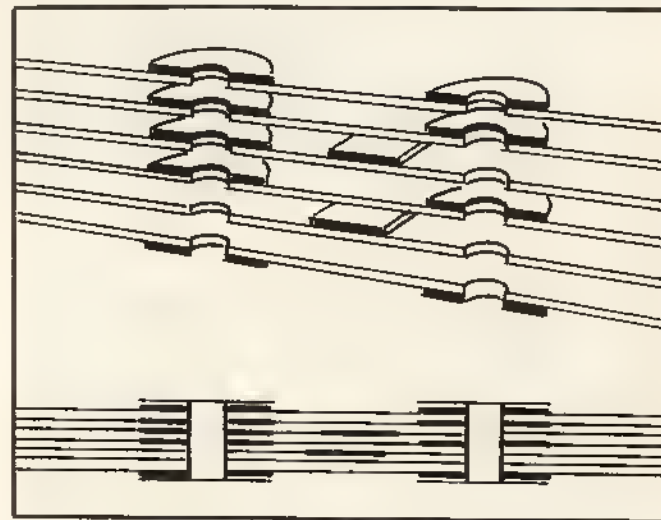


Fig. 20.3 Cutaway view of typical plated-through-hole construction.

production quantities, easier layout and an equal amount of circuitry for each layer, and design flexibility resulting from the approach which could utilize standardized grid centers.

The plated-through interconnection technique has probably enjoyed the greatest acceptance to date, since most of the very miniaturized multilayer circuits benefit from its compactness and its ability to take from three up to ten layers of interconnection without affecting the space requirements adversely.

b. Disadvantages. The user must carefully evaluate potential sources for demonstrated background capability in making through-hole connections on a production basis. Through-hole plating is one of the finer arts of printed circuit manufacture; and since it requires considerable technology, experience, statistical quality analysis, and destructive testing on almost a production line basis, the capability of producing reliable plated-through holes is difficult to attain. If there are disadvantages to the plated-through-hole technique, they might be listed as a considerably lower production yield than by techniques utilizing mechanical means, slower production, and less reparability and reusability.

7. Clearance Holes The clearance-hole technique for interconnection is probably the simplest type to manufacture, and, where space permits, it will probably enjoy economies over other methods (Fig. 20.4). Basically, the clearance-hole technique provides, in graduated diameters, a number of holes adjoined by copper pads or blivets and arranged in registration with each other between the layers of the metallic circuit patterns which are to be interconnected. The clearance holes must be cut or perforated into the layers of

circuitry prior to lamination, and the actual connection is made after lamination of the circuitry by spot-soldering or dip-soldering the multilayer board. The solder will bridge across the desired areas and make a connection from pad to pad through the clearance holes, which get progressively smaller with each layer so that there is copper on the desired layers to which the solder can attach itself.

a. **Advantages.** Among the advantages of the clearance-hole technique is relative simplicity, which can naturally be assumed to provide for a higher production yield and less

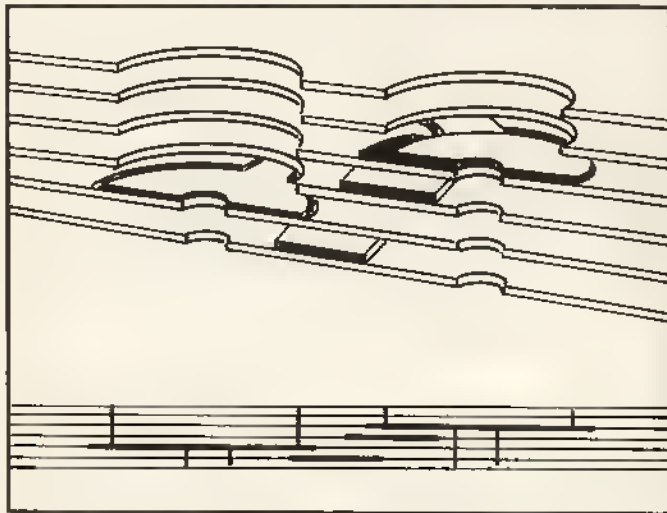


Fig. 20.4 Cutaway view of typical clearance-hole construction.

fallout. Another advantage is that, in most cases on a per square inch basis, a multilayer circuit will have a cost advantage over circuits produced by using the other techniques to be described.

It can be assumed in most cases that the clearance-hole technique will provide a more reliable through connection than several of the other techniques, since the connections lend themselves to ready examination for soundness under visual or moderate magnification check. It is probable that three-dimensional circuits utilizing clearance-hole interconnection techniques also permit more repairs and more reusability in removal or interchange of components which may be mounted thereon.

The clearance-hole technique also has advantages for short-run production in that the cost of the circuit and tolling necessary to make small quantities will frequently be less than with the other techniques, whereby more setup is required.

b. **Disadvantages.** The major disadvantages of the clearance-hole technique are inherent in the construction and are principally related to the fact that, because of the necessity for larger clearances on succeeding layers of circuitry, some of the potential miniaturization characteristics must be sacrificed. It is recommended that there be a full pad of copper around the through connection with a minimum of 0.020 in clearance at all stages of layers. Thus, with a hypothetical feed-through hole diameter of 0.040 in, the second insulating layer would have to have a hole clearance of 0.080 in, the third insulating layer would have to have a clearance hole of 0.120 in, and the insulating layer over the third layer of circuitry would have to have a clearance hole of 0.160 in, etc. Some of this disadvantage, as it may relate to the more compact plated-through-hole interconnection technique, may be offset by the fact that it is practical to start with the initial circuit layer having a feed-through hole of 0.020 in (desirable for mounting many transistors basings) and go through successive layers in 0.020 in clearance per side increments or 0.040 in overall additions per layer. It would then be possible to build up a fourth conductor layered circuit with the maximum diameter being 0.140 in and a three-layered circuit with the maximum clearance hole diameter of 0.100 in.

It is obviously not practical to set up standards for center-to-center minimums by using the clearance-hole interconnecting technique until such time as the number of layers can be determined. The center-to-center distances could, however, be assumed to grow on modular grids in increments of 0.050 in, which allows for the 0.020-in clearance on all sides plus an additional 0.010 in for tolerance.

8. **Comparison of Plated-Through Hole and Clearance Hole** Several factors determine the choice of clearance-hole or plated-through-hole boards. Some specific advantages and limitations of each type of laminate relative to each other, as well as some design criteria, are given in Tables 20.1 and 20.2. Under laboratory conditions, some of these values may be exceeded or improved. For example, special six-layer clearance-hole boards have been

TABLE 20.1 Relative Advantages and Limitations of Clearance Hole and Plated-Through Boards

Clearance hole		Plated-through	
Advantages	Limitations	Advantages	Limitations
1. Lower tooling cost for prototype or short run.	1. Higher tooling cost for large run.	1. Lower tooling costs on long-run production.	1. Tooling higher on prototype or short-run production.
2. Modules may be soldered and unsoldered more times.	2. Minimum pin spacing 0.100 in.	2. Minimum pin spacing to 0.050 in.	2. More difficult to unsolder modules for repair or replacement.
3. Reliability higher due to soldering direct to pad rather than plate through hole.	3. Increasing number of clearance holes limits number of layers.	3. Greater number of layers possible.	3. Probably somewhat lower reliability.
4. Faster production.	4. No layer-to-layer communication except at edges.	4. Provides inter-layer communication.	4. Slower production.
5. More production yield.	5. Decreasing space for circuits on succeeding layers because of clearance holes.	5. Equal space for circuitry on each layer.	5. Lower production yield rate.
	6. Cannot be produced with edge-board connector tabs for external interconnection.	6. Can be produced with edge-board connector tabs.	

TABLE 20.2 Relative Features of Multilayer Printed Wiring Interconnection Processes

Factor	Clearance hole	Plated-through
Number of layers	2 to 7	2 to 12
Minimum distance between pad centers	0.100 in	0.050 in
Minimum conductor width	0.010 in	0.005 in (0.010 in if on exposed outside layer)
Minimum conductor spacing	0.010 in	0.004 in (internal)
Minimum pad diameter	0.062 in (with 0.055-in clearance hole)	0.030 in
Minimum insulation per layer	0.004 in	0.004 in
Maximum copper conductor thickness	5 oz (0.0068 in)	5 oz (0.0068 in)
Means of external interconnection	All except edge-board connector	All

produced with a pad diameter of 0.060 in and a pin spacing of 0.075 in. It is possible to achieve plated-through laminates with pin spacings as low as 0.050 in, but once again there are many process controls.

9. Drilling One of the most important operations in producing a multilayer printed board is drilling holes which will, in a following operation, be plated through. Drilling is important, since the condition and smoothness of the hole wall and the cleanliness of annular rings of copper connected to inner conductors will determine the quality and reliability of the multilayer. The selection of the correct drill, drill speed, and feed rate controls the quality of the drilled hole. The designer of a multilayer board should try to limit the number of different drill sizes required in any one setup.

SPECIFICATIONS AND STANDARDS

Although some single- and double-sided board specifications and standards (Chap. 1) apply to multilayer boards, there are several which apply primarily to multilayer printed wiring. The most commonly used are covered briefly in the following paragraphs.

10. Institute of Printed Circuits Specifications

a. IPC-ML-910: Design and End Product Specification for Rigid Multilayer Boards. This specification covers design and end product requirements for rigid multilayered printed boards consisting of several layers of conductor patterns on insulating bases separated from each other by an insulating material (generally similar to the base material) and interconnected by a continuous metallic interlayer connection.

b. IPC-ML-950: Performance Specification for Multilayer Printed Wiring Boards. This document contains recommendations for the requirements and test methods for judging the performance and characteristics of multilayer printed boards, irrespective of their method of manufacture, when the boards are used for the mounting of components.

c. IPC-ML-975: End Product Documentation Specification for Multilayer Printed Wiring Boards. This specification covers end product documentation requirements for multilayer printed boards consisting of several layers of conductor patterns on insulating bases separated from each other by insulating material (generally similar to the base material) and interconnected by a continuous metallic interlayer connection.

11. Department of Defense Specifications

a. MIL-STD-1495: Multilayer Printed Wiring Boards for Electronic Equipment. This standard establishes design requirements governing rigid multilayer printed boards consisting of three or more conductor patterns on insulating bases separated by an insulating material and interconnected by a continuous metallic interlayer connection (plated-through hole) and design considerations for the mounting of parts and assemblies thereon. The design criteria (such as electrical spacings) contained in this standard are predicated on the requirement that end item assemblies (circuit cards) shall be conformally coated.

b. MIL-P-55617: Plastic Sheet, Thin Laminate, Metal Clad, for Printed Wiring, Primarily for Multilayer. This specification covers the requirements for fully cured, metal-clad, thin-laminated plastic sheets to be used primarily for the fabrication of multilayered and thin printed wiring for electrical and electronic circuits. The specification applies to sheets when the base laminate does not exceed 0.031 in in nominal thickness.

c. MIL-C-55636: Glass Cloth, Resin Preimpregnated, B-Stage, for Multilayer Printed Wiring Boards. This specification covers semicured (B-stage), resin preimpregnated, glass cloth (prepreg), intended for bonding together the individual layers of multilayer printed boards.

d. MIL-P-55640: Printed Wiring Boards, Multilayer, Plated-Through Hole. This specification covers rigid plated-through hole multilayer printed boards consisting of three or more conductor patterns on insulating bases separated one from the other by an insulating material and interconnected by a continuous metallic interlayer connection.

GENERAL DESIGN CONSIDERATIONS

When the printed wiring assemblies for an electronic equipment are being developed, it should be remembered that there is an optimal limit to the size and cost of any discrete unit or functional subassembly. It will be uneconomical or inefficient to increase size or complexity beyond that limit, because the unit then becomes too specialized and there-

fore not easily interchangeable or too costly to be thrown away. Investigations into various approaches to achieve a desired component density indicate that multilayer concepts lend themselves to a great many packaging requirements.

The following is a list of the factors that should be taken into account in any multilayer board design.

a. Design

1. Board size (maximum and minimum)
2. Number of layers
3. Hole sizes
4. Composite thickness
 - a. Thickness of internal layers
5. Conductor widths (internal and exposed layers)
6. Terminal area (internal and exposed layers)
7. Spacings
 - a. Center-to-center mounting holes
 - b. Terminal area to terminal area
 - c. Terminal area to conductor
 - d. Conductor to conductor
 - e. Terminal area to board edge
 - f. Conductor to board edge
 - g. Conductor to mounting hole
8. Size of grid
9. Tolerances (design)—both vertical and horizontal
10. Plating (type and thickness)
11. Drawing requirements
12. Terms and definitions

b. Performance and Properties

1. Tensile strength
2. Flexure strength
3. Vibration
4. Mechanical shock
5. Temperature cycling
6. Thermal shock
7. Moisture resistance
8. Fungus resistance
9. Salt spray (corrosion resistance)
10. Warp or twist
11. Solderability and resolderability
12. Insulation resistance (bulk and surface)
13. Dielectric withstanding voltage
14. Machinability
15. High- and low-temperature storage
16. Conductor temperature rise
17. Interlayer connection resistance (ΔR)
18. Conductor resistance
19. Composite texture
20. Flame resistance
21. High altitude
22. Continuity

12. Initial Design Development The major factor in poor manufacturing yields of printed boards is the attempt to produce boards that realistically cannot be made to meet their requirements. That situation arises from one or more of the following conditions.

1. The design of the board does not take into account sufficient manufacturing variation.
2. The manufacturing equipment does not produce the required tolerances.
3. Either the process is not under rigid control or manufacturing personnel are not following the prescribed processing standards.

Multilayer boards with hole centers spaced 0.100 in or greater have sufficient room for at least one conductor between the terminals, and that makes possible the placement of

0.020-in conductors on 0.020-in spacing and allows registration and hole location tolerances of at least ± 0.005 in, all of which are within normal printed circuit manufacturing practice. Production of such boards does not present any great difficulty; and the boards can be produced with adequate yields, which tends to lower the cost. In order to manufacture boards with hole spacings of less than 0.100 in successfully, all parameters and tolerances must be decreased. In the case of hole spacing on 0.050-in centers, they must be halved. When the design requires 0.010-in-wide conductors and spacing, ± 0.001 -in tolerance on layer thickness, or ± 0.002 -in tolerance on hole location and layer registration, the price goes up.

13. Overall Design-Production Cycle An important aspect of system production with which the system design engineer must be acquainted is the lead time required for the first completed system to be developed, manufactured, and tested. It is a very frequent occurrence that the length of this period is underestimated.

The usual sequence starts after the basic logic design has been made and the subdivision of the system into discrete units has been completed. At that point the mechanical drawing for each board, or, if all the boards have an identical layout, the master drawing, is prepared. The logic diagram and mechanical drawing for each board must have the same coding of modules to facilitate circuit layout and assembly after manufacture of the boards. From the mechanical drawing, a master artwork template on an enlarged scale which contains all the terminal points of the board may be made. Copies of the enlarged master can then be used to generate the circuit layout. In an automated system the standard board pattern or characteristics would be coded and stored in the computer. In a manual design system a rough feasibility layout can be made by using single-line conductors. The layout does not require that the conductors be laid out very accurately; it simply guides future artwork preparation by showing which terminal areas are connected together. At this stage, very close engineering supervision and liaison may be necessary to ensure as much as possible that interconnections will exhibit the required electrical characteristics, that interference is minimized, and that all critical circuits are isolated. After successful translation of all the required interconnections to the circuit layout, a careful double check should be made. Automated systems use similar techniques, except that the proper wiring-rule checks are usually built into the computer programs.

Depending upon the size and complexity of the multilayer boards, the circuit layout period will take anywhere from one to four weeks per board type including checking. The time required to produce the accurate enlarged master from the mechanical drawing is usually about 2 weeks. In an automated system a comparable time might be 2 days on the computer-interactive graphic display, and then 6 h on a numerically controlled plotter.

The need for the highest-quality artwork for multilayer boards cannot be emphasized too strongly. The difficulty of producing multilayer boards with all the necessary processing tolerances does not permit the complication of artwork tolerances adding to the problem.

Some multilayer board manufacturers build products such that a mislocation of a terminal area anywhere on the board on the 1:1 glass by more than ± 0.002 in is cause for rejection of the glass. That is an example of the demands placed on preparation of artwork for multilayer boards.

The manufacturing cycle for the boards themselves is usually 3 to 5 weeks. Thus the total elapsed time from the moment when the electrical and mechanical design of a system has been firmed until the last board of a given system is ready for assembly is a minimum of 4 weeks and can be as long as 2 months. Changes in logic design which must be incorporated during the design and prototype manufacturing stage certainly do not contribute to any schedule improvements and should be minimized.

Another problem with multilayer systems is that there is no assurance that the system will operate as specified until the first complete unit is assembled and tested. The hand-wired breadboards do not exactly reproduce the conditions of the completed system, since the parameters of conductors in multilayer boards are quite different from those of wires on breadboards and the electrical characteristics in the actual boards might change accordingly. Complete simulation systems that make the probability of success on the first attempt somewhat better have been developed in the last few years.

14. Tolerance Analysis The calculation of the parameters associated with the sum of various tolerances is required to ensure that the multilayer board will "go together," i.e.,

perform a tolerance analysis. Two methods of tolerance analysis are in popular use; they are worst case and statistical. In worst case analysis, the various tolerances are simply assumed to take on the maximum values consistent with obtaining the most mismatch. For example, let us consider a component lead which is required to fit in a drilled hole. Both items have plus and minus tolerances on the nominal dimensions. In worst case analysis it is required that the maximum allowed lead diameter fit into the smallest hole. If the lead and hole are designed accordingly, the design is 100 percent safe. However, in the event of encountering the minimum lead-maximum hole situation, there might be an exceedingly loose fit.

In statistical analysis the assumption is made that the tolerances of any particular parameter are normally distributed about the specified values. Then the total tolerance variation is the root mean square sum of the individual tolerances.

Figure 20.5 and its associated table, Table 20.3, show the fundamental characteristics of multilayer boards and typical values for basic board features.

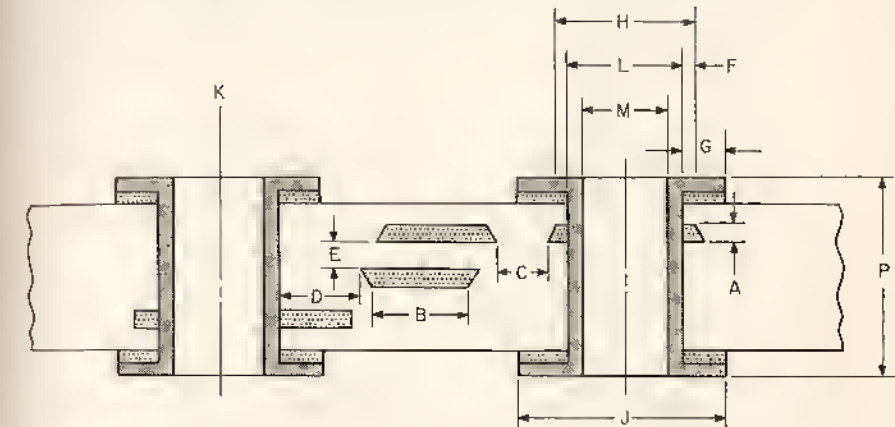


Fig. 20.5 Fundamental characteristics of multilayer boards.

Dimensions and Tolerances for Multilayer Printed Wiring

Symbol	Characteristic	Class		
		I	II	III
A	Conductor thickness ^a and tolerance	IPC-CF-150	IPC-CF-150	IPC-CF-150
B	Conductor width	0.020 min.	0.101 min.	0.005 min.
B _{tol}	Conductor width tolerance ^b	—	—	—
C	Coplanar conductor spacing	0.015 min.	0.010 min.	0.005 min.
D	Conductor-to-hole spacing	0.010 min.	0.010 min.	0.010 min.
E	Layer-to-layer spacing	0.004 min.	0.003 min.	0.002 min.
F	Internal layer annular ring	0.001 min.	0.001 min.	0.001 min.
G	External layer annular ring	0.002 min.	0.002 min.	0.002 min.
H	Internal layer terminal area	$H_{min} = B_{tol} (-) + 2F + K + L_{max} + N + 2R_{max}$		
I	External layer terminal area	$J_{min} = B_{tol} (-) + 2G + K + L_{max} + N + 2R_{max}$		
J	Hole location tolerance ^c	0.014 diam.	0.010 diam.	0.006 diam.
K _{tol}	Unplated hole diameter tolerance ^d	—	—	—
L _{min}	Plated hole diameter, min.	0.014 diam. TP	0.012 diam. TP	0.010 diam. TP
M _{tol}	Plated hole diameter tolerance ^e	—	—	—
N	Layer registration ^f	—	—	—
P _{tol}	Board thickness tolerance ^g	—	—	—

REFERENCE: IPC-ML-910

^aAll dimensions and tolerances are in inches unless otherwise specified.

^bSee IPC-ML-910, paragraph 3.1.2.3.

^cFor board dimensions up to 6 in.

^dSee IPC-ML-910, paragraph 3.1.5.2.

^eL 101 plus 2 times minimum hole plating.

^fFor distances within a 6-in square.

^g $\pm 10\%$ of P_{nom} and 0.007 min.

For military applications the data in Table 20.4, from MIL-STD-1495, serve as a guide to the tolerances and allowances used in the design process. Because of tolerance buildup, the trade-offs involved in arriving at the permissible limits for each particular tolerance for a particular design should be recognized. The data in Table 20.4 are intended to show the increasing difficulty of producing boards with tighter tolerances, but

TABLE 20.3 Dimensions and Tolerances for Multilayer Printed Wiring
(Reference IPC-ML-910)

Symbol	Characteristic	Class I	Class II	Class III
A	Conductor thickness and tolerance	IPC-CF-150	IPC-CF-150	IPC-CF-150
B	Conductor width	0.020 min	0.101 min	0.005 min
B_{tol}	Conductor width tolerance	(See IPC-ML-910, paragraph 3.1.2.3)		
C	Coplanar conductor spacing	0.015 min	0.010 min	0.005 min
D	Conductor-to-hole spacing	0.010 min	0.010 min	0.010 min
E	Layer-to-layer spacing	0.004 min	0.003 min	0.002 min
F	Internal layer annular ring	0.001 min	0.001 min	0.001 min
G	External layer annular ring	0.002 min	0.002 min	0.002 min
H	Internal layer terminal area	$H_{min} = B_{tol}(-) + 2F + K + L_{max} + N + 2R_{max}$		
J	External layer terminal area	$J_{min} = B_{tol}(-) + 2G + K + L_{max} + N + 2R_{max}$		
K	Hole location tolerance	0.014 diam	0.010 diam	0.006 diam
		(For board dimensions up to 6 in)		
L_{tol}	Unplated hole diameter tolerance	(See IPC-ML-910, paragraph 3.1.5.2)		
M_{min}	Plated hole diameter, minimum	$\frac{1}{8} P_{max}$	$\frac{1}{4} P_{max}$	$\frac{1}{10} P_{max}$
M_{tol}	Plated hole diameter tolerance	$(L_{tol} + 2 \times \text{minimum hole plating})$		
P_{tol}	Board thickness tolerance	$(\pm 10\% \text{ of } P_{nom} \text{ and } \pm 0.007 \text{ min})$		

NOTE: All dimensions and tolerances are in inches unless otherwise specified.

TABLE 20.4 Composite Board Design Guidance¹

	Preferred	Standard	Reduced producibility
Number of conductor layers (maximum) ²	6	12	20
Thickness of total board (maximum), in	0.100	0.150	0.200
Thickness of dielectric (minimum), in	0.008	0.006	0.004
Minimum conductor width (or figure 2 value, ³ whichever is greater), in			
Internal	0.015	0.010	0.010
External	0.020	0.015	0.010
Terminal area diameter tolerance and conductor width tolerance (add to the end item requirements), in			
Internal	+0.005	+0.004	+0.002
External	+0.0075	+0.005	+0.003
Minimum conductor and terminal area spacing (or table 1 value, ³ whichever is greater), in	0.020	0.010	0.005 ⁴
Conductor spacing tolerance (add to end item requirements), in			
Internal	+0.004	+0.002	+0.001
External	+0.0075	+0.005	+0.003
Feature location tolerance (master pattern, material movement, and registration), rtp—in			
Longest board dimension, 12 in or less	0.008	0.007	0.006
Longest board dimension, over 12 in	0.010	0.009	0.008
Master pattern accuracy, rtp—in			
For the layer	0.004	0.003	0.002
For all layers	0.005	0.004	0.003
Board thickness-to-hole diameter (maximum)	3:1	4:1	5:1
Hole location tolerance, rtp—in	0.007	0.005	0.003
Drilled hole tolerance bilateral, minimum, in			
Minimum hole diameter			
Up to 0.032	0.004	0.002	0.002
0.033 to 0.063	0.006	0.004	0.002
0.064 to 0.188	0.008	0.006	0.004

¹ For 2 oz/ft² copper.

² The number of conductor layers should be the optimum for the required board function and good producibility.

³ "Figure 2" and "table 1" referred to are in MIL-STD-1495.

⁴ For internal layers only. rtp = radius of true position.

they do not express the limits attainable or permissible for any single aspect of board design. That is, the data should not be interpreted as end item board requirements.

MECHANICAL DESIGN FEATURES

15. Thickness and Size A multilayer board should be no thicker than 3 times the diameter of the smallest plated-through hole. Although it is possible to have the board up to 5 times thicker than the diameter, cost will be much higher and reliability will be reduced. This restriction on thickness is the result of the fluid kinetics of plating baths, either copper or solder.

The designer should specify the minimum thickness of the B stage. The distance between adjacent copper layers should not be less than 0.004 in. The minimum thickness between copper layer and adjacent core should be specified also, but it should not be less than 0.0006 in (Fig. 20.6). Beyond that, it is wise to let the board maker use his own preferred methods for building a board at minimum cost. As before, his methods depend on his inventory and the processes that he has developed.

In order to reduce manufacturing costs, it is common practice to process a multiplicity of smaller boards on a larger panel and then separate them in the last steps of the manufacturing cycle. This practice can be used in the manufacture of multilayer boards also. If the tolerances are wider, larger panels with more small individual circuits can be processed through manufacturing without any difficulties. When closer terminal area spacing and tight tolerances are demanded, the size of such panels must be reduced to ensure accuracy of registration and hole locations. Therefore, a smaller number of multiple circuits on the panel can be processed simultaneously. That is reflected in the manufacturing costs, and the combined increase due to tighter tolerances and smaller panels can be as much as 80 percent more in the cost of individual circuits.

The final or finished board dimensions, the length and width as well as the thickness, affect board cost. Most copper-clad base material is available in 24 × 36-in sheets or in multiples of 1-ft increments. A 1-in margin must be allowed around the final size of the board to accommodate tooling holes. Then limit the final size of the board so that a good yield is obtained from each standard-sized sheet. The purchaser always pays for the scrap; and the amount of scrap can be considerable, as Fig. 20.7 shows.

Another useful purpose of the margin around the finished multilayer board is for

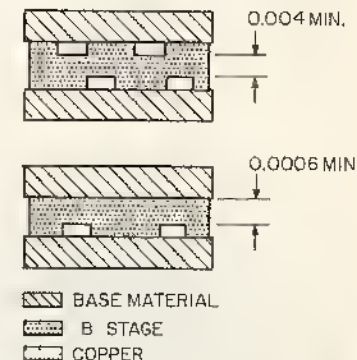


Fig. 20.6 Minimum B-stage dimensions.

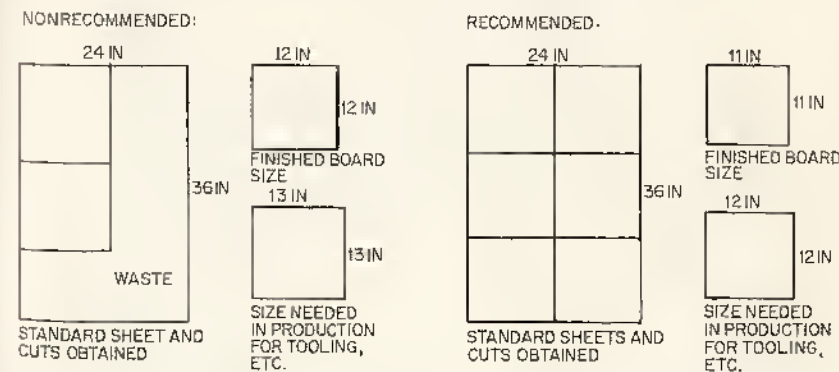


Fig. 20.7 Adjusting finished board size to the standard sizes of clad sheet material will reduce the waste for which the purchaser pays.

sampling. Extra holes and lines should be made in the margin so that the sample can be tested and sectioned through for examination after processing to determine condition of the finished board. Such samples, known as coupons, will provide an excellent indication of the integrity of the entire board.

The final board thickness will depend upon the number of conductor layers and on the electrical layer-to-layer spacing requirements of the design. In multilayer printed wiring the increase in cost is not directly proportional to the increase in the number of conductor layers. For example, doubling of the number of layers from five to ten will probably increase costs by only 30 percent. In many cases additional layers may so simplify the design and reduce conductor density and increase conductor size that costs are actually lowered. However, if the number of conductor layers exceeds ten, the extra-layer costs may increase at a rapid rate.

There is no standard rule for the optimum thickness of printed wiring nor for the number of multilayer conductive layers. The difference in base laminate costs due to variations in thickness has a small effect on the finished printed wiring cost.

Occasionally the limiting factor for printed wiring thickness is the diameter of the smallest hole, especially when the holes are plated through.

16. Holes and Terminal Areas

a. **Plated-through-hole Size (Fig. 20.8).** A plated-through hole must be sufficiently large to allow the required deposit of metal on its wall. Minimum hole diameters should be three-

fourths of the board thickness after plating. By that ratio, between 0.0005 and 0.002 in of metal can be deposited in the wall of each hole with standard plating equipment. Metal deposited upon the surface will be up to two times the amount deposited in the hole. Smaller holes can be plated through, but this ratio of metal deposited on the surface to that deposited within the holes increases.

The plated-through hole size is related to the total board thickness by the sophistication of the manufacturing plating technology. There is an upper limit to the ratio of the multilayer printed wiring thickness to plated-through hole size (plating ratio), as previously mentioned. The minimum plated-through-hole size is thus defined by

the overall board thickness, component lead diameter, and the applicable plating ratio. Generally, the strength and reliability of a through hole will increase with size.

The minimum plated-through-hole size is often related to the multilayer board thicknesses by the degree of sophistication of the plating process. In general, long narrow holes are more difficult to plate than are holes of large diameter in relatively thin boards.

b. **Terminal Area (Pad) and Hole Size.** Pad size has a direct relation to the producibility of the multilayer board. A good rule of thumb is to have the pad at least twice the diameter of the hole to be drilled through it, but in today's high-density technology that luxury is rarely possible. The drilled hole must be from 0.004 to 0.007 in larger than the required nominal size to allow for hole size reduction after plating through the hole.

Numerically controlled drills are capable of making between 40 and 180 hits (drill strokes) per minute. If there are holes of several sizes in the board, remember that it takes approximately as long to change a drill as it would to drill 1700 holes. Therefore, the smaller the number of hole sizes, the less the drilling downtime and the lower the cost.

Hole size is another area that affects cost. If the diameter of a hole is over 0.125 in, the hole will probably need to be drilled by hand while the board is held in a special fixture. That involves higher cost and reduced accuracy.

ELECTRICAL DESIGN FACTORS

The electrical factors which affect the design of a multilayer circuit are basically the electrical requirements of the system of which the circuit is to become a part. The

conductor thickness, width, and spacing are primarily functions of the voltages and currents required by the system. In addition, conductor spacing, location, and routing may be affected by the types of signals being carried when those signals may interfere with each other or be affected by stray capacitance. This factor also may control insulation thickness between layers and even dictate shielding between layers, between conductors, or both.

Because multilayer circuitry is comprised of a series of individual discretely designed and etched circuit boards, the basic design parameters that apply to the individual layers are the same as those that apply to conventional one- or two-sided circuitry. When slight differences occur, they are usually beneficial and result from the effective encapsulation of the individual circuits during the laminating process.

17. Conductor Spacing and Clearances The separation (spacing) between adjacent conductors must be sufficient to ensure electrical isolation. In addition, if components are to be soldered to terminal areas, the separation must be adequate to prevent solder bridging between the terminal area and other external conductors. The nominal value of this spacing has an associated tolerance due to the location and conductor width tolerances inherent in artwork and fabrication. For that reason, spacing is normally specified as a minimum value to ensure that the desired separation is maintained regardless of tolerances.

Multilayer printed conductor spacing is the same as for single- and double-sided printed boards; the table pertaining to coated boards is used for internal conductor spacing values. The one exception is that spacing of 0.005 in is allowed for internal conductors of 0 to 15 V.

Frequently it is required to interconnect conductors on two separate layers of a multilayer board with plated-through holes which are separated by an intermediate conductor layer without shorting to the intermediate layer. Thus arises a need to keep an area concentric to the plated-through hole in the intermediate layer free of conductive material. In the case of a voltage or current distribution plane, simply etch out a circular area of copper concentric to the plated-through hole of sufficient diameter to allow for any misregistration during multilayer board fabrication to ensure the minimum space requirements on the board are met. The diameter of the clearance area must be larger than the maximum outside diameter of the plated-through hole by an amount sufficient to meet the minimum space requirement and allow for production tolerances.

18. Capacitance It should be noted that actual capacitive coupling is a complex value. Each conductor of a group located in close proximity will contribute some cross-coupling to the other conductors, and therefore calculated values might be exceeded considerably in actual boards. To minimize such capacitive coupling, it is advisable to alternate the direction of conductor routing on adjacent layers. The basic capacitor equation serves as a useful approximation:

$$C \text{ (pF/in)} = \frac{\text{conductor width (in)} \times 10^{-8} \times \text{dielectric constant}}{4.45 \times \text{dielectric separation (in)} \times 10^{-9}}$$

This equation is generally in close agreement with empirical values obtained, but it tends to give a value somewhat on the lower side of actual measurements.

The capacitive coupling between conductors can be minimized by limiting the length of conductors running in the same vertical plane on two layers. Caution must be exercised with circuits which are located over the shield or ground planes, since in such cases the entire length of a conductor is capacitive-coupled to the plane and allowance must be made in design for it.

Four different test patterns (Fig. 20.9) were used to obtain the capacitance data in Figs. 20.10 to 20.13. The test patterns were:

Pattern 1. Three sets of conductors with varying conductor spacing, one side solid-shielded.

Pattern 2. Three sets of conductors with varying conductor spacing, both sides solid-shielded.

Pattern 3. Three sets of conductors with varying conductor spacing, each conductor individually shielded on both sides.

Pattern 4. Three sets of conductors with varying conductor spacing, unshielded.

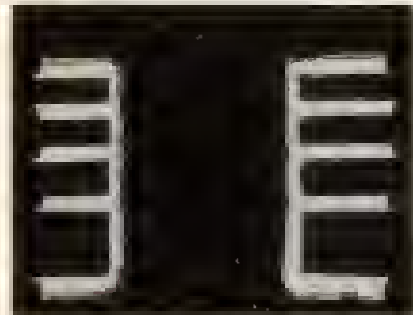


Fig. 20.8 Cross section of plated-through hole.

Multilayer circuit capacitance values are controlled by careful design, routing, and spacing of conductors and thickness of insulation.

Additional tests have indicated that 0.030-in-wide conductor lines spaced 0.004 in apart had an average capacitance of 7 pF/in. (Variations in fabrication can alter that value by 10 to 15 percent.) When capacitance is critical, shielding as shown in Fig. 20.9 can be included to minimize its effect.

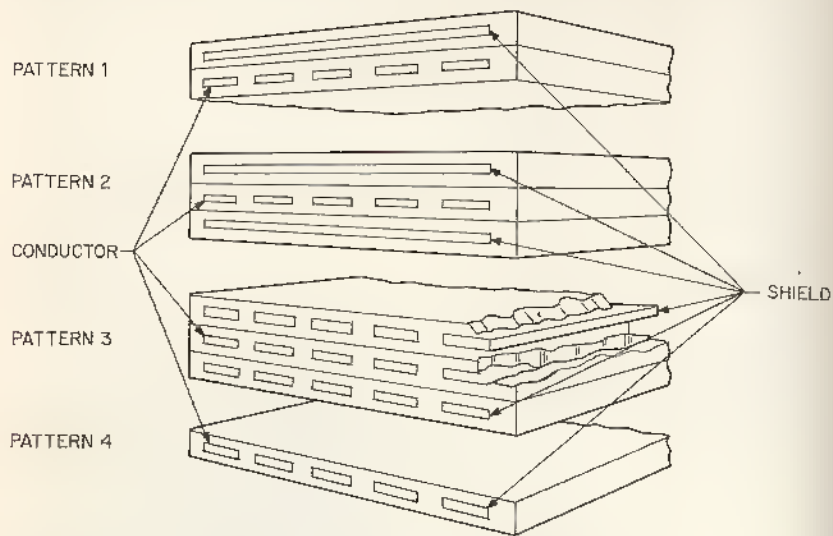


Fig. 20.9 Test patterns of capacitance shielding.

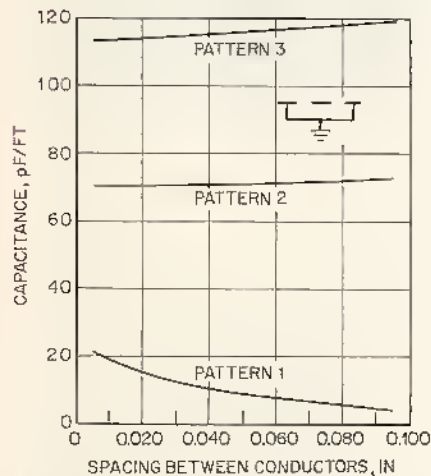


Fig. 20.10 Capacitance between a center conductor and two adjacent grounded conductors in the same plane.

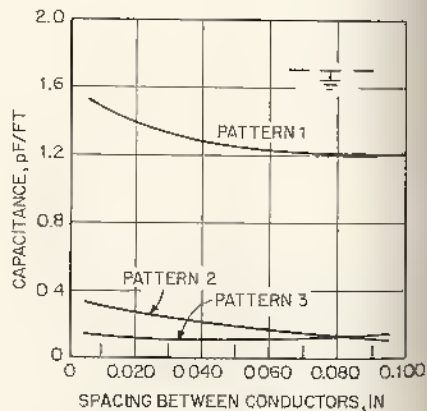


Fig. 20.11 Capacitance between two alternate conductors with the center conductor grounded.

LAYOUT

Three classes of multilayer printed boards have been established by the Institute of Printed Circuits; see Table 20.3. They range from the most difficult to produce (class I) to the easiest (class III). Line and board size tolerances, hole size and location tolerances,

and physical requirements are typical factors which determine how a particular board is classified.

It is highly recommended that the outer layers of a multilayer board be designed to have no circuitry and only pads. The reason is that the most common process problem is "resist circuit breaks," in which the resist has not acted as desired. It is therefore desirable to reduce the number of areas to be processed with resist after the board has been laminated. When laminated, the board represents a rather expensive assembly, and risks should be minimized during the processing that follows. Having only pads on the outer layers will limit a major risk.

The preparation of the layout is a very interesting topological problem. Several papers on the mathematical background for an optimum conductor routing and the use of computers for that purpose have been presented. Some electronic companies have already successfully computerized a multilayer layout.

It appears that the number of variables and restrictions necessary for computerized layout preparation is quite large and that it requires the services of a very complex

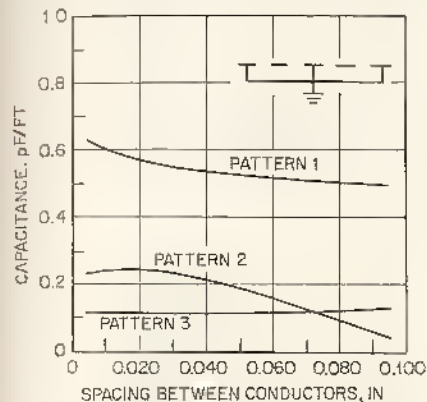


Fig. 20.12 Capacitance between two alternate conductors with the center conductor and each adjacent conductor grounded.

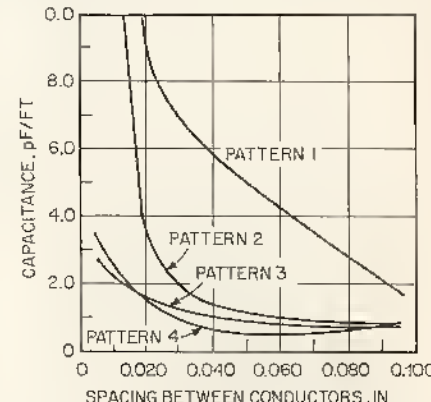


Fig. 20.13 Capacitance between two adjacent conductors.

installation. Besides, the software preparation for such programs is quite time-consuming and difficult. Therefore, the average engineer who wants to design a multilayer board will, for some time to come, have to make the layout by manual operation. A general procedure for manual layout operation is as follows:

To prepare the circuitry layout for individual layers, a number of thin Mylar sheets are placed over the terminal point master. The number of these sheets is equivalent to the expected number of layers, and the sheets are taped together so that each one is easily accessible. The overlay package is then placed over a light table and layout work can start. The routing of approximately 80 percent of the interconnections in a fairly complicated board goes rather fast; it is the last 20 percent that cause problems because a path must be cleared for them among the interconnections already laid out. This occasionally requires the transfer of existing connections from one layer to another to clear a path for the new ones.

It should be noted here that one man has to work on one board from start to finish of the layout; the work cannot be subdivided among a number of drafters. The checking should be done by someone other than the designer.

The physical layout of a multilayer printed board (MLB) will depend, among other things, on the dimensions and tolerances associated with most, if not all, of the following:

1. Line width
2. Impedance value and tolerance control, if controlled impedance is required
3. Individual layer thickness
4. Number of layers
5. Layer function
6. Total MLB thickness

7. Plated-through-hole size
 8. Pad size
 9. Grid dimension
 10. Clearance area diameter
 11. Minimum space between conductive surfaces
- The fabrication of these items will be dependent upon
12. Artwork fabrication accuracy
 13. MLB production or fabrication tolerances

In determining the parameters (items 1 to 11) for a multilayer board, it is best to start with a grid and set of fabrication tolerances (items 12 and 13) and then compute items 1 to 11. If the values are compatible, everything fits. Otherwise, modifications which usually result in having the MLBs fabricated to tighter tolerances and correspondingly higher cost will have to be made.

Modification of the grid is not usually practical if a judicious choice was made originally. Popular grid sizes are usually multiples of 0.025 in. That is sometimes dictated by the component manufacturers, who usually offer fixed lead devices on 0.025-in grid centers. Other grid dimensions may also be used.

Sometimes it is desirable to be able to place the center of a pad surrounding a plated-through hole and the center of a signal line on adjacent grid positions. That implies the pad surrounding plated-through holes could be placed only on alternate grid positions.

All of the procedures involved in the generation of a multilayer board design can be summarized in the following six basic steps. (In an automated system computer programs and numerically controlled plotters perform many of these functions.)

Step 1. Make the early decisions:

- Develop the logic diagram.
- Select component types.
- Determine packaging, interconnection, and component attachment methods.

Step 2. Partition the system:

- Establish the number of input-output points per board.
- Decide on pin or finger terminations.
- Determine the maximum number of modules per board.

Step 3. Start physical design of board:

- Determine hole sizes.
- Evaluate conductor routing.
- Decide on discrete component location, when applicable.

Step 4. Prepare formal drawing of board:

- Include preceding information plus new details such as mounting holes.
- Indicate forbidden areas.
- Investigate need for heat sinks.

Step 5. Code each module and termination:

- Assign numbers to each element of drawing with corresponding numbers on logic diagram.
- Identify modules with heaviest interconnection traffic and relocate holes where necessary.

Step 6. Prepare artwork:

- Produce enlarged masters of each board.
- Determine, manually or with computer, optimum conductor routing for each layer.
- Check out final interconnection design.
- Draw or tape enlarged masters of each layer.
- Reduce masters and prepare glass negatives.

19. Early Decisions At the earliest stage of the design, an important decision must be made about the division of the entire system into individual boards in order to make the best use of space, achieve the least costly throwaway or replacement level, ensure the greatest design flexibility, and reduce weight, overall cost, and number of interconnections. When the physical dimensions of a system are given, the only problem for the design engineer is to make certain that all required components, modules, and boards will fit in the specified volume. When the configuration of a system has not been defined, the problem of selecting the optimum board size becomes acute. An increase in board size, with a corresponding increase in the interconnection complexity, increases the cost of a board, not in direct proportion, but at a much slower rate.

A different approach is to limit the board size by interconnecting only a small but repetitive subfunction, such as an adder, shift register, or logic unit. With such a design, a system will consist of a large number of smaller boards.

20. Input-Output Input-output terminations have a strong influence on multilayer board design. If the number of points on a board that require connection to the outside world is equivalent to the number of internal connections, then the use of multilayer boards will be quite inefficient and some other packaging method should be used. A good rule of thumb is that in a multilayer board no more than 20 percent of the total number of terminal points should be devoted to input-outputs; the smaller the percent the better the design. The present trend to increase the number of integrated circuits on a board decreases the number of input-output points per package.

Since in most designs the output-input terminals are located close to one side of the board or on one edge (in the case of finger terminations), sufficient room must be allocated to accommodate an adequate number of those terminations. There is a practical limit to the density of terminal points in multilayer boards. In the case of pin-type connectors, which usually have two or more rows of pins, the pin separation should be no less than 0.1 in and pin diameter no greater than 0.03 in; otherwise, there will be no room to bring connectors between the first row of pins and the next.

In the case of finger-type terminations, it is just possible to accommodate a row of single-sided fingers on 0.05-in centers. If the fingers are located on both sides of the board, the minimum center-to-center locations must be 0.1 in; otherwise, the plated-through holes will connect the fingers from both sides and it will be impossible to bring separate signals to the fingers on each side of the board.

There are some advantages in using pin-type connectors to bring input-outputs into multilayer boards. The finger types limit the overall thickness of the board, which may present occasional problems in locating all the desired interconnections within the predetermined number of layers. There may also be problems if electrical requirements call for greater vertical separation between layers and the overall thickness of the board is limited by the mating connector. In an emergency it would be easier to use a pin-type connector with more pins. That change will usually require a minimum of additional space on the board, but it is impossible to add extra fingers when all positions available on the board edge have already been used.

The maximum number of termination points which can be placed on a given board determines, in many cases, the maximum number of modules a board can handle. To determine the number of input-outputs, encircle a portion of the logic diagram to be placed on a single board and then count the number of connections crossed by such a hypothetical line. Of course, avoid encircling more modules than there is room for physical accommodation on the board. This method of isolating the subsystem with a minimum number of input-output connections automatically groups the functionally most highly interrelated modules on the same board. Conversely, it will determine the maximum number of modules that fit on a single board.

21. Conductor and Terminal Area Layout Widths should be selected that produce the desired conductors after final artwork reduction. As conductor widths and spacings are varied by manufacturing tolerances, minimum line and spacing dimensions and allowances for tolerances must be considered. In preparing the layout, care must be exercised to assure that adequate allowances are made for artwork preparation and manufacturing.

Terminal areas (pads) should be treated in the same manner as conductors. Terminal areas are usually round, although square pads with a hole in the center are sometimes used. When a pad size is selected, the outside dimension (diameter) should be sufficient to assure a minimum annular ring after all tolerances have been taken into account. In general, the largest pad possible, considering available space, should be used regardless of hole size.

If a pad around certain holes is undesirable, the diameter of the pad selected for the artwork should be smaller than the finished hole. It will then be removed by drilling.

Where ground or voltage planes appear as inner layers, the clearance holes required to eliminate unwanted connection to the plated-through holes must be clearly delineated. This is essential because these holes are fabricated in the plane prior to laminating, whereas all other holes are drilled after lamination of the multilayer structure.

22. Layout Feasibility Concepts After the board size has been determined, all module positions distributed, pad or hole location fixed, and input-output methods selected, the formal drawing of the board can be started. The drawing should contain all the above information, as well as information about other holes such as mounting holes to secure the boards to the frames or to secure connectors or other hardware to the board. It is important to indicate the existence of any forbidden areas, in which no conductors should be placed, during layout and artwork preparation lest conductors on inner layers be cut through by mounting holes drilled in the final operation prior to assembly.

Before full design information can be included in the drawing, some consideration must be given to the appearance of the surface of the board. The surface will depend greatly on the type of component connection selected for the system. If flat packs are used and connected by parallel gap welding or some form of solder reflow techniques, adequate tabs must be provided on the surface. Since the welding or soldering operations are apt to weaken the bond of the tab to the plastic substrate, the tabs should never be shorter than 0.1 in to achieve sufficient anchoring. When surface tabs are used for connection of flat packs, the plated-through holes which bring the signals from internal layers can be staggered to provide more room for conductor routing. Dual in-line packages have leads placed on 0.1-in centers to allow easier conductor routing between holes.

Holes in multilayer boards are often very closely spaced, and there is not much room between holes on the surface for sizable pads. Therefore, it may be necessary to use "landless holes," the soldering of which is done to the barrel of the hole. Landless holes have the plating contained entirely on the hole walls. The plating overhang around the hole orifice forms a small metallic ring. Landless holes have minimized the occurrence of solder bridging between closely spaced terminal areas and neighboring conductors, a frequent problem in boards with very small spacings. It is also highly recommended that no conductors be placed on the surface of boards connecting integrated circuits. The tolerances of plated-up conductors on the surface are much wider than those on internal layers and in any locations that might present electrical problems and difficulties in handling or assembling such boards. The only exceptions are situations in which high-frequency requirements demand a microstrip configuration for impedance matching. Then the conductors must be located on the surface of the board.

By having all interconnections on internal layers, the multilayers provide ample room for placing heat sinks on the surface. For flat-pack or dual in-line mounting, continuous strips can be placed directly under the body of the package and extended either to selected output points or to any edge of the board for heat extraction. That feature becomes very important when heat cannot be removed by convection and is entirely dependent upon conduction, as in space. Usually such heat sinks are produced during fabrication of the board, i.e., they will consist of copper foil overlaid with whatever materials are used in plating through the holes. The overall thickness of heat sinks manufactured by this process will be no more than 0.006 in. If that cross section is insufficient, then the heat sinks can be fabricated separately from thicker foils and bonded to the board surface.

The geometry of flat and dual in-line packages lends itself to a much more efficient heat sink design than TO-5 cans. With TO-5 cans it is very difficult to place heat sinks underneath the body. The only solution is to place the ground plane on the surface; then the grounded pin of the module acts as the heat-transfer link to the plane and through it to the outside. With such construction there is the possibility that the board will be more apt to warp because it has a heavy metal layer on one surface.

From the logic diagram it is fairly clear which modules have the most interconnection traffic between them, and they should be grouped in close proximity. Modules which have critical electrical connections between them should be closely placed. Obviously, the modules with the most input-output signals should be located close to the input-output area.

At this point it is hardly possible to forecast the complete conductor routing plan and thus arrive at an optimum module distribution on the board. Simply adhering to the above rules is usually sufficient to make a reasonable distribution of modules on a given board. Occasionally, more efficient interconnections can be achieved by relocating some modules from their preassigned position during the layout phases of the design.

A somewhat different situation exists in regard to input-output points. The exact pin locations of all signal outputs should not be defined at this stage. If both module locations

and output positions have been defined before the layout is made, the result may be a highly inefficient conductor routing scheme requiring many more layers than one in which at least some degree of freedom for the layout is available.

Before actual layout of the circuitry can start, a few rules about conductor routing must be developed. In multilayer boards it is advisable to use conductors as wide as the design permits. If conductors are made very narrow, the etching tolerance can be a substantial percent of the conductor cross section and might create some electrical disturbances. Preparation of the artwork can be accelerated if a uniform conductor width is used throughout the entire board, but some narrowing down in critical areas is permitted occasionally. Also, if power distribution buses instead of planes are used, they should be made wide to minimize the voltage drop.

Spacing considerations are of utmost importance. During layer lamination processing, two parallel conductors on the same layer will not move relative to each other, but the entire layer might shift a little. If provisions for adequate spacing between conductors and drilled holes are not made, the possibility of shorts or low resistances between various signals is greatly increased. If any trade-offs in spacing are necessary, they should be made at the expense of spacing between adjacent conductors on the same layer, so long as electrical parameters permit, and never between the conductor and terminal areas. It is advisable to calculate from the beginning how many conductors of a given width, properly spaced, can be brought between two terminal points in various areas of the board and then stick to those ground rules during the layout of the conductors and never exceed the specified conductor number.

23. Master Drawing Preparation The most basic and possibly the most important consideration for accurate and economical production of multilayer printed boards is the initial preparation of a correct and complete part drawing. The drawing is used by the manufacturer in determining characteristics of the part and the most efficient method of production. The same drawing will be utilized by the quality control and inspection departments of both the user and manufacturer to establish the quality, reliability, and acceptance of the finished part. The information contained on the part drawing should be as follows:

1. An outline drawing of the part showing all dimensions and tolerances.
2. A complete description of the base material (that is, G-10, G-10FR, or other selected material).
3. A cross section of the desired end item indicating layers (by number) and critical dimensions (Fig. 20.14).
4. Circuit pattern images for each layer identified by layer number (optional but desirable).
5. Thickness of copper, by layer, taking into consideration the plated-through hole process, which will increase the copper thickness of the outside layers.
6. A table showing hole diameters, tolerances, number of holes, and a coding system so that holes shown in the hole table can be identified on the multilayer circuit board (Table 20.5). (Large numbers of small holes of the same size need not be counted, but the quantity of holes occurring in the lowest number is an aid in eliminating errors in manufacture.)
7. A note stating which holes are plated through.
8. All machining dimensions with correct tolerances.
9. Any nomenclature or markings other than the circuit pattern.
10. Name and number of any other specifications that may apply to the part.

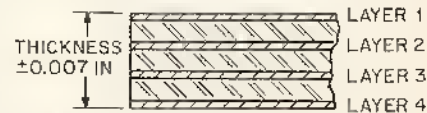


Fig. 20.14 Cross section of multilayer printed board.

TABLE 20.5 Typical Table Showing Proper Hole Designation.

Hole	Quantity	Diameter	Tolerance
A	2	0.375	± 0.005
B	13	0.093	± 0.002
C	7	0.125	± 0.003
D	All other	0.040	± 0.003

It is necessary that the circuit patterns be positioned with respect to the outside edges of the finished board. Positioning may be accomplished in a variety of ways, one of which is as follows: Locate two or more tooling holes within the board configuration, preferably close to the edge. These holes are placed on each layer of the artwork, and the circuit pattern is positioned with respect to the holes. Not only are the tooling holes used in preparing the artwork, they are necessary for checking layer-to-layer registration and machining of the multilayer board. All dimensioning should revolve around these holes (Fig. 20.15).

Although two tooling holes are used by several companies, many designers prefer using three tooling holes to guard against a reversed image on one or more circuit layers. The

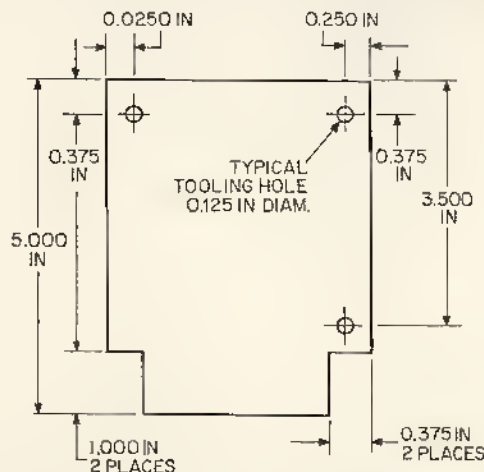


Fig. 20.15 Typical dimensioning with respect to tooling holes.

third tooling hole can be of particular value when a high degree of symmetry in circuit and hole pattern exists.

24. Digitizing Many companies today have semiautomated systems. The design is done manually, but the information is extracted and put into a form to operate artwork-generating equipment. Digitizing is the technique used. The primary function of digitizing is to extract the xy coordinate information manually from the design layout through the use of equipment called a digitizer. The layout is used as a template for the preparation of artwork or as a means of accomplishing the coding required for a numerically controlled plotter.

The digitizer consists of an output unit and a table equipped with a reading head mounted on an xy carriage. The motion of the head is coupled to encoding devices. The encoders feed digital information on the xy coordinate position of the reading head to the output unit. The output unit converts that information to the proper code as interpreted by the photoplotter and punches it into paper tape, cards, or magnetic tape on command from the operator.

The operator places the design layout on the digitizing table and manually moves the reading head (cursor) along the conductors and terminal areas on the layout. That can be considered a tracing operation. As the operator progresses from point to point, he or she presses a record button at each grid the coordinates of which are to be recorded. A trace overlay can be used to assist the operator on missing digitizing points.

There are many different orders in which information is digitized. Many companies put identification information at the start of the NC tape or cards. A usual procedure is to code all of the terminal areas first, grouped into sizes, and then the conductors. Standard symbols for specific patterns, board outlines, connectors, etc. are sometimes coded in advance and inserted into the NC data stream when needed. This reduces digitizing effort.

The terminal area (hole) location (xy coordinate) information may be converted to

provide fabrication information for drilling the printed board. Conversion is accomplished by means of a computer processing sequence for specific drilling equipment. Such a sequence would create an NC drill tape or a tabulated coordinate listing for use in producing the NC tape. The final output is composed into a completed job package and becomes ready for the photoplotting sequence in the development of the artwork.

Because of the wide range of forms of information either created or required to satisfy each company's needs, no single piece of an equipment ensemble can solve all the problems. Therefore, the need arises to implement a comprehensive man-machine interface for desired results.

Depending on the complexity of the design layout, the digitizing steps can run from 1 to 60 h to complete a single printed board design. That operation, therefore, becomes the pacing item within the design department. Changes in or revisions of previously digitized data become another pacing item. It is usually speeded up by means of computer programs which can add and delete information and thereby allow a fast turnaround for photoplotter completion.

ARTWORK GENERATION

After the layout is completed and checked out, actual artwork preparation is started. The preparation can be by the conventional method of reproducing the master pad layout for each layer on Mylar, placing it over the pencil sketch which shows the interconnections, and then taping the conductors in the same fashion as for regular boards. That part of the job can be distributed among many drafters, which will reduce lead time. After the artwork is complete and the conductor widths and spacing are checked out, the masters are reduced by camera to 1:1 size and glass or film negatives are made. The negatives serve as tools for the production of the layers. The artwork operation usually goes faster than the circuit layout.

Today many companies use artwork-generating machines which can transfer the information from the layer layout sheets by digitizing on a tape and then supplying the tape to numerically controlled artwork generators that produce the final 1:1 masters very accurately.

The exactitude required on completed multilayer parts is so great that the tolerances necessary for the manufacturing processes use almost all available space and leave practically no room for mistakes in the artwork. Ideally, the 1:1 masters should have the location of terminal areas, line widths, and spacings held within 0.001 in. of true. Therefore, the work on the enlarged artwork masters must be performed with the utmost care and the most accurate drafting equipment available must be used.

It should be evident that very serious attention must be paid to all steps in the design and preparation of artwork and that the multilayer board design is a rather long process. Sufficient time must be allowed for those steps to ensure that satisfactory boards will be manufactured.

The minimum space cannot be the same on the artwork as on the end-product board, since the conductor pattern width and the location of the pattern on the artwork have associated tolerances. Also, the conductor width has a fabrication tolerance. If the tolerance on the artwork allows the conductors to come closer together than nominal and the fabrication tolerances on conductor widths are positive (wider than nominal), the minimum space requirement on the end-product board can be violated. Therefore, an associated tolerance is needed on the artwork to compensate for those factors.

25. Registration One of the most critical parameters in producing multilayer boards is registration between layers. Proper registration of layers in the finished product depends upon the precision of the artwork, which cannot be overemphasized. The use of the following sequence in laying out multilayer artwork is a way to ensure good registration:

1. Prepare a master layout. It should show the tooling holes and the complete hole (pad) pattern positioned dimensionally to them. At least two reduction targets and the exact final (reduced) dimension between them also should be included.
2. Photograph (1:1) the master layout and make as many copies as layers.
3. Lay out the conductor pattern for the internal layers and eliminate any unwanted pads (holes).
4. Lay out the conductor pattern for the two external layers.
5. Check registration of all layers.

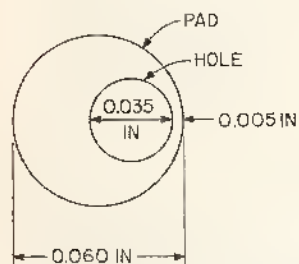
In laying out the artwork, consideration must be given to many items to assure production of a satisfactory end product.

TOOLING

No amount of material and process control can compensate for inaccuracies in the original artwork. A good artwork and tooling system is essential if adequate registration is to be achieved on the finished board. Table 20.6 outlines errors resulting from tooling and other

TABLE 20.6 Tooling and Other Errors (Absolute); Reduced to 1:1 on 17 × 11 = in Board

Tooling	Error, in	(Error) ²	Datum
Reference grid	±0.001		Center edge
Grid pinning	±0.0005		Center edge
Camera	±0.002		Center
Camera pinning	±0.0007		Center
	±0.0042	$17.6 \times 10^{-6} \text{ in}^2$	
Material instability			
Original artwork	±0.001		Center edge
Photographic film	±0.0007		Center
Film processing	±0.0005		Center
Laminate	±0.004	$16.0 \times 10^{-6} \text{ in}^2$	Center
Process			
Pad position	±0.0015		Center edge
Drilling	±0.001		Center
	±0.025	$6.25 \times 10^{-6} \text{ in}^2$	
		$41.6 \times 10^{-6} \text{ in}^2$	
RMS error (radial)		±0.0065 in	



sources. In the table we see a list of features which contribute to the final degree of misregister. The values shown, which are absolute with respect to a theoretical grid and are scaled to apply to a 17 × 11-in panel, are those which can be expected even when the artwork is prepared and used in a controlled environment, the material movement is minimized by the use of center panel and center edge datums, good-quality photographic and drilling equipment is used, and pinned transfer is used throughout the entire sequence.

The resultant of any system may be expressed in practical terms as the pad diameter required to contain a hole and leave a specified clearance between the edge of the hole and the edge of the pad. The figure associated with Table 20.6 shows that, if a 0.006-in clearance is required, the pad diameter must exceed the hole diameter by a minimum of 0.025 in.

Artwork and tooling systems are expensive to install and should obviously not be overspecified. However, in making the choice, one should assess a system's ability to cope with future design and manufacturing requirements.

26. Corner Marks Corner marks and part outlines are useful features for the finished product, especially when hand-tooling does not exist. As a minimum, corner marks should

appear on the artwork for each layer, for toolmaking, and for registration checking purposes.

27. Reduction Targets Each piece of artwork should have two reduction targets along with the exact final (reduced) dimension in between. The dimension will be used to determine whether the artwork has been reduced to proper size.

TEST COUPON

It is recommended that a simple test coupon be incorporated into each panel processed in order that certain destructive tests may be carried out.

The design of the coupon and the tests carried out will depend to some extent on the board specification and the processes used, but the coupon can be quite small and would normally be expected to include continuity, solderability, and microsection patterns. The continuity pattern may be monitored for resistance and intermittance under thermal or mechanically induced stress. The solderability and microsectioning patterns may be used to evaluate surface solderability, the resistance of the plating and interlayer bonds to solder immersion, and plating thickness and adhesion and also to give a subjective but very useful assessment of joint quality.

MARKING

Markings such as nomenclature, part number, and component symbols are frequently desired on multilayer printed boards. Such markings can be made either by etching simultaneously with the conductor pattern or by printing in ink.

Etched markings are usually lower in cost, provided they are large enough. When etching is used, the letters, numbers, or symbols must be sufficiently large to withstand processing, because they are subject to the same variations as conductors. Also, care must be exercised that the marking does not reduce the space between conductors below the specified minimum. Etched markings should be included on the original artwork.

Markings printed by using epoxy-based ink may be smaller than etched markings. Small printed markings will be easier to read, and the ink is as good an insulator as the base laminate. Also, ink markings remain readable after dip soldering, since solder will not adhere to ink. Markings may be ink-printed on the base laminate, the conductor pattern, or both. Since a separate printing operation is required, a separate piece of artwork containing the desired markings is required. If markings are required on both sides of the board, two pieces of artwork are needed.

Multilayer Materials Selection and Specifications

CHARLES G. HENNINGSEN

Vice President, Insulectro, Mountain View, California

Introduction	21-2
Multilayer Laminate Manufacture and Quality Control	21-2
Laminate and Prepreg Evaluation, Specification, and Quality Control	21-2
1. Surface and Appearance Standards	21-2
2. Color Standardization	21-3
3. Peel Strength	21-3
4. Solder Resistance	21-3
5. Electrical Testing	21-3
6. Dimensional Stability	21-4
7. Testing of Prepreg	21-4
8. Volatile Content	21-4
9. Resin Content	21-5
10. Resin Flow	21-5
11. Tack Time	21-5
12. Cured Thickness	21-5
13. Visual Inspection of Prepreg	21-5
Special Concerns for Multilayer Materials	21-6
14. Mechanical Properties of Clad Multilayer Laminates	21-7
15. Electrical Properties of Multilayer Clad Materials	21-8
General Problems	21-8
16. Moisture Absorption	21-8
17. Properties of Prepreg Materials	21-8
18. Storage of Prepreg Materials	21-10

INTRODUCTION

Metal-clad laminates used in the manufacture of multilayer boards are generally restricted to a base thickness of less than 0.031 in. These so-called thin laminates for multilayer printed boards have some variance in properties from conventional standard laminated materials. Therefore, it is advisable to use different specifications and design parameters in describing and specifying them. This chapter will be concerned only with properties other than those discussed in Chapter 2 on standard laminated materials. It therefore can be assumed that properties not discussed in this chapter are common to both thin and standard materials.

MULTILAYER LAMINATE MANUFACTURE AND QUALITY CONTROL

The manufacture of thin multilayer laminates is almost identical with the manufacture of the thicker counterparts. There are, however, several areas in which most laminators vary from standard processes. First, in thinner metal-clad laminates, thinner layers of base material, usually glass cloth, are used. That means more careful control of the treating process is required both in maintaining the resin-to-glass ratio and also in the partial curing of the prepreg material. Because multilayer boards require a more critical application of thin materials, the entire treating and pressing operation is usually enclosed in a "clean room" atmosphere to control more of the minor defects which would not affect the thicker board. In the laminate buildup area, more care is taken to eliminate dust particles from the press pans and from the foil prior to lamination.

On the unclad side of thin single-sided laminates, a textured surface is often obtained by roughening the steel press pans or by using combinations of papers or foil pressed against the unclad surface. The textured surface increases the bondable area of the unclad side and therefore allows greater bondability when the multilayer board is pressed. Most thin multilayer materials are produced to a much tighter thickness tolerance than is necessary on standard materials. The tighter tolerance is obtained by more closely controlling the resin content of the base material, the thickness tolerance of the glass, and the flatness variation of each press pan. Each thickness of base material from 0.002 to 0.030 in usually requires a different combination of glass cloths to make up the final thickness. Therefore, in the construction of thinner materials, many different fine-weave cloths are employed, depending upon the final thickness required. Removal from the press, trimming to size, and inspecting and packing require a great deal of care and handling owing to the ease of breakage of thin clad materials. Generally, thin clad laminates are shipped in smaller sheet sizes to minimize the customer's breakage, and the materials are often shipped in cut-to-size panels. In final inspection, the laminate is subject to more rigid standards. The base material is more carefully inspected for thickness, as well as base continuity, since thinner materials can permit electrical breakdown if uniformity of the base laminate is not maintained.

LAMINATE AND PREPREG EVALUATION, SPECIFICATION, AND QUALITY CONTROL

It is important to evaluate and develop a specification for thin materials in the same manner as for standard materials. It is equally essential to establish the specification on the basis of the multilayer manufacturing process and electrical and mechanical design criteria to minimize the inconsistencies of multilayer board production. Many of the standards outlined for general laminates can also be used in establishing those criteria, but there are some exceptions or modifications which must be made when thin clad materials and prepregs are evaluated.

1. Surface and Appearance Standards Pits and dents in the clad copper surface have been outlined from MIL-P-13949E in Chap. 2. The military specification for thin laminate metal clad is MIL-P-55617A, and it defines the following point system for measuring pits and dents in the clad surface:

a. Grade A. The maximum point count for pits and dents combined in any square foot of surface must be less than 30. The point system used is as follows:

Longest Dimension, in	Point Value
0.005 to 0.010	1
0.011 to 0.020	2
0.021 to 0.030	4
Over 0.031	30

b. Grade B. The maximum point total for pits and dents combined in any square foot is defined as no pit with the longest dimension greater than 0.015 in and no more than three pits greater than 0.005 in in any square foot. There should be no scratches greater than 140 μ in in depth, with the exception of within $\frac{1}{4}$ in of the edge of full-size sheets.

It is also recommended that the user consider specifying the surface roughness or finish of the unclad side of single-sided multilayer materials, since an extremely smooth finish even without release agents can prove difficult to bond in the multilayer laminating process.

2. Color Standardization Because the finished multilayer board is made up of several layers of thin material, the user may find it important to specify a color standard. Most laminators today use an undyed or pigmented resin to minimize opacity. The coupling agent, usually a silane on the glass cloth, and the basic coloration of the epoxy resin themselves provide most of the final color of the thin material. Usually, by working with a laminator, the user can develop a coloration range which will produce the minimum of variance in color on the finished multilayer board. The alloy treatment on the copper also can add coloration, depending on the alloy and the amount of it imbedded in the epoxy.

3. Peel Strength The test and test pattern for peel strength of thin foil-clad materials is the same as that outlined for standard laminates, but thin materials must be bonded to a rigid surface, such as $\frac{1}{16}$ -in aluminum plate, as specified in MIL-P-55617A. If the bonding is not done, the angle between the copper and the base material will not be 90° consistently during the peel test and irregular peel strengths will be determined.

4. Solder Resistance Since thin laminates for multilayer boards have a larger surface-to-volume ratio than standard materials have, they exhibit a higher degree of water absorption. Generally, therefore, it is recommended that the following procedure be used in testing the solderability of thin materials. All of the copper can be removed, or a solder test pattern can be used.

1. Dry the samples in an air-circulating oven at 150°C for at least 1½ h.
2. Place specimens in a desiccator and allow to cool to room temperature.
3. Immerse the specifications edgewise or at 45° in a solder pot of 500°F for 20 s or the prescribed test period.
4. Examine visually for evidence of discoloration, loss of surface resin, blistering, meandering, delamination, etc.

If the user does not choose to oven-dry the materials prior to solder testing, he should remember that the amount of moisture absorbed will directly affect capacity to resist solder temperatures.

5. Electrical Testing Thin materials should be oven-dried before they are tested electrically. Since they have a large surface-to-volume ratio, an increased amount of water absorption will lower volume and surface resistivity. The samples should be placed in an air-circulating oven maintained at 105°C for a minimum of 1½ h and then cooled to room temperature in a desiccator. All of the electrical tests outlined in Chap. 2 can now be followed. It is also recommended that the user consider adopting a step-by-step voltage breakdown test, since thin laminate materials are much more subject to voltage breakdown through the base materials. The test can be performed as follows:

1. Electrodes to be used are usually metal cylinders 2 in in diameter and 1 in in length. Two-inch copper circles may be etched in the panels to be tested, depending on the type of test equipment used.

2. Dielectric strength of most insulating materials varies with temperature and humidity; therefore, test conditions should duplicate the final conditions that the multilayer board will encounter. Test samples should be kept in a humidity chamber long enough to reach uniform temperature and humidity before step-by-step voltage is applied.

3. Voltage should be applied and increased at a uniform rate from zero to the value

- a. Sheet Form
- 1 Imbedded dirt or stain. No more than one per ft² if less than 0.010 in. None over 0.010 in allowed.
 - 2 Loose dirt. Essentially free of loose dirt.
 - 3 Handling creases. No more than two not exceeding 2 in, no more than one not exceeding 4 in, and none exceeding 4 in allowed in 1 ft². No broken strands of fibers allowed.
 - 4 Cut edges. Frayed ends not to exceed 0.125 in.
 - 5 Loose or missing strings. No more than one missing or loose string in 1 ft².
 - 6 Splices. No splices allowed.
 - 7 Cuts or tears. No cuts or tears allowed.
 - 8 Holes in material. None over 0.062 in in diameter allowed.
 - 9 Knots. Knots caused by wadded thread are not allowed.
 - 10 Resin. Resin must appear uniform over the entire sheet. (No screen-door effect.)
- b. Roll Form All of the standards for the sheet form apply except for the following:
- 1 Handling creases. A crease 1 ft long is allowed.
 - 2 Splices. Five splices in a 100-ft roll are allowed.

TABLE 21.2 Designation of Clad and Prepreg Multilayer Materials

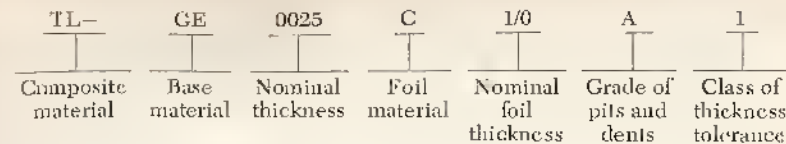
NEMA grade	Military designation MIL-P-55617A	Resin system	Base	Color	Description
G-10	GE	Epoxy	Glass	Translucent green or white	General purpose
FR-4	GF	Epoxy	Glass	Translucent green or white	Epoxy-glass with self-extinguishing resin system generally used for both G-10 and FR-4 applications
Polyimide	None	Polyimide	Glass	Translucent dark brown	Polyimide glass has a high continuous operating temperature with high property retention at temperature; particularly attractive for multilayer applications because of its low α or axile dimensional expansion

SPECIAL CONCERNS FOR MULTILAYER MATERIALS

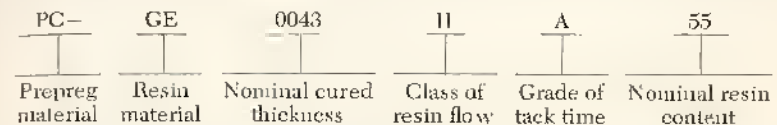
Listed in this section are the unique characteristics of multilayer materials. Consult Chap. 2 on standard laminates for all other properties. The materials to be discussed are given in Table 21.2. The most common method of designating clad multilayer materials is described in MIL-P-55617A.

TABLE 21.3 Thickness Tolerance of Multilayer Laminate

Thickness of base laminate, in	Tolerance of base laminate, \pm in	
	Class 1, standard tolerance	Class 2, close tolerance
0.001 to 0.0045	0.0010	0.00075
0.0046 to 0.006	0.0015	0.0010
0.0061 to 0.012	0.0020	0.0015
0.013 to 0.020	0.0025	0.0020
0.021 to 0.030	0.0030	0.0025



The method used in designating prepreg and B-stage materials is described in MIL-G-55636A.



14. Mechanical Properties of Clad Multilayer Laminates

a. Thickness Tolerance. Nominal thickness and tolerance of base laminate less the foil are specified in Table 21.3. At least 90 percent of the area of the sheet should be within the tolerance given, and at no point should the thickness tolerance vary from the nominal by a value greater than 125 percent of the specified tolerance. Table 21.3 is commonly used throughout the industry; it is taken from MIL-P-55617A. Class 1 is standard tolerance, and class 2 is close tolerance as defined in the specification.

TABLE 21.4 Peel Strengths of Multilayer Laminates

Material	Condition A*		Condition E*, 1/125	
	1 oz	2 oz	1 oz	2 oz
G-10	6	8	5	6
FR-4	6	8	5	6
Polyimide	6	8	6	8
Polyimide (postcure 450°F for 16 h)	10	12	9	10

*Condition A means the material as received and with no special conditioning. Condition E means after a temperature conditioning of 1 h at 125°C.

b. Peel Strength. Table 21.4 lists typical peel strengths both in room temperature conditions and in elevated temperatures. Since postcure of polyimide laminate significantly affects the peel, postcure peel strengths also are included.

c. Dimensional Stability. The α -axis for coefficient of expansion of multilayer materials in the thickness dimension is a critical aspect of multilayer design and fabrication. Figure 21.1 approximates the α -axis expansion over a wide temperature range.

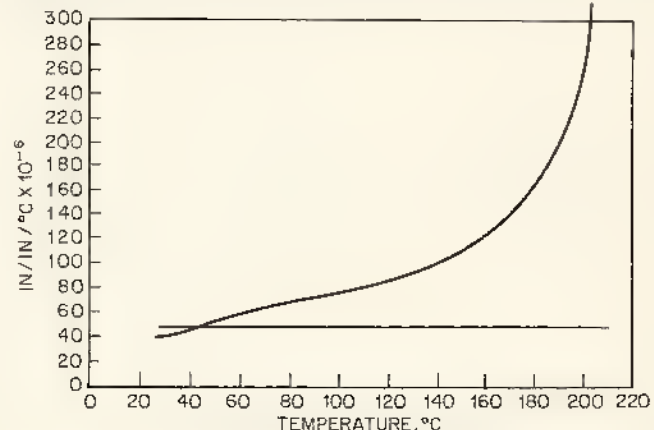


Fig. 21.1 Axile, or z-axis coefficient of thermal expansion.

15. Electrical Properties of Multilayer Clad Materials

a. **Dielectric Strength.** The electrical strength perpendicular to laminations, the average minimum volts per 0.001 in, is G-10, 750; FR-4, 750; polyimide glass, 750.

b. **Dielectric Constant.** The dielectric constant of the final multilayer package varies with not only the thin materials but also with the resin content of the prepreg. Because the dielectric constant of glass cloth is approximately 6.11 and the dielectric constant of epoxy resin is approximately 3.45, the ratio of the two is the final dielectric constant. Figure 21.2 illustrates the effect on dielectric constant as the resin content of the multilayer package is varied.

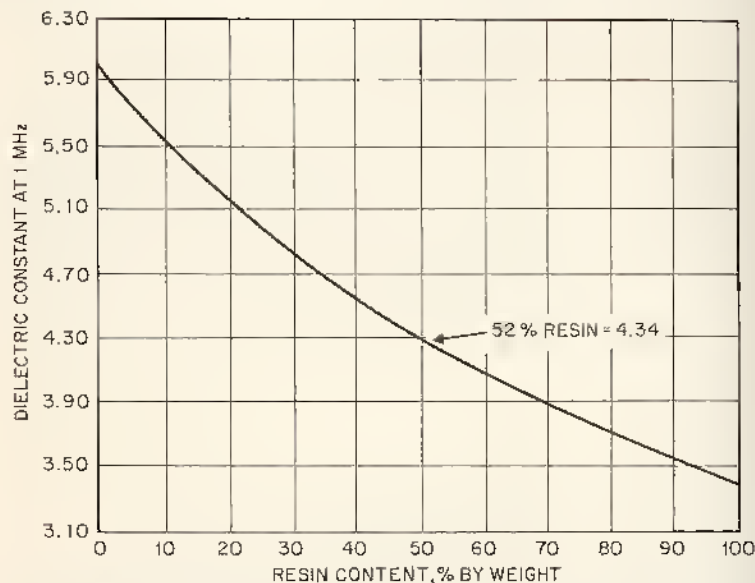


Fig. 21.2 Dielectric constant vs. resin content, FR-4 multilayer; glass $\kappa = 6.11$, resin $\kappa = 3.45$.

c. **Voltage Breakdown vs. Resin Content.** In finished multilayer boards the amount of resin retained between opposing copper runners determines the voltage breakdown. Many manufacturers recommend that 30 percent resin be so retained, as in Fig. 21.3, and an electrical strength of 400 V/0.001 in thereby maintained. Figure 21.4 illustrates electrical strength vs. percent of resin retained between runners at various pressing pressures.

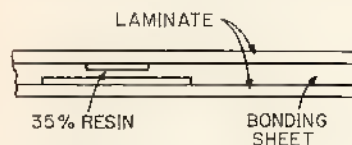


Fig. 21.3 Resin retention between opposing copper runners.

GENERAL PROBLEMS

16. Moisture Absorption Because of the greater surface area of thin clad laminates, the author recommends a bake cycle to drive out moisture before multilayer lamination and before the solder operation. Multilayer boards are much more hygroscopic than standard materials, and Table 21.5 illustrates solder blister resistance at various

degrees of humidity conditioning. A bake cycle of several hours can greatly increase solder blister resistance and thus the reliability of the finished multilayer board.

17. Properties of Prepreg Materials The following properties are typical of epoxy prepregs. Various manufacturers may vary the amount of resin flow and gel time in the particular prepreg construction, but the most common properties used in multilayer packages are given in Table 21.6.

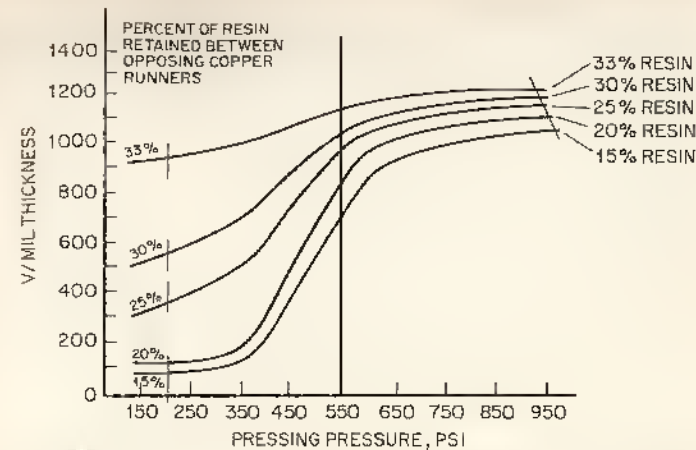


Fig. 21.4 Electrical strength vs. percent of resin retained between runners at various pressing pressures.

TABLE 21.5 Solder Blister Resistance vs. Water Pickup

Condition	Time, s, to blister at 500°F
1. Initial	340
After 1 wk at 72°F, 35% RH	160
After 2 wk at 72°F, 35% RH	120
After 3 wk at 72°F, 35% RH	30
2. Initial	180
After 4 wk in a desiccator	200
After 4 wk at 72°F, 20% RH	125
After 4 wk at 72°F, 90% RH	34
3. Initial	277
After 2 wk at 72°F, 50% RH	39
After 4 wk at 72°F, 50% RH	30
After 8 wk at 72°F, 50% RH	30
4. Initial	405
After 4 wk at 72°F, 50% RH + 2 h at 110°C	57
After 4 wk at 72°F, 50% RH + 6 h at 110°C	55
After 4 wk at 72°F, 50% RH + 16 h at 110°C	70

TABLE 21.6 Epoxy Prepreg Properties

Glass cloth style no.	Cloth thickness, in	Typical prepreg thickness nominal, in	One or two pressed layers thickness average, in	Typical resin (wet) content range, %	Typical resin flow, %	Typical gel time, s	Volatile content maximum, %
104	0.0010 ±0.0002	0.0015	0.0010– 0.0015	70–75 ±5	25–40 ±5	90–120 ±10	0.3–0.5
108	0.0020 ±0.0002	0.0035– 0.0045	0.0025– 0.0030	60–65 ±5	20–40 ±5	90–120 ±10	0.3–0.5
112	0.0030 ±0.0003	0.0045– 0.0055	0.0035– 0.0040	55–60 ±5	25–40 ±5	90–140 ±10	0.3–0.5
113	0.0030 ±0.0003	0.0045– 0.0055	0.0040– 0.0045	50–55 ±5	20–40 ±5	90–140 ±10	0.3–0.5
116	0.0040 ±0.0004	0.0060– 0.0065	0.0040– 0.0050	50–60 ±5	25–40 ±5	90–120 ±10	0.3–0.5
128	0.0070 ±0.0007	0.0080– 0.0085	0.0065– 0.0075	35–45 ±5	15–25 ±5	80–120 ±10	0.3–0.5

18. Storage of Prepreg Materials Since all prepregs are subject to cure at increased temperature, they must be stored in a temperature and humidity environment that has known and controlled maximums. Recommendations vary from manufacturer to manufacturer, but the following standards will apply in general.

a. **Epoxy Prepreg.** The following apply at 50 percent relative humidity or less.

1. Maximum time three months at room temperature (77°F).
2. Maximum six months at refrigerator temperature (35°F).
3. Indefinite at freezer temperature (10°F).

b. **Polyimide Prepreg**

1. Maximum three months at room temperature (77°F).
2. Maximum six months at refrigerator temperature (35°F).
3. Nine months freezer temperature (10°F).

In addition to the above, ultraviolet light or excessive radiation may sometimes act as a catalytic environment for some prepregs. It is also essential to minimize moisture absorption, so prepreg should be enclosed in a moistureproof container. The author further suggests that, prior to pressing multilayer boards, prepreg be given a prebake cycle to drive out all moisture absorbed during the storage period.

Chapter 22

Fabricating Multilayer Circuits

V. J. GRONDAHL

Rockwell International, Cedar Rapids, Iowa

Multilayer Board Process Description	22-1
1. Circuit Registration System	22-6
2. Lamination	22-7
3. Lamination Fixtures	22-8
Drilling Plated-Through Holes	22-8
4. Common Drilling Problems	22-9
5. Causes and Mechanisms of Drilling Problems	22-9
6. Drilling Equipment	22-11
7. Deburring	22-11
8. Plated-through-Hole Cleaning	22-12
Electroless Plating	22-12
9. Typical Problems	22-12
10. Outer Circuit Preparation	22-13
Electroplating	22-13
11. Electroplating Problems	22-13
Inspection of Multilayer Boards	22-14
12. In-process Inspection	22-14
13. Completed Visual Inspection	22-15
14. Completed Electrical Inspection	22-15
Specifications	22-16
References	22-16

MULTILAYER BOARD PROCESS DESCRIPTION

Conductive material generally consists of thin layers of copper bonded to the dielectric material, though Kovar, nickel, and aluminum also are used. An appropriate circuit pattern is etched into each of the thin, solid layers of copper. Dielectric material exposed by etching is the fully cured C-stage glass-epoxy. Semicured sheets of glass-epoxy called B stage are used as the adhesive to bond etched circuit layers together. Bonding or

lamination is performed in presses through application of heat and pressure. Interconnection between layers is performed by first drilling a hole which penetrates circuit features to be connected electrically. Following drilling, copper is electroplated into the holes to provide electrical continuity between the circuit features, and the board is returned to final shape.

Fabrication processes will vary among multilayer board (MLB) manufacturers, but the following steps provide an outline of a good basic process. The example includes steps required for a nickel-gold edge connector type of multilayer board, since that is a very common requirement.

Process step	Description
1. Raw material inspection.	Check C-stage material samples for dimensional stability, copper peel strength, embedded foreign particles, etc. Check B-stage material samples for resin flow, resin content, gel time, volatile content, epoxy type, and glass weave type.
2. Shear C stage material into panel sizes.	Panel size should be based on several points of view: <i>a.</i> Minimum scrap from the standard raw material sheet size. <i>b.</i> Number of circuit images possible per panel size. <i>c.</i> Lamination fixture sizes available or desired. <i>d.</i> Raw material dimensional stability. <i>e.</i> Production volume of a given MLB.
3. Material identification marking.	Once sheared, raw material is difficult to identify for copper clad and dielectric thickness. Storage in marked bins is possible but presents material control difficulties. A stamped code along a panel edge is recommended.
4. Drill racking holes in panel.	This is an optional step based on an individual facility's material handling system. Hole tolerances are not critical.
5. Extra material cure.	Some raw material suppliers give their material a final cure bake after lamination. Baking at this point by the MLB fabricator is an extra epoxy cure step which can improve dimensional stability of the material.
6. Registration pinholes.	Registration system holes, or hole-and-slet combinations. These features must be accurately located, since they directly affect circuit-to-circuit registration between layers of the MLB.
7. Raw material inventory.	Temporary holding area until all material required for an order has received the prior six steps.
8. Raw material inventory release.	Material for a given order released in a complete package.
9. Material scrub.	Material is mechanically and/or chemically cleaned to remove oxides, oxide conversion coatings, fingerprints, etc. from surface of panels. Surfaces must be clean and dry or photoresist will not adhere to the panel.
10. Apply photoresist. (Photoresist is available in both positive- and negative-acting polymerization types. This discussion considers only positive-acting resists—those which polymerize when exposed. Also, this discussion is limited to solvent base film resists, though it is recognized that liquid resists and alkaline base film resists are commonly available and used by MLB fabricators).	Photoresist is applied to a panel surface so that an artwork circuit image may be transferred to the blank. That must be done in a controlled room for best results for several reasons. <i>a.</i> Photoresist is polymerized by light of a certain wavelength; therefore, lighting within the room must be selected to eliminate that particular type of light source. <i>b.</i> Dust and dirt particles between the clean copper surface and the photoresist can result in etched-out circuit features in subsequent pro-

cessing and the scrapping of that piece of material. Therefore, a clean, positive-pressure filtered-air environment is recommended for this step.

An artwork tool is placed beneath a light source of the proper wavelength to polymerize the photoresist on the photoresist-coated panel. Clear areas in the artwork expose the photoresist to light and that sets off a photo polymerization process within the photoresist. Exposed areas are resistant to etching, plating, and developing chemicals, whereas the unexposed areas of photoresist may be easily dissolved and removed by a methyl chloride (solvent base resist) developing solution and water spray rinse. Accurate, repeatable transfer of the image to the blank is provided by pins aligning registration holes in the artwork with those in the blanks. Artwork should carry an image identification number which is transferred to a panel border area.

Photoresist-coated and exposed blanks are developed by washing unexposed photoresist areas in a methyl chloride (solvent base resist) water-rinse system for removal. That exposes bare copper, which is etched from the buried, or inner, elements of an MLB prior to lamination and plated on an external surface, or outer, circuit following lamination.

All exposed copper is etched from the panel. NOTE: Area around the registration holes should always be coated with an etch resist to prevent etching of the copper. That maintains material strength and support for layer-to-layer circuit relations based on the registration holes.

Photoresist has served its purpose and must be removed before lamination.

Following etching, inspection should be made for etched outlines, (circuit opens), unetched areas (circuit shorts), and registration accuracy of circuit pattern to registration holes recommended. The above problems can require rework or be cause of scrap of inner panels.

Copper and epoxy surfaces must be mechanically and/or chemically cleaned to remove contamination and roughen surfaces to promote adhesion during lamination.

Water, solvents, and chemicals used in cleaning blanks must be haked out to prevent bond line voids and delamination during the lamination process. Circuit panels in this condition should be held in a clean controlled environment (50 percent RH maximum) for not more than 8 h prior to lamination for best results. Also, they should be handled only by operators wearing clean protective gloves.

B stage is sheared and punched to match the size of circuit panels being laminated. B stage should be stored in a cooler (50°F, 50 percent RH). Care and cleanliness in handling B stage is very important. Operators should handle B stage only when wearing clean white protective gloves. Placing B stage in a vacuum chamber

t1. Artwork image transfer.

12. Develop artwork image.

13. Etch inner circuit patterns. (This example is for a subtractive, or copper removal, process. An additive copper process would start with a bare dielectric panel. Exposed areas on the photoresist-coated inner circuits would be sensitized and plated with copper instead of etched.)

14. Remove exposed photoresist.

15. Inner circuit inspection.

16. Scrub for lamination. NOTE: Some fabricators follow this step with a heavy oxide surface treatment to increase copper surface area and promote lamination bond strength.

17. Bake prior to lamination.

18. Shear B stage and punch registration pinholes.

19. Lay up for lamination. See Fig. 22.1.

will remove water vapor and volatiles entrapped during storage.

Lamination is performed by using fixtures with registration pins to maintain circuit panel relations to one another. A typical lay-up consists of:

- a. Bottom lamination fixture.
- b. Sheet of release material.
- c. Bottom circuit panel (an etched outer bottom copper layer, dielectric and etched inner top layer of copper).
- d. Three sheets of B stage.
- e. Inner circuit panel (double clad copper, etched both sides).
- f. Three sheets of B stage.

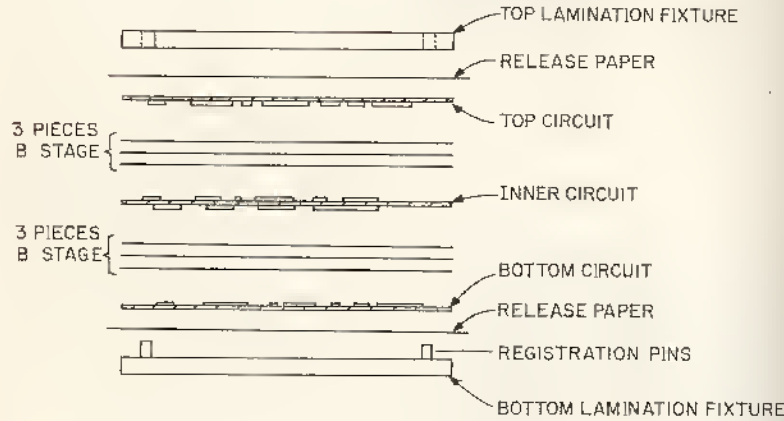


Fig. 22.1 Typical multilayer board lamination process lay-up.

- g. Top circuit panel—an etched inner (bottom) copper layer, dielectric, and unetched copper outer (top) layer.
 - h. Sheet of release material.
 - i. Top lamination fixture.
- Laminates may be multiple-stacked in the fixtures during laminations.

20. Lamination.

Application of heat and pressure in the laminate fixture package converts B-stage semicured material between etched circuits to fully cured C-stage epoxy. During the process, the B-stage resin becomes a liquid adhesive which fills all etched circuit pattern voids and bonds the layers together.

21. Remove laminate from fixtures.

Registration pins should be pushed out and the top lamination plate carefully removed. Care should be taken to prevent scratching or marring surfaces of either laminate or lamination fixture.

22. Laminate trim.

Shear excess epoxy flash from perimeter of laminate and drill any plugged racking holes.

23. Mark identification number on laminated blank.

Identification numbers on internal etched circuits are no longer accessible because of lamination. An external surface of the laminated panel should be marked with an identification number.

24. Post-laminate bake.

Additional curing for laminate bond resin.

25. Drill plated-through holes.

Holes to be plated through are drilled in laminates. Laminates may be multiple-stacked dur-

26. Deburr holes.

ing drilling depending on the relations of drill hole size to circuit feature and the overall panel thickness.

Laminates are hand-sanded in run through a deburring machine to remove drilling burrs. Particles left by deburring must be removed by pressurized air or water rinse. Do not use an air source which may contain oil particles.

27. Hole cleaning* (etchback).

Vapor honing of holes or chemical cleaning with concentrated sulfuric acid dip can be used to remove epoxy smear from inner circuit layers exposed during drilling. Etchback consists of multiple chemical processing steps to include hydrofluoric acid for removal of exposed glass fiber ends plus epoxy removal with concentrated sulfuric acid. Rinse and bake to neutralize and drive off retained solutions in panels following these processes. Registration pinholes must be protected from these solutions to prevent enlargement and inaccuracy.

28. Plated-through-hole sensitization* (electroless copper).

This step consists of a series of chemical cleaning steps, rinses, and a sensitizing catalyst bath followed by an electroless copper bath. The sensitizing catalyst activates the exposed epoxy surfaces within the drilled holes to promote adhesion of the electroless copper plating. Electroless copper is plated to a thickness of 10 to 20 μ in. At this point the panel is electroplated with copper to build up plating in the hole to a typical 0.0002- to 0.0005-in thickness.

29. Scrub for outer photoresist application.

Mechanical and/or chemical cleaning of copper surface to promote adhesion of photoresist. Same process as step 9. Masking over registration pinholes should be removed before scrubbing.

30. Apply photoresist.

This repeats step 10.

31. Artwork image transfer.

This repeats step 11. The artwork can be either a clear image or a black image depending on whether the panel has been electroplated following electroless copper.

32. Develop outer artwork image.

This repeats step 12, except that the exposed copper is the final board circuit pattern and will be plated up to final thickness rather than etched.

33. Electroplate copper.*

This step consists of a series of chemical cleaning steps and rinses followed by the electroplating bath. During the electroplating step, a low dc voltage is applied between copper anodes in the plating bath and the panels, which function as cathodes. The panels are mounted in racks with electrical connections to the cathode to form the circuit. The applied current must be compatible with the amount of exposed circuitry to be plated. Current densities in the range of 10 to 40 A/R² of exposed copper area are common. Copper-plated thickness is usually 0.001 in minimum inside the plated hole.

34. Electroplate tin-lead.*

Tin-lead is the most common surface overplate employed on MLBs. Following copper electroplating, boards are routinely rinsed and cleaned prior to going into the tin-lead bath. Tin-lead is electroplated in a manner similar to copper in step 33. Tin-lead plating thickness will vary, with specific board requirements, from 0.0003 to 0.0015 in.

35. Strip photoresist. Photoresist has served its purpose as a plating mask and must be removed to permit etching of the bare copper.
36. Etch copper. Exposed copper must be removed to leave only the tin-lead-plated outer MLB surface circuit pattern. Tin-lead plating acts as an etch resist.
37. Mask circuit board for edge connector plating.* Chemically resistant plating tape applied with pressure from rubber backup rollers is commonly used to provide masking of circuitry above the edge connector fingers. Tape must prevent stripping and plating solutions from wicking up along vinyl traces.
38. Strip tin-lead. Tin-lead plating must be removed from the edge connector pattern to expose bare copper.
39. Plate nickel-gold on edge connector finger pattern.* Nickel is commonly used as a wear-resistant base for the gold overplate in connector patterns. Plating follows preparatory clean and rinse steps. A separate plating line for this process is common. The gold-plating tank should be surrounded with a backup tank to catch the expensive gold solution in the event of primary tank rupture and to comply with the Occupational Safety and Health Act (OSHA) regulations.
40. Remove plating tape. Physically remove the plating tape. Any residual adhesive may be removed by a solvent clean and rinse.
41. Reflow tin-lead. Tin-lead reflow is a step which reduces the microscopic surface area of electroplated tin-lead so that less surface oxidation can occur. Reflow is accomplished by fluxing, preheating, and dipping panels in a hot-oil (450°F) bath or exposing them to infrared heat. The heating actually causes melting of the tin-lead.
42. Scrub after reflow. Solder flux and oil particles must be removed by a thorough scrub and rinse cycle.
43. Drill nonplated through holes. Most MLBs have holes in which plating is undesirable. This drilling operation must be performed with care to prevent marring the surface circuitry.
44. Contour board to final shape. Cutting individual boards from the panel is usually done in a milling machine rather than a punching operation to minimize the possibility of delamination of a completed board.
45. Marking boards. At a minimum, MLBs get an ink-stamped serial number. In many cases, silk-screened identification of holes and features on the board also is required.
46. Final inspection. Inspection can include any or all of the following:
- Visual defect inspection.
 - Dimensional inspection.
 -
 - Cross section of a coupon or test hole from the panel to check plating thickness.

*During these steps, panels must be mounted in racks.

The sequence of these fabrication steps will vary somewhat among fabricators, but the greatest difference will be in the equipment and facilities used in MLB production.

1. Circuit Registration System Circuit registration is the common denominator to all steps in any particular MLB fabrication facility. Each layer of an MLB must be located as

accurately as possible in relation to all other layers. Otherwise, during drilling, an out-of-location feature can be improperly attached to a plated-through hole, which will cause an internal short or multiple internal shorts and a scrap MLB. Registration accuracy does not include the processing effects of photoresist exposure, material dimensional stability, etching, and copper thicknesses or machine accuracy from drilling, etc.

Registration system accuracy is the tolerance variable between alignment pins and holes on machines using or generating those features.

It is important that the registration system be carefully considered because:

- It is additive to processing variables and overall fabrication accuracy.
- It can be handled with accuracy and consistency from machine to machine.
- If it is not considered properly at an early stage of setting up a facility, it becomes increasingly difficult to change because of the many specialty tools and fixtures which are generated.

How accurate is "as accurate as possible" circuit registration for a good MLB fabricator? An overall registration system accuracy of less than ± 0.002 in is achievable and desirable.

2. Lamination Lamination is the first MLB process step in which process errors cause scrapping more than an individual item of material plus labor. Special material and process control will pay off in reduced scrap. Whether the material considered is glass-base epoxy, polyimide, or Teflon,* rigid cleanliness precautions are mandatory.

a. Cleanliness Precaution. Circuit laminates should be scrubbed and baked to drive out solvents and moisture prior to lamination. Lamination bonding material (B stage) should be stored in a chamber with controlled temperature (50°F) and humidity (50 percent RH maximum). Prior to lamination, the material should be placed in a vacuum chamber to remove any excess volatile matter. Both B and C stage should be handled only by operators wearing clean gloves to prevent fingerprinting and oil smudges prior to lamination. Cutting sheets of B stage to size in preparation for lamination should take place in a clean, dustfree room. The room should contain a filtered positive-pressure air supply to keep dirt particles out and also a controlled temperature and relative humidity (72°F and 50 percent RH maximum).

b. Laminate Scrubbing. MLB laminates are subject to three scrub operations prior to:

- Inner laminate photoresist lamination
- Lamination
- Outer circuit photoresist lamination

The first and last scrub operations are important in that a good photoresist bond to the copper surface is essential for image transfer and chemical solution resistance. However, the scrub prior to lamination is critical to a successful interlaminate bond.

Cleaning is performed by mechanical scrubbing and chemical treatment means with equipment ranging from hand-brush-hand-dip operations to sophisticated conveyorized machines. The purpose of cleaning is to remove surface contaminants, burrs, or protective coatings from the copper. Presence of those items detracts from laminate bond quality. Physical work is usually required to remove the contaminants, and cleaning is therefore commonly a combination of mechanical scrubbing followed by chemical cleaning and is practically never done exclusively by chemical means. Rotary brushes with a pumice and water slurry and a water rinse followed by a 10 percent HCl solution dip is probably the most widely used scrubbing process. Proprietary abrasive brushes with water flush also are used successfully.

c. Material Testing. It is important to check B-stage lots† in the facilities lamination presses. Three to six pieces of B stage should be used in the standard facility lay-up (Fig. 22.2), to make a series of test laminates.

For two-stage press cycles, the application of high pressure should be varied for several laminates, starting with a 2-min low-pressure cycle and increasing the time until high pressure is applied in 2-min increments. If possible, the B-stage resin temperature should be monitored at the time of high-pressure application with a thermocouple inserted into the edge of the bond line.

Sample laminates will range from resin-sticking due to too early an application of

*Registered trademark, E. I. duPont de Nemours & Company.

†See Glossary.

pressure (low-viscosity resin and high resin flow) to trapped air bubble conditions due to too late an application of high pressure (high-viscosity resin and inadequate resin flow). Select a time or, preferably, a B-stage temperature which yields the best visual quality laminate as the operating point for that particular lot of B stage.

For single-stage press cycles, a similar procedure may be used but with different variables. If initial B-stage sample laminates contain air bubbles, increase the platen preheat temperature by some amount (20°, but not greater than the laminating temperature of 350°F). If excessive resin starvation occurs, start adding one sheet of cardboard to each side of the lamination fixtures (or reduce the platen preheat temperature) until sample laminates are of uniform thickness and good visual appearance. Good visual appearance means a resin flash approximately 1½ to 3 in wide around the perimeter of the sample, no exposed glass bundles, no voids or bubbles, and a smooth glossy surface appearance.

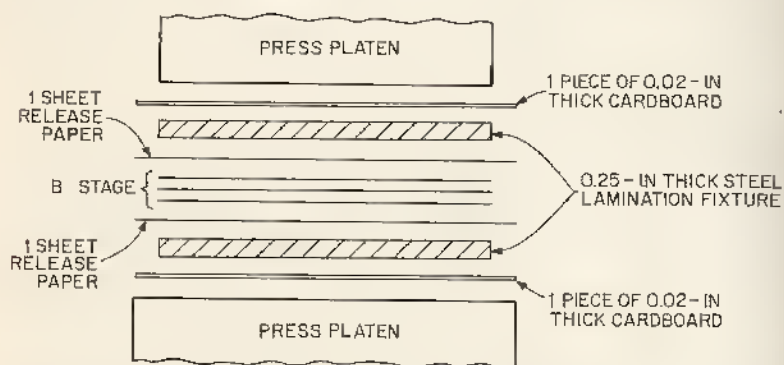


Fig. 22.2 Typical lamination press lay-up to check B stage.

3. Lamination Fixtures Lamination fixtures are most commonly made of steel, since steel approximates the thermal expansion rate of glass-copper-epoxy resin composite materials below the glass transition temperature* (approximately 270°F). The lamination fixture must contain at least two registration pins to align the MLB circuit laminates. Special MLBs of many thin dielectric layers may require layer-to-layer registration aid through the use of multiple-pin lamination fixtures. Multiple-pin fixtures contain holes about the periphery of a blank spaced at 3 to 5 in. Circuit laminates and B stage must have a similar hole pattern so that a series of close-fitting pins can be inserted to prevent movement of circuits during lamination.

Lamination fixture surface imperfections are transferred to the laminate in the lamination process. Scratches, dents, or raised areas produce the opposite contour on the laminate. The surface irregularities can cause poor resist adhesion and excess copper (shorts) or etched-out conditions (broken circuit lines) on the MLB outer circuit pattern.

Lamination fixtures must also be flat (within 0.001 in/in of length or better is recommended). Out-of-flat lamination fixtures produce nonuniform pressure on the laminate and cause nonuniform thickness laminates or possibly a movement of internal circuit feature locations.

DRILLING PLATED-THROUGH HOLES

Drilling holes in the composite epoxy-glass copper material primarily used for MLB fabrication is an area which contains as many as or more identifiable variables than any other process step. Drill machine technology has progressed rapidly with the use of MLBs. MLBs are interconnect systems which rely on the drilled, plated-through hole as the means of making a connection between circuit layers. To produce the vast number of holes required, drill machine capacity has been improved in several areas:

1. Hole production rate. Table travel speeds of over 200 in/min and spindle speeds of over 80,000 rev/min allow hole production rates of 200 to 300 holes/min.

*See Glossary.

2. Hole accuracy. Along with increased hole production rates, hole location accuracy tolerance is of the order of ± 0.001 in or better.

3. Drill machine control system. Computer control, numerical control tapes, and combinations of the two drill table control modes offer a wide range of flexibility and automation to the hole-drilling operation.

Hole-drilling capability is judged for the most part in four areas described below. Following the descriptions are some cause-and-effect relations.

4. Common Drilling Problems

a. Epoxy Smear. Drilling generates, on the drill face surfaces, local temperatures which can melt the epoxy resin. Good drilling minimizes the effect through a satisfactory combination of feed rate, drill speed, and drill configuration. Poor drilling through a different combination of the same parameters may leave a thin coating of epoxy across the face of each internal copper feature penetrated. The epoxy is an insulator, and if not removed, it will insulate hole plating from internal MLB circuitry. The result will be an open connection and a cause for a scrap board.

Epoxy smear can also be caused by only partially cured B stage in the bond line. A posthake following lamination is good insurance against this cause of epoxy smear.

b. Burrs. Burrs can occur on top and bottom of the MLB. Burrs are usually caused by (1) dull drills, (2) poor backup material, (3) inadequate pressure foot on the drill machine, or (4) the wrong speed and feed combination on the drill.

c. Hole Quality or Appearance. Hole quality refers to the straightness of the sides of drilled hole walls. It is a function of drill feed and speed as the drill bit does the work of penetrating, cutting, and removing material from the hole.

d. Nail Heads. Nail heading refers to an effect drilling sometimes produces on internal layers of copper. Copper planes will be widened to twice normal thickness and acquire the appearance of a nail. Drill sharpness, feed and speed rate, material physical properties, and drill point configuration can contribute to nail heading during drilling. Nail heads can also be produced by drills which hit the edge of a circuit feature.

Reasons for these problem areas overlap so much that some discussion of how the problem is caused plus some recommended conditions are in order.

5. Causes and Mechanisms of Drilling Problems Among the problems most frequently described above is proper feed and speed rate.

a. Drill Feeds and Speeds. Drilling is a material cutting and removal process. Feed and speed should be matched to (1) minimize drill bit contact with the material and (2) allow cutting rather than extrusion of the material to take place. Copper is a soft material which has poor machining qualities. Rather than cut and form chips cleanly, it tends to smear at the tool cutting edge and to generate friction heat. The same situation occurs during drilling. Drill bit shape also enters into consideration here. A good rule of thumb for drill feeds and speeds is to use a combination which will penetrate a 0.002-in-thick copper layer in one drill revolution. Faster than that resembles a punching operation (which also can reduce drill life), and slower causes excessive heat buildup in the operation. Drill speeds above 50,000 rpm are readily available on new drill machines and generally yield straighter hole walls. Straight sidewalls present less possibility of entrapped gas bubbles and plating voids in holes during the plating process.

b. Drill Shape. Drill helix and point angles (including angle) are available in an almost infinite variety of combinations. Helix angles vary from 20° to 50° and point angles from 60° to 130° (Figure 22.3).

With so many variables it is difficult to make general statements, but some basic principles may be discussed. During drilling, work performed at the tip of the drill is mostly extrusion of material. Material cutting becomes more efficient along the tool cutting edge as full drill diameter is approached. Tool heat causes epoxy to melt and

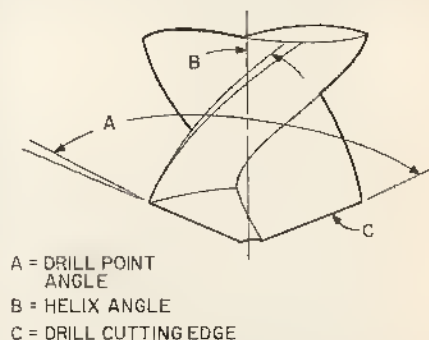


Fig. 22.3 Typical drill point.

smear. Drill energy is converted to heat, which has three places to be dissipated: (1) drill bit, (2) workpiece, and (3) chip. Since drilling generates heat which can cause epoxy smear, one method of reducing heat buildup is to remove the drill chip load as rapidly as possible. A faster (20 rather than 50°) helix angle will provide faster chip removal. Carried too far at the cutting edge, however, the helix angle effectively becomes the rake angle

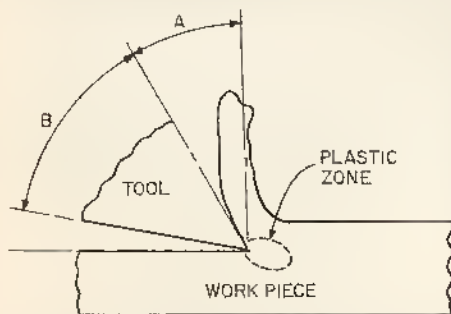


Fig. 22.4 Drill bit, section view.

(Fig. 22.4). A larger rake angle, corresponding to a slow helix angle, creates a larger material plastic zone, which generates more heat in the drilling operation. A helix angle of about 30° is a good compromise between a minimal plastic zone and quick chip removal. The importance of quick chip removal can be seen in Fig. 22.5, which shows up to 70 percent of drilling heat located in the chip.

Drill point angles of 118° are commonly used with good results. When softer materials are drilled, a sharper point will provide a higher ratio of cutting edge to compressive downthrust on

the material which is being cut by the drill. That should result in less copper extrusion.

c. Drill Sharpness and Sharpening. Drill sharpening is routinely performed to extend the life of solid carbide drills. Important aspects of drill sharpening are:

1. Straight uniform cutting edge; no nicks (Fig. 22.6).
2. Sharp end of cutting edge; no nicks or grinding marks (Fig. 22.6).
3. Surface finish of cutting edge; highly polished to reduce friction.
4. Flute relief angle of 7 to 12°.

d. Drill Life. Drill life is sometimes defined in terms of holes drilled. Heat has been shown to have the greatest effect on drill life (Fig. 22.7). The exact source of heat is not universally agreed upon. Neither copper nor glass fiber cuts easily.

Operators can generally get an indication of wornout drills by the amount of surface burr raised. Drills which have penetrated 10,000 oz of copper are usually ready for resharpening. Considering an average MLB to contain at least 10 oz of copper, average drill life is no more than 1000 holes between sharpenings.

Carbide drills are fragile and must be handled carefully, but they stand up much better than high-speed steel drills to the wear and heat problems associated with drilling MLBs.

e. Backup Material. Backup material should, for best results, be the same as the material being drilled. That minimizes the tool wear on the tip, transition vibration wear on the drill, and drag-back of different material into the drilled hole.

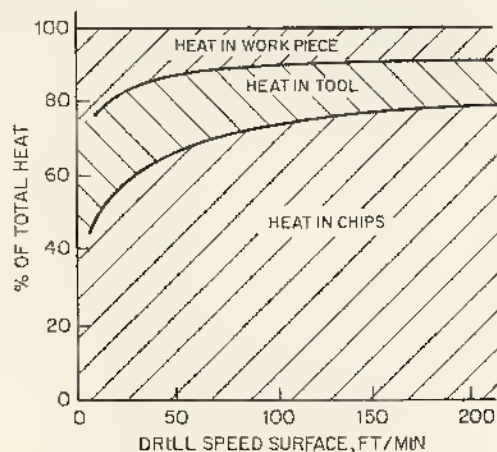


Fig. 22.5 Drilling heat distribution.¹

f. Drill Spindle Pressure Feet. During drilling, the workpiece must be clamped securely to the drill table close to where the hole is being drilled. That function is performed by a part of the drill machine. Lack of the function causes severe drilling burrs, drill breakage, and even delamination of surface copper on the MLB.

6. Drilling Equipment. Highly sophisticated drill machines are available. Some noteworthy features offered by these machines are:

1. Mechanical drill feed mechanisms to provide variable entry feed and extraction rate profiles.
2. Drilling speeds up to 90,000 rpm.
3. Drilling rates per spindle of over 300 holes/min.
4. Digitizing auxiliary units to make tapes from artwork.
5. Computer memory for storing a repeated hole pattern.
6. Computer memory for rotating stored hole patterns.
7. Drill table sizes capable of drilling 24- and 36-in board sizes.
8. Drilled hole location accuracy of better than ± 0.001 in.
9. High unit pressure feet close to the drill.

Drilled hole equipment technology has advanced very rapidly. Drilled hole sizes below 0.013-in diameter become very difficult owing to reduced strength in the drill for protection against handling, vibration, transition from one material to another, etc. However, circuit feature size reduction is always attractive to designers if fabricators can provide the method.

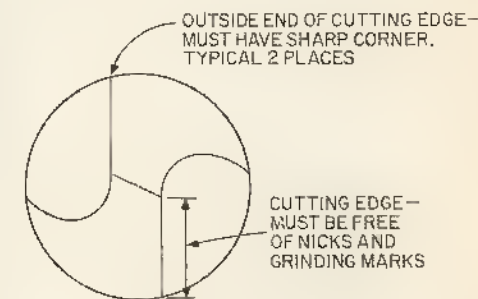


Fig. 22.6 End view of drill.

7. Deburring. Following drilling, holes are mechanically deburred on both top and bottom surfaces. Deburring is performed manually with orbital type hand sanders and fine (400- to 600-grit) sandpaper or in special machines. Machines most frequently use composition abrasive brushes operated either wet or dry. Rotational and endwise oscillatory motion is used to perform deburring on all 360° of hole circumference. Residue from the operation is flushed from the board with water or an air knife system.

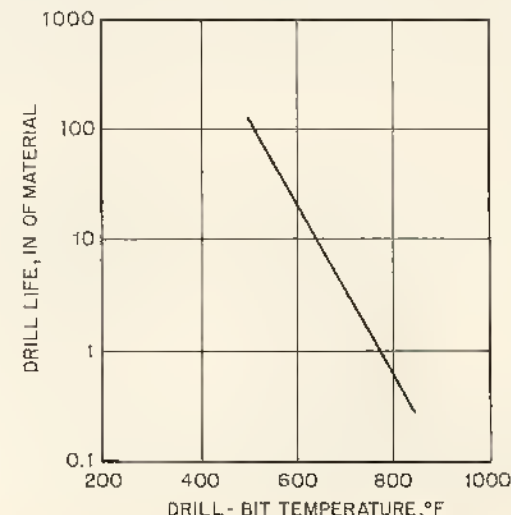


Fig. 22.7 Effect of temperature on drill life.²

8. Plated-through-Hole Cleaning Plated-through-hole cleaning is a preparatory step to metalizing the internal surfaces of a drilled hole. The internal surfaces can exhibit numerous particles and imperfections plus epoxy smear which must be removed to provide good plating adhesion between the epoxy, glass, and copper constituents of a drilled hole. Particles—as well as controlled amounts of the hole constituents—can be removed by chemical or mechanical processes.

a. Mechanical Hole Cleaning. Mechanical hole cleaning is preferred by some MLB fabricators because no strong chemicals are used in the process. It consists of manual or conveyorized vapor honing equipment. High-pressure jets of water and pumice slurry are oscillated across the surface of a blank to, in effect, deburr the inside of each hole. The vapor blast is followed by a water rinse and air knife drying cycle. It is successfully used by a number of companies fabricating MLBs. Fewer operational precautions are involved than with chemical cleaning.

b. Chemical Hole Cleaning. Three categories of chemical hole cleaning are (1) smear removal, (2) chemical cleaning, and (3) etchback. They remove respectively more material from the drilled hole. The amount of material removed by each process is in the following ranges:

1. **Smear removal.** Concentrated (greater than 90 percent concentration) sulfuric acid bath to remove epoxy in the range of 0- to 0.0003-in material.

2. **Chemical-clean.** Concentrated sulfuric acid bath to remove 0.0003- to 0.001-in material.

3. **Etchback.** Concentrated sulfuric acid bath to remove epoxy followed by concentrated hydrofluoric acid to etch back exposed glass fibers. Dip cycles can be varied and repeated to provide material removal in the range of 0.001 to 0.003 in. Chemical etchback is a hazardous process but is commonly used with good results. A bake cycle following etchback is recommended to drive out any retained solutions.

Chemical etchback installation requires positive ventilation and protective clothing to be worn by operators.

c. Plated-through Holes. Plated-through holes perform the interconnect function which makes an MLB such a useful design tool. At the start of the process, drilled holes exist between discrete circuit layers. Upon completion of a relatively short processing cycle, thousands of permanent electrical interconnections are formed to provide electrical continuity throughout the MLB.

ELECTROLESS PLATING

Electroless plating is a process whereby sites on the surface of a nonconductive material are "sensitized and activated" so that metal ions (copper in this instance) will plate out and form a conductive metal coating over the previously nonconductive material. Sensitizing and activating functions have been combined into a single catalyst solution which is available from all major chemical suppliers, as is the electroless copper bath itself.

Some typical problems and causes related to electroless plating are as follows:

9. Typical Problems

Problem	Potential Causes	Potential Solutions
1. No copper in hole	<i>a.</i> Skipped plating operation	<i>a.</i> Analyze baths regularly; maintain limits recommended by chemical suppliers
	<i>b.</i> Depleted anodes	<i>b.</i> Check and replace anodes regularly when surface area drops below approximately 40 percent of original anode.
	<i>c.</i> Intermittent rectifier operation	<i>c.</i> Section boards daily to check plating results.
	<i>d.</i> Depleted plating solution	<i>d.</i> Replenish solution
	<i>e.</i> Oil in holes from agitation air supply	<i>e.</i> Use turbine air source
2. Burrs in holes	<i>a.</i> Dull drills	<i>a.</i> Limit drills to 10,000 oz of copper

3. Voids in plated-through holes

b. Drill feed rate too great

c. Poor deburring and hole-cleaning operation

a. Poor cleaning and rinsing; surface of G-10 doesn't get sensitized

b. Depleted plating solution

c. Rough drilling traps gas

b. Use deburring machine with multiple direction action and air knife

a. Use agitation with clean and rinse steps

b. Use heated cleaning and rinse solutions

c. Use sharp drills

10. Outer Circuit Preparation At this point the MLB is totally covered with electroless copper followed by a thin (approximately 0.0002- to 0.0005-in) coating of electroplated copper. The fabricator has the option of electroplating the entire copper thickness (typically 0.001-in-thick in holes) prior to defining the outer surface pattern. That would

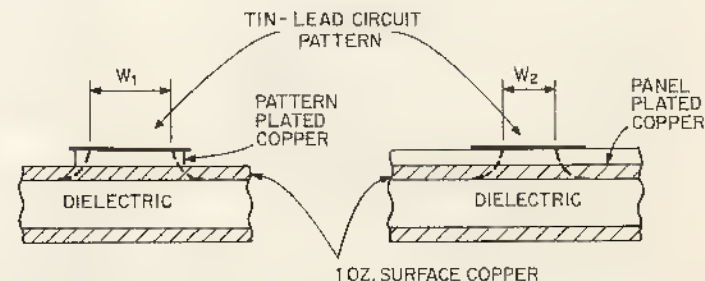


Fig. 22.8 Reduction of circuit lines; pattern vs. panel-plated copper process (positive resist). Line width remaining, W_2 , is narrower than W_1 by twice the thickness of the panel-plated copper.

be called panel-plating. One reason for not doing it is copper anode cost; another is finished line width reduction during etching (Fig. 22.8).

The outer circuit pattern is defined by "black line" artwork. Protected areas of resist are developed away to reveal the surface circuitry and plated-through-hole pattern. Electroplated copper will be deposited on the exposed circuitry pattern and in the holes to a typical thickness of 0.001 in minimum on the hole walls. Following copper plating the panel will immediately be plated with tin-lead, which makes the circuit solderable and also acts as an etch resist.

ELECTROPLATING

In order to plate to a given thickness, it is desirable to know plating deposition rates at various plating current densities. The rates can be measured, as a plating aid, for all copper and electroplating baths. Surface plating area plus internal plated-hole circumferential area must also be known to select an appropriate current density and plating time.

11. Electroplating Problems Electroplating processes have many potential problem areas. Some common problems and their solutions are the following:

Problem	Cause	Solution
1. Nonuniform plating thickness over panel	<i>a.</i> High current density on isolated surface features will plate very thick	<i>a.</i> Shield edges of blank during plating to move current field lines in plating bath <i>b.</i> Add artificial surface features just outside board surface pattern to distribute plating current density

2. Cracks between hole barrel and surface copper plating	a. Excessive brightness in copper bath	a. Carbon-filter to reduce brightness levels b. Routinely check plated copper tensile strength and ductility a. Drill feed rate should be 0.001-0.002 in/rev of drill b. Replace drills after 10,000 oz of copper drilled c. Replace etchback bath
3. Epoxy smear	a. Drill feed rate too slow b. Dull drills c. Etchback solution not functioning properly	a. Use 40 A/ft ² max plating current b. Use agitation and heated cleaning solutions
4. Nodular plating	a. Excessive plating current b. Poor cleaning, drilling, sanding; particles in holes c. Particles in plating bath	a. Filter plating bath
5. Debris in holes	a. Poor cleaning after drilling; plating to chips b. Copper peeling from dielectric; poor cleaning and no sensitizing of basic material c. Dirty plating bath	a. Use air blast after drilling to clean holes b. Use agitation and heated cleaning solutions c. Use adequate number of rinses d. Perform regular internal filtering of copper plating baths

INSPECTION OF MULTILAYER BOARDS

Inspection requirements for MLBs vary with customers primarily along military vs. individual company procurement specification lines.

Military MLB inspection requirements are defined in Mil-P-55640 and its revisions. Individual company requirements can be most anything, depending on the application. Excluding environmental testing, inspection is commonly performed in five categories to monitor and evaluate MLB quality:

1. In-process inspections
2. Completed MLB visual inspection
3. Completed MLB dimensional inspection
4. Coupon sample evaluation
5. Completed MLB electrical inspection

The first three of these inspections are routinely performed by the fabricator as part of the operation cost. The latter two items are usually negotiated items depending on fabricator's capability and buyer's requirements.

12. In-process Inspection In-process inspection is essential to MLB production. Since errors are not usually reworkable, inspection points should be established to minimize scrap loss and also to be used as an indicator of process control. Typical inspection points include:

1. Artwork tool damage and accuracy check.
2. Visual check of photoresist lamination for wrinkles or foreign particles.
3. Visual scan after photoresist develop for areas of poor image transfer or deposits of solid photoresist on the pattern during develop cycle.
4. Insure registration hole to circuit pattern location after stripping photoresists from etched circuits. Check for shorts (poor etch), broken lines, and foreign particle inclusion in raw material dielectric. Electrical inspection for opens and shorts in the circuit pattern may also be performed at this point.
5. Check after hole cleaning for debris in holes and off location inner circuits.

6. Check during plating for plating thickness by inspecting buildup in plated-through holes with plug gauges.

7. Check plating adhesion after plating by using tape test.

8. Inspect blanks after outer circuit etching for lamination defects.

13. Completed Visual Inspection The multilayer board is inspected at this time for imperfections on the outer circuit pattern such as reduced line width, etched-out spots in the tin-lead plating, peeled plating, foreign particles, inclusion in the laminate, lamination flaws, shorted circuit features, and spots of copper remaining on the MLB surface. Inspections is typically performed by using low-power (5 to 10x) magnifying glasses or with closed-circuit television scanners.

Rework is rarely possible at this point, but in some cases, flaws have little if any functional impact on MLB usage and are acceptable.

a. **Completed MLB Dimensional Inspection.** Multilayer boards are inspected for online dimension, thickness, hole size, and flatness.

b. **Coupon Sample Evaluation.** Coupons, when used, are test patterns located singly or in multiples on every layer of an MLB. They are used to inspect plating thickness, dielectric thickness, plated-through-hole quality, drilling quality, layer-to-layer registration, and thermal shock resistance of the MLB. These characteristics are inspected by mounting a segment of the coupon in clear plastic. Plastic and board segment are progressively sanded and polished to section one or more holes along their centerlines. High-powered microscopic (200 to 400x) inspection allows measurement of plating thickness and observation for defects described.

14. Completed Electrical Inspection Electrical inspection requirements are becoming more common with MLB usage. It is enough of a problem to have an assembly which does not work because of component failure, without the added possibility the MLB is faulty. Several common characteristics of a MLB may be checked:

1. Dielectric voltage breakdown. Arcing through thin dielectric material.
2. Open circuits. Internal broken lines.
3. Low-resistance shorts. Internal copper bridging between circuits.
4. High-resistance shorts. Internal shorts due to contaminants left on inner circuit surfaces or wicking and entrapment of chemical solutions between circuit features.
5. Controlled capacitance between circuit layers.
6. Controlled impedance of circuit lines.

Continuity test machines are used to perform the first four types of electrical test. Point-to-point interconnect information from the MLB schematic is placed on tape or in computer memory of the continuity test machine. A handling fixture consisting of spring-loaded electrical contacts in a pattern which matches the MLB circuit pattern is also part of the continuity tester. It compares the interconnect information with results measured through contact with the MLB. Contact with the MLB can be made with individual handling fixtures for a specific MLB or with a large universal fixture with contacts mounted on a grid pattern.

Continuity testing is programmable to test for different levels of: (1) voltage breakdown potential, (2) high-resistance shorts, and (3) low-resistance shorts. Varying those parameters influences test time, which will range from 500 to 5000 contacts per minute that can be checked. Continuity testing can be performed on two-layer boards and inner circuits of MLBs.

Inspection of MLBs is an essential part of production, but no amount of inspection can improve a product. Product improvement and high quality are results of process control.

a. **Process Control.** Process control differs from quality control and inspection in that it looks for potential causes rather than the effects of process problems. The process causes can range from fading light intensity on an exposure machine to impurity levels in plating baths, bath temperatures, acid concentration in an etchback system, or the pH level of a deionized water supply. Those are typical examples of the many process steps which contain observable and measurable characteristics. Understanding the limits of each process step which must be maintained to produce not marginal, but consistently good results of that process step is the first step toward establishing process control. Vendors of supplies are one source of information; they know the strong points of their own products and also the weak points of their competitors' products. Supplemented by first-hand product experience, that is one approach to establishing process step limits.

Second, process characteristics must be measured or monitored routinely to observe their relations to established limits. Appropriate action must be taken to restore a process step as it exceeds a limit condition, even though product quality may not suffer immediately.

Next, on a long-term (several months) time base, plot process step characteristics to show trends not observable in simple routine measurements. As a corollary, scrap loss records on a similar time base for each process step can assist in general troubleshooting activity or help define areas in which process or equipment improvement can reduce operating costs.

Finally, the charts are useful only if they are used, so hang them up where they are noticeable to stimulate interest and action among operators, foremen, inspectors, and managers.

SPECIFICATIONS

MLCB Raw Material Specifications

a. Military

- Mil-G-55636 Glass Cloth, Resin Impregnated (B-stage)
Mil-P-55617 Plastic Sheet, Thin Laminate, Metal Clad

b. Commercial

- IPC-L-110A Preimpregnated, B-Stage Epoxy Glass Cloth for Multilayer Printed Circuit Boards
IPC-L-120 Inspection Procedure for Chemical Processing Suitability of Copper Clad Epoxy-Glass Laminates

REFERENCES

1. J. K. Hagge, Drilling of Multi-layer Printed Circuit Boards, Collins Radio Group of Rockwell International, Company Report 1974.
2. "Metal Cutting Tool Handbook," Metal Cutting Tool Institute, 1965, pp. 10-40.
3. G. C. Everstine, A Theory of Machine Fiber-reinforced Materials, *J. Composite Materials*, vol. 5, pp. 94-106, 1971.
4. A. O. Schmidt, Distribution of Heat Generated in Drilling, *Trans. ASME*, vol. 71, pp. 245-252, 1949.
5. R. E. Weiss, R. D. Rust, and C. J. Bartlett, Bell Laboratories, Whippany, N.J., report on The Generation and Removal of Epoxy Smear for the Institute of Printed Circuits Proceedings, Spring Meeting, 1974.
6. M. F. DeVries, Drill Temperature as a Performance Criterion, ASTM paper MR 68-193, 1968.

Chapter 23

Laminating

V. J. GRONDAHL

Rockwell International, Cedar Rapids, Iowa

Introduction	23-1
Materials	23-2
1. Epoxy Resin	23-2
2. Glass Cloth	23-2
3. B-Stage Material	23-3
4. G-10 and FR-4 Copper-clad Raw Material	23-4
5. Miscellaneous Multilayer Board Materials	23-5
Lamination Fixtures and Tooling	23-6
6. Registration Pin Systems	23-6
7. Lamination Fixtures	23-6
8. Lamination Presses	23-7
9. Lamination Press Auxiliary Equipment and Procedure	23-9
Lamina Preparation	23-10
10. B-Stage Characteristics	23-10
11. B-Stage Storage and Handling Prior to Lamination	23-11
12. Circuit Laminate Preparation Prior to Lamination	23-11
13. Lamina Lay-up Procedures	23-11
14. Two-Stage Pressure Cycle With either a Hot or Cold Start	23-13
15. Single-Stage Pressure Cycle With either a Hot or Cold Start	23-14
16. Vacuum Lamination	23-15
17. Cycle Type Comparison	23-15
Lamination Process Troubleshooting	23-16
18. Lamination Problems—Causes and Effects	23-16
SPI Prepreg-reinforced Plastics Committee Test Method	23-18
References	23-18

INTRODUCTION

This chapter deals with the materials, processes, and equipment involved in laminating multilayer boards. The laminating process marks the major difference between two-layer

and multilayer boards (MLBs). It consists of a number of individual process steps and related equipment. The process steps and the equipment involved explain the increased cost normally experienced when a circuit design must go from a one- or two-layer to a multilayer board with three or more layers. In return the designer has at his disposal an extremely versatile design tool.

MATERIALS

The materials most commonly used for multilayer boards are G-10 and FR-4 grade epoxy resins, woven glass fabric matrix material, and electrolytic sheet copper. Special-purpose materials are also laminated to meet specialized electrical or environmental equipment parameters.

1. Epoxy Resin

a. Definition. In a broad sense, the term "epoxy" refers to a chemical group consisting of an oxygen atom bonded with two carbon atoms already united in some other way.¹ The simplest epoxy is a three-membered ring to which the term α -epoxy or 1,2-epoxy is applied. Ethylene oxide (I) is an example of this type. The terms 1,3- and 1,4-epoxy are applied to trimethylene oxide (II) and tetrahydrofuran (III). The bonding structures are shown in Fig. 23.1.

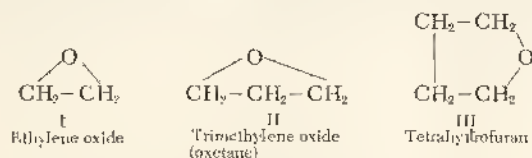


Fig. 23.1 Epoxy structures.

Epoxy resins commonly used for printed board materials are those whose molecular structure consists of multiple α -epoxy groups.

b. Characteristics. The low-viscosity and easy-cure properties of epoxy resins used for MLBs simplify manufacture of the basic raw materials. The epoxy resin curing agent system can be modified readily to provide special properties such as room-temperature latency, high-temperature resistance, and flame retardancy. Room-temperature latency is of particular interest, since that feature is used to formulate a dry-handling, semicured epoxy adhesive resin used to bond circuit layers together in MLBs. Availability of the semicured material, called B stage, has contributed to the rapid use and ease of manufacturing MLBs.

Epoxy resins as a material category possess excellent electrical, mechanical, and chemical properties. The general and (cured state) properties vary little. However, the exact chemical composition—curing agents, stabilizing agents, and flame-retardant chemicals—is usually proprietary and unavailable from the resin suppliers. That becomes very important to the fabricator of multilayer circuit boards, who is using a semicured material for intracircuit layer bonding and usually from more than one supplier. Reaction of the semicured resin to MLB fabrication steps can vary substantially with the supplier.

2. Glass Cloth Woven glass cloth is used as the support vehicle for epoxy resins in the manufacture of printed board raw material and for structural strength. Continuous rolls of glass cloth fabric are drawn through vats of liquid resin, sizing rollers, and curing ovens to produce dry semicured rolls and sheets of epoxy-resin-impregnated glass-base material.

The glass cloth is of interest to the MLB fabricator and the lamination process primarily for reason of its impact on material dimensional stability of the individual circuit layers within an MLB. A glass cloth characteristic which has an influence on material stability includes weave style. Glass cloth is available in a wide variety of weaves, although there is a continuing trend toward standardization by the raw material (copper-clad glass-epoxy) suppliers, as well as the glass weavers.

"Weave style" is a term which includes subvariables of:

1. Woven fabric thickness.

2. Weave construction, the number of glass bundles per inch running "with the grain" vs. the number of glass bundles per inch running "across the grain." Major weave direction, or with the grain, is also called the warp direction of the fabric. Minor weave direction, or across the grain, is also known as the fill direction of the fabric.

3. Glass filament size and fabric weight per square yard.

4. Weave type; the glass filaments within a bundle may be individual strands or wrapped strands.

Weave style can influence material dimensional stability by the following means:

1. Heavier fabrics (thick and tight weave) tend to exhibit physical characteristics approaching those of glass, which is dimensionally very stable. Heavier fabrics also are less subject to distortion and stressing during application of the epoxy resin and copper sheet material.

2. Weaving and treating stresses, during weaving, glass filament bundles in the fill direction are subjected to less bending, since they are carried back and forth across the width of the material in an unobstructed path. Glass bundles running in the warp or major weave direction are alternately bent up and down over the fill bundles. The warp direction therefore has more "kinks" or direction changes, which appear to act as springs in subsequent processing steps of etching and lamination.

a. Glass Filament Surface Treatment. Simple application of epoxy resin to glass cloth is not adequate to yield the maximum benefits which this composite material has to offer. In all cases an adhesion promoter, usually applied by the glass weaver, is used. Two types of adhesion promoters are commonly available:²

1. Primers, available under various tradenames. Primers promote adhesion by providing a boundary layer of adhesive around the glass filaments.

2. Chemical coupling agents, most commonly used are silane coupling agents, which are available in various compositions for specific applications. They promote glass-to-epoxy adhesion by providing chemical bonds both to the inorganic glass and the organic resin materials.

Advantages of the chemical coupling agent adhesion promoter over primers include:

1. Less susceptibility of glass-filled epoxy to wick moisture and chemical solutions along glass filament bundles.

2. Improved material strength through transfer of stresses to high-strength glass fibers.

Glass fabric within the copper-clad raw material is often overlooked as having any influence on the printed board fabricator's product. In fact, it plays an extremely important part in overall material stability, moisture and chemical solution absorption, and fabrication problems such as movement of inner circuit layers during lamination.

3. B-Stage Material

a. Description. Semicured glass-cloth-reinforced epoxy resin is commonly referred to as B stage and sometimes as prepreg. In the semicured state the B-stage epoxy resin has not been fully polymerized; therefore, although it may be handled easily, it may also be returned to the liquid state by application of heat and pressure. That feature makes B-stage resin a convenient bonding material for multilayer boards.

Technically, B stage is available in all of the glass weave styles; but for bonding layers in multilayer boards, the thinner lightweight fabrics are most commonly used because they carry a higher resin-to-glass ratio. That characteristic helps provide a void-free laminated bond, since more resin is available to fill circuit patterns where copper has been removed.

B stage is available in both standard G-10 and flame-retardant FR-4 material types. Both types are widely used in the lamination of multilayer boards.

b. B-Stage Resin Flow. Resin flow is a characteristic used to define B stage in categories for special uses. In the B stage it is controlled by the degree of semicure applied to the epoxy-treated glass cloth and by the curing and stabilizing agents used in the basic epoxy compound. The amounts of resin flow for B stages are described in Table 23.1.

c. No-flow B Stage. No-flow B-stage material has rather specialized uses. Resin flow is essentially zero, although a rim up to approximately 0.03 in of resin can be induced around the edge of a no-flow bond line. Precontoured parts such as heat sinks are commonly bonded to printed boards by using no-flow B stage.

d. Low-flow B Stage. Low-flow B stage is used primarily by fabricators of the fully cured

glass-filled epoxy, copper-clad raw material that is employed throughout the printed board industry. Laminated raw material from low-flow B stage has two characteristics which explain this wide use:

1. Economy of resin, there is just enough resin flow to drive out air voids in the finished raw material.
2. Thickness uniformity of the raw material. Although several factors influence thickness uniformity, minimized resin flow from the B-stage lay-up is one of the more important considerations.

e. High-flow B Stage. Both medium- and high-flow B stages are used extensively in the lamination of multilayer boards. There are two reasons for this:

1. Adequate amounts of resin are available in the high-flow material to fill etched circuit void areas and flush entrapped air bubbles from the bond line. Air bubbles and voids in the bond line are usually cause for scrappage of the MLB.
2. Resin flow can be promoted at lower lamination pressures, which in turn generate lower internal forces on the softened inner circuit layers and thereby reduce the likelihood of circuit features shifting or floating during the lamination cycle.

TABLE 23.1 MIL-G-55636A Designations

Category*	Class	Resin in flow, percent
No flow	I	20 or less
Low flow	II	21 to 30
Medium flow	III	31 to 45
High flow	IV	Greater than 45

*Resin flow is measured by standard procedures such as the Society of Plastics Industry test method, the MIL-G-55636A method, or similar specialized methods used by suppliers and users of B-stage material.

4. G-10 and FR-4 Copper-clad Raw Material Single- and two-sided printed boards are fabricated from a variety of paper-base and phenolic grades of material with layers of copper on the outside layers. Multilayer boards, on the other hand, are fabricated almost exclusively from G-10 grade glass-filled epoxy, or a flame-retardant type of the same material known as FR-4. Good electrical, mechanical, and chemical resistance characteristics are the reason for the high usage of G-10 and FR-4. The material is available in many physical conditions:

1. Copper-clad one side
2. Copper-clad both sides
3. No copper cladding
4. Varied copper-clad thickness
5. Varied dielectric material thicknesses
6. Varied glass weave styles within the material

Most common dielectric (cured epoxy-glass material) thicknesses are available from 0.004 in thick to 0.062 in thick. Thicknesses of 0.0025 in and above 0.062 in also are available, but they are usually special situations.

The G-10 or FR-4 fully cured material is commonly referred to as C-stage material. During the heat and pressure of lamination, it will soften; but it will not become a liquid again. Excessive lamination temperature will turn the laminated board brown, or char it.

5. Miscellaneous Multilayer Board Materials In addition to the predominantly used G-10 or FR-4 material, a number of other materials are used in the multilayer board industry. Some of them and their advantages and disadvantages are discussed in the following paragraphs.

Polyimide is a high-temperature material utilizing a glass cloth base, and it is available as B-stage or copper-clad C-stage material. Maximum service temperature is of the order of 500°F compared with about 275°F for G-10. (The temperature varies.) The high service temperature and glass transition temperature (Fig. 23.2) should extend polyimide use into multilayer board applications, where many high-temperature thermal cycles would eventually create Z-axis expansion which would fracture plated-through-hole barrels in standard G-10 or FR-4 material. The low Z expansion feature may be of particular use in solving such problems in MLBs greater than approximately 0.08 in thick. Electrical

characteristics are similar to those of G-10 and FR-4, with improved chemical resistance and somewhat less mechanical flexural strength. Polyimide also appears to be a more dimensionally stable material. Features which should improve with increased industry usage include:

1. Cost and availability. Sources of the resin are limited. Cost is two to three times that of G-10 raw material.

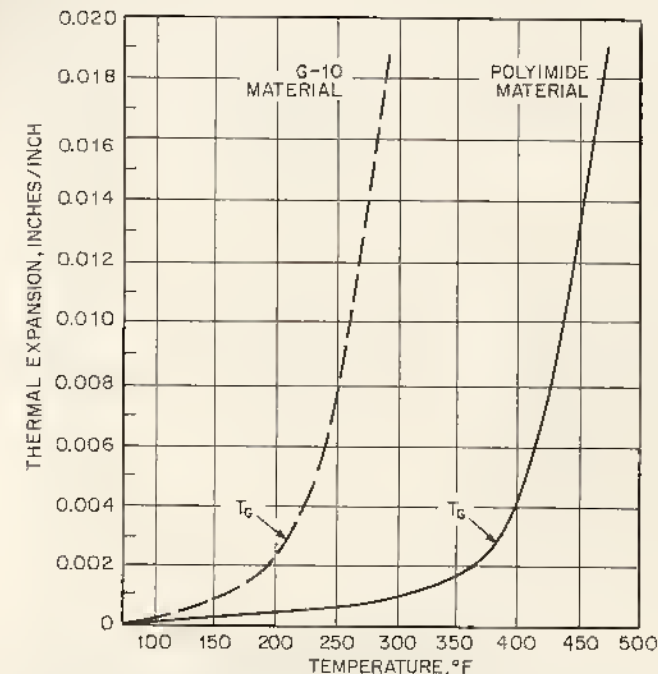


Fig. 23.2 Comparison of typical glass transition temperatures (T_g); G-10 vs. polyimides.

2. Copper adhesion strength. Typical copper peel strengths are in the 4- to 6-psi range as compared with 12 to 16 psi for G-10.

3. Lamination temperatures and cure cycles. Obtaining laminated bonds of consistent quality can be a production problem. Lamination is performed at higher temperatures (425°F) than for G-10 and FR-4 (350°F). In addition, an extended postcure cycle (up to 16h) is recommended for polyimide laminates.

Despite those drawbacks, there are a number of suppliers of polyimide raw material some of whom are working on an epoxy-polyimide alloy as well. The area has considerable potential as the operating environments for multilayer boards increase in temperature.

Polyphenylene oxide (PPO) has excellent low-loss electrical properties for high-frequency radar and communications equipment applications. PPO is either a cast or extruded material with no glass fabric base material. It is available with single- or double-clad copper sheet on its external surfaces. Lamination is readily performed by using polyethylene or irradiated polyolefin sheet material.

The major problem with fabricating multilayer boards from PPO is dimensional change within the material during lamination. Dimensional changes are random in direction and are of the order of 0.002 to 0.003 in/in of material size. Special compensating techniques such as annealing and double exposing and etching steps are required to utilize the material. Given those difficulties, plus a 5:1 cost premium over G-10, PPO has a limited but important area of MLB application.

Glass-base Teflon,* with a random filler and glass cloth base, has excellent low-loss

*Registered trade name E. I. du Pont de Nemour & Company.

electrical characteristics which make it useful for high-frequency radar and communications equipment application. In the woven glass-fabric base material the dielectric constant will vary with weave direction, sometimes creating a requirement for the random glass-fiber-filled material. Teflon is a high temperature (600°F) material, but its use in multilayer board applications has been slow owing to several processing difficulties:

1. Teflon-to-glass filament adhesion. Teflon does not penetrate the glass bundles well or adhere to the glass filaments. Therefore, during processing through chemical solutions, the glass fiber bundles wick solution which is retained or released at other processing steps. Reducing the Teflon part's temperature before introducing it to a bath will tend to minimize the wicking action.

2. Teflon chemical composition. Teflon's molecules are based on extremely strong double-valence bonds. The result is a material difficult to prepare chemically or mechanically to promote surface plating. A highly combustible etching solution called Tetra-Etch* does perform a surface etch on Teflon, which permits boards to be built with plated-through holes.

3. Lamination. Bonding or laminating of Teflon circuits together to form multilayer board also requires treatment of surfaces with Tetra-Etch for a reliable process.

Bonding materials can be the bc G-10 B stage, polyethylene sheet, or thin (0.003-in) sheets of Teflon raised to melting temperature during lamination.

Flexible Materials. One- and two-sided copper-clad polyimide material has been available for some time and is used in many equipment applications. Features associated with the material include:

1. Conformance to equipment contour for improved package interconnect density.
2. Reels supply, which makes it readily adaptable to highly automated in-line processing.
3. Enough thinness to allow punched holes and stamped contours, again making it amenable to in-line processing.

Beyond those applications, flex cables are being used for special multilayer board applications and as parts of multilayer board-interconnect hybrid packages.

LAMINATION FIXTURES AND TOOLING

Successful multilayer circuit board lamination technique really requires only two things to occur: (1) Layer-to-layer registration of circuit pads—which are eventually interconnected by drilling and electroplating—must be maintained. (2) Laminate bond lines must be void-free and must resist thermal shock requirements. Uniform and predictable thickness of finished laminate is desirable as well.

Layer-to-layer circuit registration is usually maintained by the use of tooling pins in the material outside the circuit area.

6. Registration Pin Systems Aside from the fact that every company making multilayer boards uses registration pins, there is little if any commonality of tooling pin systems. Within a given company, however, the tooling pin system becomes the common denominator for all process steps from shearing the initial material panel size to routing the finished boards from the panel. Selection of a registration system which provides both versatility and accuracy can be one of most important decisions the MLB fabricator can make.

Some registration pin systems with their advantages and disadvantages are noted in Table 23.2. Whatever registration pin system is selected, it must be remembered that it is the reference base between the machines and the circuit pattern throughout the MLB fabrication process.

7. Lamination Fixtures Ideally, lamination fixtures should have an expansion rate which would exactly match laminate reaction to heat and pressure. That would minimize induced thermal stresses between registration pins in the fixtures and the laminate material. However, since the laminates are a composite of copper, glass, and epoxy and since the lamination temperature exceeds the glass transition temperature of the material, the laminate is a real unknown entity when it comes to predicting thermal expansion

phenomena. Glass base epoxy laminates do approximate the thermal expansion rates of steel quite well up to the glass transition temperature (approximately 265°F), and therefore steel provides a compatible fixture material with a minimum of induced thermal stresses throughout the lamination cycle.

Lamination fixtures may have as few as two pins or a series of pins around their border (matching holes in the laminate layers required) on approximately 3-in centers. The technique of multiple pinning may not guarantee perfect layer-to-layer registration, but it

TABLE 23.2

System	Advantage	Disadvantage
1. Two pins or more along one edge of panel.	Allows unrestricted range of lamination fixture and panel sizes without change to basic registration system.	Material shrinkage effects are maximized with movement away from registration pins.
2. Two pins centered on panel.	Material shrinkage effects are minimized since material is symmetrically located on pins.	Difficult system to implement with many panel sizes, particularly for the artwork and expose operations. May require a "movable" secondary pin with some accuracy loss for drilling various panel sizes.

has been successfully used in conjunction with special press cycles and laminate borders to reduce misregistration problems in lamination.

For vertical stability and laminate registration, lamination plates should be thicker than the registration pin diameter. Surfaces of lamination plates should be examined for scratches which can cause dents and broken line problems of outer laminate circuits during subsequent process steps. To protect press platens, registration pin length should be less than the combined thickness of the top and bottom lamination fixture plus the thinnest laminate fabricated (Fig. 23.3); that is, T must always be greater than L .

Lamination pins and fixtures are straightforward elements required for lamination. Since they are constantly used and reused, however, the plant process engineer must assign a high priority to the total handling system surrounding them.

8. Lamination Presses Lamination presses for the production of multilayer circuit boards fall in two categories: (1) electrically heated and (2) steam-heated. Actual press pressure may be provided by a closed-loop hydraulic pump system for both types of presses.

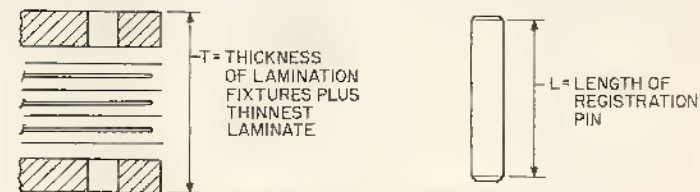


Fig. 23.3 Laminate buildup vs. registration pin length.

*Registered trade name, W. L. Gore and Associates, Inc.

Presses used for lamination of raw material may also use a steam-hydraulic system to provide lamination force.

Some characteristic advantages and disadvantages of steam vs. electrically heated presses related to multilayer board fabrication are listed in Table 23.3. Press platen heat-up curves will be discussed further in connection with lamination press cycles.

TABLE 23.3

Type of press	Advantage	Disadvantage
Steam-heated	Lower operating cost More platens per press Fast platen temperature heat-up rate Lower thermal inertia in platens Uniform heat rise curve	Higher initial cost for boiler and steam pipe installation More system to require maintenance Usually limited to maximum temperature of 400°F
Electrically heated	Low installation cost Low maintenance cost Higher platen temperature attainable	Higher operating cost Slow platen temperature heat-up rate* Fewer platens per press

*A way to overcome this disadvantage with electric presses is to use two presses. One press is operated hot all the time, the other cold. Laminates are loaded into the hot press for a heat-and-cure cycle under pressure. Pressure is released and laminates are transferred to the cold press, where they are cooled under pressure. In addition to energy savings, more than twice as many press loads can be run on the two machines as on one press.

9. Lamination Press Auxiliary Equipment and Procedures

a. Thermocone Welder. Lamination presses will on occasion malfunction or have problems which change their characteristic heat-up-rate curve. That usually results in scrap laminates. A small 26-gauge thermocouple temperature detector embedded in a laminate bond line of a laminate within each press opening can pinpoint those problems. The welder is a simple low-voltage power supply used to prepare the thermocouple joint as shown in Fig. 23.4.

b. Temperature Recorder. Thermocouples are connected to a recorder and inserted into

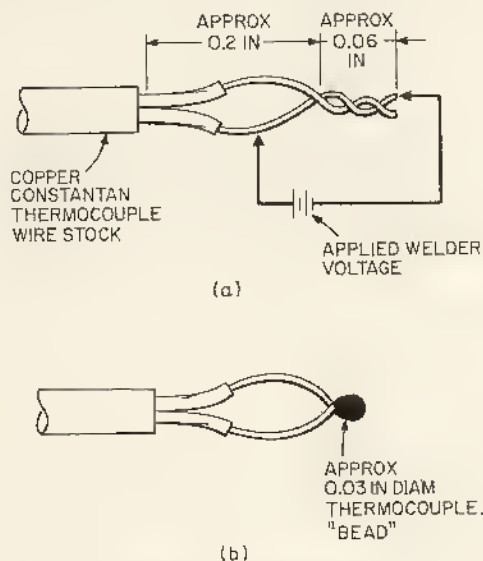


Fig. 23.4 Thermocouple. (a) Before welding; (b) after welding.

the edge of a laminate bond line to monitor bond line temperature. The press may be manually operated at a desired bond line temperature, or the recorder can be set up to apply high pressure automatically at a predetermined bond line temperature. Note that this set up would primarily apply to fabricators who use a two-stage press cycle, although it would provide useful process monitoring information.

c. Pressure Transducer. Lamination press hydraulic systems and pressure gauges can malfunction on occasion. The most direct method of detecting problems of this nature is to

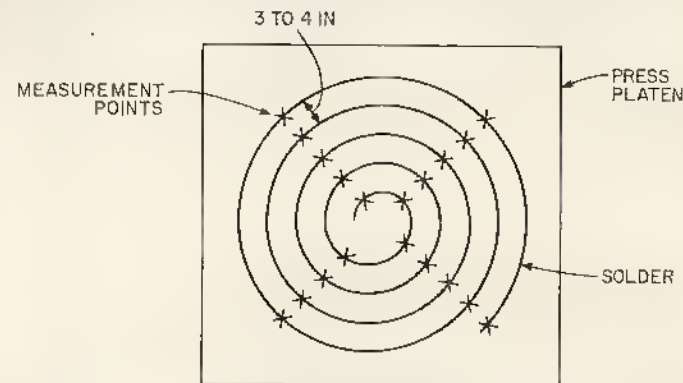


Fig. 23.5 Solder spiral.

use a pressure load cell directly in the press opening to calibrate applied load vs. pressure or force gauge readings. Alternate methods of platen force calibration are sometimes used but give less reliable results. Typical would be use of the reasonably consistent force-deflection characteristics of lengths of standard-size drawn tube material. It is recommended that presses be checked for applied load at the platen periodically on at least a 2- to 4-month schedule.

d. Platen Flatness Check. Press platens should be parallel and flat within 0.004 in in order to produce consistent-quality multilayer boards. A simple way to check that is to start with a cold press and lay a spiral of solder (approximately 0.06 in in diameter) across the face of press platen. Solder spirals should be approximately 3 to 4 in apart as shown in Fig. 23.5. Straight pieces of solder work equally well.

Apply enough load to the press to produce approximately 0.02- to 0.03-in deflection on the solder and then, with a micrometer, prepare a cross-the-corners profile of measurements on the solder. If measurements vary more than 0.005 in, it is likely that a corresponding or even increased variation will be noticed on laminates produced in the press. Press manufacturers can usually correct such problems by adjustments or rework, although extensively warped platens must sometimes be replaced.

e. Platen Heating Uniformity Check. Nonuniform heat distribution (greater than $\pm 3^\circ\text{F}$ variation of surface temperature) on press platen surfaces can be a problem on both steam and electric presses. Greater temperature variation can cause voids in the laminate bond line. In general, a 2-in outside border zone of any platen has a greater heat loss than the central portion of the platen and should not be used for lamination. Two methods of checking platen temperature uniformity are to prepare a grid pattern (2 to 3 in) of thermocouples or temperature-indicating paint dots. Either pattern should be on a piece of nonmetallic material approximately $\frac{1}{8}$ in thick.

Cover the thermocouples or paint dots with a sheet of release material and two sheets of 0.02-in-thick chipboard material (Fig. 23.6). This lay-up is loaded into a preheated (340°F) press for 4 min and then removed. Thermocouples should be connected to a recorder to read temperature just prior to removal of the lay-up from the press.

The test procedure described here may be varied depending on a given set of equipment. The purpose of the test is to leave the lay-up in the press long enough to get a temperature indication resulting from a point on the adjacent platen, with as little influence as possible from the opposite platen and the surrounding area of the adjacent platen. With the lay-up and temperatures described, monitored temperatures should be in

the 300 to 310°F range. Temperatures on a press platen surface area should be within 6°F. Both platens of a press opening must be checked. The checkout procedure is recommended annually, or when lamination problems become prevalent.

f. Press-loading Equipment. Depending upon size and number of press openings, a variety of carts, conveyors, and elevator lifts can be employed to transport and ensure accurate placement of the lamination fixtures in the press openings.

g. Registration Pin Removal. Following lamination, registration pins are surrounded by resin from the laminate. It is important that pins and mating holes in fixtures be coated with spray or liquid mold release material prior to being used. Removal of the pins is then readily accomplished with an air- or hydraulically assisted press. Hand-operated arbor press also may be used, but hammer and pin removal methods are not recommended because of the possibility of marring lamination fixture features.

h. Lamination Fixture Maintenance. Lamination fixtures should be routinely protected from resin squeeze-out from the laminate package. Protection is secured by the use of spray or liquid mold release applied to registration pins and the insertion of a sheet of release material between each half of the lamination fixture and the laminate. Also,

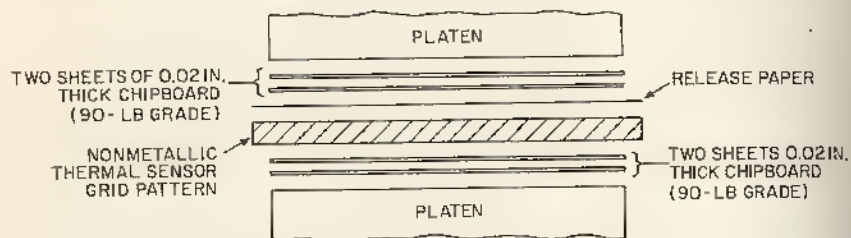


Fig. 23.6 Temperature distribution buildup.

between use cycles, the fixtures should be inspected for droplets of resin on either surface which will cause localized high-pressure points or depression in the surface of the laminate. Excess epoxy may be removed by solvents or mechanical means.

There are, no doubt, other specialized techniques relating to lamination which should be considered for each fabricator's facility. Operational techniques which can be employed to make the lamination process more consistent should be given careful consideration. Lamination is the first multilayer board process step which requires a complete restart owing to a process problem. Also, since several laminates are usually processed in a given press load, the importance of a consistent lamination process, skilled operators, and good equipment can not be overemphasized.

LAMINATE PREPARATION

10. B-Stage Characteristics In order to maintain a consistent lamination process, the multilayer board fabricator must be concerned with at least six specific characteristics:

1. Glass weave type. This determines bond line thickness.
2. Resin type. C-10 and FR-4 can produce different laminate bond strengths.
3. Resin flow. (a) Too high a flow may result in a resin-starved bond line. (b) Too low a flow may result in too thick a bond line or voids in the bond line.
4. Resin content. (a) Too low a resin content may result in a resin-starved bond line. (b) Too high a resin content may result in too thick a bond line.
5. Gel time. "Gel time" and "tack time" are commonly used interchangeably. Both terms are meant to indicate the point when a B-stage resin has been heated, gone back to a liquid state, and is beginning to resolidify into a solid thermoset condition. Measurement consistency is a function of test apparatus and some judgment on part of the test operator.
6. Volatile content. Volatiles consisting of retained solvents or moisture in B stage must be controlled at less than 1.0 percent by weight to prevent formation of gas bubbles and voids in laminate bond lines.

The preceding characteristics and the procedures for measuring them have been defined in an attempt to allow consistent testing by the B-stage suppliers and users. Standard test procedures are available from the Society of Plastics Industry, Inc. and

military specification Mil-G-55635A. Despite the standards, inconsistent results between users and suppliers can still occur owing to operator techniques, test equipment differences not specified in the standards, and sampling techniques.

From the use standpoint it is essential that the user establish a well-defined and controlled method of testing the B stage used in fabricating multilayer boards. That is very important, since from even a single B-stage supplier the B resin constituents can vary and the resin itself can come from different sources to the B-stage manufacturer. Testing should be done on a sampling procedure for each resin batch prepared by the B-stage supplier.

As mentioned previously, there are standards for testing the various characteristics of B stage, but the standards are not truly representative of the lamination process for most multilayer board fabricators. Differences occur in lamination fixtures, types of presses, types of lay-ups, types of press control, etc. Therefore, in addition to standard testing, it is good shop practice for each new batch of B stage, to make a representative size laminate of several pieces of B stage to observe the way it responds to a normal press cycle. Resin starvation or resin voids in the sample will require adjustments to the press cycle or possible rejection of the material.

11. B-Stage Storage and Handling Prior to Lamination B stage is very susceptible to moisture, solvent, dirt, and oil particle contamination. Any of these foreign materials are likely to produce spots of delamination within a laminate bond line. In addition, time and temperature gradually promote a change from the semicured to the fully cured epoxy state.

To protect and maintain material usefulness, the following shop practices and conditions should be observed.

1. Material should be packaged by the supplier in airtight containers (sealed plastic bags).
2. Material should be stored in a cool dry environment (less than 50°F and 50 percent RH).
3. Material should be handled only by operators wearing clean cotton gloves.
4. Material should be cut to size in positive-pressure room.
5. Material should be placed in a vacuum chamber to remove volatiles prior to use.

Observation of those practices will resolve many lamination process problems.

12. Circuit Laminate Preparation Prior to Lamination Circuit laminates are the circuit patterns etched in copper-clad to C-stage glass-filled epoxy material. Laminates may have circuitry on one or both sides.

Prior to lamination with B stage, the laminates should undergo some combination of the following preparatory steps:

1. Chemical-clean. Dilute HCl (10 percent) dip followed by water rinse and forced-air drying.
2. Scrub, rinse, and dry. This is either an acid pumice and water slurry scrub or a composite material brush scrub followed by a water rinse and forced-air drying system.
3. Oxide treatment.
4. Bake cycle (optional). One-hour bake at 250°F to remove moisture from material absorbed during cleaning steps.

Laminates to be bonded are treated by some fabricators with a copper-oxidizing treatment to promote adhesion strength. Most chemical suppliers for the printed circuit industry have proprietary brands of this material available. The treatment appears to create improved delamination resistance with the B stage through a combination of increased surface area and chemical bonds between epoxy molecules and the copper oxide.

Following the bake-out step, circuit laminates are ready for the lay-up operation, in which items discussed to this point are brought together: lamination fixtures, registration pins, release paper, and B stage.

13. Laminate Lay-up Procedures. Laminate lay-ups should be performed in a controlled environment room for best results. Desirable conditions include:

1. Positive-pressure air
2. Filtered air source

3. Daily cleaning of the room
4. Clean protective gloves and lint-free gowns and caps for operators
5. Controlled temperature and humidity (70°F, 50 percent RH).
6. Precleaned lamination fixtures with mold release on registration pins and holes prior to entering the room

A typical laminate lay-up as it would be in the press ready for lamination is shown in Fig. 23.7.

To increase press output, laminates can be stacked double or more, depending on thickness up to approximately a 0.150-in-thick stack, by the addition of 0.020- to 0.030-in steel plates between laminate stacks. The 0.150 in is not an absolute dimension; it is a function of heat transfer rate through the total lay-up plus alignment capability of the

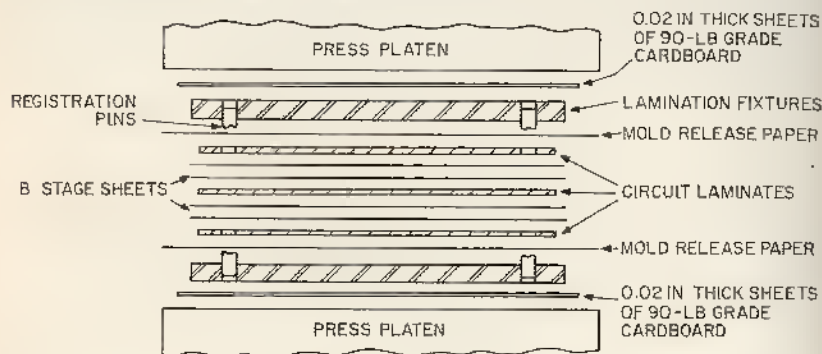


Fig. 23.7 Typical laminate lay-up.

registration pin system. A laminate stack is defined as the lay-up plus its two sheets of release paper. The steel plates are necessary to prevent circuit image transfer from one stack to the next.

Amount of B stage in the bond line will vary because of the amount of etched circuit to be filled and circuit copper thickness. Considering only B stage made of 108 glass (0.002

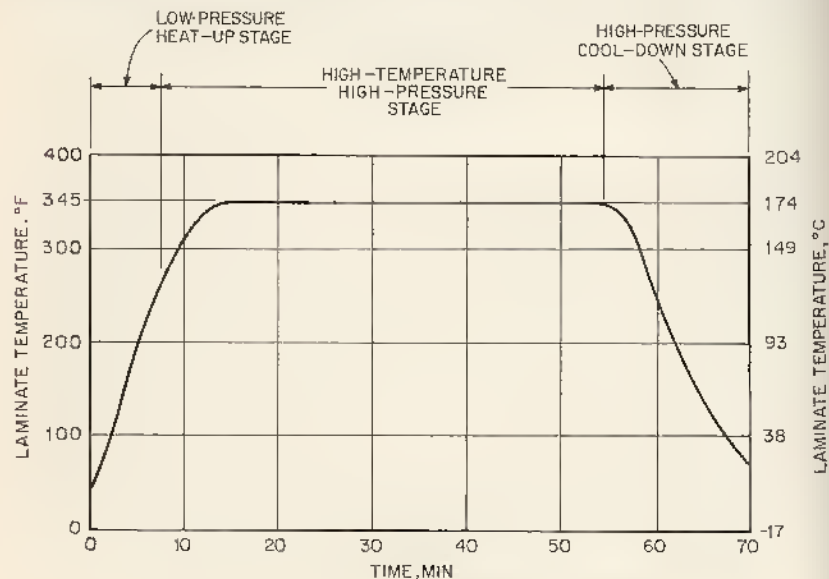


Fig. 23.8 Typical two-stage pressure lamination cycle.

to 0.0025 in thick before lamination) a bond line should contain a minimum of two sheets of B stage; plus one more sheet for each 2 oz of copper circuitry in the bond line.

Lamination fixtures should be loaded in the press so that laminate area is symmetrically loaded about the center of the press platen. Off-center loading can cause nonuniform laminate thickness and even damage to press support rods. Within a given press opening, laminates should be of the same or equivalent final thickness to avoid nonuniform platen loading.

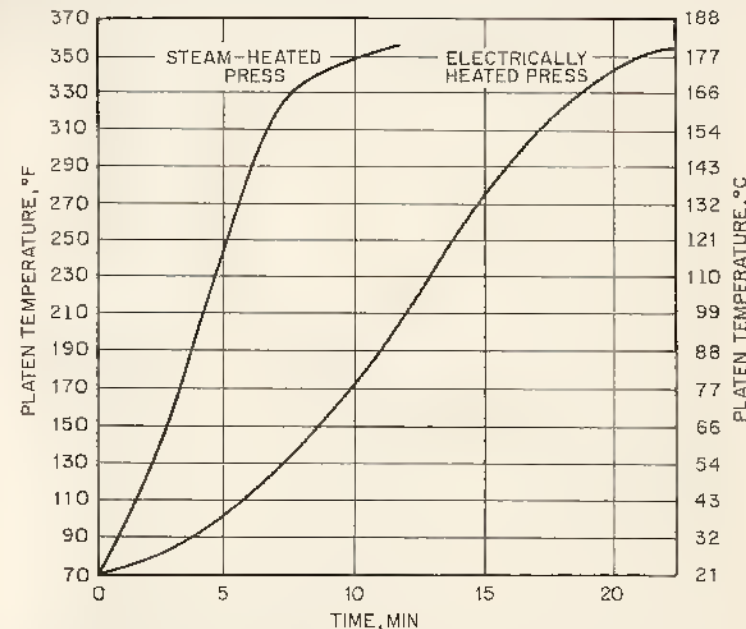


Fig. 23.9 Typical steam-heated and electrically heated press heat-up curves.

a. Lamination Press Cycles. Before considering the press cycles, it is appropriate to consider the requirements of a good laminate bond. Bond line requirements are reasonably consistent:

1. No delamination after thermal shock. (Several definitions of thermal shock exist. A representative one is 550°F solder float for 10 s for a 2 × 3-in-square specimen.)
2. No bubbles or voids in the bond line.
3. No resin starvation (white glass fabric pattern in B stage visible) in the bond line.
4. No dirt or foreign particles in the bond line.
5. It is imperative to have good layer-to-layer circuit registration to allow drilled holes to hit internal circuit pads.

To accomplish these desirable laminate bond features, a number of press cycles are available.

14. Two-Stage Pressure Cycle With either a Hot or Cold Start Two-stage pressure cycles will vary as to exact pressures, but in general they have a low pressure (2- to 25-psi*) dwell time when the B stage is melted. Low-viscosity resin wets, fills, and drives air out of the etched copper laminates. The high-pressure application point in the cycle is critical. Too early results in excess resin squeeze-out and resin starvation; too late results in laminate bond line voids and registration shift because of increased resin viscosity.

Cold start press cycles (Fig. 23.8) are associated with steam-heated presses, and hot-start press cycles are associated with electric presses. To help understand the reason, see the typical platen heat-up rate in electric and steam presses, Fig. 23.9, and the B-stage

*This represents an industry survey range. Personal experience recommends a 10-psi maximum.

resin viscosity curve, Fig. 23.10. Electric presses can take twice as long to heat as steam presses.

B-stage resin is cured by the combination of temperature and time, or thermal energy, to which it is exposed. Viscosity of the resin is also a function of the absorbed thermal energy. Figure 23.10 shows that, during a fast press heat-up cycle, there will be a longer period of time T_1 when the resin has a useful viscosity for fabricating multilayer boards. A slower heat-up cycle such as on the electric press presents a shorter period of time T_2 when the resin has an acceptable viscosity for fabricating multilayer boards. Therefore,

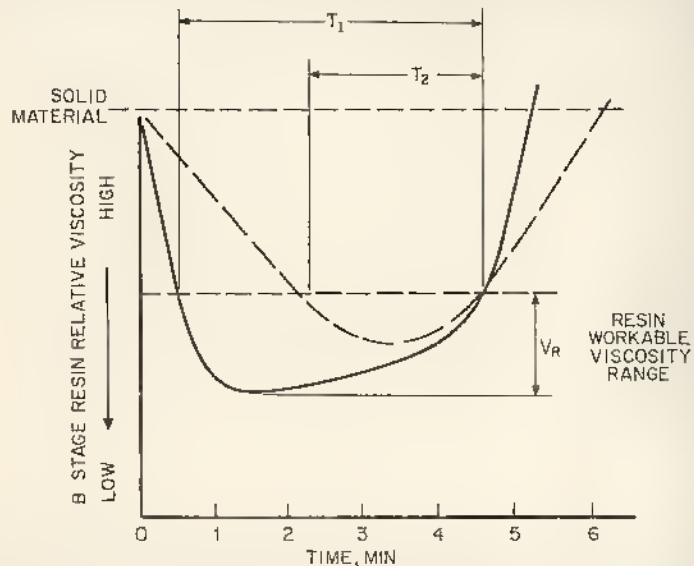


Fig. 23.10 Typical high-flow B-stage viscosity curves during lamination. Solid line indicates B-stage viscosity curve for fast-heat-up press. Broken line indicates B-stage viscosity curve for slow-heat-up press.

electric presses are usually preheated either partially or to full temperature for MLB lamination use. To prevent excessive resin runoff from a full-temperature preheated press, the heat-up cycle can be slowed by inserting a sheet or two of 0.020-in-thick cardboard between the press platens and the lamination fixture.

One way to obtain successful lamination results is to prepare several test laminates of new B stage by using the production press and lamination fixtures. Prepare the laminates, and increase low-pressure cycle times by 2-min increments for from 2 to 8 min. The short cycle time should produce excessive resin flow and resin starvation, whereas the long cycle time should show the presence of voids and air bubbles. Select a low-pressure cycle time in midrange which gives good visual results in the laminate. The visual results will be supported by consistent resin flow and resin content measurements.

The above discussion is primarily applicable to high-flow B stages used under low-pressure conditions for the fabrication of multilayer circuit boards. Low pressures are used to minimize hydraulic side loads on circuitry within the bond line and resultant movement or lamination shift of internal board circuitry.

15. Single-Stage Pressure Cycle With either a Hot or Cold Start See Fig. 23.11. Single-stage pressure cycles have long been used with low-flow-grade B stage. The B stage, in turn, is less frequently used for multilayer board work but more often for the production of C-stage material. Press operation is simpler, and higher press pressures (300 to 1000 psi) minimize the variables which are of concern in typical MLB work.

Suppliers of B stage are now offering material formulated for use in multilayer board fabrication which does not require the two-stage press cycle. Single-stage pressure cycles offer the benefit of simplified lamination cycle controls, which in turn can improve

laminated product consistency. The B stage can be used with either a hot- or cold-start press cycle.

16. Vacuum Lamination Vacuum lamination is an experimental process which has been tried by several multilayer board manufacturers. The purpose of the process is to promote removal of gases and air while using a medium- or low-flow B stage. Reduced resin flow

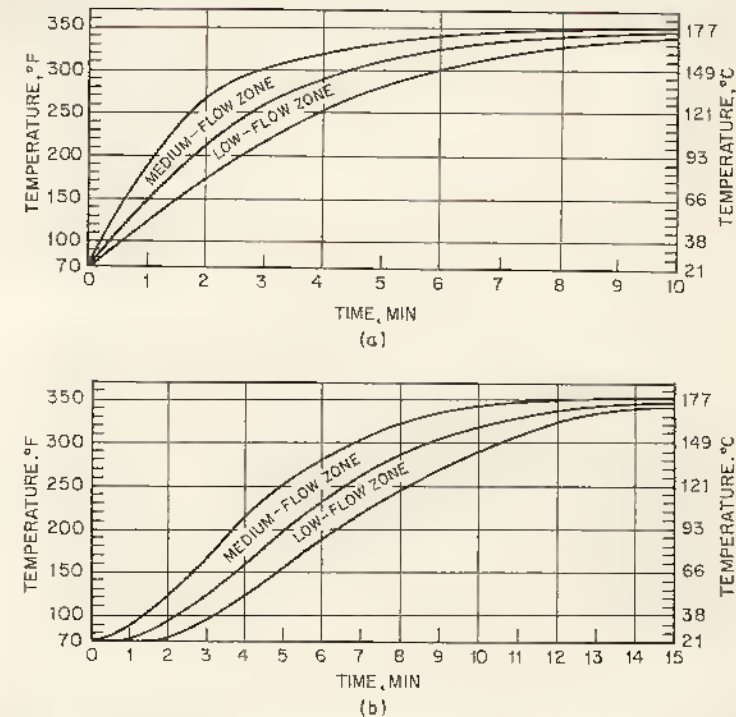


Fig. 23.11 Prepreg system, suggested thermal profile. (a) Hot press cycle; (b) cold press cycle. The suggested thermal profiles were computed to a minimum fill based on approximately 30 percent copper circuit density. (Fortin Laminate Inc.)

with that type of B stage would also provide uniform laminate thickness. Implementation of the process requires special equipment: compressors, gauges, and vacuum frame seals for the press openings. The requirements for and use of vacuum lamination has decreased with the availability of high-flow B-stage resin systems.

17. Cycle Type Comparison To summarize the discussion on lamination cycles, considerably more space has been devoted to the two-stage press cycle because it has been in use longer. Single-stage press cycles have been associated primarily with raw material (C stage) fabrication, which is not the primary concern of this chapter. Single-stage press cycle B stage for multilayer board fabrication is being produced and offers the advantage of a simplified lamination to the printed board industry.

Even with the benefits of a simplified lamination process, lamination involves many variables of materials, suppliers, and equipment. To minimize the task of resolving lamination problems when they occur or when a new lamination capability is initiated, it is essential the production facility equipment be used and observed. Detailed lamination process information and troubleshooting assistance is available from most B-stage suppliers, but the user must understand the effects of his specific facility lamination fixtures, press controls, etc. on the B stage through preparation of sample B-stage laminates.

LAMINATION PROCESS TROUBLESHOOTING

Lamination process problems are extremely costly to the multilayer board fabricator because they are not always obvious at the completion of the process. Often it is not until the board is through final pattern etching that lamination problems are evident. To prevent exposure to losses that can amount to thousands of dollars in a few days time, some manufacturers routinely (day or shift) etch the outer copper from a board which has just been laminated to check for laminate quality by both visual test and thermal shock resistance. If production volume warrants, that practice can save money by reducing scrap losses.

When problems do occur, many causes and effects are of an overlapping nature and are difficult to analyze. Some lamination problems and their potential causes and effects, are described in the following section.

18. Lamination Problems—Causes and Effects

1. Resin starvation (bond line appears white; glass weave is visible) may result from excessive pressure in low-pressure cycle, which produces high resin flow when B stage is in the fluid state. Bond lines and laminate may be too thin.

Corrective Action:

Check low-pressure calculations. Check gauge (hydraulic pressure) settings and calibrations. Calibrate press low pressure with load cell or crush tubing comparison test.

2. Resin starvation and a thin laminate may result from excessive high pressure.

Corrective Action: Reduce pressure, but not below 150 psi. The problem is more likely to occur on small board with short B-stage flow dimension.

3. Resin starvation and resulting thin laminates may occur because of application of high pressure at wrong time or temperature following precure (low-pressure) cycle.

Corrective Action: Prepare set of B-stage laminates (six plies each) at 2-min intervals from resin starvation condition to air bubble condition. Adjust application of high-pressure time to yield B-stage laminate of good visual quality (no resin starvation). Test panels must be laminated in the same type of fixture and with the same amount of cardboard padding as actual lay-ups.

4. Resin starvation occurs because of too few plies or no B stage in the laminate.

Corrective Action: If not an obvious situation, section the laminate, or perform a burn-out to determine the amount of B stage.

5. Resin starvation may occur because a B-stage lot has too low a resin content, too long a gel time, or too high a resin flow.

Corrective Action: Any combination of the conditions may be accommodated by adjusting the time and temperature at which high pressure is applied. See description 3.

6. Air voids and thick laminates may occur if pressure is too low during the low-pressure cycle.

Corrective Action: Check low-pressure calculation and calibration.

7. Air voids and thick laminates may occur if high pressure is too low in lamination cycle, or if press went into high pressure at too high a temperature. Low pressure doesn't produce enough flow to fill or carry out air voids.

Corrective Action: Check high-pressure calculation and calibration. Press may not go into high-pressure cycle if recorder or pressure control timer malfunctions.

8. Air voids and thick laminates may occur if high pressure is applied too late in the low-pressure cycle and the resin is too viscous to flow properly. That can also cause movement of circuitry on inner layers.

Corrective Action: Prepare a flow curve vs. time curve for the B stage being used and see if press controls are set properly. (Voids indicate too long a pressure control timer setting.) If a flow curve doesn't exist, prepare one as described in 3.

9. Air voids may be caused by excessive volatile content or grease on B stage.

Corrective Action: Place B stage in vacuum chamber for 2 h at 27-in-Hg vacuum and then retest for volatility. Discard out-of-specification, volatile or grease-spotted material.

10. Air voids in laminates may be caused by poor cleaning of layers which leaves contaminant particles or grease spots on surface. Resin doesn't adhere in dirty spots and leaves masses of delamination.

Corrective Action: Check output of circuit layers from scrubber for consistently poor

cleaning. New brushes or air supply may be required on scrubber. Look for source of accidental contamination in layer such as falling on floor, etc.

11. Air voids may be caused by low press pressure, which will cause a low resin flow condition. Insufficient pressure yields low resin flow and air voids or delamination in board.

Corrective Action: Check on steam valve obstruction and boiler operation.

12. Surface dents and resin on surface of laminate may be caused by particles of resin left on press pans, by particles of dust or B stage collecting on release paper, or by creases in release paper. B stage on release paper can create the spots of resin on lamination fixtures, along with stray particles picked up when laminates are removed and lamination fixtures are readied for reuse.

Corrective Action: Clean press pans with resin solvent. Perform lay-up in positive-pressure room to reduce the amount of airborne dirt that enters the lay-up area.

13. Air voids and a thick laminate may occur if press temperature is low in the high-pressure cycle.

Corrective Action: Verify low temperature by monitoring with a thermocouple on the recorder. Perform maintenance check of the press heating system if the thermocouple shows low temperature.

14. Inner circuit shift can be caused by excessive pressure during low-pressure stage of press cycle.

Corrective Action: Check press pressure calibration. Check rate of pressure application; 10,000 to 15,000 lb/s has worked satisfactorily.

15. Placement of more than one lamination fixture in a press opening requires all combinations of fixtures plus lay-ups to be the same thickness. Differences greater than approximately 0.008 in can show up as nonuniform-thickness laminates or laminates with inner shift.

Corrective Action: Avoid the practice of different laminates being laminated within a single press opening. Make sure all laminates in the press had the same number of pieces of B stage and laminate layers. Take special precautions and use shim sheets to obtain uniform total thickness.

16. Platens on some multiple-opening presses are essentially "floating" or not guided for parallelism by structural members. Nonuniform laminates and inner circuit shift can be produced by nonuniform centering of lamination fixtures on the platens.

Corrective Action: Use symmetrical loading of identical size laminates. If laminates are smaller than the fixture, make sure it's the laminate rather than the fixture which is loaded in a symmetrical manner.

17. Dirt particles and pieces of foreign matter accidentally included in a lay-up will cause small blisters or spots in a laminate. That is usually detectable by a local increase in laminate thickness.

Corrective Action: Maintain clean lay-up area and be observant during lay-up procedure.

18. Incorrect amount of B stage in a bond line or a B-stage lot with too short a gel time will give thick laminates. Also, heavy glass weave in B stage will give a thick laminate.

Corrective Action: Check lay-up for correct amount and glass fabric style of B stage. Recheck B-stage gel time.

19. Board warp and twist may be caused by nonsymmetrical lay-ups—more heavy-weave glass material layers on one side of laminate than other. It can also be caused by mixed glass fabric major weave directions and by removal of laminate from the press before the curing cycle is completed.

Corrective Action: Always use material with the same major weave direction for a given layup. Laminates should receive an approximate 45-min cure cycle at 350°F followed by a cool-down to approximately 100°F in 20 min. Boards should be designed with, as nearly as possible, glass weave style symmetry about the centerline of a board.

20. Inadequate thermal shock resistance can be caused by poor laminate cleaning, improper handling of laminates, excessive conversion coating thickness on copper, or wrong type of B stage.

Corrective Action:

Identify the problem by high-magnification observation of delaminated surfaces.

Check brushes on scrubber.

Review handling procedures.

Consider multiple scrub steps or new equipment.

Consider chemical cleaning step after mechanical scrub.

Consider oxide treatment of circuit layers.

Perform flammability test on B stage. FR-4 B stage will not sustain combustion and will usually delaminate easier in thermal shock unless inners have been oxide-treated.

SPL PREPREG-REINFORCED PLASTICS COMMITTEE—TEST METHOD

Flow of Preimpregnated Products

1. Scope

1.1 This method is designed for use in determining the flow of preimpregnated products used in molding flat panels and shapes. Parts are laminated at pressures from a few pounds per square inch to several thousand pounds and at temperatures from 180 to 400°F. To have correlation with molding performance, it is essential that the flow test be run at a temperature and pressure in the neighborhood of the actual conditions. Therefore, the temperature and pressure should be chosen from paragraph 1.2 which is closest to the operating conditions.

1.2 Test Conditions

Test No.	A	B	C	D	E	F	G
Temp °F.	225	225	250	250	300	300	300
Pressure #/sq. in.	15	100	15	100	15	100	1000
Test No.	H	I	J	K	L	M	
Temp °F.	325	325	325	350	350	350	
Pressure /sq. in.	15	100	1000	15	100	1000	

2. Test Specimens

2.1 The test specimens shall be cut 4 in. square (16 square inches area). The specimens shall be cut on the bias.

2.2 The number of specimens used for each flow test will vary with the weight per unit area of the product. The number of 4 in. square plies taken is that which will give a total weight nearest to 25 grams.

2.3 A strip shall be cut the full width of the cloth or paper, i.e., from one edge to the other. The specimens shall then be cut from this strip. The length of the strip will depend on the number of specimens necessary to give the required 25 gram weight. The specimens shall be cut from the center and edge.

3 Apparatus

3.1 A square metal cutting template 4 in. × 4 in.

3.2 Balance capable of weighing to 0.01 grams.

3.3 Dow Corning DC-20 release agent or equivalent.

3.4 Two No. 430 stainless steel polished #4 finish press plates 16 gauge × 6 in. × 6 in.

3.5 Hydraulic press with one opening containing accurately machined, leveled and heated platens. These platens shall be capable of maintaining a temperature within 5°F of the test temperature in paragraph 1.2 over the entire area of the platen except for an area 1 in. from the edge of the platen. The press shall be equipped with a gauge or gauges to read force of line pressure required to give the psi called for in paragraph 1.2 and be calibrated to be accurate to within 5 percent. (Refer to press manufacturer for calculations). The pump or other closing device must be capable of closing the press and applying full pressure on the specimens within ten seconds of the time they are inserted in the press.

3.6 Timer

4. Standard Conditions

4.1 Standard conditions shall be 73.4 plus or minus 10°F, and 50 plus or minus 8 percent relative humidity. Specimens shall be stored for at least 4 hours at standard conditions prior to test.

5. Procedure

5.1 Preparation of new caul plates. Rinse plates in MEK or equivalent solvent, then scrub with a mild abrasive cleaner such as Bon Ami, then water rinse and dry. Dilute DC-20 1 part to 8 parts MEK or toluol. With a clean rag apply a light coat of diluted DC-20 and bake in the heated press or oven for at least 1 hour at 325°F or above. With a clean rag wipe off excess release agent. The plates are now broken in. In between

flow tests normally all that is required is an occasional light application of DC-20 solution.

5.2 Cut the required number of 4 in. × 4 in specimens to give a total weight nearest to 25 grams.

5.3 Weigh all specimens together to the nearest 0.01 gram, record weight as W_1 .

5.4 Place the specimens in the center between caul plates maintained at standard temperature.

5.5 Adjust press temperature to that chosen from paragraph 1.2.

5.6 Insert the assembly of plates and specimens in the hydraulic press, in the center of the opening. Close the press and apply pressure called for in paragraph 1.2.

5.7 Cure sample minimum of 5 minutes. Some materials will require longer to enter the resin to the point where the squeeze-out or flash will be cured enough to break off and the resin on the surface of the laminate itself will be cured enough not to stick to the caul plates.

5.8 Remove laminate, allow to cool to room temperature, break off flash with fingers, if any remains, then scrape with dull knife or back of a knife handle being careful not to remove any glass.

5.9 Weigh sample to nearest 0.01 gram and record weight as W_2 .

6. Reports and Calculations

The flow percentage shall be reported to the nearest 0.1% and calculated as follows:

$$\text{Percent flow} = \frac{W_1 - W_2}{W_1} \times 100$$

REFERENCES

1. Henry Lee and Kris Neville, "Handbook of Epoxy Resins," McGraw-Hill Book Company, New York, 1967, chap. 1
2. "Silane Coupling Agents," Dow Corning Corporation, 1970.
3. Frank Thatcher, The Mica Corporation.
4. Fortin Laminating Division, Monogram Industries, Inc.

Section **7**

Flexible Printed Wiring

Chapter 24

Flexible Printed Wiring Design Layout

J. R. CANNIZZARO

Digital Equipment Corp., Maynard, Massachusetts

Introduction	24-4
Advantages of Flexible Printed Wiring	24-4
Limitations and Disadvantages of Flexible Printed Wiring	24-5
Classification	24-5
Class I, Single-Sided Flexible Printed Wiring	24-6
1. Class Ia. Single-Access Uncovered	24-6
2. Class Ib. Single-Access Covered	24-6
3. Class Ic. Double-Access Uncovered	24-6
4. Class Id. Double-Access Covered	24-7
Class II, Double-Sided Flexible Printed Wiring	24-8
Class III, Multilayer Flexible Printed Wiring	24-8
5. Class III, Type I. Flexible Base Material; End Product Flexed	24-10
6. Class III, Type II. Flexible Base Material; End Product Not Flexed	24-12
7. Class III, Type III. Flexible Base; End Product Formable	24-12
8. Class III, Type IV. Flexible and Rigid-Base Materials	24-12
Interfacial Connection	24-13
Design	24-13
9. Interconnection Assessment	24-13
10. Layout Development	24-13
11. Processing Parameters and Tolerances	24-16
12. Access Windows	24-18
13. Conductors	24-17
Electrical Design of Flexible Printed Wiring	24-18
Design of Flexible Wiring with Transmission Line Properties	24-22
14. Characteristic Impedance	24-24
15. Microstrip Configuration	24-24
16. Strip Lines	24-25

Electrical Parameters	24-25
17. Capacitance	24-25
18. Time Delay	24-27
19. Cross Talk	24-28
20. Attenuation	24-29
Mechanical Design for High-bending-stress Applications	24-29
Bibliography	24-34

INTRODUCTION

The present space-age semiconductor industry is constantly trying to increase circuit density, reliability, and miniaturization and meet stringent economic demands, as illustrated in the transition of integrated circuits from flat packs to MSI and LSI packages. To keep pace, and to provide the necessary interdependent relations with these semiconductor devices, the printed wiring industry expanded tremendously in the late 1960s and early 1970s, and along with that growth came the necessary advanced technology. Starting with the now commonplace two-sided, plated-through-hole boards, dimensional tolerances and material specifications were tightened, and this provided the groundwork for multilayer interconnection boards and flexible printed wiring (FPW) products.

Flexible printed wiring is widely used in electronic packaging of computers, telephones and telephone equipment, relays, gyros, missiles, computers, automotive instrument panels, and even in blanket heaters. Future applications will be widespread and limited only by the imagination of design engineers.

ADVANTAGES OF FLEXIBLE PRINTED WIRING

a. Flexibility. The obvious advantage of utilizing a flexible dielectric material presents the design of circuits which may travel in the x , y , and z planes or may be coiled as in retractable drawer applications. Self-retracting FPW interconnections can and have been designed to be flexed as much as 7×10^6 times without failure.

b. Volume Reduction. The use of FPW will reduce the space required for interconnection in a given application as compared with discrete wire cables. The ease of being formed to the contour of the box, the thin, flat cross section, and the reduced conductor size permit more efficient packaging, which is manifest in decreased interconnection volume.

c. Weight Reduction. The lower-volume design capability, the use of multiple conductors in close spacing, the use of smaller-cross-section conductors (based on capability to design to current-carrying capacity rather than mechanical strength) all add up to a lower overall interconnection weight.

d. Wiring Consistency. The use of FPW eliminates wiring errors inherent in discrete wire harnesses. Once the conductor-producing artwork is approved, all subsequent copies will be identical. That alone results in assembly line savings due to elimination of rework.

e. Increased Reliability. The overall system reliability of the electronic box is increased when FPW interconnections are employed. That is primarily due to the reduction of elimination of interfacial connections (solder joints, welds, etc.) because of the capability of x , y , and z -plane wiring. Also, when the FPW is designed properly with adequate strain relief, etc., the interconnection device virtually eliminates conductor breakage or open circuits. The encapsulation effect of FPW design provides a rugged package which is resistant to environmental conditions.

f. Electrical Design Freedom. FPW can be designed to have transmission line properties by controlling the capacitance C , inductance L , and characteristic impedance Z_0 . Those properties are a function of the conductor line width, conductor thickness, dielectric separation (i.e., distance from ground plane in microstrip and tri-plate designs), and the dielectric constant of the base material. The consistent location of individual conductors maintains consistency of such electrical parameters as noise and capacitive coupling.

g. Self-termination. By definition, FPW products are preterminated as in a plated-through hole, eyelet, etc. This feature eliminates any wire preparation and tinning and

thereby makes a substantial savings in assembly cost possible. By keying the FPW device in symmetrical connector patterns, misplacement can be avoided.

h. Mass-termination Techniques. The termination of the FPW device can be performed en masse. For example, in terminating a plated-through-hole FPW device to a pin connector, all terminations can be performed by dip or wave soldering in lieu of hand soldering a discrete wire cable.

i. Custom Material Applications. FPW devices may be fabricated from a variety of commercially available materials to suit the specific application. For example, in a simple low-cost interconnection application, an inexpensive vinyl polymer may be utilized; but in a sophisticated military application in which the device must possess high-temperature and transmission line properties the use of fluorinated or polyimide polymers may be dictated.

j. Overall Interconnection Cost Reduction. The reduction of overall interconnection costs is manifested by the following features of FPW devices:

1. Prelocation of conductors eliminates costly wiring errors and rework.
2. Replacement, if required, is easier.
3. Termination costs are reduced by mass soldering, welding, etc.
4. Reduction of brackets, wire clamps, etc., is made possible by custom design of circuitry. FPW may be bonded to structure walls to eliminate attachment hardware.
5. Shielding can be inexpensively applied to FPW device.

k. Continuous Process Capability. Because flexible dielectric materials are available in continuous rolls, FPW can be manufactured in a continuous web process with a significant savings in production costs.

LIMITATIONS AND DISADVANTAGES OF FLEXIBLE PRINTED WIRING

Coupled with the many advantages of FPW interconnections are some inherent limitations and disadvantages which the designer must bear in mind prior to designing an FPW into a system. They are enumerated below:

a. High Nonrecurring Start-up (NRSU) Costs. Since the FPW device is custom-designed and manufactured for a specific application, the initial costs will be relatively high. These costs are mainly in circuit design, layout, and artwork. Generally it would not be wise to use FPW in low-volume-oriented applications unless special requirements dictate their use.

b. Engineering Change is Difficult and Costly. Because of the custom nature of the FPW device, changes in circuit routing or periphery are difficult and somewhat costly because the layout, drawing, master artwork, and working artwork must be changed. If possible, those changes could be incorporated in the FPW device by the use of jumpers, etc.

c. Difficult Repairability. Repair of an FPW device is difficult, since the protective insulation cover coat or base material must be removed and replaced after repair.

d. Size Limited. Since FPW is usually manufactured in batch processes, the overall size is limited by equipment constraints. A 30 × 30-in FPW panel, for example, would be considered to be quite large.

e. Handling Damages. Assembly personnel must be thoroughly trained in the care and handling of FPW to preclude damage even prior to actual system use. In some designs, bending stresses imposed on the manufacturing line can severely decrease effective life of the system.

f. Soldering and Rework. Soldering and rework of the FPW device must be performed by experienced personnel to prevent damage to circuits in the form of lifted lands or pads, lines, blistering, etc.

CLASSIFICATION

The term "flexible printed wiring" encompasses a plurality of separate and distinct types of wiring products. In all cases FPW consists of random copper conductors on a flexible dielectric base material. The number of copper conductors employed in the design sets the class of the product. The generic categories of FPW are identical with those of rigid printed wiring products.

Class I. Single-sided flexible printed wiring

Class II. Double-sided flexible printed wiring

Class III. Multilayer flexible printed wiring

This section will present definitions and a breakdown of current and potential applications of the three classes of FPW and the specifications governing designs and performance.

CLASS I. SINGLE-SIDED FLEXIBLE PRINTED WIRING

Single-sided flexible printed wiring contains only one conductive plane and may or may not have an overcoating, here to be referred to as a cover coat.

The dielectric materials used to manufacture class I FPW products vary with the intended application. Those generally utilized are vinyls, polyethylene terephthalate (Mylar[®]), fluorocarbons (Teflon[®]), flexible epoxy-glass, flexible epoxy-polyester, polyimides (Kapton[®]) and fluorocarbon-polyimides.

If one were to consider use and not shape of the cover coat as a variable, then Class I FPW could be further subdivided in the following manner:

Class Ia. Single-access uncovered

Class Ib. Single-access covered

Class Ic. Double-access uncovered

Class Id. Double-access covered

A description of these subclasses is presented below:

1. Class Ia. Single-Access Uncovered Generally, a class Ia product is the least expensive type, owing to its basic configuration of a conductor over a base dielectric without a cover coat. A typical schematic and cross section of this type of FPW is shown in Fig. 24.1.

a. Applications. Class Ia wiring is generally utilized in non-strategic environmentally protected applications. Interconnections are accomplished by soldering, welding, or pressure. A high-volume application of this type of wiring was in the Bell Systems Touchtone-Tripline[®] handset. The early version of that telephone utilized class Ia wiring in conjunction with a rigid board. A subsequent redesign eliminated the rigid board in favor of class IIa, double-sided plated-through-hole FPW. Several important characteristics of the flexible dielectric utilized in the telephone application were that the dielectric required high insulation resistance, flame retardance, good tear strength, dimensional stability, and low cost. For that application an epoxy-polyester material was selected. It is interesting to note in the class IIa version that components were mounted directly on the flexible substrate even though the substrate was not bonded to a rigid support member. That was accomplished by clamping the FPW in such a way that it was, itself, held rigidly. Application of class Ia and IIa FPW in telephones is illustrated in Fig. 24.2.

2. Class Ib. Single-Access Covered Class Ib FPW is similar to class Ia, but it employs a precut cover coat. The customizing of the cover coat may be in the form of clearance holes or stopping the cover coat short of the termination area, as shown in Fig. 24-3 (p. 24-8).

3. Class Ic. Double-Access Uncovered Class Ic FPW permits access to the termination pad from either the top or bottom side. To permit that, the base dielectric is fabricated with access holes in the pad area. The access holes may be prepunched on location, etched, or mechanically removed from the base dielectric. A schematic representation of class Ic FPW is shown in Fig. 24.4.

a. Applications. Class Ic FPW is frequently utilized in high-production applications in which inexpensive interconnections are required and mounting from both sides is necessary. Application quantities should be high, since tooling will be relatively expensive. A specific example of the use of class Ic wiring is the interconnection of two rigid boards with an intermediate welded connection in a computer core memory application. In this application polyamide-imide-rolled copper FPW was used for the interconnection. The polyamide-imide material was chemically removed at the termination sites.

[®]Trademark of E. I. du Pont de Nemours & Company. The types and properties of flexible materials will be covered in the next chapter.

[®]Registered trade name of the Bell System.

4. Class Id. Double-Access Covered Class Id FPW is similar to class Ic, but the cover coat has access holes also to permit terminating from either side of the wiring device while maintaining a cover coat. Class Id FPW is fabricated from two layers of a dielectric material and one metal conductor layer as shown in Fig. 24.5.

a. Application. Class Id FPW devices are the most versatile. They are utilized where the cover coat is required for insulation from hardware, environmental protection, and insulation from itself and termination access is needed from both sides.

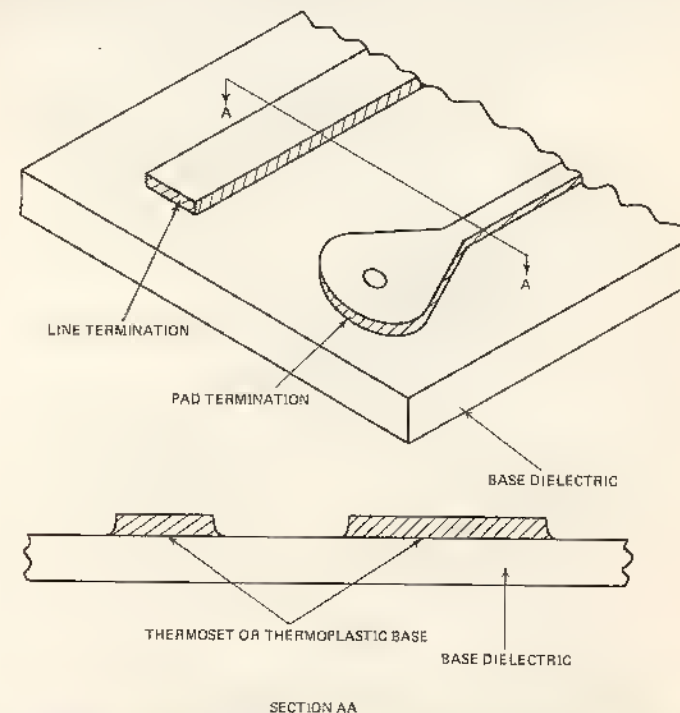


Fig. 24.1 Class Ia single-access uncovered flexible printed wiring.

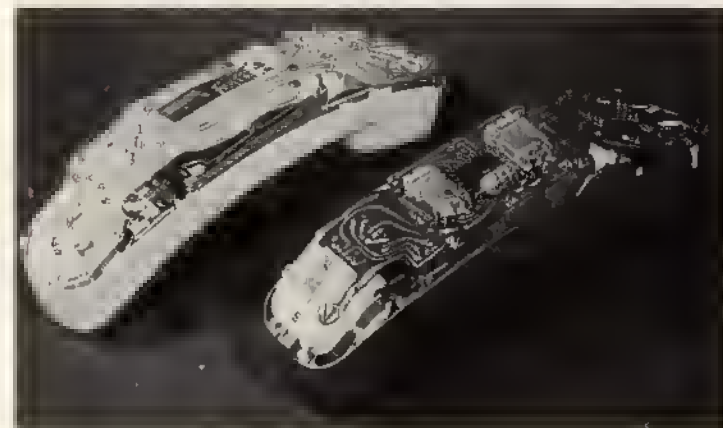


Fig. 24.2 The application of flexible printed wiring in telephone handsets.

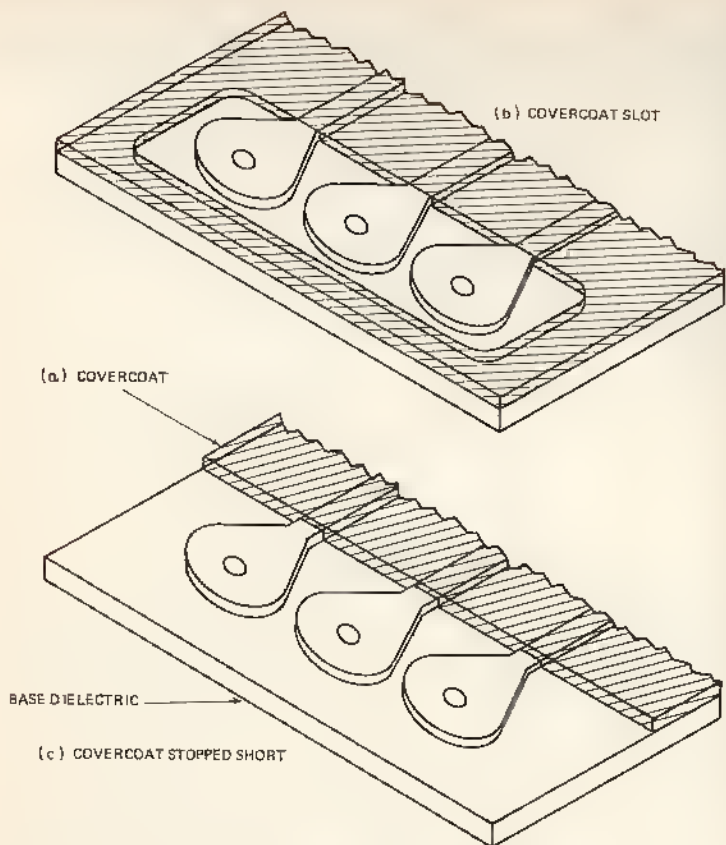


Fig. 24.3 Class 1b single-access covered flexible printed wiring.

CLASS II. DOUBLE-SIDED FLEXIBLE PRINTED WIRING

Flexible printed wiring which contains two conductive planes which may or may not be interfacially connected is referred to as class II double-sided flexible printed wiring. The applications and advantages of class II FPW are similar to those of single-sided printed wiring except for the added advantage of increased wiring density per unit area. The subclassification for class II FPW is based upon the existence or absence of interfacial connections and cover coating as shown below:

- Class IIa. Noninterfacial connection, uncovered
- Class IIb. Noninterfacial connection, covered
- Class IIc. Interfacial connection, uncovered
- Class IId. Interfacial connection, covered

As previously stated, the advantages and applications are similar to those of class I FPW, but with increased wiring density. A schematic representation of the class II type of wiring is shown in Fig. 24.6.

CLASS III. MULTILAYER FLEXIBLE PRINTED WIRING

One of the latest achievements in the printed wiring industry has been that of combining the advantages of flexible and multilayer printed wiring to form multilayer flexible printed wiring (MFPW). The packaging technology provides, among other advantages, high-density, three-dimensional circuit interconnection capability in lieu of the conventional xy planar approach.

Multilayer flexible printed wiring is used in the packaging of electronics where high electrical performance is mandatory and space and volume are at a premium, as in aerospace applications. An example of a high-density MFPW application is shown in Fig. 24.7. In that avionics computer application, two MFPW boards manufactured from a combination of epoxy-glass and polyimide materials are bonded to a heat sink assembly

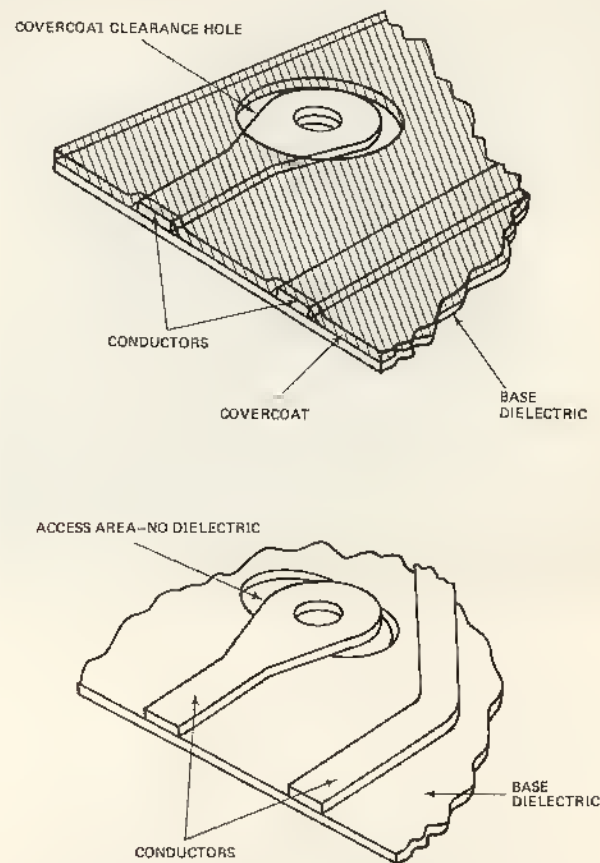


Fig. 24.4 Class 1c double-access uncovered flexible printed wiring.

which will subsequently interconnect the third-level back panel of the computer with the input-output connectors in the rack box. Multilayer flexible printed wiring is an end product which consists of three or more conductive layers on flexible dielectric bases. The bases are bonded to form a monolithic or solid mass, which may have an insulator applied over the outer conductors.

Multilayer flexible printed wiring is further subdivided into the following types:

Class III, type I. Multilayer wiring constructed on flexible dielectric base materials in which the end product is intended to be flexed.

Class III, type II. Multilayer wiring constructed on flexible dielectric base materials in which the end product is not intended to be flexed.

Class III, type III. Multilayer wiring manufactured from flexible dielectrics in which the end product must be formable, but not flexed continuously.

Class III, type IV. Hybrid printed wiring products manufactured from a combination of rigid and flexible materials.

The applications for multilayer flexible printed wiring has been primarily in military rack boxes, but it is believed that the packaging concept will be employed in commercial

applications in which dielectric performance and high packaging density are required. Some theoretical, actual, and potential MFPW applications are discussed in the following paragraphs.

5. Class III, Type I. Flexible Base Material; End Product Flexed An MFPW device which must have transmission line properties and be flexed in its intended usage may be constructed as shown in Fig. 24.8. It consists of a number of single microstrip FPW's bounded together at the ends but left unbanded in the center section to provide a high degree of flexibility.

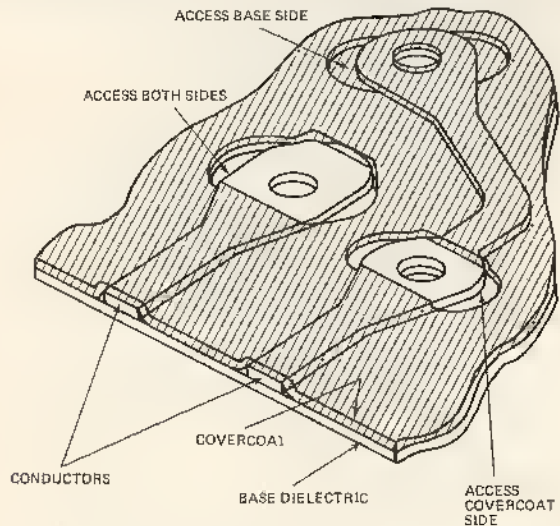


Fig. 24.5 Class Id double-access covered flexible printed wiring.

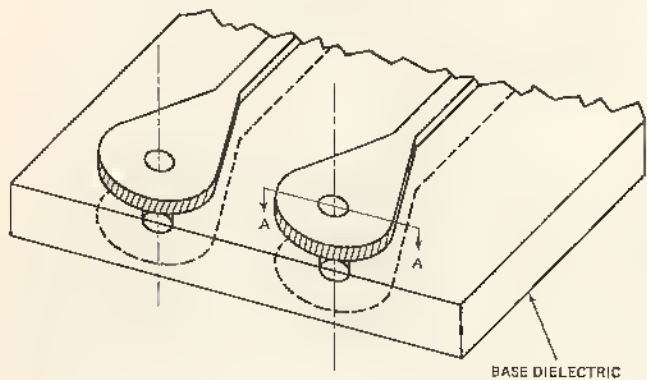


Fig. 24.6 Class II flexible printed wiring.

To maintain the desired electrical characteristics such as matching the characteristic impedance (Z_0) to the rigid boards it interconnects, the MFPW device is designed to have a signal over a ground plane in each of the individual wiring layers. To provide a high

degree of flexibility, the wiring layers could be coated with a thin, suitable coating such as polyimide in lieu of a thicker laminated cover coat. Plated-through holes could provide the necessary Z-plane interconnections between the flexible wiring layers. This MFPW device is best used where flexibility, increased reliability, and high density are essential design requirements.



Fig. 24.7 Multilayer flexible printed circuit assembly (IBM).

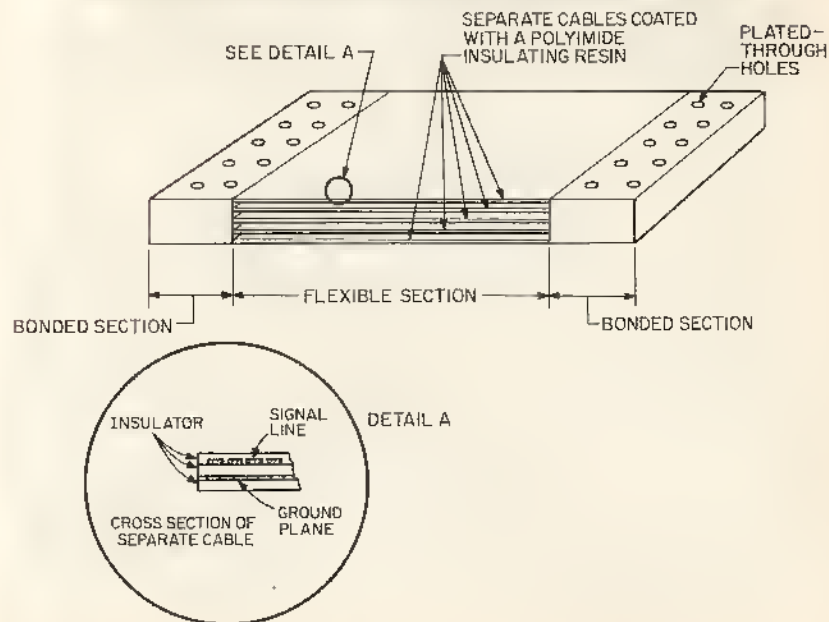


Fig. 24.8 Multilayer flexible printed wiring, class III, type I.

6. Class III, Type II. Flexible Base Material; End Product Not Flexed This class of MFPW consists of multilayer boards that are fabricated from flexible dielectrical materials such as polyimide film but lose their inherent flexibility when laminated. This type of board is employed when the design calls for maximum utilization of film dielectric properties such as low dielectric constant, uniform dielectric, lower weight, and continuous process capability. For example, multilayer boards fabricated from a polyimide film dielectric weigh approximately one-third less than a comparable epoxy-glass board.

7. Class III, Type III. Flexible Base Material; End Product Formable This type of MFPW is manufactured from flexible dielectric materials, but because of electrical design constraints such as a conductor resistance requirement, which dictated the use of a heavy-weight conductor material, and impedance or capacitance requirements, which called for a thick dielectric separation between signals and ground, the end product, although manufactured from flexible materials, is formable in its end application. The term "formable" is defined as the capability of the MFPW device to be shaped into a desired contour and not further flexed in its application.

Class III, type III, MFPW was utilized in an avionics application for intrabox wiring. The requirement was for strip-line or tri-plate design with low conductor resistance, an extremely low capacitive coupling or circuit noise, and the capability of being formed in a smooth-flowing 90° bend at the interconnection end. That wiring task was accomplished by manufacturing the MFPW from polyimide film material because of its high temperature resistance, flexibility, and good overall electrical and mechanical properties. A cross

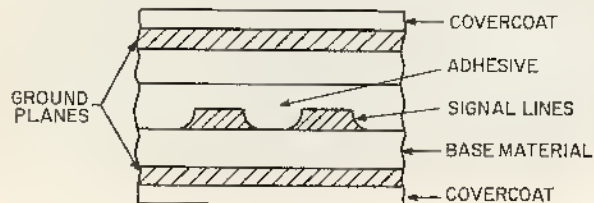


Fig. 24.9 Cross section of class III, type III, multilayer flexible printed wiring.

section of the wiring is shown in Fig. 24.9. To accomplish all the interconnections in this section of the box, the wiring was subdivided among multiple multilayer flexible wiring devices and taped together to form a printed wiring harness. The harness is shown in Fig. 24.10.

8. Class III, Type IV. Flexible and Rigid-Base Materials This type of wiring is usually in the form of a rigid board which contains integral flexible printed wiring layers. The layers may be laminated within the confines of a rigid multilayer board for specific electrical requirements or extend about the periphery of the board to provide Z-plane wiring capability. This type of wiring is gaining increased use in applications in which ultrahigh packaging density, good electrical properties, high reliability, and tight volumetric constraints are essential.

A series of hybrid MFPW devices were designed for use in a flying military electronics application in which weight and volume were of primary importance. To comply with the weight and volume constraints, the internal packaging had to be extremely dense. In addition to the high circuit density, all signal transmission lines had to be shielded to provide a minimum of cross talk or noise. Shielded, discrete wires could not be physically or economically packaged in the system. Thus, to provide the interconnections, hybrid MFPW was employed. The devices incorporated shielded signal lines in flat, strip-line flexible printed wiring as an integral part of the rigid printed board. A schematic representation of this class III, type IV device is shown in Fig. 24.11a, and the actual board is shown in b. At a higher level of operation, after complete fabrication, a 90° S bend was formed in the board to provide the Z-plane interconnection path and provide a stress-strain relief to the solder joints during X-, Y-, and Z-plane vibrational stresses. A formed board is shown in Fig. 24.12.

INTERFACIAL CONNECTION

Interfacial connections are most commonly made by plated-through-hole techniques. The approach lends itself to mass production and can assure a solderable surface by the use of a tin-lead overplate, electroless tin, or a lacquer or flux coating. Other systems such as eyelets and brazed tubelets also are employed for interfacial connections. The proper choice is dependent upon the intended application, operating environment, and economic constraints.

In applications in which maximum joint reliability is required, an interfacial connection (class IIc or d) via a plated-through hole or tubelet-eyelet is recommended over a noninterfacial connection joint.

The schematic in Fig. 24.13 depicts the differences in terminating class IIb and class IIc designs. It is readily apparent that the plated-through-hole termination will provide a stronger and superior interconnection joint.

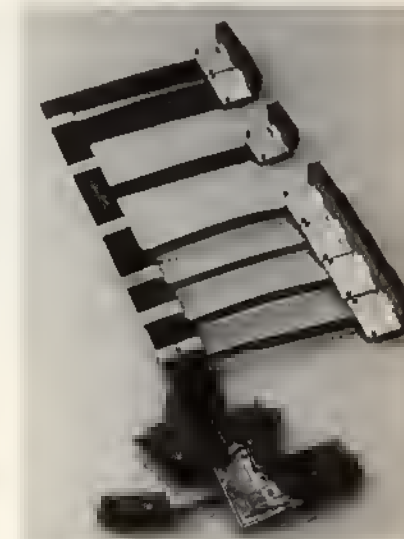


Fig. 24.10 Multilayer flexible printed wiring harness.

DESIGN

9. Interconnection Assessment In assessing a potential application for the use of FPW, the designer must analyze the interconnection task in terms of the following:

a. Applicability and Feasibility. The interconnection must be analyzed by asking these questions: Is FPW feasible? Will there still be too many discrete wires which cannot be converted after FPW is designed into the system? The questions must be answered before proceeding. If feasibility is established, the next question must be addressed.

b. Cost. Assuming an appropriate FPW design, what is the cost differential of the approach as compared with discrete wiring? In this analysis the total installed cost of the interconnection must be considered, not simply the cost of FPW vs. discrete wiring. Another valuable consideration is the number of units to be built. Generally, its high nonrecurring start-up costs (artwork, drill programs, drill tapes, tooling, etc.) will eliminate FPW from small production runs, but it will show a pronounced cost savings in large production runs owing to their overall savings in assembly.

c. Function. Assuming that feasibility and cost considerations favor the use of FPW, the designer should evaluate the circuit's purpose and whether that purpose could be as well served by other wiring techniques at a lower cost. For example, the use of many individual layers of FPW could be traded off against the use of one multilayer printed board. If after a thorough analysis in which the above parameters are addressed and the interconnection system favors the use of FPW, the following design considerations should be employed.

10. Layout Development By its basic definition, FPW is generally custom-designed for the application. Thus the first item that should be generated is the wire list and the physical layout.

a. Wire List. Anyone can generate a wire list in which random points are interconnected in no orderly manner. That type of layout could suffice for discrete wiring, but it could kill a potential FPW design. In the design of FPW, the designer must keep circuit crossover points to a minimum to reduce the number of circuit layers. Interconnection costs are directly proportional to the number of conductive planes required. Reduce the number of layers at points of greatest wiring concentration, as at input or output connec-

tors. First, find, identify, and list all common terminals. Then arrange the layers to minimize crossovers at the area of highest density. Continue crossover reductions to less dense conductor areas. Various colored pencils can be utilized to interconnect all common wiring paths. Once the connections are made with the colored lines, multiple contacts per conductor are evident, and the number of jumper wires is established.

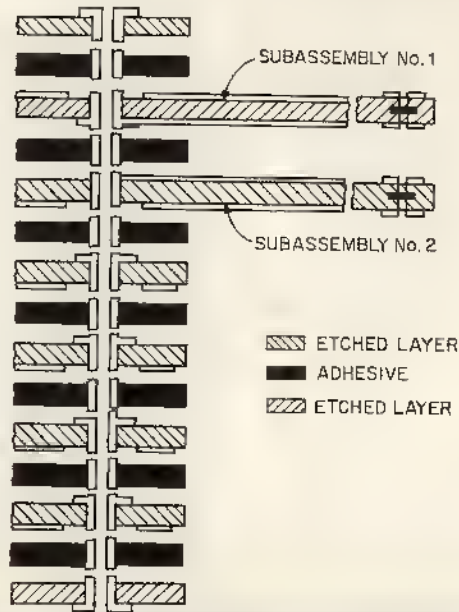


Fig. 24.11 (a) Schematic representation of a hybrid multilayer flexible printed wiring device. (b) Multilayer flexible printed wiring, class III, type IV.

The following exercise shows how the number of layers can be reduced in the design of a typical circuit: In Fig. 24.14, a row-by-row analysis of the 12 wires feeding a connector in four layers of FPW shows that no conductors in row A—one layer of circuitry required—

cross other conductors in that row, that two conductors from row B—two layers of circuitry required—cross two other conductors from that row, and that no conductors in row C—one layer of circuitry required—cross other conductors in that row.

Fig. 24.14 shows how 12 wires coming from various points on an assembly feed to a connector through four layers of flexible circuitry. Figure 24.15 shows one solution to the



Fig. 24.12 Z-plane interconnection form provided in hybrid multilayer flexible printed board.

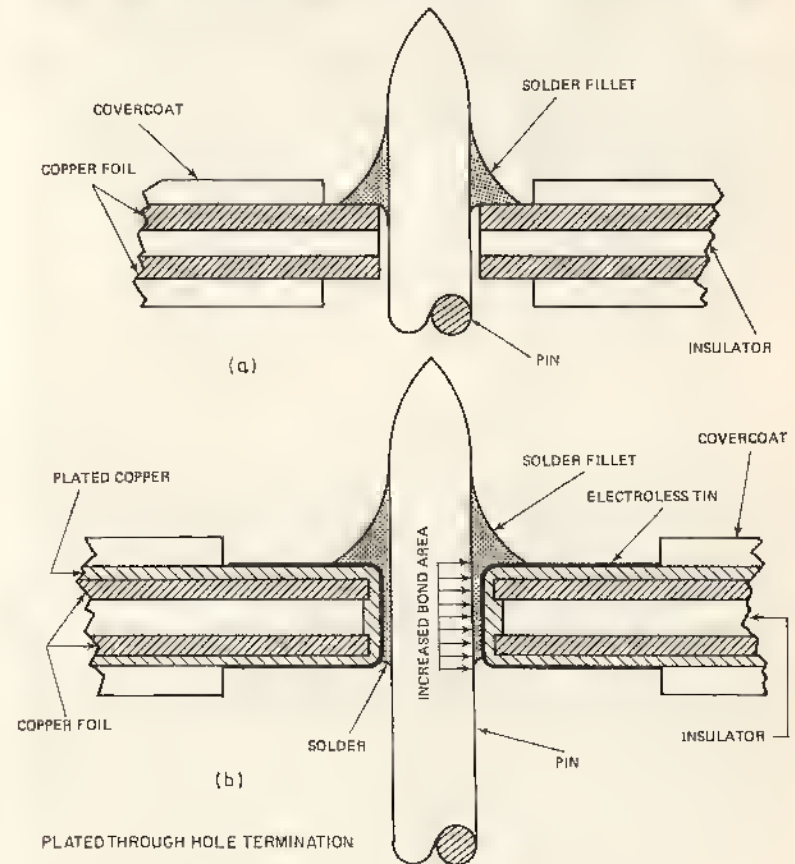


Fig. 24.13 Noninterfacial vs. interfacial joint termination.

crossover problem: three layers instead of four at the cost of increased circuit size. Figure 24.16 shows a better way to get by with three layers of circuitry as compact as possible. Figure 24.17 shows the best circuit layout in a single compact layer. Thus the basic principle of fanning out from the areas of greatest conductor density to areas of lesser density provides the best arrangement mechanically and at lowest cost.

Shielding may affect mechanical flexibility of the circuit, for as the shielding increases, flexibility usually decreases. For applications that require repeated flexing of single- or double-shielded wiring you must determine where the flexing will occur and the number of bendings required during the operating lifetime. In complex layouts in which compound curves and physical interruptions are encountered, as in a high-density electronic structure, a paper mock-up of the flexible wiring should be made. The mock-up should be used to develop the shape and size of the flexible wiring. This prefitting of the circuitry

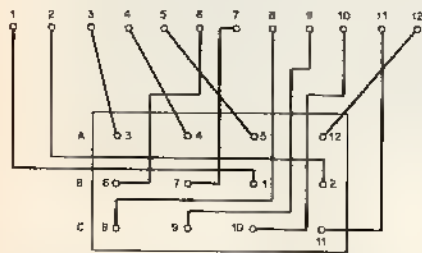


Fig. 24.14 A row-by-row analysis of the 12 wires feeding a connector in 4 layers of flexible printed circuitry shows that no conductors in row A—one layer of circuitry required—cross other conductors in that row, that two conductors from row B—two layers of circuitry required—cross two other conductors from that row, and that no conductors in row C—one layer of circuitry required—cross other conductors in that row.

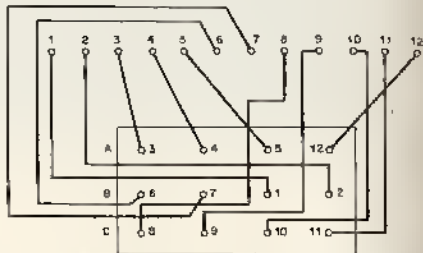


Fig. 24.15 Solve the crossover problem in row B shown in Fig. 24.14 by rerouting the conductors from connector pins 6 and 7 to swing around the conductors from pins 1 and 2. Interconnection now requires only three layers of circuitry, but the circuit size for the conductor must be increased to swing around the end.

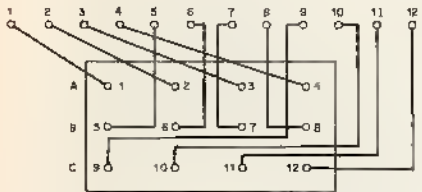


Fig. 24.16 Rearranging the pin layout of the assembly not only removes the crossovers in row B to eliminate one layer of circuitry to a total of three but makes each layer most compact.

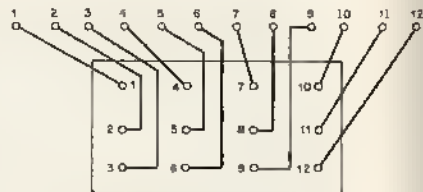


Fig. 24.17 All connections in this final design occur in one compact layer.

can avoid possible mistakes in the original design. An example of a paper mock-up is shown in Fig. 24.18.

b. Pad Design. The pad diameter or shape should always be maximized without violating the minimum spacing requirements. If possible, additional bonding area in the form of ears should be incorporated in the pad design, but potential shorts should be avoided. Flexible wiring pads should always be generously filleted or tear-dropped to minimize stress concentrations at the pad-line interface. A variety of pad shapes are shown in Fig. 24.19. When applicable, use the following minimum limits:

1. When a pad is at least $3\times$ hole size, specify a minimum of 0.005 in of metal around the hole.

2. When a pad is $2\times$ hole size, metal must be visible all around the hole and a 0.005 in area must be at least 270° around the hole.

3. Tie the pads down with ears, but avoid creating shorts.
4. Fillet or tear-drop pad design and fillet all sharp heads to minimize stress concentrations.
5. Specify all hole sizes and hole-to-hole location to ± 0.005 in tolerance.

11. Processing Parameters and Tolerances The data in Tables 24.1 to 24.3 are presented as a guide to the designer. The values may vary slightly from shop to shop, and should be verified.

TABLE 24.1 Line Widths and Tolerance as a Function of Copper Foil Thickness

Copper foil thickness, oz/ft ²	Line width minimum, in	Line and pad tolerance, in
1	0.010	± 0.003
2	0.015	± 0.005
3	0.020	± 0.006
4	0.030	± 0.008

TABLE 24.2 Conductor Spacing as a Function of Copper Foil Thickness

Copper foil thickness	Minimum spacing between conductors (in)
1	0.015
2	0.020
3	0.030
4	0.040

TABLE 24.3 Minimum Edge Distance as a Function of Copper Foil Thickness

Copper foil thickness oz/ft ²	Minimum edge distance in
1	0.025
2	0.040
3	0.050
4	0.060

12. Access Windows Access windows should be designed to overlap the pads as much as possible to provide additional holddown for the pads, provide strength for the lines, and present additional repair cycles. If possible, design the FPW to have all access windows on the cover-coat side. The wiring becomes more expensive if access is required from the base material side of the flex wiring device. However, if access is required on the base material side, it can be accomplished by mechanical or chemical milling (bare padding) or by a folding technique which reverses the cover coat clearance side as shown in Fig. 24.20.

13. Conductors All design charts and calculations hold only for single-conductor cases. The values thus derived must be derated if more than one conductor in a FPW will be carrying significant current and contributing to the heat load. A general rule of thumb is that single-conductor current capabilities should be derated by 20 percent if two conductors will be carrying equal current and by 50 percent if fifteen conductors carry equal current. Figures 24.21 to 24.25 further define the current capabilities of flexible printed wiring when such factors as number of conductors, conductor spacing, and heat sinking are considered. Data for the design charts of Figs. 24.21 to 24.25 were obtained from measurements on a standard FPW device with the following parameters, only one of which was varied for each series of tests:

Conductor material	Roller copper
Conductor width	0.060 in
Conductor thickness	0.0027 in (2 oz)
Conductor spacing	0.060 in
Number of conductors	10
Insulation thickness	0.005-in base and 0.005-in covercoat

All measurements were made with a thermocouple bridge using 0.010-in thermocouple wire to reduce heat transfer down the leads to a minimum. Equilibrium was maintained before measurement. Location of the thermocouples, unless otherwise specified, was at the assumed hot spot (middle of cable span and on surface of insulation directly over current-carrying conductors). All measurements were made at approximately 25°C from ambient temperature unless otherwise specified without previous conditioning of the samples other than standard processing procedures.

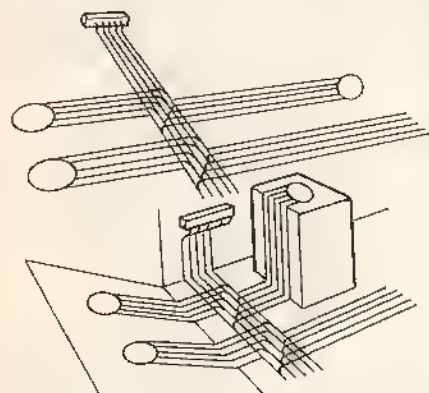


Fig. 24.18 Paper mock-up in the flat, fitted to the product or prototype, helps avoid alternations that may be necessary when original flexible printed wiring design fails to allow for obstructions.

in the manufacture and handling of flexible printed wiring. The test is conducted by performing IR measurements between conductors on a layer and/or between the layers in

the assumed hot spot (middle of cable span and on surface of insulation directly over current-carrying conductors). All measurements were made at approximately 25°C from ambient temperature unless otherwise specified without previous conditioning of the samples other than standard processing procedures.

ELECTRICAL DESIGN OF FLEXIBLE PRINTED WIRING

The electrical characteristics of flexible printed wiring are controlled by the configuration of the conductors, ground plane(s), if any, and the dielectric properties of the base and cover coat materials.

a. *Insulation Resistance.* Insulation resistance (IR) is one of the most critical parameters from both a design and materials standpoint. IR tests are utilized in determining the raw materials properties, as well as controlling quality of the end product, since they are indicators of moisture and other contaminants commonly found

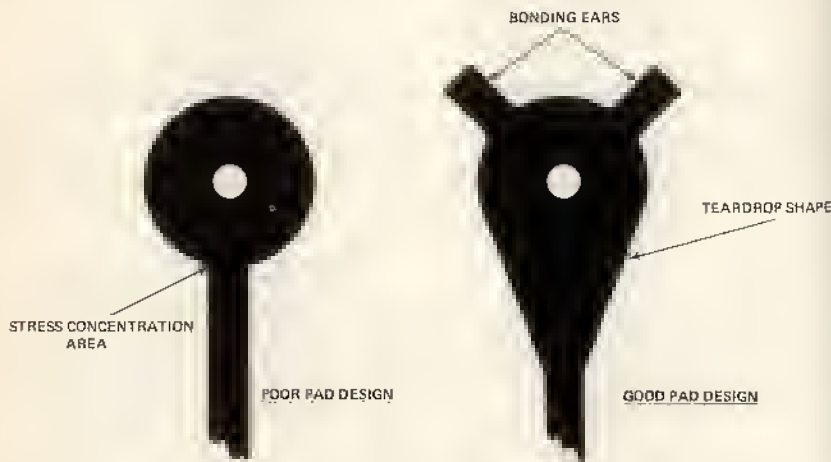


Fig. 24.19 Flexible printed wiring pad design.

a double-sided or multilayer board after a specified time in a humidity and temperature cycle. A voltage of 100 V ac is applied during the test, and measurements are made immediately thereafter.

Cover-coated flexible wiring retain their high-IR values even under severe humidity conditions. Wiring that is not cover coated usually retains high-IR values provided the surfaces are clean and free from flux, fingerprints, etc. The insulation resistance of a 30-Ω, polyimide, FPW device of microstrip design and cover coated with a 0.001-in polyimide film-phenolic modified polyvinyl butyral adhesive is shown in Table 24.4. The environmental test cycle is shown in Fig. 24.26.

TABLE 24.4. Insulation Resistance of 30-Ω Microstrip Flexible Printed Wiring

(Average Readings)		
FPW device No.	Initial, Ω × 10 ¹¹	After humidity, Ω × 10 ⁹
	Line to line	
1	5	7.1
2	5	3.9
3	5	3.6
4	5	4.8
	Line to shield	
1	5	11.6
2	5	3.2
3	5	9.0
4	5	8.2

NOTE 1: All after-humidity readings were taken within 0 to 2 h after removal from humidity.
NOTE 2: Above readings are average for 45 measurements per FPW.

b. *Dielectric Withstanding Voltage.* This property of the wiring is dependent upon the air density and conductor spacing. The important factor for the designer to consider is the potential at which flashover occurs in the air above the conductors. The results in Fig. 24.27 will serve as a design guide. The dc breakdown voltage variation, as a function of conductor spacing for cover-coated flexible printed wiring, is shown in Fig. 24.28. The test pattern used for this series of tests is shown in Fig. 24.29. As indicated, increasing altitude lowers the insulating properties of air, owing to ionization, and thus greater conductor spacing is required.

c. *Determination of Conductor Size.* The size of a conductor for a given printed wiring application is based upon the following parameters:

- Current-carrying capacity
- Conductor resistance
- Maximum allowable temperature rise in conductor
- Derating effect of multiple conductors in same wiring
- Effects of mounting on current-carrying capacity
- Fusing current
- Capacitance

d. *Capacitance.* Capacitance between adjacent conductors consists of both parallel-plate capacitance C_p and fringing capacitance C_f , as shown in Fig. 24.30. Parallel-plate capacitance is generally negligible, since the conductors are very thin and thus have small areas opposing each other. Fringing capacitance is therefore the principal form of distributed capacitance between adjacent conductors in flexible printed wiring. Since some portion of the fringing field (usually about half) extends into the air outside the thin insulation, calculation of exact capacitance depends on thickness and dielectric constant of the insulation, dielectric constant of the surrounding medium, widths and thicknesses of the conductors, spacing of the conductors, presence of nearby grounds, and frequency (dielectric constants change with frequency).

Actual measurements are required to obtain accurate capacitance values. However, distributed capacitance can be roughly examined by establishing a dielectric constant of "median value" which falls between that of the insulating material and that of air. The capacitance values calculated by using that approximation serve as guidelines to the distributed capacitance which will actually exist in a given circuit configuration. The

results of such a calculation are shown in Fig. 24.31. The capacitance per foot at 1 kHz between two conductors in flexible printed wiring insulated with 0.005 in of polyester is shown. To determine capacitance values for similar flexible wiring with other insulation materials, multiply the values found from Fig. 24.31 by the proper dielectric constant ratio

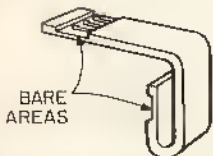
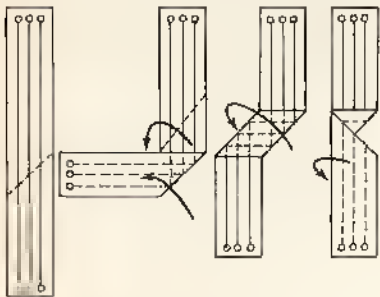


Fig. 24.20 Reverse-fold termination process to eliminate bare padding.

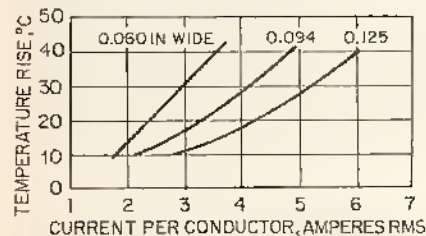


Fig. 24.21 Temperature rise in 10-conductor FPW for various currents and conductor widths at 20°C ambient temperature.

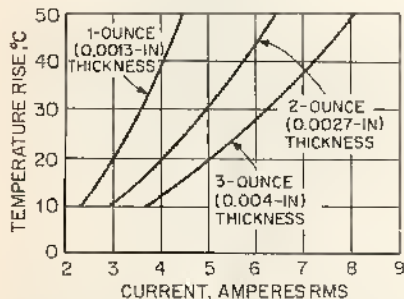


Fig. 24.22 Temperature rise in a 10-conductor FPW with only one conductor carrying current for various currents and conductor thicknesses at 20° ambient temperature.

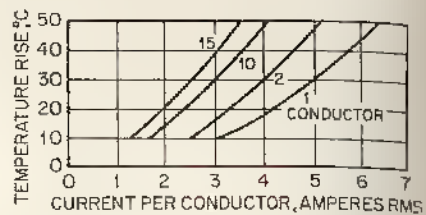


Fig. 24.23 Temperature rise in an FPW for various currents and numbers of conductors at 20°C ambient temperature.

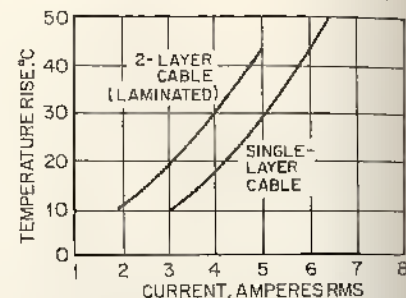


Fig. 24.24 Temperature rise in a multilayer circuit compared with rise in a single layer at 20°C ambient temperature. Only one conductor per cable carries current.

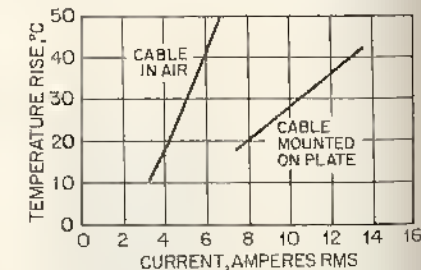


Fig. 24.25 Temperature rise in a single-conductor heat-sinked cable compared with rise in air at 20°C ambient temperature. Only one conductor carries current.

TABLE 24.5 Ratio of Dielectric Constants

Polyester	1.0
FEP Teflon	0.90
Polyethylene	0.93
Vinyl (PVC)	1.4
Polyimide (Kapton)	1.21

found from Table 24.5. The values shown in Fig. 24.31 assume a conductor pair free from the influence of other surrounding conductors or ground planes.

In practice, the isolation shown in Fig. 24.32a is seldom encountered, and measurements made between two conductors "guarded" by two other grounded conductors, in *b*, show capacitance values reduced by approximately 20 percent from those found by using *a*. The reduction occurs because some of the fringing flux lines normally joining the measured conductors now terminate on the surrounding grounded conductors and therefore do not contribute to the measured value. Capacitance between two conductors was even more drastically reduced (about 85 percent) when a grounded conductor was placed

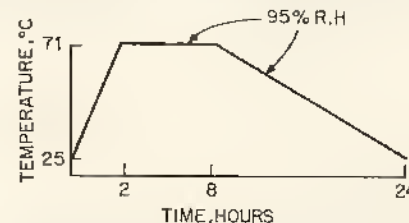


Fig. 24.26 Humidity environment test cycle: 25° to 71°C in 2 h; 71°C at 95 percent relative humidity for 6 h; 71° to 25°C at 95 percent relative humidity for 16 h; 24 h per cycle.

between the conductors and grounded conductors also were placed on either side (Fig. 24.32c). If the guard conductors of *b* and *c* are floating rather than grounded, capacitance between the measured pair will increase slightly. That situation is seldom encountered in practice, since conductors other than spares usually are not floating.

For some applications it is desirable to find the capacitance between two flat conductors which are completely enclosed by the insulation and have their wide surfaces facing each other. Figure 24.33 illustrates the geometry of the problem. Capacitance in that case can be found using one of the two following relations:

$$\text{For } \frac{w}{s} < 1 \quad C_{total} = \frac{3.68 E_r}{10_{10} \left[\frac{45}{w} \right]} \text{ pF/ft}$$

where *W* = width, ft
s = dielectric separation between conductors, ft
C = capacitance, pF/ft
 ϵ_r = effective dielectric constant

In this case, the result will be approximately 2 percent high for $\frac{w}{s} < \frac{1}{2}$ and approximately

10 percent high for $\frac{w}{s} = 1$

$$\text{For } \frac{w}{s} > 1 \quad C_{total} = 2.7 \frac{w}{s} \epsilon_r \times \left[1 + \frac{s}{\pi w} \left(1 + 2.303 \log_{10} \frac{2\pi w}{s} \right) \right] \text{ pF/ft}$$

In this case, the result will be approximately 4 percent low for $\frac{w}{s} = 2$ and approximately

10 percent low for $\frac{w}{s} = 1$.

$$\text{For } \frac{w}{s} > 1 \quad C_{total} = 2.7 \frac{w}{s} E_r \times \left[1 + \frac{s}{\pi w} \left(1 + 2.303 \log_{10} \frac{2\pi w}{s} \right) \right] \text{ pF/ft}$$

In this case, the result will be approximately 4 percent low for $\frac{w}{s} = 2$ and approximately

10 percent low for $\frac{w}{s} = 1$.

Figure 24.34 is an approximate guideline to the capacitance between two flat conductors which are exposed to air on either side of a dielectric layer. Capacitance of flat conductors to surrounding metallic shields and/or ground planes is shown in Fig. 24.35 for standard flexible printed wiring using FEP Teflon* insulation.

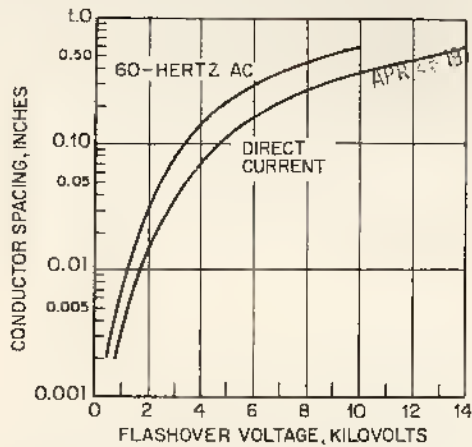


Fig. 24.27 Flashover voltage in air (Sanders Associates)

DESIGN OF FLEXIBLE PRINTED WIRING WITH TRANSMISSION LINE PROPERTIES

Flexible printed wiring has found widespread use in transmission line applications either as a replacement for coaxial wires or in initial designs of high-speed circuits. With the advent of LSI devices the use of printed wiring with transmission line properties has increased many-fold. This section will provide the designer with sufficient data to effectively design flexible printed wiring with transmission line properties.

The Institute of Printed Circuits defines and explains the term "transmission line" as follows: "Transmission line" is one signal-carrying circuit composed of conductors and dielectric material with controlled electrical characteristics used for the transmission of high-frequency or narrow-pulse type signals."

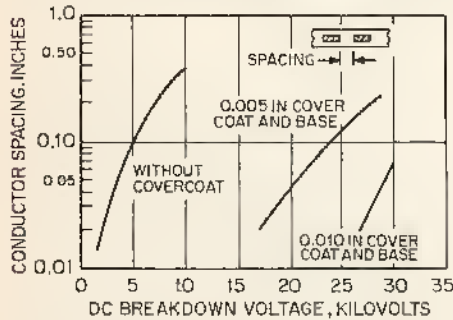


Fig. 24.28 Breakdown voltage between insulated conductors (Sanders Associates).

Two types of such transmission line configurations are utilized: "microstrip," in which the conductor is located above a single ground plane (Fig. 24.36a), and "strip-line," in which the conductor is located centrally between two ground planes (Fig. 24.36b). Important parameters of such conductors are line width, dielectric separation to ground plane, and dielectric constant of the insulating material, all of which determine the impedance, signal propagation, and cross-talk characteristics of the lines. A variety of manufacturing techniques can be used to achieve the microstrip or strip-line configuration. The most

*Trade name of E. I. du Pont de Nemours & Company.



Fig. 24.29 High-voltage test pattern (Sanders Associates).

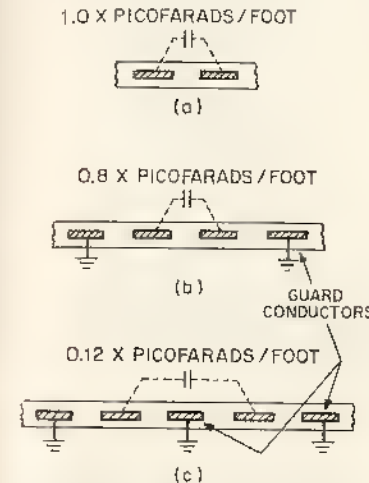


Fig. 24.32 Typical conductor shielding effect resulting from "guarding" by adjacent grounded conductors: (a) No guard conductors, (b) peripheral guard conductors, (c) interspersed and peripheral guard conductors.

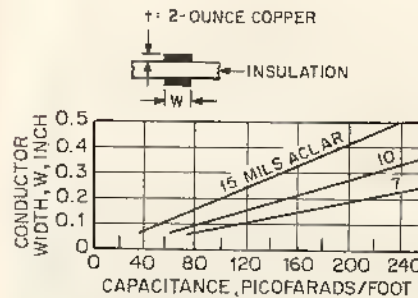


Fig. 24.34 Typical capacitance at 1 MHz between two flat conductors exposed to air on either side of a dielectric layer.

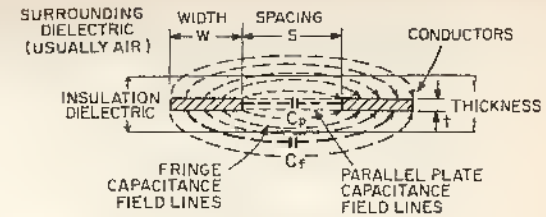


Fig. 24.30 Distributed capacitance between conductors is fringing C_f capacitances.

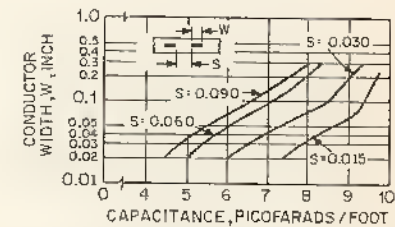


Fig. 24.31 Calculated capacitance between two adjacent parallel conductors insulated with 0.005 in. of Kel-F or polyester (heat-sealable); calculated for 2-oz conductors at 1 kHz.

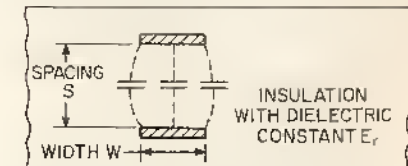


Fig. 24.33 Capacitance between two conductors with large surfaces opposing is largely parallel-plate capacitance.

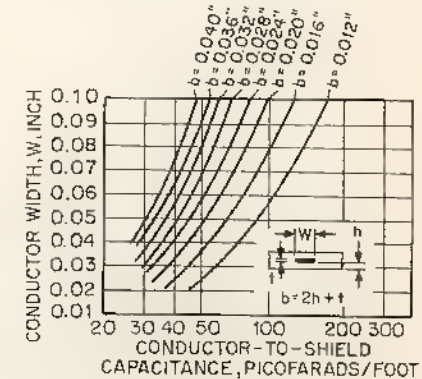


Fig. 24.35 Typical conductor-to-shield capacitance for 2-oz copper conductors insulated with FEP Teflon.

widely used method is multilayer printed boards, although microstrip is also used in double-sided flexible printed wiring. The following parameters govern the design of flexible printed wiring transmission line devices.

14. Characteristic Impedance. The single and most important electrical parameter in determining the performance of high-speed circuits is the characteristic impedance of the conductors. If Z_0 is controlled, signal reflections which result from the passage of fast

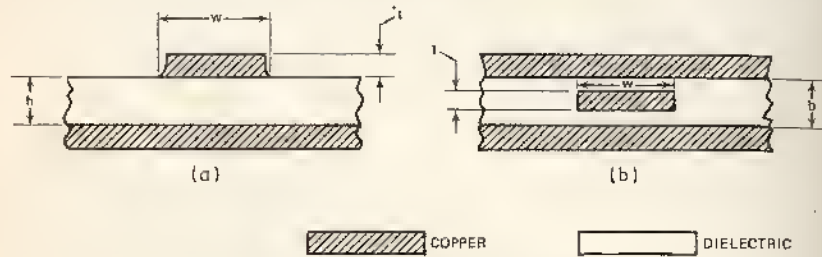


Fig. 24.36 Printed wiring transmission line configurations.

pulses through an impedance discontinuity or mismatch at the load end of the line will be avoided. The characteristic impedance is dependent upon the conductor line width w , dielectric thickness h or b , conductor thickness, dielectric constant of the insulating medium, and the configuration (microstrip or strip-line) of the device. The general equation for characteristic impedance of any high-frequency transmission line is as follows:

$$Z_0 = \sqrt{\frac{R + j\omega L}{G + j\omega C}}$$

- where Z_0 = impedance, Ω
 R = resistance per unit length of line
 L = inductance per unit length of line
 G = conductance per unit length of line
 $\omega = 2\pi f$
 C = capacitance per unit length of line

If the line is lossless, the equation is simplified:

$$Z_0 = \sqrt{\frac{L}{C}}$$

15. Microstrip Configuration A specific formula for microstrip is the following:

$$Z_0 = \frac{h}{w} \frac{377}{\epsilon_r}$$

where h and w are given in the diagram and ϵ_r is the effective dielectric constant of the material (considering the effect of air). This analytical method disregards fringing effects and leakage flux and gives validity to the notion that more reliable values may be obtained by measurement. To account for the fringing effects of microstrip transmission line, and if the analytical method must be used, the following formula is recommended:

$$Z_0 = \frac{h}{w} \frac{377}{\sqrt{\epsilon_r} [1 + (2h/\pi w)(1 + \ln \pi w/h)]}$$

If the microstrip line is embedded or covercoated as shown in Fig. 24.37, the Z_0 values will be reduced by approximately 22 percent. Typical values of Z_0 for microstrip fabricated of an epoxy-glass base material having a dielectric constant varying from 6 at 1 kHz to 5 at 25 MHz when conductors are exposed to air, are given in Fig. 24.38.

16. Strip-Lines The characteristic impedance Z_0 of strip transmission line when $w/b \gg 0.35$ can be determined by the following relation:

$$Z_0 = (60/\sqrt{\epsilon_r}) [\ln(4b/d_0)]$$

where d_0 , the effective wire diameter for square configuration, is $0.567w + 0.67t$.

Representative values of Z_0 for microstrip and strip-line transmission line wiring are presented in Fig. 24.39. The use of curves to determine strip-line parameters is preferred over the more cumbersome methods of calculation. A family of curves that has proved to be very useful for that purpose is shown in Fig. 24.40. It is applicable to a wide range of copper thicknesses.

If one were to fix the parameter t at 0.0028 in (2-oz copper), the family of curves could be redrawn to show the relations of w and b for various values of normalized characteristic impedance $\epsilon_r Z_0$. That family of curves is shown in Figure 24.41, and it is usable for any insulating material whose dielectric constant is known or measurable.

a. Specific Example. The following exercise is presented to familiarize the designer with the use of the curves in the solution of a typical design problem given these conditions:

Flexible printed wiring, cover coated

$Z_0 = 75 \Omega$

Overall thickness = 0.062 in

Material = Teflon, $\epsilon_r = 2.1$

1. Determine the proper curve in the family. Since the material has been designated as Teflon with an ϵ_r of 2.1, the suggested curve would be $\epsilon_r Z_0 = 110$.

2. Determine dielectric separation b . Knowing that the overall thickness should not exceed 0.062 in, the value of b should be set at 0.050 in. The number is based upon the difference between the overall thickness and the expected thickness of two ground shields and two cover-coat layers.

3. Determine conductor width w . The value of w as read from the graph is 0.017 in.

The construction of the strip-line in this hypothetical case is shown in Fig. 24.42 (p.24-28). The spacing between conductors is determined by FPW width, electrical parameters, and processing limitations.

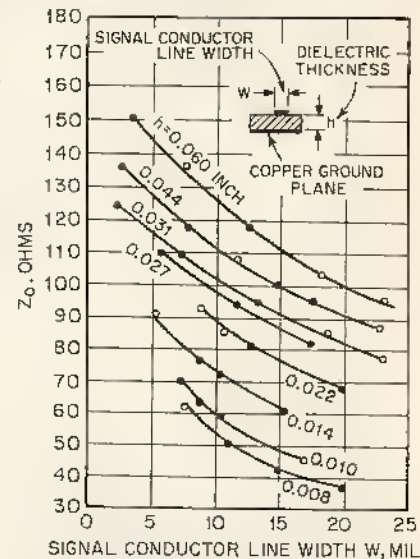


Fig. 24.38 Characteristic impedance (Z_0) of microstrip wiring as a function of line width w and dielectric thickness h . Conditions: (1) 2-oz copper conductors; (2) measurement made at 26°C (room temperature) and room humidity; (3) board material dielectric constant relatively stable with frequency varying from about 6 at 1 kHz to about 5 at 25 MHz.

ELECTRICAL PARAMETERS

17. Capacitance C Capacitance for double-sided wiring is determined by the formula

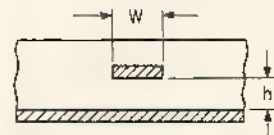


Fig. 24.37 Schematic representation of embedded or covercoated microstrip FPW.

$$C = \frac{0.2249}{d} \epsilon_r A$$

where C = capacitance, pF/in
 ϵ_r = dielectric constant relative to air = 1
 A = area of smallest electrode, in²
 d = thickness of dielectric, in

Under normal conditions, lines that are parallel and are separated by 0.062 in on 0.062-in-thick epoxy-glass ($\epsilon_r \approx 5$) will have a capacitance of 0.5 pF/in. Corresponding wiring fabricated from Teflon* will have a C of 0.3 pF/in.

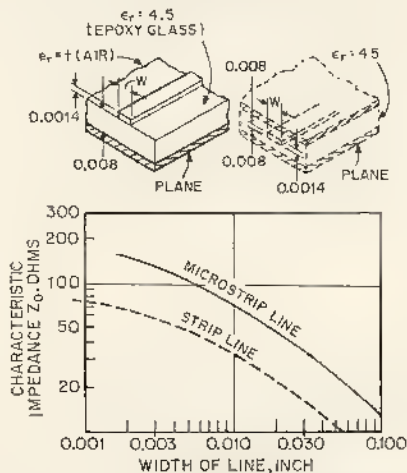


Fig. 24.39 Characteristic impedance Z_0 comparison of microstrip and strip-line transmission line wiring devices.

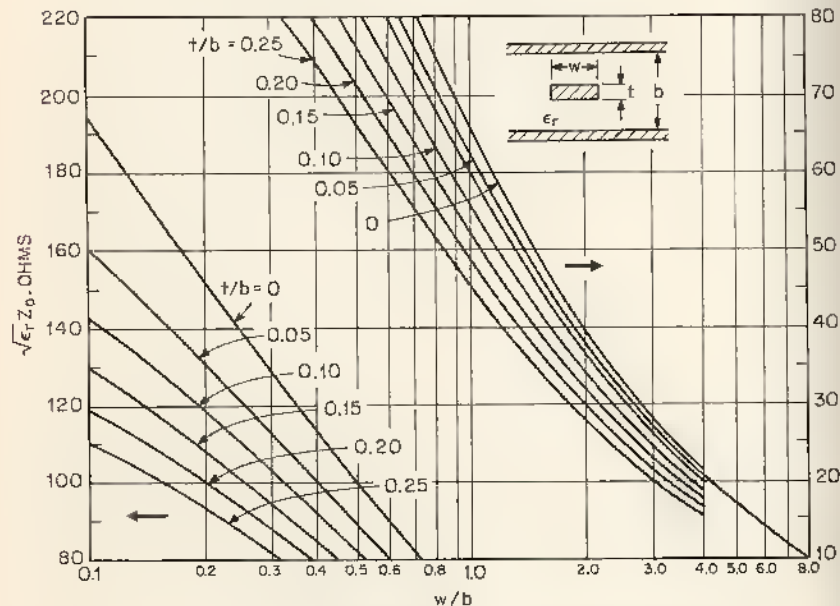


Fig. 24.40 Graph of Z_0 versus w/b for various values of t/b .

*Trade name of E. I. du Pont de Nemours & Company.

In multilayer boards, capacitance is calculated by a different formula, although the conductors are still capacitively coupled when they physically approach each other in the horizontal or vertical direction. An approximate value for the amount of capacitive coupling may be calculated by the equation

$$C = \frac{w\epsilon_r}{h4.45}$$

where ϵ_r = effective dielectric constant
 h = height, in $\times 10^{-3}$
 w = width, in $\times 10^{-3}$

for the configuration shown in Fig. 24.43.

Microstrip may be used as a single wiring layer or the outside layer of a multilayer

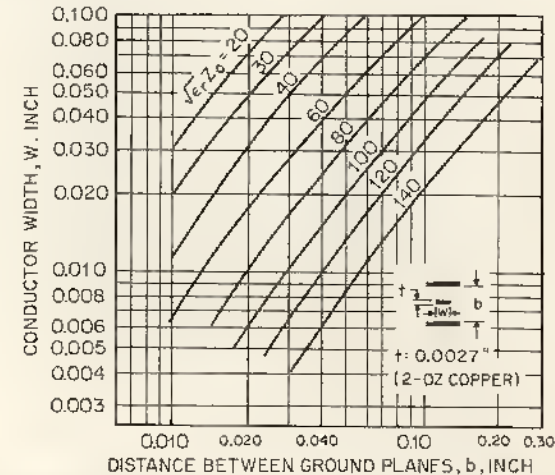


Fig. 24.41 Normalized impedance $\epsilon_r Z_0$ as a function of conductor width w and dielectric separation b in strip-line configuration.

board. Strip transmission line may be used either as one or a multitude of wiring layers. Capacitance of strip transmission line can be approximated by the equation

$$C = 0.9 \frac{w/b}{1 - (t/b)} \epsilon_r$$

18. Time Delay The delay in the signal propagation velocity in microstrip line can be calculated using the formula:

$$T_d = 1.015\sqrt{\epsilon_r} \quad \text{ns/ft}$$

where T_d = time delay per unit length of conductor
 ϵ_r = effective dielectric constant of the transmission line or the dielectric constant of the board's dielectric if the conduction is embedded

The equation is valid only when the conductor's capacitance is greater than the load capacitance that results when switching circuits are added to the line. Embedding microstrip lines, as in multilayer printed wiring, makes the circuit appear "electrically larger." That is, a signal takes longer to travel a given distance in the embedded line than in the unembedded or pure microstrip line. Therefore, embedding the lines means that critical wiring lengths must be reduced 22 percent to maintain the same time delay (length embedded = 0.78 \times length unembedded). It is apparent that line delays may approach circuit delays. Figure 24.44 shows graphically the relation between circuit and

line delays. The solid lines represent circuit delays and the broken lines represent percents of logic delays: 10, 20, 30, 40, and 50 percent. For a required maximum total delay and a given logic delay, the maximum line length can be determined.

19. Cross Talk Cross talk is the undesirable coupling of energy between signal paths. The unwanted transfer of energy between lines results from the capacitive and inductive

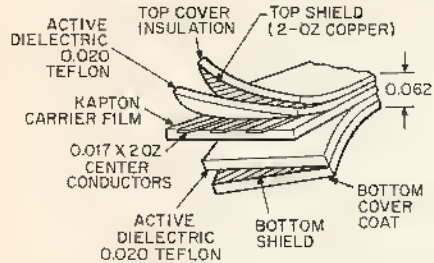


Fig. 24.42 Seventy-five-ohm strip-line flexible printed wiring device.

coupling of the lines and is a function of length of the lines, space between the lines, and the dielectric constant. The phenomenon is often a limiting factor in high-speed digital systems in which coupling between lengths of parallel lines may be the only alternative to the high wiring density which is required. Cross talk between an active line, i.e., the line carrying the voltage pulse, and a passive line in the vicinity places a spurious pulse on the passive line. If the latter pulse is of sufficient amplitude, it could falsely switch circuits on the passive line. Traditionally, before cross talk could be calculated it was necessary to calculate or measure mutual inductance, mutual capacitance, propagation velocity, and impedance of the lines. Analytical methods are unwieldy because of the effect of air on the dielectric constant and the uncertainty of fringe field effect. A method that is now available depends only on the geometric cross section of the lines and is independent of time, length, and voltage.

For the case of microstrip, then, induced voltage values due to cross talk may be calculated by use of the formulas:

$$V_B(t) = K_B [V_i(t) - V_i(t - 2l\sqrt{LC})]$$

$$V_F(t) = K_F l \frac{dV_i}{dt} (t - l\sqrt{LC})$$

where l = length of coupled region

- V_B = induced backward voltage as seen at the ends of the coupled region
- V_F = induced forward voltage as seen at the ends of the coupled region
- V_i = input voltage
- K_B = backward cross-talk constant
- K_F = forward cross-talk constant
- t = time, ns

K_B is always positive with respect to the driving signal; K_F is negative for microstrip and will be zero for homogeneous dielectric media. To determine cross-talk constants for any system, use line lengths in which $2T_d$ is significantly greater than the rise time and use the following formulas:

$$K_B = \frac{V_B}{V_i} \quad \text{and} \quad K_F = \frac{V_{Fmax}}{dV_i/dt_{max}} \frac{1}{l}$$

where l = length of coupled region

- V_B = induced backward voltage
- V_F = induced forward voltage
- V_i = input voltage

Pulse cross talk is represented by

$$V_B(t) = K_B [V_i(t) - V_i(t - l)\sqrt{LC}]$$

$$K_B = V_i \quad K_i = V_1 \quad V_i \rightarrow V_1$$

Empirical values for K_B and K_F are given in Figs. 24.45 and 24.46. A summary of cross-talk values is given in Fig. 24.47.

coupling of the lines and is a function of length of the lines, space between the lines, and the dielectric constant. The phenomenon is often a limiting factor in high-speed digital systems in which coupling between lengths of parallel lines may be the only alternative to the high wiring density which is required. Cross talk between an active line, i.e., the line carrying the voltage pulse, and a passive line in the vicinity places a spurious pulse on the passive line. If the latter pulse is of sufficient amplitude, it could falsely switch circuits on the passive line. Traditionally, before cross talk could be calculated it was necessary to calculate or

20. Attenuation Power may be lost in transmission lines by radiation, heating of the conductors, and heating of the dielectric. Heat losses in both the conductor and the dielectric increase with frequency. Conductor losses are greater the lower the characteristic impedance of the cable. The converse is true of dielectric losses. Because of its low impedance, the specified printed wiring has an attenuation higher than that of the most common coaxial cables. Typical data for a frequency range from 1 to 3000 MHz are shown in Fig. 24.48.

MECHANICAL DESIGN FOR HIGH BENDING STRESS APPLICATIONS

In many cases flexible printed wiring is utilized for such factors as conformability, weight reduction, and mass termi-

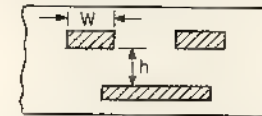


Fig. 24.43 Schematic of multilayer printed wiring.

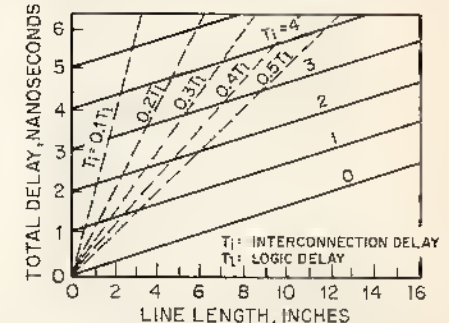


Fig. 24.44 Interconnection delay and logic delay vs. line length for a dielectric of 4.5.

nation capability as previously described. In those applications, the end uses of the devices which are flexible, is quasi-flexible, but in other applications the FPW must withstand quite severe bending stresses. This section will provide the designer with basic design philosophies in designing FPW to maximize its flex life.

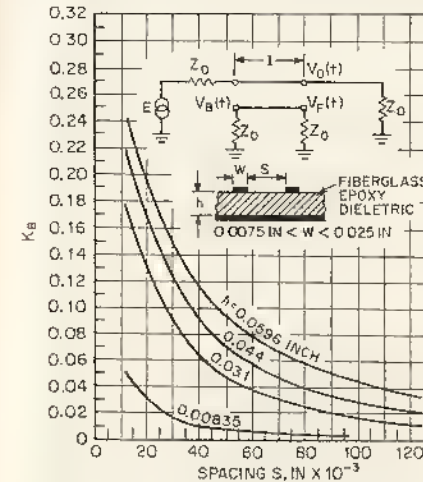


Fig. 24.45 Back cross talk constant as a function of spacing. Notes: (1) Lines terminated in Z_0 , the characteristic impedance of each line. (2) Curves valid for $0.0075 \text{ in} < W < 0.025 \text{ in}$. (3) Copper weight is 2 oz (0.0028 in thick). (4) Time delay of line = $T_d = 1.8 \text{ ns/ft}$. (5) $V_B(t) = K_B V_d t + l t_d - K_B V(t - l T_d)$.

Nominal board, in	h , in	Range of Z_0 , Ω
$\frac{1}{16}$	0.00835	36-61
$\frac{1}{32}$	0.031	75-105
$\frac{3}{64}$	0.044	86-110
$\frac{1}{8}$	0.0596	93-121

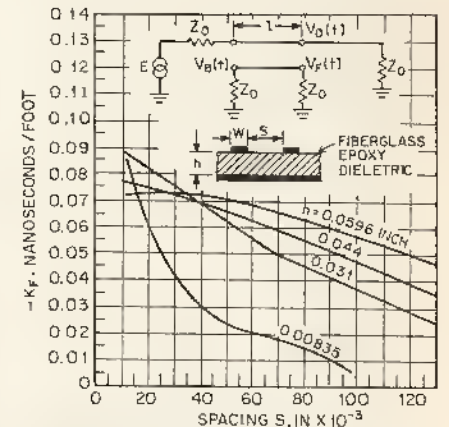


Fig. 24.46 Forward cross talk as a function of spacing. Notes: (1) Lines terminated in characteristic impedance. (2) Curves valid for $0.0075 \text{ in} < W < 0.025 \text{ in}$. (3) Time delay of line = $T_d = 1.8 \text{ ns/ft}$. (4) $V_F(t) = K_F l d[V_i(t)] / dt$.

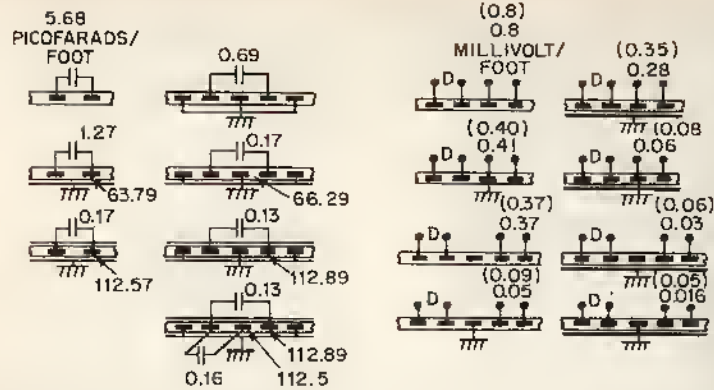


Fig. 24.47 Summary of cross talk measurements. Capacitive cross-talk measurements for various configurations of flexible printed wiring. The values in parentheses are in millivolts per foot, and they represent the pickup voltage caused by a current of 4 A at 400 Hz in the disturbing circuits (D = 120 V, 460 Hz). Suitable shielding and grounding arrangements can be used to reduce capacitance in various configurations of FPN. For example, the capacitance of 5.68 pF/ft for the configuration shown at the far left can be reduced to 0.69 pF/ft by grounding three conductors as shown in the next configuration. The addition of grounded shields reduces the capacitance still further, as can be noted. The values marked with an asterisk are for conductor-to-ground capacitance. The following table shows what happens in a specific case using 27 conductors, each 0.003 x 0.042 in, 0.075-in center-to-center spacing, and aluminum foil shields.

Shields	Disturbing conductors	Grounded conductors	Sensitive conductors	Cross talk	
				Inductive, $\mu V/A/ft$ (10 Ω)	Capacitive, $\mu V/V/ft$ (5000 Ω)
No shield	1 + 2	—	3 + 4	23	6.9
	1 + 2	3	3 + 4	23	3.4
	1 + 2	4	3 + 4	23	16.8
	1 + 2	—	4 + 5	9	3.0
	1 + 2	3	4 + 5	9	0.4
Shield on one side, $\frac{1}{8}$ -in Al strip grounded	1 + 2	—	3 + 4	23	2.3
	1 + 2	3	3 + 4	23	0.5
	1 + 2	4	3 + 4	23	5.2
	1 + 2	—	4 + 5	9	0.2
Shields both sides, $\frac{1}{8}$ -in Al strip Al foil grounded	1 + 2	—	3 + 4	23	0.4
	1 + 2	3	3 + 4	23	0.0
	1 + 2	4	3 + 4	23	0.8

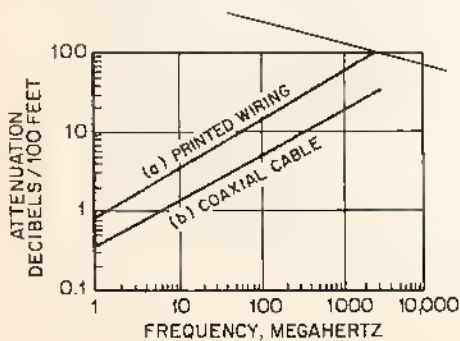


Fig. 24.48 Attenuation of wiring devices. (a) Flexible printed wiring with 0.025 x 0.0027 in copper conductors, spacing between conductors 0.020 in, polyester dielectric 0.003 in thick on top and bottom of conductors, aluminum shield on one or both sides of the cable. (b) Coaxial cable type RG-55/V (NASA-MSFC, Huntsville, Ala.).

In the bending of an FPW device, high stress could be brought forth to the conductors as shown in the single-sided case in Fig. 24.49. The most common cause of failure in that case is a tensile overstress of the conductor. That factor is further emphasized when columnar-structured electrodeposited copper is used. The tensile stress on the "solution"

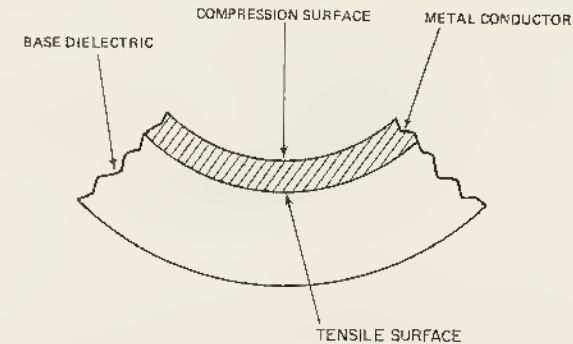


Fig. 24.49 Mechanical design for high-bending-stress applications.

side of the electrodeposited copper tends to precipitate a crack along the columnar grain structure of the foil. On the other hand, rolled copper foil is more resistant to that type of failure, primarily for reason of its higher elongation and equiaxial grain structure. A comparison of the grain structure of typical electrodeposited sulfate copper and rolled copper is shown in Fig. 24.50.

The bending stress imposed on a conductor is also a function of the distance of the copper fibers from the neutral axis of the composite laminate. The neutral axis is approximated by the imaginary symmetrical axis of the laminate. The outer fiber stress imposed on a copper conductor is proportional to the distance from the neutral axis. The closer the copper conductor to the neutral axis the lower the bending stress. That is shown in Fig. 24.51. Thus the factors which affect bending stress are radius of curvature R , distance from the neutral axis d , and modulus of elasticity E of the material. Those parameters can be arranged in the following relation:

$$S_b = \frac{Ed}{R}$$

The formula says, of course, that the bending stress S_b is directly proportional to the modulus of elasticity E of the material and the distance d from the neutral axis and is inversely proportional to the bend radius R . That relation, when applied to the design of flexible printed wiring, states:

- Minimize the bending stress by using low-modulus conductor material.
- Minimize the distance of the conductor from the neutral axis.
- Maximize the radius of circuits of the FPW.

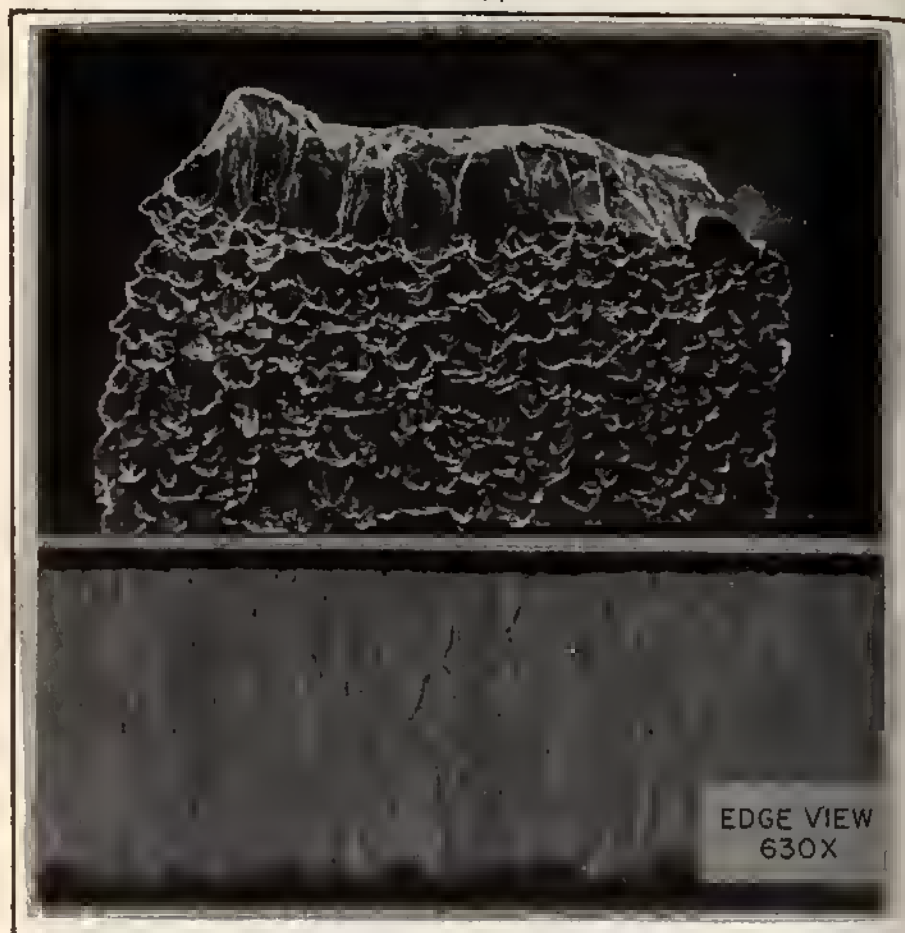
To further increase the flex life, the following factors should be considered:

a. Cover Coat. A cover coat of proper thickness will place the conductor on the neutral or stress-free axis in a single-sided application as shown in Fig. 24.52. The cover coat will also add to the overall thickness, which will increase the bend radius with a corresponding decrease in bending stress.

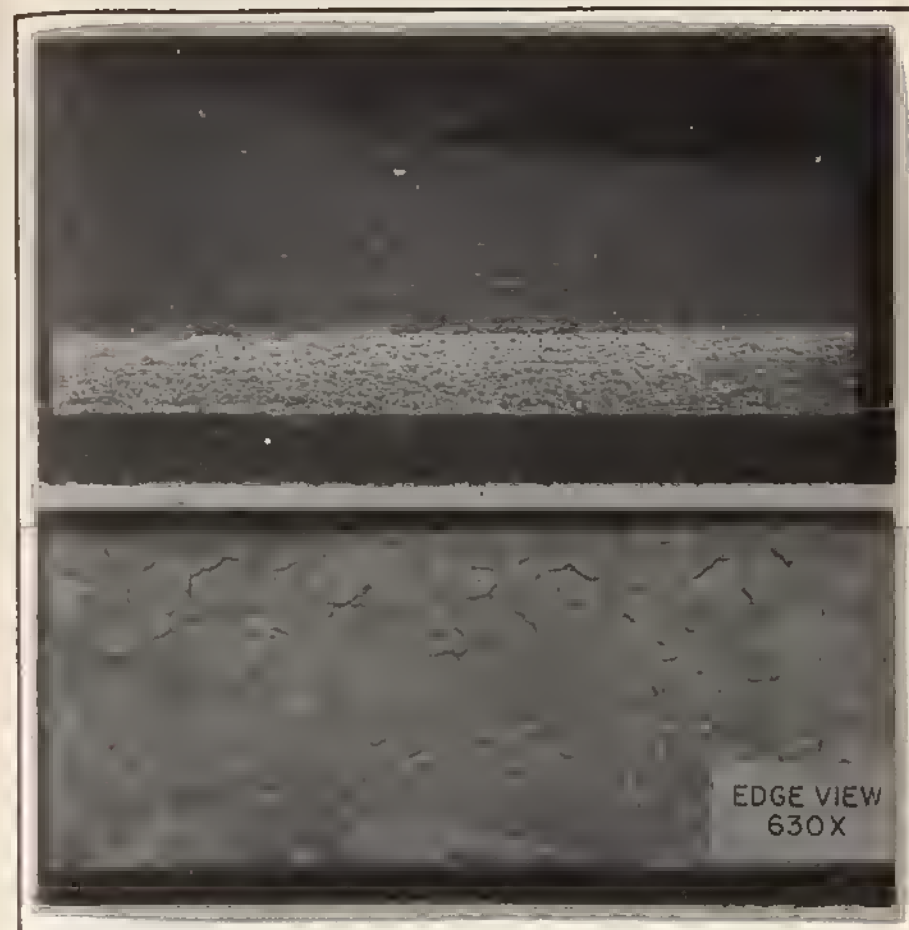
b. Rolled Copper. To achieve maximum flex life, the use of rolled annealed copper vs. electrodeposited copper should be specified. As indicated earlier, rolled copper has an equiaxial grain structure as compared to the stress-susceptible columnar grain of electrodeposited copper. Its elongation is higher than that of electrodeposited copper (28 percent vs. 19 percent for standard electrodeposited copper in 2-oz weights).

c. Oriented Base Materials. If the base material utilized has a fiber orientation, the flex wiring should be so manufactured that the orientation is perpendicular to the flexing direction, as shown in Fig. 24.53.

E.D. Copper



R.T. Copper



Copper Properties	E.D.	R.T.
Tensile St. (psi)	38,000	33,000
Elongation (% 2.0 in G.L.)	14	28
Hardness (KNOOP)	85	55
Grain Structure	Columnar	Equiaxed

Fig. 24.50 Comparison of the grain structure of electrodeposited (above) and rolled (opposite) copper foil. (Courtesy R. N. Wild, IBM.)

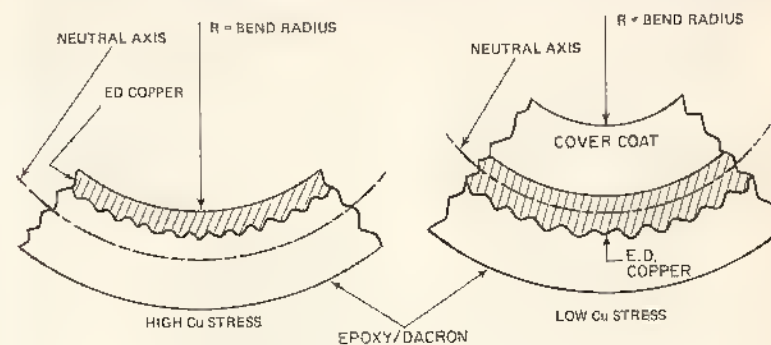


Fig. 24.51 Effect of neutral axis on conductor stress.

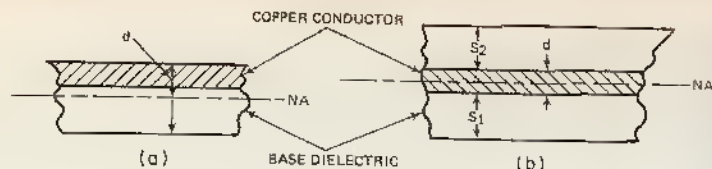


Fig. 24.52 Effect of cover coat on neutral axis location.

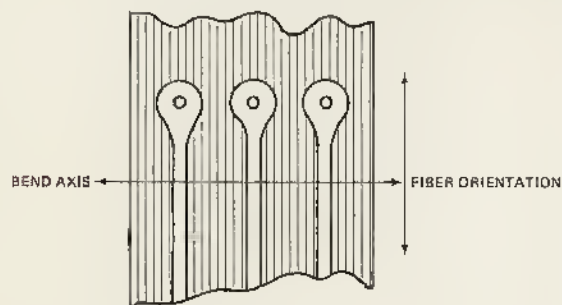


Fig. 24.53 Orientation of conductors and material directionality.

BIBLIOGRAPHY

1. R. F. Van Ness, A Total Flat Flexible Multilayer Printed Wiring Approach, *Picatinny Arsenal Tech. Report 3588*, July 1967.
2. "Copper Clad Laminates and Adhesive Coated Film," Product Bulletin, Riegel Paper Corporation.
3. "Cu Flex Flexible Laminate," Product Information, Dielectric Materials and Systems Division, 3M Company, Minneapolis, Minn.
4. "Designers Handbook Multilayer and Flexible Circuits," Methode Electronics, Inc. 1966.
5. Cannizzaro, J. R.: "Development of Kapton Polyimide Printed Circuit Cables," IBM No. 67-825-2144, IBM, Poughkeepsie, N.Y.
6. "Flexible Circuit Cable Design Guide," Teledyne Electro-Mechanisms, 1970.
7. "Flexible Flat Cable Handbook, Institute of Printed Circuits, Chicago, Ill., 1965.
8. "Flexprint Circuit Design Handbook," Saunders Associates, 1968.

Chapter 25

Flexible Printed Wiring Materials

J. R. CANNIZZARO

Digital Equipment Corp., Maynard, Massachusetts

Introduction	25-1
Base Dielectric Materials	25-2
1. Thermoplastic Materials	25-2
2. Fluorocarbons	25-2
3. Nylon, Nomax	25-8
Type H Kapton Polyimide Film	25-12
4. Cut-through and Cold-flow Film	25-15
5. Electrical Properties of Kapton® Polyimide Film	25-15
6. Conditioning Prior to Laminating Kapton Polyimide Film	25-16
7. Type F Polyimide Film	25-17
Polyparabanic Acids (PPA)	25-19
8. Mechanical Properties of PPA Film	25-20
9. Orientation of PPA Film	25-20
10. Thermal Properties of PPA Film	25-22
Polyesters	25-23
Insulating Film Cover Coats	25-26
11. Cover Coat Properties and Bonding	25-26
12. Liquid Cover Coat	25-28
13. Properties of the Cured Cover Coat	25-33
14. Cover Coats for Polyimide Circuits	25-36
15. Cover Coats for Polyester Circuits	25-42
16. Cover Coats for PVC Circuits	25-43

INTRODUCTION

The widespread use of flexible printed wiring (FPW) was made possible by the commercial availability of film and other flexible base materials. The useful applications of FPW are steadily increasing, owing to the continuing advances in plastics technology. New

films and flexible base materials are constantly being developed for flexible wiring applications, and their upper temperature limit has risen substantially in recent years. This chapter will discuss the testing methods, types, and properties of plastic base materials as they relate to FPW applications.

BASE DIELECTRIC MATERIALS

Plastic materials may be referred to and subdivided into many different categories, classifications, etc., but in general they fall into two major groups: thermosetting materials and thermoplastic materials. The present commercially available dielectric materials suitable for use in the manufacture of flexible printed wiring, grouped into thermosets and thermoplastics, will be presented in this section.

1. Thermoplastic Materials Thermoplastics differ from thermosets in that thermoplastics do not cure or set under heat as thermosets do. When heated to a flowable state, thermoplastics merely soften, and under pressure they can be extended or continuously cast into film form. Subsequent cooling will harden the film. Since thermoplastics do not cure or set, they can be remelted and rehardened many times by cooling. Thermal aging, brought about by repeated exposure to the high temperatures required for melting, eventually causes degradation of the material and so limits the number of reheat cycles. Essentially all thermoplastics are processed by heating to a soft state and applying pressure, as by injection molding, extruding, and thermoforming.

Thermosets, on the other hand, do not melt. Both thermosets and thermoplastics soften to varying degrees upon heating, but fully cured thermosets are not usually processed in that fashion. Instead, they are cured and chemically cross-linked. Thermoplastics are straight-chain polymers rather than infusible, cross-linked polymers.

2. Fluorocarbons Fluorocarbons are used quite frequently in the manufacture of flexible printed wiring devices for reason of their excellent electrical properties, which are relatively unaffected by most environments or operating conditions. The original fluorocarbon was tetrafluoroethylene, better known as TFE Teflon. Teflon, however, is a Du Pont trade name for TFE and FEP (fluorinated ethylene propylene) fluorocarbons. TFE was a chance discovery in 1938 by a du Pont research chemist, who temporarily stored some tetrafluoroethylene gas (Freon) in an empty steel cylinder and later found it transformed into an insoluble waxy, white powder. The three most commonly used fluorocarbons are described in Table 25.1. The TFE fluorocarbons possess optimal electrical properties but have the disadvantage of cold flow or creep under loading. FEP fluorocarbon is quite similar to TFE in most properties except that its useful temperature is limited to about 400°F (Table 25.2). FEP is much more easily processed than TFE. The CTFE (chlorotrifluoroethylene) material like FEP, is melt-processable. Compared with TFE, CTFE has greater tensile and compressive strength within its service temperature range. However, at the temperature extremes, CTFE does not perform as well as TFE. Electrical properties of CTFE are generally excellent, but dielectric losses are higher than those of TFE. Chemical resistance is poorer than that of TFE, but radiation resistance is better. CTFE does not have the low-friction properties of TFE. The formation reactions for TFE [polytetrafluoroethylene, $(-CF_2CF_2-)_n$] and FEP [copolymer of tetrafluoroethylene and hexafluoropropylene, $(-CF_2CF_2-CF_2CF-CF_3)_n$] are shown in Fig. 25.1.

a. Mechanical Properties. As is true of all plastics, especially thermoplastics, the mechanical properties of fluorocarbons vary with temperature. TFE fluorocarbon, perhaps the most commonly used fluorocarbon in electronics, is a semisoft plastic which exhibits some cold-flow properties. TFE and FEP are unique in their ability to retain a useful balance of flexibility and strength over a wide temperature range. TFE plastics are superior in their combination of toughness, flex resistance, and general abuse resistance. The hardness of TFE as a function of temperature is shown in Fig. 25.2. The tensile strength of TFE and FEP as a function of temperature is shown in Fig. 25.3. The overall mechanical properties of TFE as a function of temperature are shown in Table 25.2. Linear thermal expansion of TFE and FEP as a function of temperature is shown in Fig. 25.4. The mechanical properties of TFE and FEP at low temperatures are shown in Table 25.3.

TABLE 25.1 Applications and Forms of Common Fluorocarbon Materials

Material	Major application considerations	Common available forms	Typical suppliers and trade names
Fluorocarbons a. Chlorotrifluoroethylene (CTFE)	Excellent electrical properties and relatively good mechanical properties. Somewhat stiffer than TFE and FEP fluorocarbons, but does have some cold flow. Widely used in electronics, but not quite so widely as TFE and FEP. Useful to about 400°F.	Extrusions, isotactic moldings, injection moldings, film, stock shapes	3M Co. (Kel-F); Allied Chemical Corp. (Plaskon CTFE)
b. Fluorinated ethylene propylene (FEP)	Properties very similar to those of TFE, except useful temperature limited to about 400°F.	Extrusions, injection moldings, laminates, film	Du Pont (Teflon FEP)
c. Polytetrafluoroethylene (TFE)	Electrically one of the most outstanding thermoplastic materials. Exhibits very low electrical losses and very high electrical resistivity. Useful to over 500°F and to below -300°F. Excellent high-frequency dielectric. Among the best combinations of mechanical and electrical properties but relatively weak in cold-flow properties. Nearly inert chemically, as are most fluorocarbons. Very low coefficient of friction. Nonflammable.	Compression moldings, stock shapes, film	Du Pont (Teflon TFE); Allied Chemical Corp. (Halton TFE)

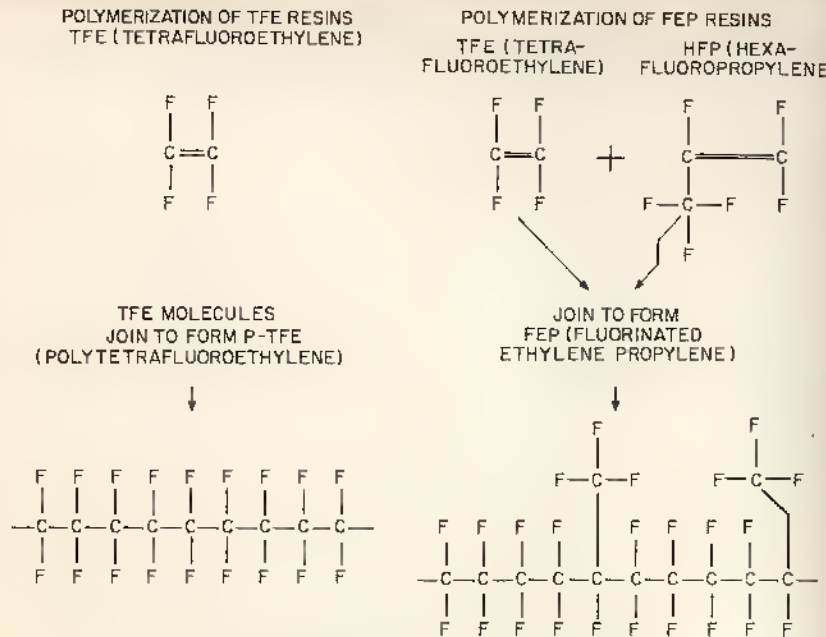


Fig. 25.1 Polymerization of Teflon® fluorocarbon resins.

b. Electrical Properties. Fluorocarbons have excellent electrical properties. TFE and FEP fluorocarbons in particular have low dielectric constants and dissipation factors which change little with temperature or frequency. That is shown in Figs. 25.5 and 26.6, which also present data for some other thermoplastics. The dielectric strength of TFE and FEP resins is high and does not vary with temperature and thermal aging. Initial

TABLE 25.2 High-temperature Mechanical Properties of TFE Fluorocarbon Resins*

Property	Temperature*		
	72° F (23° C)	212° F (100° C)	400° F (204° C)
Ultimate tensile strength, psi	3850	2500	1500
Yield strength, psi	1050 at 2%	400 at 2%	200 at 4%
Ultimate elongation, %	300	>400	360
Flexural modulus of elasticity, psi	80,700	28,700	
Flexural modulus of elasticity (35% glass-reinforced), psi	208,700	113,000	
Compressive stress, 1% strain, psi	700	290	100
Compressive stress, 5% strain, psi	1850	800	430
Compressive stress, 5% strain (15% glass-reinforced), psi	3600	1300	600
Linear expansion, in/(in)(° F)	6.90×10^{-5}	6.90×10^{-5}	8.40×10^{-5}
Linear expansion, %	0.0	0.9	2.7
Linear expansion (35% glass-reinforced), %	0.0	0.15	
Coefficient of friction		0.04 over temperature range 80 to 621° F for static loads	

*These values are typical of those for "Teflon" TFE-fluorocarbon resin. Variations may be expected from the values shown depending on the type of TFE resin used, methods of molding, and fabrication techniques employed.

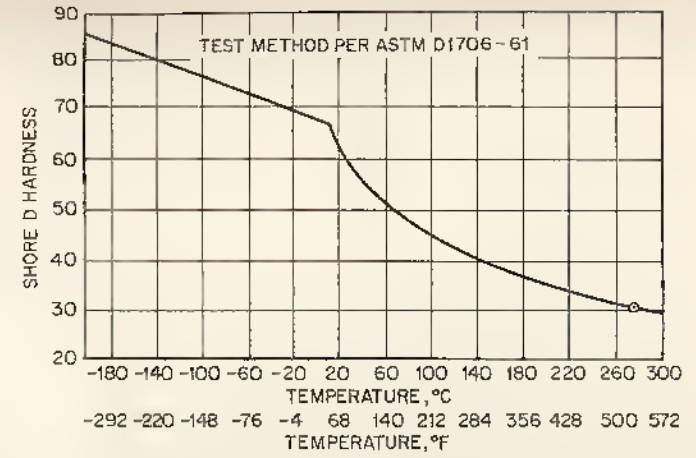


Fig. 25.2 Hardness of TFE fluorocarbon as a function of temperature.

dielectric strength is very high as measured by the ASTM short-time test. As with most dielectric materials, the value drops as thickness of the specimen increases, as shown in Fig. 25.7. Further the dielectric strength is a function of frequency, as noted in Fig. 25.8. Life at high dielectric stresses is dependent on corona discharge, as shown in Fig. 25.9. The absence of corona, as in special wire constructions, permits very high voltage stress without damage to either TFE or FEP resins. Changes in relative humidity or physical stress imposed upon the material do not diminish life at those voltage stresses.

Surface arc resistance of TFE and FEP resins is high, and it is not affected by heat

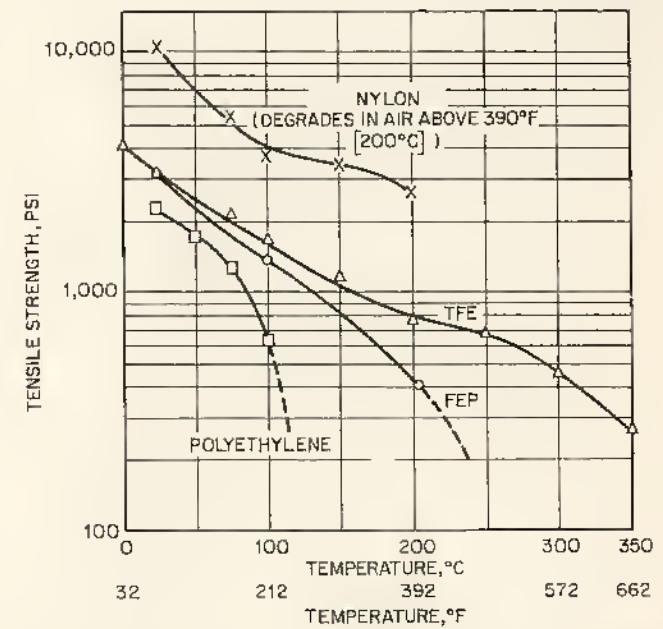


Fig. 25.3 Tensile strength of TFE, FEP, polyethylene, and nylon as a function of temperature.

TABLE 25.3 Strength Properties of TFE and FEP Fluorocarbons at Low Temperature

Mechanical property	Temperature		TFE		FEP	
	°F	°C	Unfilled*	Filled 25% glass fiber	Unfilled†	Filled 25% glass fiber
Tensile yield strength, psi × 10 ⁻³	-423	-253	17.8	4.3	23.7	15.5
	-320	-196	13.3	3.5	19.0	13.5
	-297	-183	(12.1)	(3.3)	(18.2)	
	-200	-129	7.7	(2.3)	11.4	
	-110	-79	4.7	(1.3)	5.6	
	+77	+25	1.7	0.6	2.0	2.0
Ultimate tensile strength, psi × 10 ⁻³	-423	-253	17.9	4.2	23.8	15.5
	-320	-196	14.9	3.6	18.0	13.5
	-297	-183	(13.8)	(3.5)	(17.1)	(12.9)
	-200	-129	9.1	(2.9)	12.2	(10.0)
	-110	-79	5.9	(2.2)	6.5	(7.1)
	+77	+25	4.5	1.0	4.0	2.2
Tensile modulus, psi × 10 ⁻⁶	-423	-253	0.62	0.46	0.73	1.20
	-320	-196	0.47	0.38	0.58	1.10
	-297	-183	0.45	(0.36)	0.56	(1.04)
	-200	-129	0.30	(0.29)	0.48	(0.80)
	-110	-79	0.20	(0.23)	0.30	(0.58)
	+77	+25	0.08	0.08	0.07	0.15
Elongation, %	-423	-253	3	1	5	3
	-320	-196	7	2	7	4
	-297	-183	(7)	(3)	(8)	(8)
	-200	-129	13	(13)	15	(23)
	-110	-79	31	(22)	33	(41)
	+77	+25	198	40	353	70
Flexural strength, psi × 10 ⁻³	-423	-253	28.3	8.7	36.3	28.6
	-320	-196	25.8	6.3	26.7	23.4
	-297	-183	(24.7)	(6.1)	(24.7)	(22.4)
	-200	-129	15.2	(5.2)	20.1	(17.5)
	-110	-79	6.9	(4.2)	9.6	(13.3)
	+77	+25	3.7	2.8	2.9	4.7
Flexural modulus, psi × 10 ⁻⁸	-423	-253	0.74	0.41	0.77	0.94
	-320	-196	0.68	0.36	0.68	0.85
	-297	-183	(0.66)	(0.34)	(0.68)	(0.82)
	-200	-129	0.46	(0.26)	0.57	(0.70)
	-110	-79	0.23	(0.19)	0.33	(0.59)
	+77	+25	0.17	0.04	0.20	0.38
Compressive strength, psi × 10 ⁻³	-423	-253	31.9	27.3	35.9	37.6
	-320	-196	21.1	21.3	30.0	31.9
	-297	-183	(20.2)	(20.3)	(28.8)	(30.3)
	-200	-129	16.0	(15.8)	23.4	(23.3)
	-110	-79	7.4	(11.5)	13.3	(16.7)
	+77	+25	3.7	3.0	1.5	4.2
Compressive modulus, psi × 10 ⁻⁶	-423	-253	0.90	0.99	1.02	1.50
	-320	-196	0.80	0.86	0.92	1.34
	-297	-183	(0.78)	(0.81)	(0.91)	(1.25)
	-200	-129	0.59	(0.64)	0.74	(1.01)
	-110	-79	0.28	(0.47)	0.39	(0.78)
	+77	+25	0.10	0.15	0.09	0.23
Torsion modulus of rigidity, psi × 10 ⁻²	-423	-253	322	244	964	528
	-320	-196	212	104	(320)	304
	-297	-183	(202)	(100)	(284)	(288)
	-200	-129	142	(76)	172	(220)
	-110	-79	70	(56)	96	(156)
	+77	+25	23	16	16	36

TABLE 25.3 Strength Properties of TFE and FEP Fluorocarbons at Low Temperature (Continued)

Mechanical property	Temperature		TFE		FEP	
	°F	°C	Unfilled*	Filled 25% glass fiber	Unfilled†	Filled 25% glass fiber
Notched Izod impact strength, ft-lb/in	-423	-253	1.40	1.00	1.83	1.65
	-320	-196	1.30	1.00	1.73	1.85
	-297	-183	(1.35)	(1.05)		
	-200	-129	(1.40)	(1.30)		
	-110	-79	1.50	(1.55)	>9.0	
	+77	+25	1.90	2.00	>19.5	>145

NOTE: Filled composition by volume. Interpolated data indicated by parentheses.
 *Crystallinity 41-71%.
 †Crystallinity 44-51%.

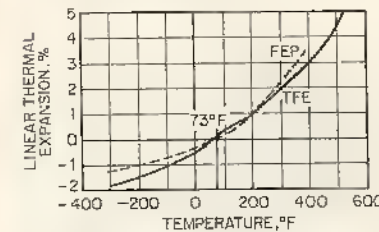


Fig. 25.4 Linear thermal expansion of TFE and FEP fluorocarbons as a function of temperature.

aging. When the resins are subjected to a surface arc in air, they do not track or form a carbonized path. When tested by the procedure of ASTM D 495, they pass the maximum time of 300 s without failure.

The unique nonstick surface of the resins helps reduce surface arc phenomena in two ways:

1. It helps prevent formation of surface contamination and thereby reduces the possibility of arcing.

2. If an arc is produced, the discharge frequently cleans the surface of the resins and increases the time before another arc.

Volume resistivity (10¹⁸ Ω·cm) and surface resistivity (10¹⁶ Ω/sq) for both FEP and TFE resins are at the top of the measurable range. Neither resistivity is affected by heat aging or temperatures up to recommended service limits.

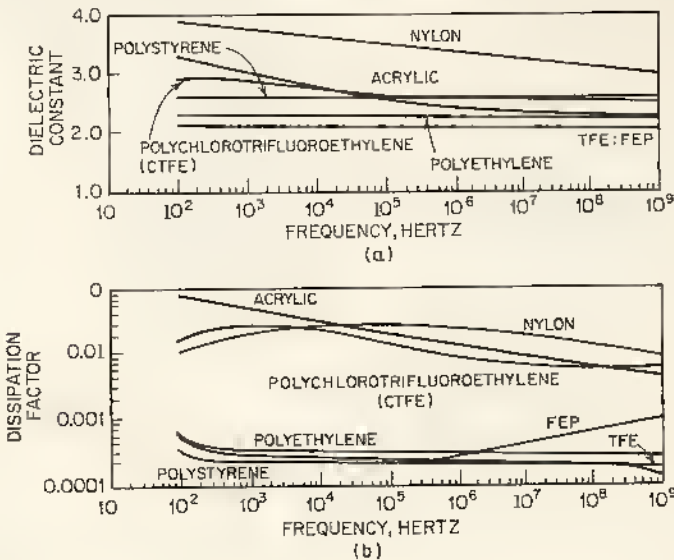


Fig. 25.5 Effect of frequency on (a) dielectric constant and (b) dissipation factor of several thermoplastic materials at 23°C.

c. Environmental Properties. There has been much discussion of the presence of oxygen. The following major findings resulted from a study made by Du Pont.

1. TFE in air:
 - a. Has a damage threshold in the range of 2 to 7×10^4 rad.
 - b. Retains 50 percent of initial tensile strength after a dose of 10^6 rad.
 - c. Retains 40 percent of initial tensile strength after a dose of 10^7 rad or more.
 - d. Retains ultimate elongation of 100 percent or more (adequate for flexible wire) for doses up to 2 to 5×10^8 rad.
2. TFE in vacuum:
 - a. Has a damage threshold in or above the range 2 to 7×10^6 rad.
 - b. Retains 50 percent of initial tensile strength after a dose of 10^7 rad or more.
 - c. Retains 40 percent of initial tensile strength after a dose of 8×10^8 rad or more.
 - d. Retains ultimate elongation of 100 percent or more for doses up to and probably beyond 2 to 5×10^9 rad.
3. FEP in air will tolerate a dose 10 to 100 times larger than TFE in air.
4. FEP in vacuum will tolerate a dose up to 10 times larger than TFE will tolerate in vacuum.
5. Testing in air after vacuum irradiation does not necessarily yield a true measure of radiation tolerance in the absence of oxygen.
6. The radiation tolerance of TFE and FEP is at least as good as that of some common electronic components, transistors in particular.
7. The radiation tolerance of TFE and FEP is much greater than that of man.
8. Since radiation produces electric charge distributions in the resin which decay with time, dielectric properties are a function of dose rate and exhibit transients. The low-frequency loss properties of TEF polymers are drastically affected by X-ray irradiation; the high-frequency loss properties considerably less so. The increases in dielectric constant and dissipation factor depend on the ambient oxygen concentration during exposure and recovery. The dielectric constant and dissipation factor of FEP remain unaffected by X-ray irradiation in vacuum at frequencies from 60 Hz to 100 kHz.

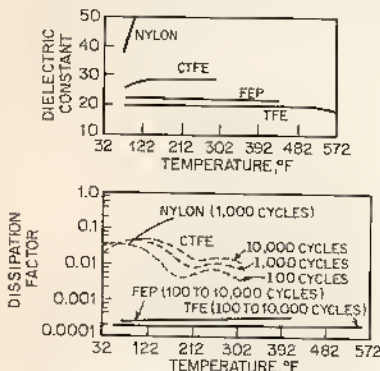


Fig. 25.6 Effect of temperature on dielectric constant and dissipation factor of fluorocarbons and nylon.

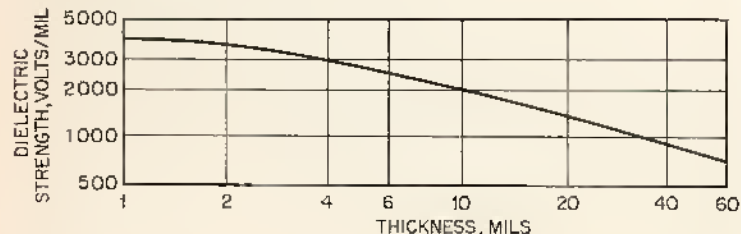


Fig. 25.7 Short-term dielectric strength of TFE fluorocarbon as a function of thickness.

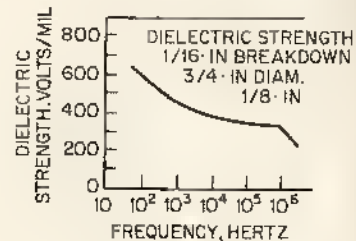


Fig. 25.8 Dielectric strength of TFE resins as a function of frequency.

9. Teflon TFE and FEP resins will not evaporate in a vacuum of 10^{-7} mm Hg. Theoretical calculations show that they will not evaporate in any anticipated space vacuum to an extent that would limit their usefulness. Although some outgassing occurs initially, all the volatiles are adsorbed atmospheric gases.

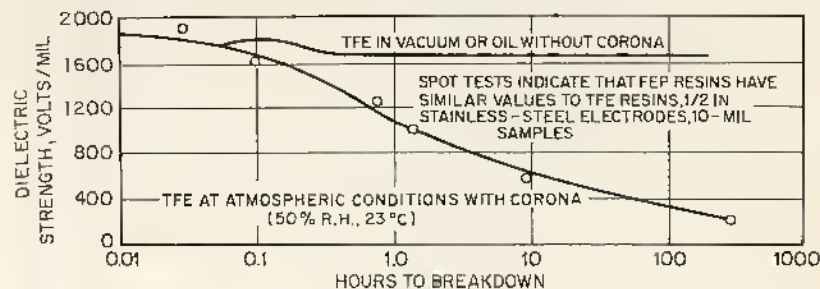


Fig. 25.9 Insulation life vs. continuously applied voltage stress for TFE and FEP fluorocarbons.

3. Nylon, Nomex The common nylons (polyamides) are not generally utilized in flexible printed wiring applications, since they are poor in electrical performance after exposure to humidity conditions. However, Nomex, a relatively new, high-temperature nylon with good overall properties, is finding increased use in flexible printed wiring.

Nomex is Du Pont's registered trademark for high-temperature-resistant nylon fibers and papers. Nomex nylon has strength and toughness characteristics like those of the familiar nylon 66 used, for example, in tires, but it differs, because of its aromatic molecular structure, in several major characteristics:

1. Outstanding thermal and chemical stability. (At 250°C the melting point of nylon 66, Nomex still retains 60 percent of its room-temperature tensile strength.)
 2. Will not melt or support combustion.
 3. Initial modulus (resistance to deformation) about four times as great as nylon 66.
- Nomex paper is composed of two different forms of the same polymer, short fibers (floc) and smaller fibrous binder particles (fibrils) as shown in Fig. 25.10. After processing on

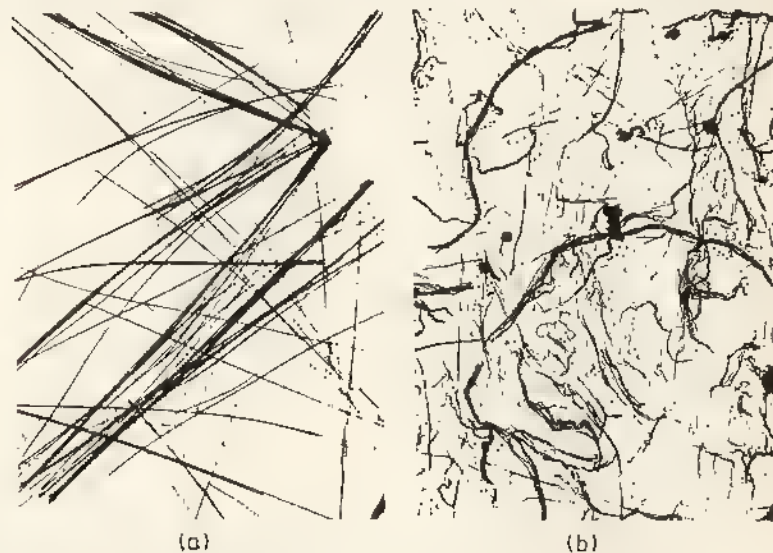


Fig. 25.10 Components of Nomex® nylon paper: (a) one-quarter-inch floc fibers (medium magnification); (b) fibril binder particles (highly magnified).

conventional papermaking equipment, the two constituents are permanently bonded together. No additional binders, filler, or sizes are used.

Nomex paper is available in calendered (type 410) and uncalendered (type 411) forms. Type 410 paper is available in a wide range of thicknesses from 0.002 to 0.030 in. The specific gravity of type 410 paper ranges from 0.75 to 1.1, depending on thickness. The paper is relatively nonporous, and its color may range from white to buff. Type 411 paper also is available in a range of thicknesses from 0.006 to 0.042 in. It has a relatively porous structure (specific gravity about 0.3) which is useful in certain applications where a soft, compressible, nonconducting packing or cushioning material is required or where the insulation must be completely impregnated with a resin and mechanical properties are of secondary importance. Nomex paper is recognized by Underwriters Laboratories as a 220°C material, and it conforms to MIL-I-24204 as a 220°C insulating material.

a. **Electrical Properties.** Important electrical properties of Nomex type 410 paper at room temperature and 50 percent relative humidity are shown in Table 25.4. The volume (Ω -cm) and surface (Ω /sq) resistivities of Nomex paper at room temperature are better than 10^{16} . Those values decrease with increasing temperature, but they still exceed 10^{12} at 250°C.

TABLE 25.4 Typical Electrical Properties of Nomex® Type 410 Nylon Papers

Property	Thickness, in							
	0.002	0.003	0.005	0.007	0.010	0.015	0.020	0.030
Dielectric Strength, V/in $\times 10^{-3}$								
AC rapid rise	450	500	600	750	750	800	700	600
AC 1-min hold					600			
AC 1-h hold					500			
DC rapid rise					1200			
Dielectric constant, 10^3 Hz†	2.0	2.1	2.3	2.5	2.6	3.0	3.3	3.4
Dissipation factor, 10^3 Hz†	0.007	0.008	0.010	0.012	0.014	0.016	0.018	0.020

*ASTM D-149, using 2-in-diameter electrodes

†ASTM D-150, using 1-in-diameter electrodes under 20 psi pressure

The resistivity of Nomex nylon paper is independent of thickness, as shown in Fig. 25.11. The dielectric constant and dissipation factor of Nomex type 410 paper increase slightly up to 250°C. The values for a range of frequencies and paper thicknesses are illustrated in Figs. 25.12 to 25.15. The dielectric strength of Nomex paper is unaffected by variations in temperature up to 225°C and remains at about 95 percent of its room temperature value at 250°C.

Nomex paper also retains good electrical properties in the presence of moisture, as shown in Table 25.5. The dielectric strength is essentially constant for Nomex papers conditioned at 0 and 95 percent relative humidity. Dielectric constant, dissipation factor, and volume resistivity are slightly affected over this same range.

In common with all other insulating materials, Nomex paper is gradually eroded under attack by corona discharges. Figure 25.16 shows the effect of corona on the dielectric life of Nomex relative to several other insulating materials. At stress levels from 200 to 400 V/mil the voltage endurance of Nomex is intermediate between values obtained for two inorganic type insulating materials. The data were obtained in an accelerated test, at a frequency of 360 Hz, as described in ASTM 2275. At a frequency of 60 Hz, the average life of Nomex type 410 paper, at a stress of 400 V/mil, is approximately six times that at 360 Hz. Measurements were made on 0.010-in specimens at room temperature and 50 percent relative humidity. The corona inception levels in this electrode assembly, for all materials evaluated, were in the range of 80 to 110 V/mil.

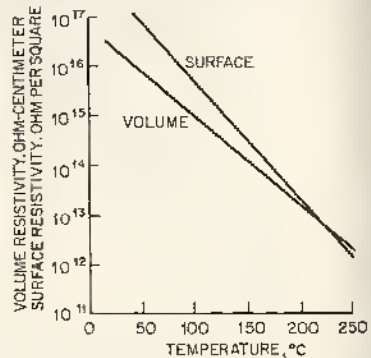


Fig. 25.11 Resistivity vs. temperature for Nomex® type 410 nylon paper.

TABLE 25.5 Electrical Properties of Nomex® Nylon Paper vs. Relative Humidity (10-mil, Type 410 paper)

Relative humidity, %	Dielectric strength, V/in $\times 10^{-3}$	Dielectric constant, 10^3 Hz	Dissipation factor, 10^3 Hz	Volume resistivity, Ω -cm
Oven dry	750	2.6	0.013	6×10^{16}
50	720	2.9	0.014	2×10^{16}
95	690	3.5	0.025	2×10^{11}

b. **Mechanical Properties.** Typical physical properties for various thicknesses of Nomex type 410 nylon paper at room temperature and 50 percent relative humidity are shown in Table 25.6. Since Nomex paper is anisotropic, its tensile and tear properties are listed for

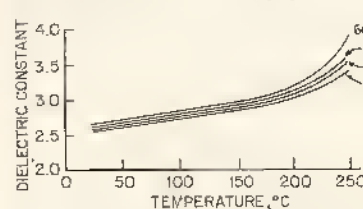


Fig. 25.12 Dielectric constant vs. temperature for Nomex® 0.010-in type 410 nylon paper.

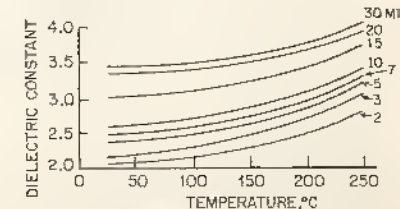


Fig. 25.13 Dielectric constant for various thicknesses of Nomex® type 410 nylon paper at 10^3 Hz.

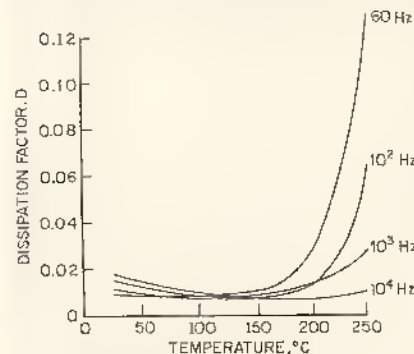


Fig. 25.14 Dissipation factor vs. temperature for Nomex® 0.010-in type 410 nylon paper.

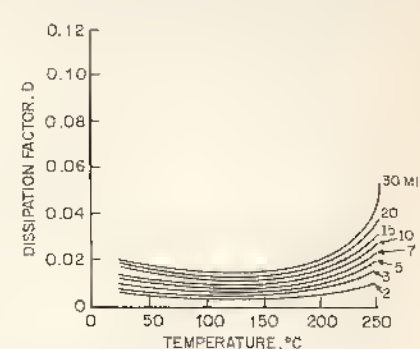


Fig. 25.15 Dissipation factor for various thicknesses of Nomex® type 410 nylon paper at 10^3 Hz.

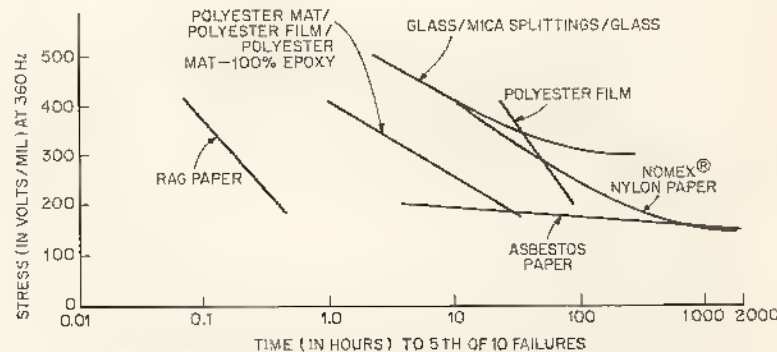


Fig. 25.16 Voltage endurance of various insulating materials (ASTM 2275).

both the machine direction (MD) and the cross direction (XD). The tensile strength of type 410 paper decreases gradually with increasing temperature, but it remains at about two thirds of its room-temperature value at 225°C (Fig. 25.17). Figure 25.18 shows that the break elongation is also essentially constant up to 200°C. Retention of excellent physical

TABLE 25.6 Typical Physical Properties of Nomex® Type 410 Nylon Paper

Property	Thickness, in								ASTM test number	
	0.002	0.003	0.005	0.007	0.010	0.015	0.020	0.030		
Tensile strength, lb/in	MD	22	37	70	120	170	270	370	550	D-828-60
	XD	13	23	45	68	100	180	250	380	D-828-60
Elongation, %	MD	10	12	16	19	24	24	25	25	D-828-60
	XD	8	10	14	16	20	21	21	22	D-828-60
Elmendorf tear, g	MD	95	150	260	380	550	850	1100	2000	D-689
	XD	140	250	450	620	900	1400	2100	2700	D-689
Finch edge tear, lb/in	MD	20	40	70	110	130	150	160	220	D-827-47
	XD	10	15	25	30	60	60	60	100	D-828-47
Shrinkage at 300°C, %	MD	1.6	1.2	0.9	1.1	1.5	1.4	1.4	1.4	
	XD	1.9	1.6	1.3	1.0	1.0	0.8	0.8	0.8	
Thermal conductivity, Btu·in/(hr)(ft²)(°F)					0.76		0.78			
Basis weight, oz/yd²		1.2	1.9	3.2	5.1	7.3	11	15	23	

MD = machine direction
XD = cross machine direction

properties at operating temperatures enables type 410 paper to withstand the mechanical stresses and vibration often encountered in electrical machines. Because of its nature (low-density, uncalendered) type 411 paper has less strength and toughness than Nomex type 410 paper. Typical physical properties for various thicknesses of Nomex type 411 paper are shown in Table 25.7.

TABLE 25.7 Typical Physical Properties of Nomex® Type 411 Nylon Paper

Property	Thickness, in					ASTM test number	
	0.006	0.009	0.016	0.025	0.042		
Tensile strength, lb/in	MD	10	17	23	44	67	D-828-60
	XD	5	9	16	22	32	D-828-60
Elongation, %	MD	3	3	3	3	3	D-828-60
	XD	5	5	5	5	5	D-828-60
Elmendorf tear, g	MD	120	200	320	520	590	D-689
	XD	150	240	340	620	740	D-689
Finch edge tear, lb/in	MD	8	15	25	46	66	D-827-47
	XD	5	10	15	24	38	D-827-47
Shrinkage at 240°C, %	MD		1.3	0.8			
	XD		1.2	1.6			
Basis weight, oz/yd²		1.2	1.9	3.2	5.1	7.3	

MD = machine direction
XD = cross machine direction

Nomex paper also exhibits good dimensional stability. The shrinkage of type 410 paper is less than 2 percent at 300°C in both the machine and cross direction (Table 25.6). Although type 411 paper does not have the dimensional stability at 300°C of type 410, it has a shrinkage of less than 2 percent in either direction at temperatures up to 240°C (Table 25.7). Dimensional changes due to increasing relative humidity are less than 1 percent at 65 percent RH and less than 2 percent at 95 percent RH for Nomex type 410 nylon paper, as shown in Table 25.8.

c. Thermal Stability. The usefulness of Nomex high-temperature-resistant nylon paper in the electrical industry stems largely from the paper's capability of maintaining its excellent combination of electrical and mechanical properties over long periods of continuous exposure at high temperatures. That is illustrated in Figs. 25.19 to 25.21, which reflect the usual linear relation between the logarithm of the aging time and the reciprocal of the absolute temperature. The lines and their corresponding equations were obtained from statistical analysis of the properties of Nomex paper which had been exposed for various lengths of time (up to one year) at 260, 280, 300, and 320°C. Conversion of the

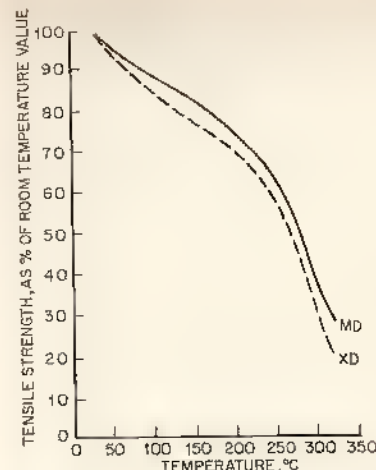
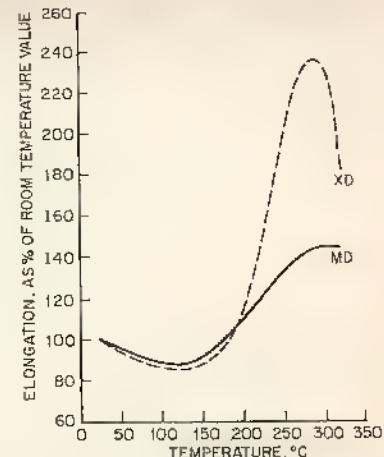


Fig. 25.17 Tensile strength vs. temperature for Nomex® 0.010-in. type 410 nylon paper.



25.18 Elongation temperature for Nomex® 0.010-in. type 410 nylon paper.

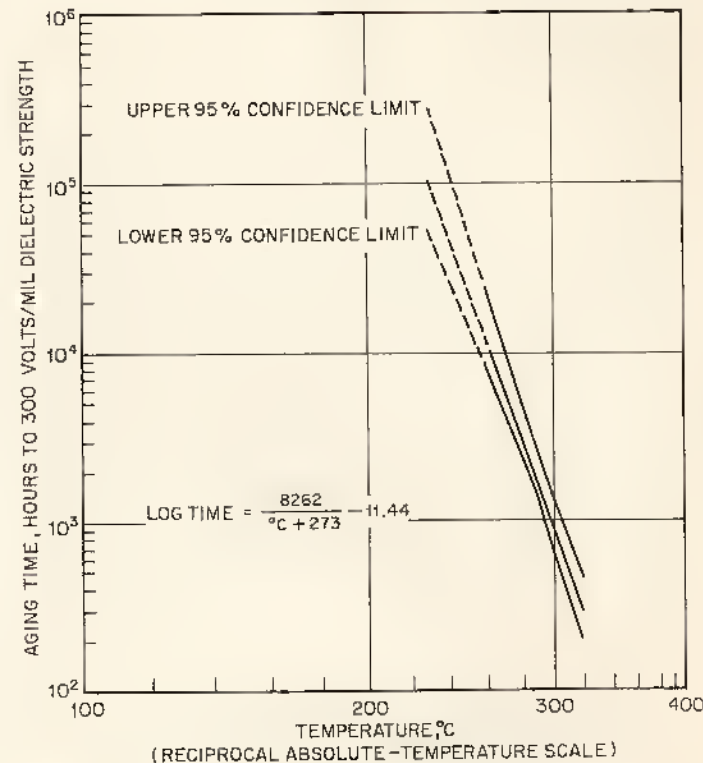


Fig. 25.19 Useful life vs. temperature for Nomex® 0.010-in. type 410 nylon paper.

normal symmetrical confidence limits for the measured electrical and mechanical properties into limits for time as a function of temperature results in the asymmetrical curves shown in the figures. Extrapolation of the data to lower temperatures (dashed lines)

TABLE 25.8 Dimensional Stability of Nomex® Type 410 Nylon Paper to Changes in Relative Humidity

Relative humidity, %	0.003-in Paper			0.010-in Paper		
	Regain, %	Expansion, %		Regain, %	Expansion, %	
		MD	XD		MD	XD
Oven dry	0	0	0	0	0	0
50	2.9	0.4	0.5	3.5	0.4	0.5
65	4.9	0.6	0.5	5.1	0.6	0.9
95	7.7	0.9	1.6	8.4	1.1	1.8

MD = machine direction
XD = cross machine direction

indicates that Nomex nylon paper can be expected to maintain a dielectric strength of at least 300 V/mil, as well as approximately 50 percent of its initial tensile strength and break elongation, after 10 years of continuous service at 220°C. Thus the indicated long-term temperature resistance of Nomex paper exceeds the minimum requirements for class H materials (continuous operation at 180°C).

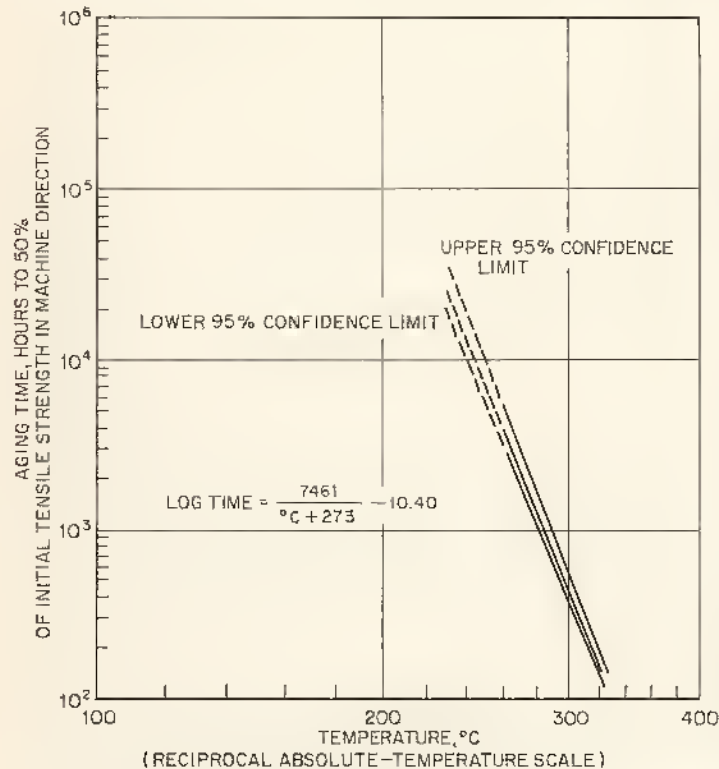


Fig. 25.20 Useful life vs. temperature for Nomex® 0.010-in, type 410 nylon paper.

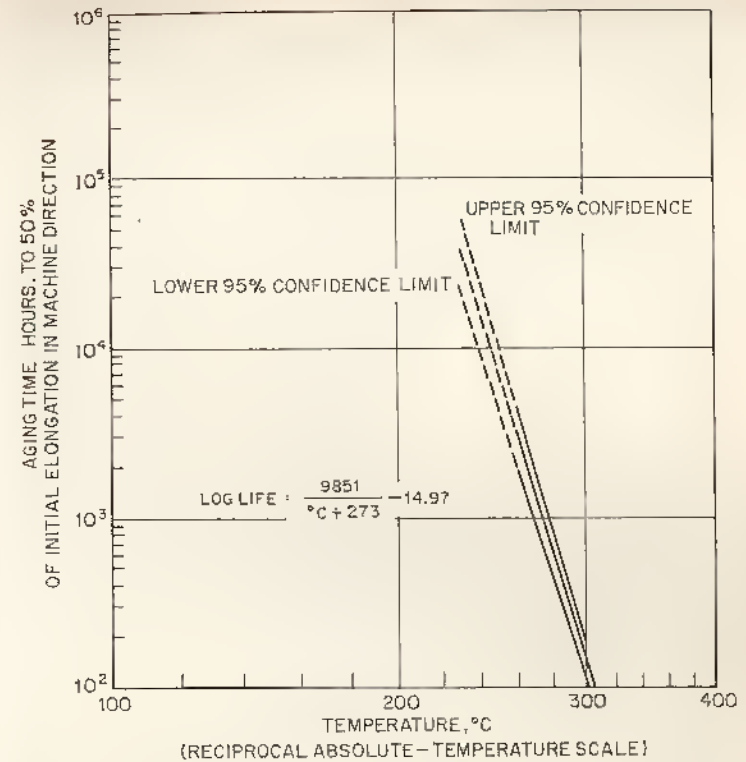


Fig. 25.21 Useful life vs. temperature for Nomex® 0.010-in, type 410 nylon paper.

d. **Chemical Resistance.** The basic chemical structure which is responsible for the thermal stability of Nomex nylon paper also gives the paper outstanding stability and resistance to attack in the presence of chemicals. Table 25.9 shows the effects of some commonly encountered acids, alkalis, and halogenated hydrocarbons at room temperature and elevated temperatures on the tensile strength of Nomex.

Exposure of Nomex nylon paper to the common solvents such as ketones, alcohols, and hydrocarbons has no effect on the mechanical properties of the paper. In addition, the solvents have no discernible effect upon the dimensional stability of the paper either at room temperature or after subsequent exposure at 285°C.

TABLE 25.9 Chemical Resistance of Nomex® Nylon

Chemical	Temperature, °C	Exposure time	Tensile strength retained, %
70% H ₂ SO ₄	21	200 h	100
70% H ₂ SO ₄	95	8 h	50
70% HNO ₃	21	100 h	50
10% NaOH	21	100 h	100
10% NaOH	95	8 h	50
Freon F-11®, F-12®, F-22®	25	150 days	100
Freon F-11	150	150 days	95
Freon F-12	150	150 days	100
Freon F-22	150	150 days	80
Askarel	200	13 days	100

TYPE H KAPTON POLYIMIDE FILM

The polyimides are a class of chemical compounds. The word "imide" refers to the cyclic structure characterized by a double-bond linkage between one nitrogen atom (N) and two carbonyl groups. In a polymer the latter are usually attached to a benzene ring as shown in Fig. 25.22. A polyimide which is presently commercially available is obtained by a polycondensation reaction between an aromatic tetracarboxylic acid anhydride and an aromatic diamine which yields a polyamic acid intermediate. The acid is then dehydrated to yield a high-molecular-weight polyimide. Many variations and properties are possible by using a variety of amino compounds. In the case of Kapton, * 4,4' diamine diphenyl ether, R is used as the aromatic amine. The reaction proceeds as shown in Fig. 25.23. The resultant compound, in film form, is transparent, amber-colored, flexible, flame-resistant, and infusible (chars at 1500°F). It possesses excellent chemical resistance (except for strong inorganic acids and bases), and it has a useful operating range

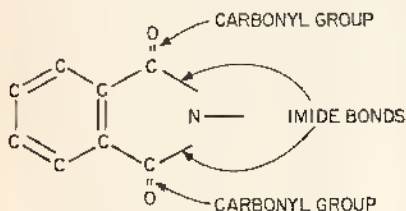


Fig. 25.22 Polyimide structure.

from -430 to 780°F. Its dielectric properties, such as dielectric strength and dissipation factor, are excellent and remain constant over a wide range of temperature and frequency.

The development of polyimide film materials has provided the necessary impetus to the design and manufacture of FPW devices. Shortly after and coincidentally with the introduction of this excellent engineering thermoplastic, FPW took a big jump in applications varying from complicated aerospace computers to camera interconnections. The properties of Kapton polyimide film are as indicated below.

a. Physical and Thermal Properties. The polyimide films retain their physical properties over a wide temperature range. They have been used in field applications in which the environmental temperatures have been as low as -269°C and as high as 400°C, but most of the data fall in the -195 to 200°C range. Summaries of the physical and thermal properties of Kapton polyimide film are given in Tables 25.10 and 25.11, respectively.

*Kapton is a registered trademark of E. I. du Pont de Nemours & Company.

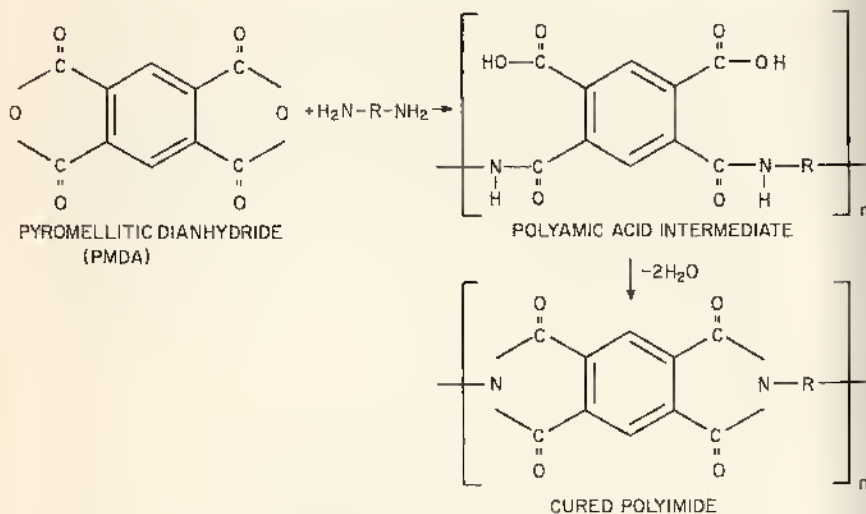


Fig. 25.23 Polyimide formation reaction.

TABLE 25.10 Typical Physical Properties of Kapton Polyimide Film

Property	Typical values at			Test method
	-195°C	25°C	200°C	
Ultimate tensile strength, MD	35,000 psi	25,000 psi	17,000 psi	ASTM D-882-64T
Yield point, MD		10,000 psi at 3%	6000 psi at 3%	ASTM D-882-64T
Stress to produce 5% elongation, MD		13,000 psi	8500 psi	ASTM D-882-64T
Ultimate elongation, MD	2%	70%	90%	ASTM D-882-64T
Tensile modulus, MD	510,000 psi	430,000 psi	260,000 psi	ASTM D-882-64T
Impact strength		6 kg-cm/in × 10 ⁻³		Du Pont Pneumatic Impact Test
Folding endurance (MIT)		10,000 cycles		ASTM D-2176-63T
Tear strength, propagating (Elmendorf)		8 g/in × 10 ⁻³		ASTM D-1922-61T
Tear strength, initial (Graves)		510 g/in × 10 ⁻³		ASTM D-1004-61
Bursting test (Mullen)		75 psi		ASTM D-774-63T
Density		1.42 g/cc		ASTM D1505-63T
Coefficient of friction kinetic (film-to-film)		0.42		ASTM D-1894-63
Refractive index (Becke line)		1.78		Encyclopaedic Dictionary of Physics, Volume I

TABLE 25.11 Typical Thermal Properties of Kapton® Polyimide Film

Property	Typical values	Test condition	Test method
Melting point	None		
Zero strength temperature	815°C	20-psi load for 5 s	Hot bar (Du Pont test)
Coefficient of linear expansion	2.0×10^{-5} in/(in°C)	(-) 14°C to 38°C	ASTM D-696-44
Coefficient of thermal conductivity	3.72×10^{-4} cal·cm/(cm ²)(s)(°C)	25°C	Model TC-1000
	3.89×10^{-4} cal·cm/(cm ²)(s)(°C)	75°C	Twin Heatmaster Comparative Tester
	4.26×10^{-4} cal·cm/(cm ²)(s)(°C)	200°C	
	4.51×10^{-4} cal·cm/(cm ²)(s)(°C)	300°C	
Specific heat	0.261	cal/(g)(°C)	Differential calorimetry
Flammability	Self-extinguishing		
Heat-sealable	No		

b. Tensile Stress-Strain. A series of tensile stress-strain curves for 0.001-in polyimide film as a function of temperature is shown in Fig. 25.24.

The usual values of tensile strength, tensile modulus, and ultimate elongation at various temperatures can be obtained from the typical stress-strain curves shown. Such properties as tensile strength and modulus have an inverse relation with temperature, and elongation peaks to a maximum value at 200 to 250°C. Other factors such as humidity, film thickness, and Instron elongation rate were found to have only a negligible effect on the shape of the 25°C curve.

c. Hydrolytic Stability. Kapton polyimide film is made by a condensation reaction; therefore, its properties are affected by water. Although long-term exposure to boiling water, as shown in Figs. 25.25 and 25.26, will reduce the level of film properties, sufficient tensile and elongation remain to ensure good mechanical performance. A decrease in the temperature and the water concentration will reduce the rate of Kapton property reduction, and higher temperatures and pressures will increase it.

d. Dimensional Stability. The dimensional stability of Kapton polyimide film depends on two factors: the normal coefficient of thermal expansion, and the residual stresses placed in the film during manufacture. The latter cause Kapton to shrink on its first exposure to elevated temperatures as indicated in the bar graphs in Fig. 25.27. Once the

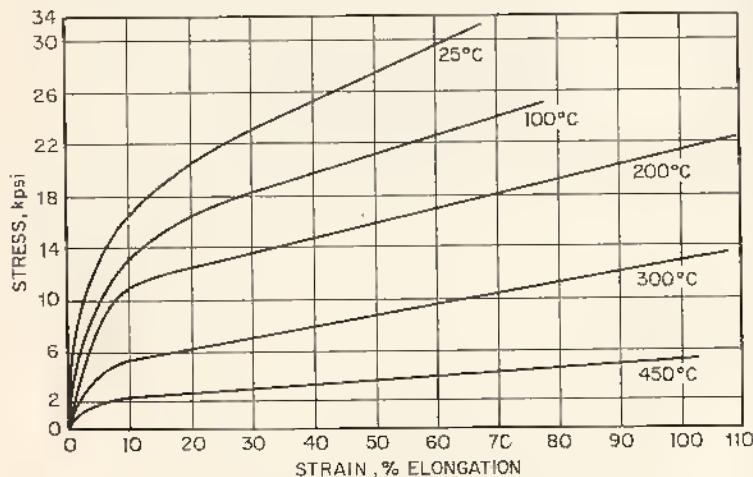


Fig. 25.24 Tensile stress-strain curves of Kapton® polyimide film.

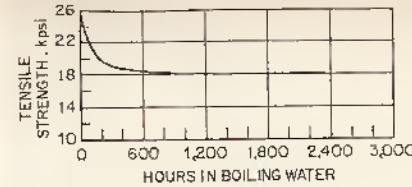


Fig. 25.25 Tensile strength of Kapton® polyimide film after exposure to water at 100°C.

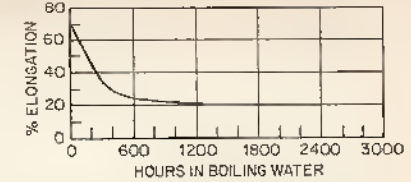


Fig. 25.26 Ultimate elongation of Kapton® 0.001-in, type H polyimide film after exposure in water at 100°C.

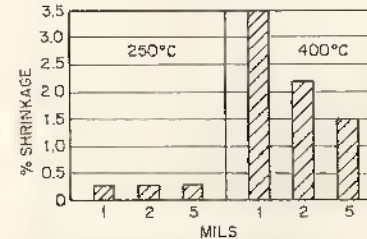


Fig. 25.27 Residual shrinkage vs. exposure temperature and gauge for 0.001-in, type H Kapton® polyimide film.

TABLE 25.12 Thermal Coefficient of Expansion of Kapton® Polyimide Film Thermally Exposed

Temperature range, °C	K, in/(in)(°C) × 10 ⁶
23-100	1.80
100-200	3.10
200-300	4.85
300-400	7.75
23-400	4.55

film has been exposed, the normal values for thermal expansion listed in Table 25.12 can be expected.

The dimensional changes of the film as a function of temperature and orientation and after etching in ferric chloride are shown in Tables 25.13 and 25.14. The shrinkage rates of

TABLE 25.13 Dimensional Changes of Kapton® Polyimide Film at 150°C and After Etching

Conditioning	0.001-in film				0.002-in film			
	Mean		2σ		Mean		2σ	
	MD	XD	MD	XD	MD	XD	MD	XD
150°C for 30 min	1.01	1.18	0.29	0.48	1.14	1.29	0.32	0.32
150°C for 30 min + recycle 150°C for 30 min	0.10	0.09	0.18	0.16	0.10	0.08	0.12	0.14
Etchant (FeCl ₃)	0.47	0.56	0.16	0.32	0.48	0.51	0.16	0.26
150°C for 30 min + etchant	0.14	0.12	0.26	0.24	0.18	0.16	0.10	0.24

MD = machine direction
XD = cross machine direction

TABLE 25.14 Dimensional Changes of Kapton® Polyimide Film at 250°C and After Etching

Conditioning	0.001-in film				0.002-in film			
	Mean		2σ		Mean		2σ	
	MD	XD	MD	XD	MD	XD	MD	XD
250°C for 30 min	2.17	2.30	0.46	1.01	2.26	2.26	0.48	0.80
250°C for 30 min + recycle 150°C for 30 min	-0.04	-0.03	0.12	0.10	0.00	0.02	0.08	0.16
Etchant (FeCl ₃)	0.47	0.56	0.16	0.32	0.48	0.51	0.16	0.26
250°C for 30 min + etchant	0.06	0.01	0.16	0.24	0.07	0.08	0.16	0.18

MD = machine direction
XD = cross machine direction

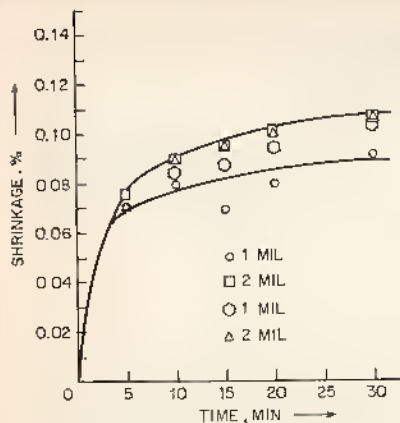


Fig. 25.28 Shrinkage rate of Kapton® polyimide film at 150°C in machine direction.

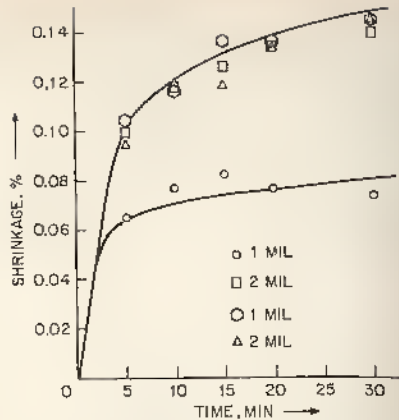


Fig. 25.29 Shrinkage rate of Kapton® polyimide film at 150°C in cross-machine direction.

Kapton in the machine direction (MD) and cross machine direction (XD) at 150°C are shown in Figs. 25.28 and 25.29. The dimensional change of Kapton film in the MD and XD as a function of relative humidity is shown in Fig. 25.30.

4. Cut-through and Cold-flow Film Most organic films exhibit a tendency to flow or thin out under high compressive stresses, especially at elevated temperatures. Kapton polyimide film possesses an extremely high resistance to such stresses. Test procedures described in ASTM D-876-61 have been adapted to flat films to provide the data in Figs. 25.31 and 25.32. Stresses range from an infinitely high point load to 12,000 psi at cut-through for a 0.001-in film.

Kapton polyimide film is subject to oxidative degradation. Therefore, its useful life is a function of both temperature and oxygen concentration in the test environment. The effect of those factors is shown in Figs. 25.33 to 25.36 and Table 25.15.

5. Electrical Properties of Kapton Polyimide Film The most common electrical properties of Kapton polyimide film in various gauges are given in Table 25.16. They are measured at 25°C and 50 percent relative humidity. The effects of such factors as humidity, temperature, and frequency on these basic values are shown in forthcoming tables.

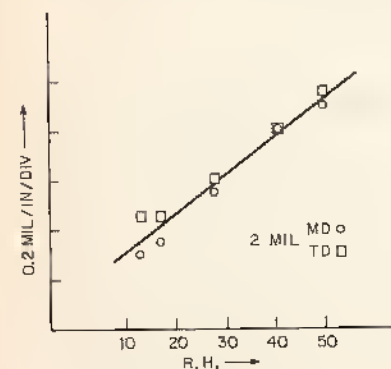


Fig. 25.30 Dimensional change of Kapton® polyimide film vs. relative humidity.

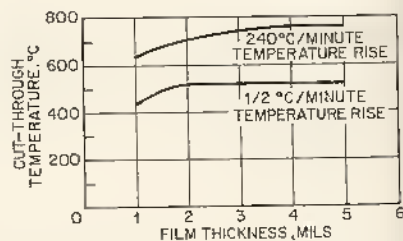


Fig. 25.31 Cut-through temperature of Kapton® polyimide film vs. rate of temperature rise and gauge.

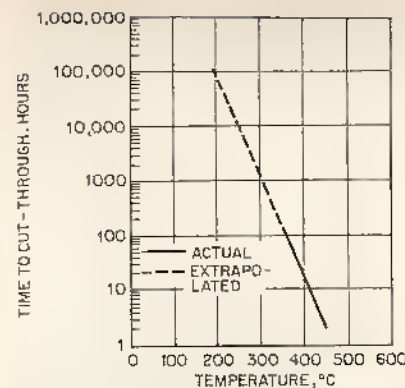


Fig. 25.32 Resistance to cut-through temperature of Kapton® polyimide film vs. temperature.

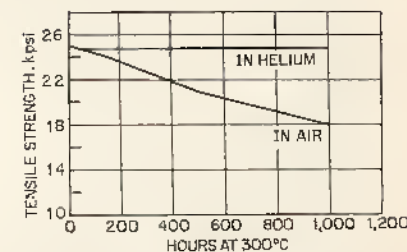


Fig. 25.33 Tensile strength of 0.001-in Kapton® polyimide film vs. aging at 300°C.

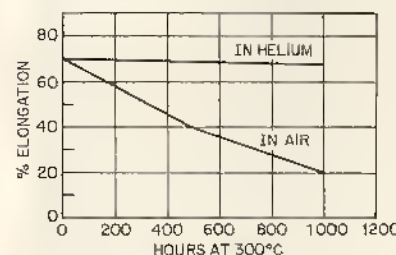


Fig. 25.34 Ultimate elongation of 0.001-in Kapton® polyimide film vs. aging at 300°C.

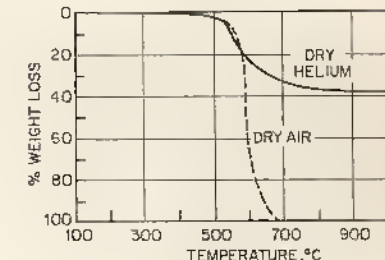


Fig. 25.35 Weight loss of 0.001-in Kapton® polyimide film at a temperature rise of 3°C/min.

a. Effect of Humidity. Because the water content of Kapton polyimide film can affect the film's electrical properties, electrical measurements are presented for 0.001-in film after

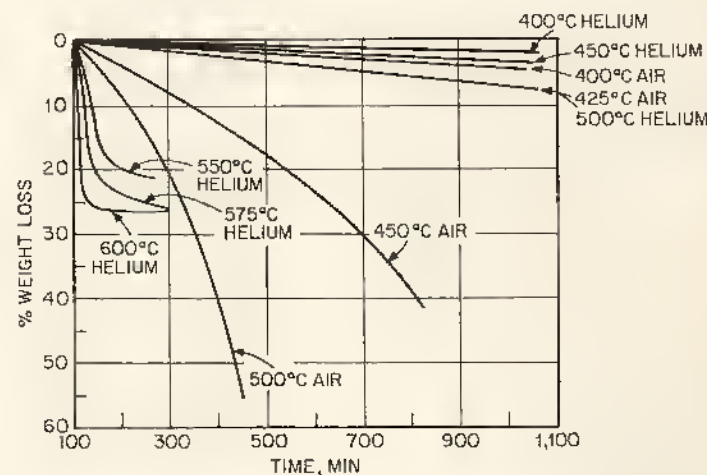


Fig. 25.36 Isothermal weight loss of 0.001-in Kapton® polyimide film.

exposure to environments of various relative humidities at 25°C. The amount of moisture absorbed in Kapton polyimide film depends upon the relative humidity, as shown in Fig. 25.37. Table 25.17 shows the effect of humidity on the electrical properties of Kapton polyimide film. The effect of temperature on the electrical properties of Kapton polyimide film is shown in Figs. 25.38 to 25.41.

b. Effect of Frequency. The effects of frequency on the value of the dielectric constant and dissipation factor of Kapton polyimide film at various isotherms are shown in Figs. 25.42 and 25.43.

TABLE 25.15 Time Required for Reduction in Ultimate Elongation from 70 to 1 Percent of 0.001-Inch Kapton® Polyimide Film

Temperature, °C	Environment	
	Air	Helium
450	2 h	22 h
425	5 h	3½ days
400	12 h	2 weeks
375	2 days	2 months
350	6 days	1 year
300	3 months	
275	1 year	
250	8 years	

6. Conditioning Prior to Laminating Kapton Polyimide Film In making and processing copper laminates and flexible printed wiring, it must be remembered that, under normal plant conditions, Kapton contains small amounts of moisture absorbed from the atmosphere. The moisture does not have a major effect on film properties, but it must be eliminated during high-temperature manufacturing and processing steps to avoid blisters and delamination in laminates and circuits.

During lamination to copper and any subsequent high-temperature processing steps,

TABLE 25.16 Typical Electrical Properties of Kapton® Polyimide Film

Property	Typical value	Test condition	Test method
Dielectric strength	0.001 in	7000 V/in × 10 ⁻³	60 cycles
	0.002 in	5400 V/in × 10 ⁻³	¼ in. electrodes
	0.003 in	4600 V/in × 10 ⁻³	
	0.005 in	3600 V/in × 10 ⁻³	
Dielectric constant	0.001 in	3.5	1 kHz
	0.002 in	3.6	
	0.003 in	3.7	
	0.005 in	3.7	
Dissipation factor	0.001 in	0.0025	1 kHz
	0.002 in	0.0025	
	0.003 in	0.0025	
	0.005 in	0.0027	
Volume resistivity	0.001 in	1 × 10 ¹⁸ Ω·cm	125 V
	0.002 in	8 × 10 ¹⁷ Ω·cm	
	0.003 in	5 × 10 ¹⁷ Ω·cm	
	0.005 in	1 × 10 ¹⁷ Ω·cm	
Corona threshold voltage	0.001 in	465 V	60 cycles
	0.002 in	550 V	¼ in. electrodes
	0.003 in	630 V	
	0.005 in	800 V	
	0.005 in H, 0.002 in FEP,		
	0.005 in H, 0.0005 varnish	1600 V	

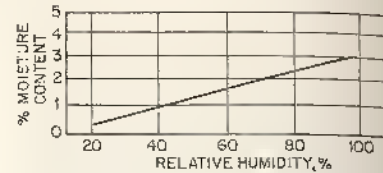


Fig. 25.37 Moisture content of Kapton® polyimide film vs. typical relative humidity values at 23°C.

TABLE 25.17 Relative Humidity vs. Electrical Properties of Kapton® Polyimide Film

Relative humidity, %*	AC dielectric strength, V/in × 10 ⁻³	Dielectric constant	Dissipation factor
0	7800	3.0	0.0018
30	7300	3.3	0.0021
50	7000	3.5	0.0025
80	6500	3.7	0.0037
100	6200	3.9	0.0047

* For calculations involving absolute water content, 50% RH in our study is equal to 1.3% water in the film and 100% RH is equal to 2.9% water, the maximum absorption possible regardless of the driving force.

moisture must be dried from Kapton to avoid blistering and delamination of the film from the copper. Under severe high-temperature lamination condition, embrittlement of the Kapton can also occur. The rate at which moisture is absorbed or dried from Kapton depends on the temperature, relative humidity, film thickness, and adhesive coating. Sheets of uncoated Kapton type II conditioned at 23°C, 100 percent RH and then placed in a 23°C, 0 percent RH dry box lost moisture at the rates shown in Table 25.18.

7. Type F Polyimide Film Since the polyimide film, Kapton II, is non-heat-sealable, although classified as a thermoplastic, a film combining the properties of Teflon, FEP, and Kapton polyimide was developed. The combination film possesses the basic properties of FEP and polyimide and has the added advantage of being heat-sealable. That feature

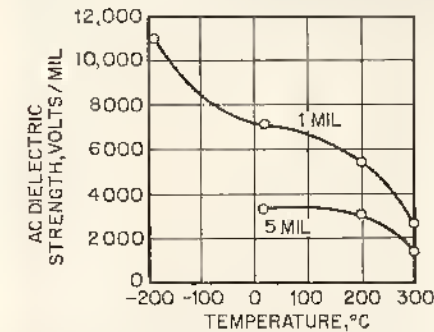


Fig. 25.38 AC dielectric strength of 0.001-in Kapton® polyimide film vs. temperature.

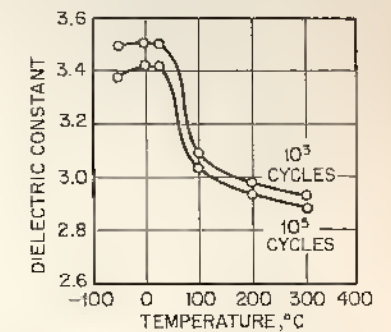


Fig. 25.39 Dielectric constant of 0.001-in Kapton® polyimide film vs. temperature.

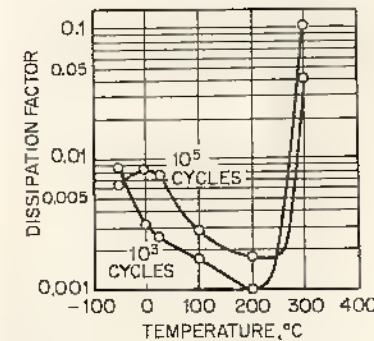


Fig. 25.40 Dissipation Factor of 0.001-in Kapton® polyimide film vs. temperature.

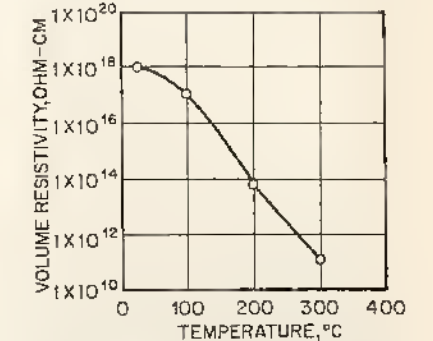


Fig. 25.41 Volume resistivity of 0.001-in Kapton® polyimide film vs. temperature.

permits the direct bonding of Kapton F to itself or to other thermally stable dielectrics.

TABLE 25.18 Rate of Drying of Kapton® Polyimide Film at 23°C

Film thickness, in	Time for half of moisture to leave film, min
0.001	3
0.002	10
0.003	21
0.005	54

At present there are six Kapton, type F, polyimide-FEP films available. The combinations and their Du Pont code identification numbers are listed in Table 25.19. The physical, thermal, chemical and electrical properties of Kapton, type F are summarized in Table 25.20.

Films should be laminated as soon as possible after drying, since reabsorption of moisture by Kapton polyimide film is fairly rapid. For example, sheets of dried one-side-coated Kapton can lie left in a 70 percent RH atmosphere for up to only about 10 min before enough moisture is reabsorbed to cause blisters during lamination. Two-side-coated film reabsorbs enough moisture to cause blistered laminates after 30 to 60 min. It

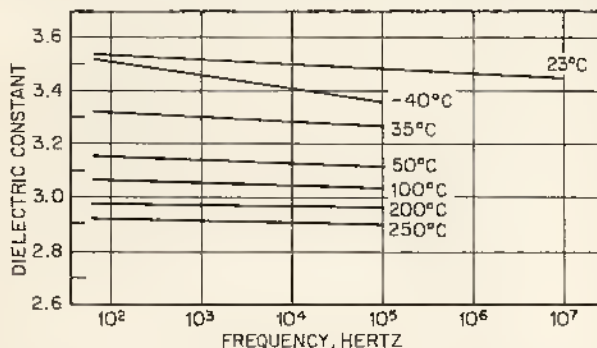


Fig. 25.42 Dielectric constant vs. frequency for 0.001-in Kapton® polyimide film.

TABLE 25.19 Commercially Available Constructions of Kapton® Type F Film

Code ident. No.*	Nominal thickness, in	Nominal construction in $\times 10^{-3}$				Area factor ft ² /lb
		Teflon FEP	Kapton type H	Teflon FEP	Standard widths, in†	
150F019	0.0015	0	1	½	¾-6	77
200F919	0.002	½	1	½	¾-6	54
200F011	0.002	0	1	1	¾-6	54
300F929	0.003	½	2	½	¾-6	39
300F021	0.003	0	2	1	¾-6	39
400F022	0.004	0	2	2	¾-6	27

* Since a number of polyimide Teflon combinations add up to the same total gauge, it is necessary to distinguish between them. In the three-digit system used, the middle digit represents the nominal thickness of the base Kapton in mils. The first and third digits represent the nominal thickness of the Teflon FEP in mils. The symbol used to represent ½ mil is 9. Example: 300F021 is a 3-mil structure consisting of 2-mil base film with 1 mil of Teflon FEP on one side. 300F929 is a 3-mil structure consisting of 2-mil base film with ½-mil coatings of Teflon FEP on both sides.

† Wider widths are available.

TABLE 25.20 Properties of Kapton® Type F Film

Property	019	919	011	929	021	022
Physical and thermal properties						
Ultimate tensile strength, MD, psi	17,000	14,000	14,000	17,000	17,000	14,000
25°C	11,000	9,000	9,000	11,000	11,000	9,000
200°C						
Yield point, MD, at 3%, psi	7300	5900	5900	7300	7300	5900
25°C	4000	3000	3000	4000	4000	3000
200°C						
Stress at 5% elongation, MD, psi	9000	7500	7500	9000	9000	7500
25°C	5500	4000	4000	5500	5500	4000
200°C						
Ultimate elongation, MD						
25°C	75%	90%	75%	>80%	90%	105%
200°C	85%	106%	95%		105%	115%
Tensile modulus, MD, psi						
25°C	320,000	250,000	250,000	320,000	320,000	250,000
200°C	173,000	130,000	130,000	173,000	173,000	130,000
Impact strength at 25°C						
kg-cm	7	8	8	14	14	16
kg-cm/in $\times 10^{-3}$	4.6	4	4	4.6	4.6	4
Tear strength—propagating (Elmendorf)						
g/in $\times 10^{-3}$	20	30	60	51	57	40
g	43	60	60	43	43	60
Tear strength—initial (Graves)						
g	650	800				
g/in $\times 10^{-3}$	435	400				
Weight % polyimide	57	40	40	51	57	40
Weight % FEP	43	60	60	43	43	60
Density	1.67	1.79	1.79	1.67	1.67	1.79
Chemical properties						
Moisture absorption at 25°C						
50% RH	0.8%	0.4%	0.4%	0.8%	0.8%	0.4%
98% RH	1.7%	1.2%	1.2%	1.7%	1.7%	1.2%
Water vapor permeability						
g/(100 in ²)(24 h)	0.57	0.31	0.31	0.57	0.57	0.31
g/(100 in ²)(24 h)/(in $\times 10^{-2}$)	0.85	0.62	0.62	0.85	0.85	0.62
Electrical properties						
Short-term dielectric strength						
Total V	6,300	9,000	8,000	11,400	10,600	12,700
V/in $\times 10^{-3}$	4,200	4,500	4,000	3,800	3,500	3,900
Dielectric constant	3.0	2.7	2.7	3.1	3.1	2.8
Dissipation factor	0.0014	0.0011	0.0011	0.0011	0.0011	0.0008
Volume resistivity, Ω -cm						
25°C	10 ¹⁶	10 ¹⁶	10 ¹⁶	8 $\times 10^{16}$	7 $\times 10^{17}$	7 $\times 10^{17}$
200°C	10 ¹⁴	10 ¹⁴	10 ¹⁴	9 $\times 10^{15}$	7 $\times 10^{16}$	8 $\times 10^{16}$

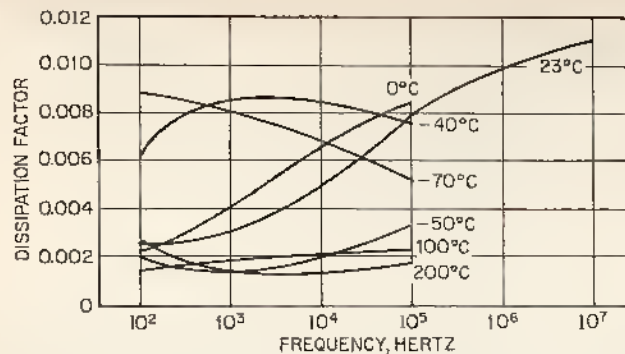


Fig. 25.43 Dissipation factor vs. frequency for 0.001-in Kapton® type H polyimide film.

may be feasible to dry full rolls of Kapton polyimide film prior to processing. Limited test data indicate 250°F for one or two days may be satisfactory to dry film in roll form.

It has also been found that the amount of blistering in laminates to copper depends on the time taken to bring the film up to laminating temperature, the release sheet used, and the copper treatment and whether the laminate is cooled under pressure. In some cases laminating techniques have produced blister-free copper laminates and printed circuits without predrying the film. In those cases, predrying is not needed unless problems are encountered with blistering.

Table 25.21 shows the drying conditions necessary for the indicated constructions of Kapton type F film. Table 25.22 presents the minimum times required to dry copper-clad laminates of Kapton type F film, prior to dip soldering, to prevent blistering.

TABLE 25.21 Drying Conditions for Kapton® Type F Prior to Lamination to Copper

Drying temperature, °F	Drying time, min, for	
	400F031 0.003 in H, 0.001 in FFP	500F131 0.001 in FEP, 0.003 in H, 0.001 in FEP
250	5	45
300		30
350	1	20
400		10

TABLE 25.22 Drying Conditions for Kapton® Type F Prior to Dip Soldering

Drying temperature, °F	Drying time, min, for	
	400F031—Copper 0.003 in H, 0.001 in FEP, 1 oz copper	500F131—Copper 0.001 in FEP, 0.003 in H, 0.001 in FEP, 1 oz copper
250	15	40
300	10	20
350	6	10
400	2	4

POLYPARABANIC ACIDS

A new class of polymers referred to as polyparabanic acids (PPA) has been developed by Exxon Chemical Company. Although as of this writing the materials are not commercially available, their properties will be presented. The materials have many potential applications in the fabrication of flexible printed wiring. PPA polymers are true thermoplastics

of the general structure shown in Fig. 25.44. The polymers are essentially amorphous, and can be cast into film having excellent mechanical properties and thermal resistance.

Many of the properties of PPA-E and PPA-M are essentially the same, but each material has certain advantages. Both materials are noncrystalline and have high glass transition temperatures that enable them to maintain good mechanical properties up to 500°F. Properties which apply, independently of fabricated forms, are given in Table 25.23. As shown in Table 25.24, both PPA-E and PPA-M are soluble in dipolar aprotic solvents. PPA-M is soluble in several other types of solvent. Mixed solvent-nonsolvent systems, such as a 60-40 mixture of cyclohexanone and methyl ethyl ketone for PPA-M, can also be used.

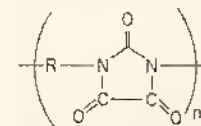


Fig. 25.44 General chemical structure of polyparabanic acids where R = aromatic, aliphatic, alicyclic, etc.

TABLE 25.23 Physical Properties of Polyparabanic Acids

Property	PPA-M	PPA-E
Glass transition	554°F	570°F
Specific gravity, g/mL	1.30	1.38
Crystallinity	None	None
Color	Amber	Amber
Clarity	Transparent	Transparent
Refractive index	1.65	1.65

Figure 25.45 shows typical stress-strain curves from low to high temperatures. The -50°F curve still displays a yield point characteristic of tough plastics, and films remain flexible in liquid nitrogen. At the other extreme, PPA's strength at 500°F is comparable with that of polytetrafluoroethylene at room temperature.

8. Mechanical Properties of PPA Films Typical mechanical properties of PPA films are compared with those of commercial films in Table 25.25. A rigid aromatic backbone and high softening point also enable these resins to maintain a high degree of mechanical integrity at elevated temperatures, as illustrated in Figs. 25.46 to 25.48. Dimensional stability as shown by creep (Fig. 25.49) is very good.

TABLE 25.24 Solvents for and Chemical Resistance of Polyparabanic Acids

Solvent	PPA-M	PPA-E
Dimethylformamide	S*	S
Dimethylacetamide	S	S
Dimethylsulfoxide	S	S
Hexamethylphosphoramide	S	S
N-Methylpyrrolidone	S	S
Butyrolactone	S	†
Cyclohexanone	S	†
Cyclopentanone	S	†
Benzonitrile	S	†
Acetophenone	S	†
Isophorone	S	†
Epichlorohydrin	S	†
Dimethylsulfoxide	S	S
Dioxolane	S	†
Chemical agent		
Acids (nonoxidizing)	No effect	No effect
Bases	Slow attack	Slow attack
Hydrocarbons	No effect	No effect
Halogenated hydrocarbons	No effect	No effect
Ethers	No effect	No effect
Aliphatic ketones	Some swell	No effect

*S = soluble
†I = insoluble

TABLE 25.25 Comparison of Mechanical Properties of Films

Property	Temp., °p	PPA-M	PPA-E	Oriented* PET	Pf†	PTFE‡	ASTM test
Tensile yield, psi	75 400	14,000 4500	15,000 4500	12,000 <10,000	10,000-14,000 6000	2000 200	D-882-64T D-882-64T
Yield strain, %	75	8-9	8-9	4	3-4		D-882-64T
Ultimate elongation, %	75	10-20	15-60	60-165	35-70	300	D-882-64T
Tensile modulus, psi	75 400	330,000 250,000	380,000 260,000	550,000 >200,000	430,000 260,000	58,000 Very low	D-882-64T D-882-64T
Folding endurance, MIT, cycles	75	10,000	>200,000	>200,000	10,000		D-2176-63T
Tear strength, propagating, g/in × 10 ⁻³	75	7-10	7-10	12-27	8	10-100	D-1922-61T
Tear strength, initial, lb/in	75	900	1000	1000-3000	1100	270	D-1004-66T
Bursting test, Mullen, psi	75	20-30	30-40	30-80	75		D-774-63T
H ₂ O absorption, %, 24 h	75	1.9	1.1	<0.8	2.9	Nil	D-570-63
Oriented film, MD							
Tensile strength, psi	75	30,000	40,000	24-50,000			D-882-64T
Stress to produce 5% elongation, psi	400	14,000					
Tensile modulus, psi	75	18,000	21,000	13,000			D-882-64T
	75	650,000	700,000	550,000			

*Commercial polyethylene terephthalate
 †Commercial aromatic polyimide
 ‡Commercial polytetrafluoroethylene

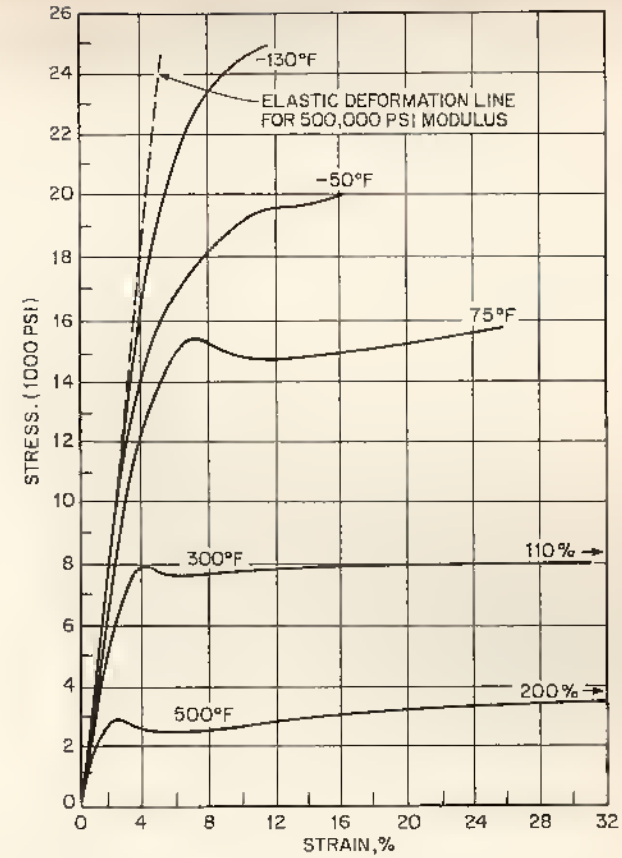


Fig. 25.45 Stress-strain curves of unoriented PPA films as a function of temperature (stress is based on the original cross section).

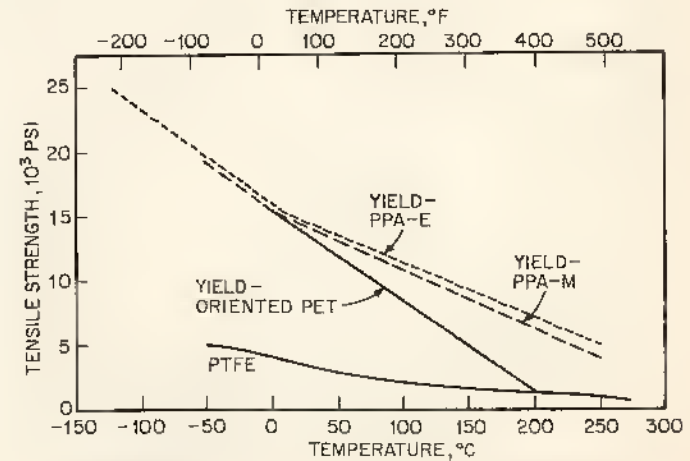


Fig. 25.46 Tensile strength of unoriented films vs. temperature.

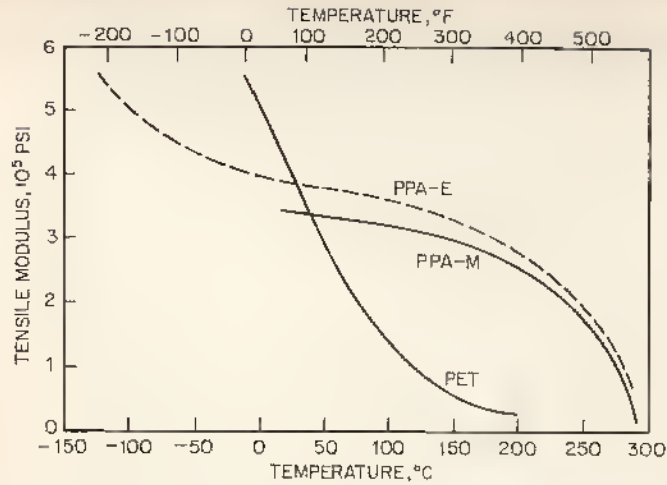


Fig. 25.47 Tensile modulus vs. temperature of unoriented films.

Another aspect of dimensional stability, thermal shrinkage, is shown in Fig. 25.50. Because of a lack of orientation and built-in stresses in the film, the permanent dimensional change on thermal cycling is extremely low. Such films are nearly isotropic.

9. Orientation of PPA Film Because of their large tensile elongations exhibited at elevated temperatures (Fig. 25.48) and their linear nature, the high-molecular-weight PPA resins

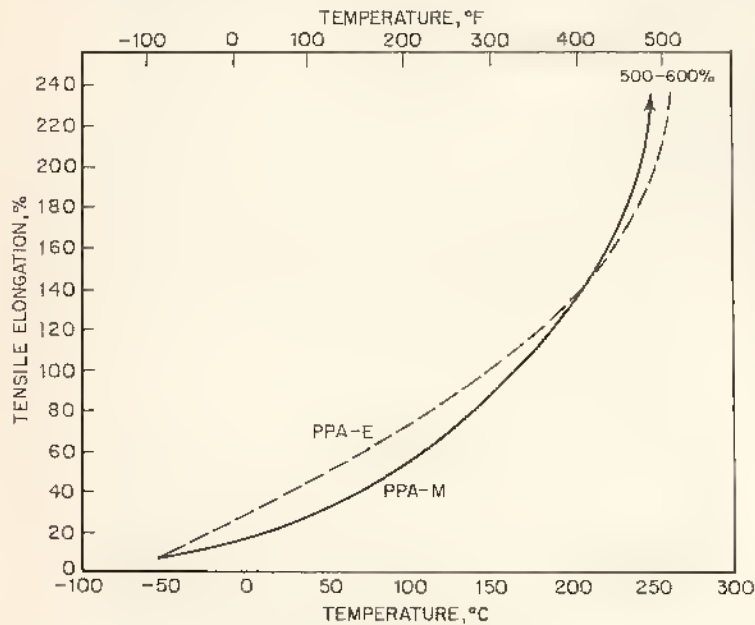


Fig. 25.48 Tensile strain vs. temperature of unoriented PPA films.

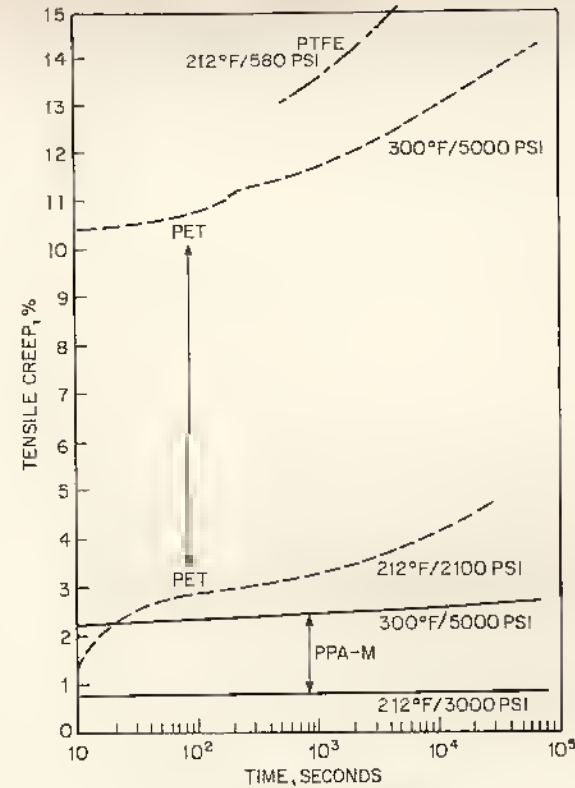


Fig. 25.49 Tensile creep of polymer films.

TABLE 25.26 Comparison of Thermal Properties of Films

Property	PPA-M	PPA-E	PET*	PI†	PTFE‡	ASTM method
Heat distortion temperature, °F (2% elongation)	520°	540°	—	—	—	TMA
Melting point, °F	None	None	480°	None	320	—
Coefficient of linear expansion × 10 ⁻⁶ in/(in)(°C)(at 125° F)	4.3	4.1	1.7	2.0	19	TMA
Coefficient of thermal conductivity, Btu·in/(h)(ft ²)(°F)	2.2	2.2	1.1	1.1	1.8	Cenco-Finch
Flammability, 0.005 in	Self-exting.	Self-exting.	Slow burning to self-exting.	Self-exting.	Nonflammable	D-1433-58
Shrinkage, % at 150°C	0.05	0.05	1.5	<0.3	2.2	—
Maximum short-term-use temperature, °F	550°	570°	~375°	>750°	570	—

*Commercial polyethylene terephthalate

†Commercial aromatic polyimide

‡Commercial polytetrafluoroethylene

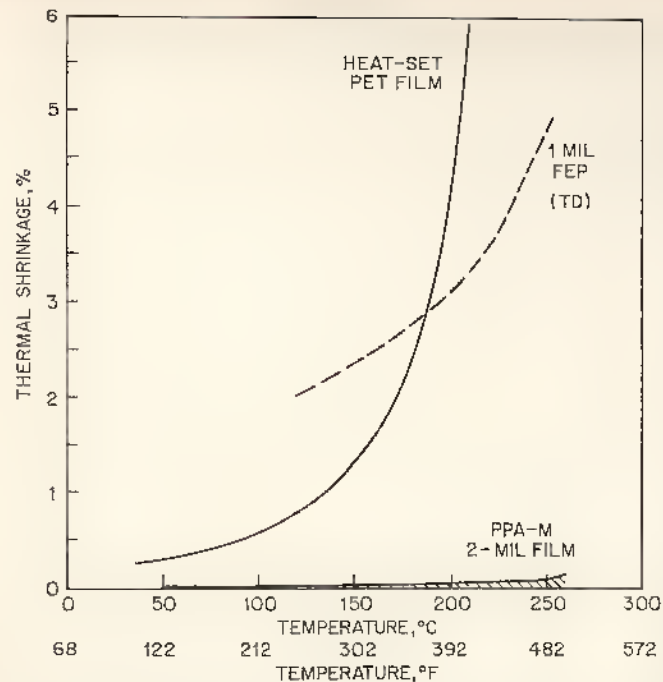


Fig. 25.50 Shrinkage of films as a function of temperature (30 min exposure at each temperature).

can be stretched in such a way as to produce highly oriented films. Orientation produces enhancement of tensile properties (Table 25.25), stiffness, improved flexibility, and greater toughness, but films retain low thermal shrinkage. Uniaxially oriented films also

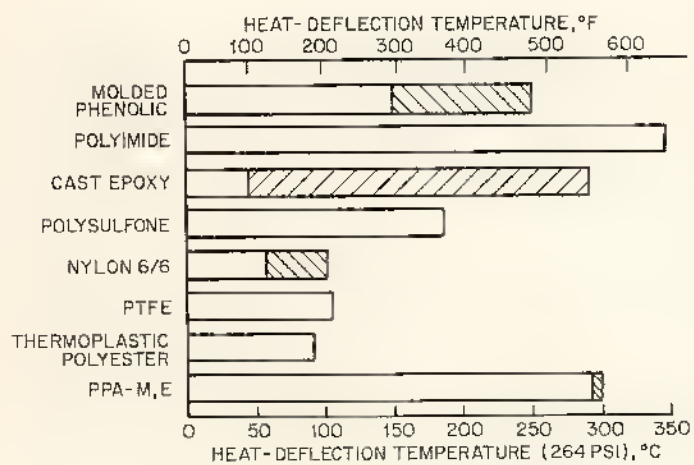


Fig. 25.51 Heat-deflection temperatures for various PPA polymers.

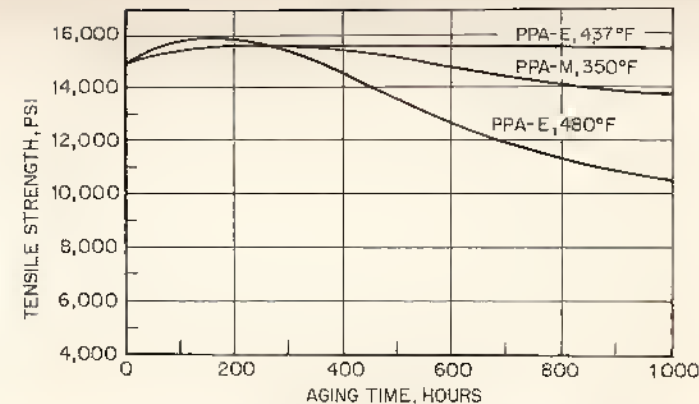


Fig. 25.52 Retention of strength during oven aging for PPA-E and PPA-M films. Samples were tested at 73°F.

possess low thermal expansion coefficients, 2×10^{-7} in/in·°C and lower. Oriented films typically have tensile strengths of over 40,000 psi and yet retain high clarity owing to the lack of crystallinity. Such films should find uses in applications requiring high strength, low shrinkage, or very low expansion coefficients such as flexible printed wiring applications.

10. Thermal Properties of PPA Film Thermal properties of PPA films are tabulated in Table 25.26. The high heat conductivity is especially important in mechanical or electrical application in which rapid heat dissipation is desirable. The heat-deflection temperature of PPA polymers is far above that of most amorphous and crystalline polymers and ranks with that of thermosets (Fig. 25.51). The high softening point gives films and laminates resistance to liquid solder baths up to 500°F and results in good resistance to cut-through in electrical applications.

Thermogravimetric curves in nitrogen show a high level of thermal stability. Long-term thermo-oxidative resistance is illustrated in Fig. 25.52. PPA-E possesses the greater oxidative resistance with a tensile strength retention of over 10,000 psi after 1000 h at 480°F. PPA-M is somewhat less stable than PPA-E in air. Room-temperature electrical properties of PPA-E, PPA-M, polyethylene terephthalate (PET), and polyimide (PI) films are presented for comparison in Table 25.27. PPA dielectric properties are similar to those of other aromatic, highly polar materials such as polyimide. PPA-E and PPA-M will probably fall between PET and PI in thermal ratings. Figures 25.53 to 25.56 give

TABLE 25.27 Electrical Properties of 0.001-inch Films at 25°C (50% RH)

Property	PPA-M	PPA-E	PET*	PI†	ASTM method
Dielectric constant, 10 ³ Hz	3.82	3.60	3.2	3.60	D-150-65T
Dissipation factor, 10 ³ Hz	0.0040	0.0027	0.0050	0.0028	D-150-65T
Dielectric strength V/in × 10 ⁻³ , air	6000	6000	7500	7000	D-149-64
Dielectric strength V/in × 10 ⁻³ , oil	6700	6100	—	—	D-149-64
Volume resistivity, Ω-cm	>10 ¹⁷	>10 ¹⁷	>10 ¹⁷	>10 ¹⁷	D-257-66
Surface resistivity, Ω	>10 ¹⁶	>10 ¹⁶	—	>10 ¹⁶	—
Arc resistance, s	125	125	—	182	D-495

* Commercial polyethylene terephthalate

† Commercial polyimide

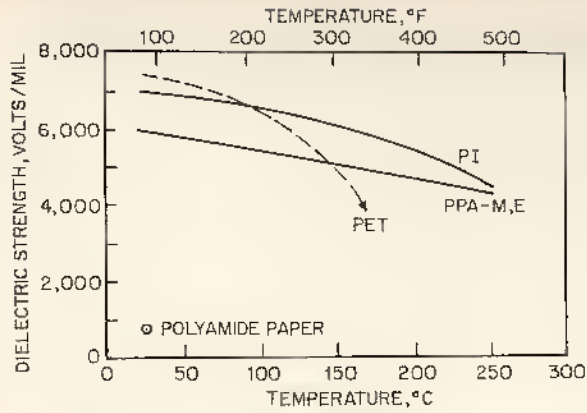


Fig. 25.53 AC dielectric strength vs. temperature for 0.001-in films at 50 percent relative humidity.

electrical properties vs. temperature up to 500°F. PPA film properties are much less affected by temperature than PET films are, and their temperature dependence is similar to that of PI film. PPA films combine high dielectric strength—4000 V/in $\times 10^{-3}$ at 500°F—with high mechanical strength.

The dissipation factors and dielectric constants over a range of frequencies are shown in Figs. 25.57 and 25.58. Dielectric constant remains steady to 10⁶ Hz and higher, and it is little affected by temperature. As with all polar resins, the dissipation factor does increase in that frequency range.

PPA films should be attractive because of their higher thermal capabilities relative to PET and lower cost relative to PI film. Stability toward solder baths without shrinking means that PPA films should perform well as substrates for flexible printed wiring.

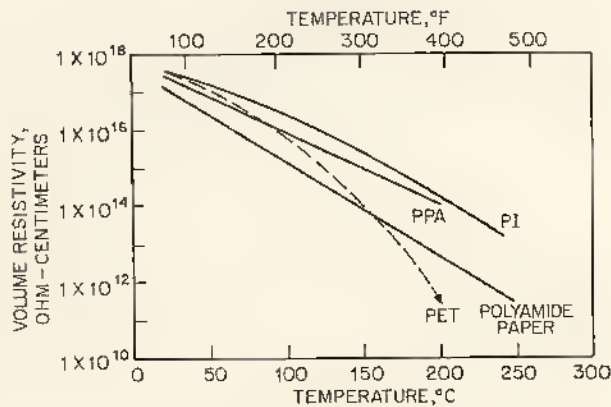


Fig. 25.54 Volume resistivity vs. temperature.

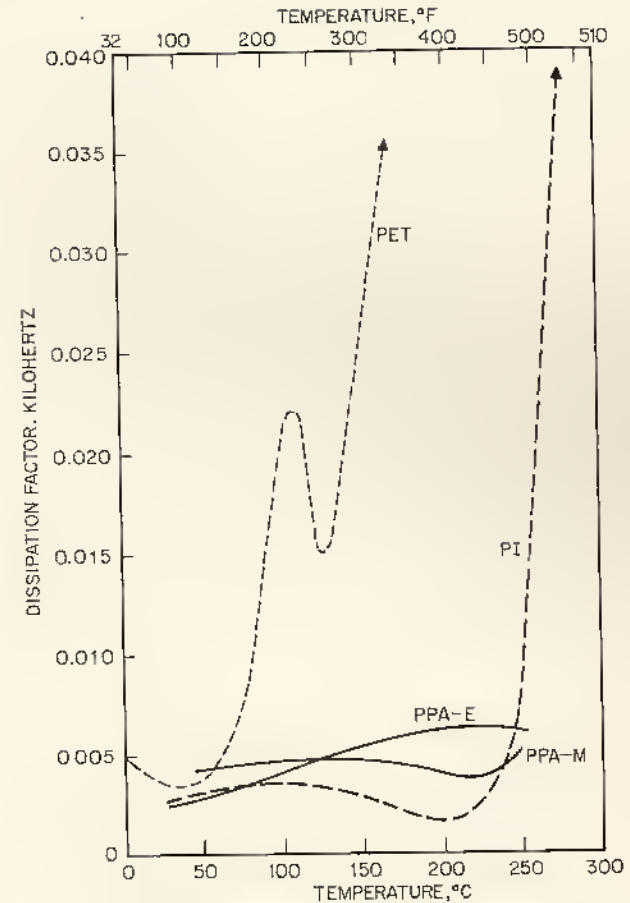


Fig. 25.55 Dissipation factor of films vs. temperature.

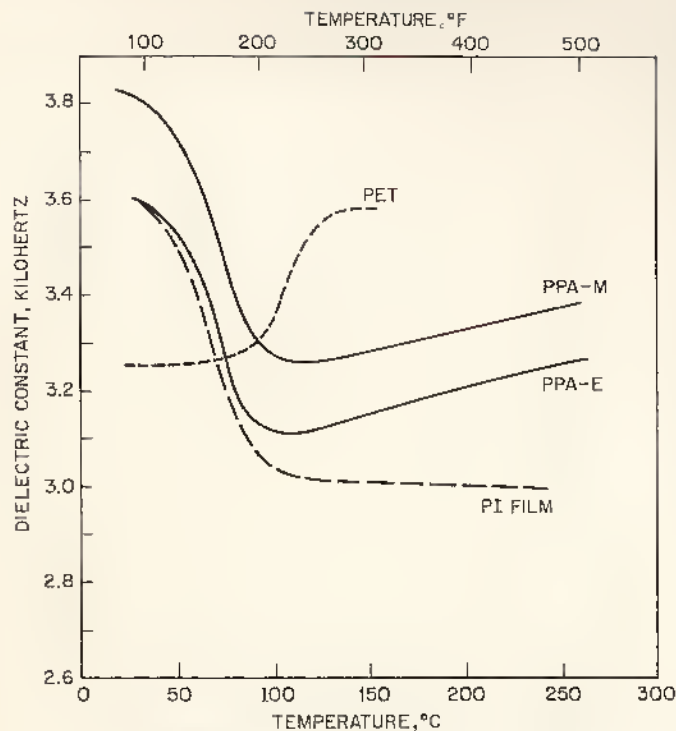


Fig. 25.56 Dielectric constant of films vs. temperature.

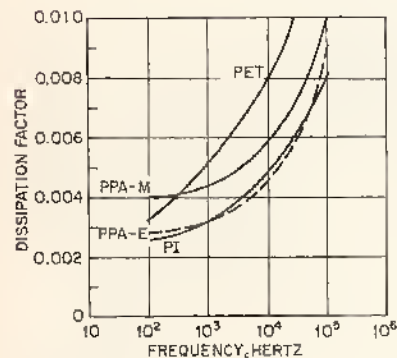


Fig. 25.57 Dissipation factor of films vs. frequency at 25°C.

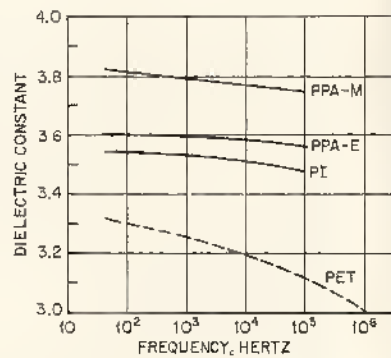


Fig. 25.58 Dielectric constant of films vs. frequency (25°C).

POLYESTERS

The most significant polyester used in the manufacture of flexible printed wiring is that referred to as Mylar, which chemically is polyethylene terephthalate. It is formed by the condensation reaction of ethylene glycol and terephthalic acid, and it is available only in film form. Its unusual balance of properties makes it especially useful in the electrical and electronics fields as a replacement for heavier, conventional insulating materials. Mylar polyester film has a dielectric strength of 7000 V/in $\times 10^{-3}$ for 0.001-in film, a tensile strength of 23,000 psi, and excellent resistance to most chemicals and moisture; it can withstand temperature extremes from -60 to 150°C. It remains flexible and does not become brittle with age. Mylar can be laminated, embossed, metalized, punched, dyed, or coated. Adhesives are available for laminating it to itself and practically any other material. The film can also be coated with heat-sealable, friction, or abrasive materials. Heavier gauges can be shaped in different ways as by stamping or vacuum forming.

There are eight types of Mylar polyester film. Each is manufactured in roll or sheet form in a range of thicknesses from 0.00025 to 0.0140 in and in widths from 1/2 to 60 to 72 in depending on gauge and type. It is important to know the correct type in order to specify properly. The individual types are as follows:

1. Type A. General-purpose; has exceptional high-temperature durability properties. It is available up to 0.014 in thick. This type is normally used in motors, wire and cable, laminations, and flexible printed wiring.
2. Type C. Has a higher level of insulation resistance than type A films; it also has more clarity and sparkle than type A film. It has been especially designed for capacitors, and it is also normally used for metallizing and metallic var.
3. Type D. Has high transparency in the heavier gauges. It is used in cartography and drafting and for graphic arts applications where appearance and dimensional stability are important.
4. Type HS. Used for the jackets of capacitors. It shrinks 30 percent in both directions when heated to 100°C. The film remains strong and tough after shrinking.
5. Type M. Has a heat-sealable polymer coating and is highly moistureproof. Like all types of Mylar polyester film, it does not get brittle with age and has a long shelf life.
6. Type S. For stationery supplies, book jackets, and sheet protectors. The film is very transparent and has a brilliant surface.
7. Type T. Has an unusually high tensile strength in the longitudinal direction. It has more longitudinal stiffness than equivalent gauges of the other films. It is used for magnetic and pressure-sensitive tapes.
8. Type W. Resistant to the sun's ultraviolet radiation. It was specifically developed for exterior glazing applications. Greenhouses made with it last four to seven years in sun or snow.

The important physical, thermal, and electrical properties of Mylar are given in Tables 25.28 and 25.29. The properties of Mylar (polyester) and Kapton (polyimide) are similar except for the higher temperature capability of Kapton. A comparison of the properties of Mylar and Kapton is given in Table 25.30.

INSULATING FILM COVER COATS

Once the flexible printed wiring has been etched or formed, it is usually necessary to cover it with an insulating film to protect the wiring from moisture, contamination, and damage. Normally the covering is accomplished by bonding an adhesive-coated dielectric film to the surface. Less commonly, fusible dielectric films are fused to the surface or liquid conformal coatings are laid down and cured.

11. Cover-Coat Properties and Bonding Consideration must be given to the desired properties of the cover coat film and adhesive. The film properties are listed in Table 25.31.

The base laminate and cover-coat lamination steps may employ either melt or adhesive bonding. In melt or fusion bonding, the thermoplastic film is adhered to the metal foil in a platen press or roll laminator. Sufficient heat and pressure are applied to melt the film momentarily. Then the film is cooled (under pressure) to resolidify it. The copper or other metal foil requires a specially treated surface to give good adhesion by melt bonding.

TABLE 25.28 Important Properties of Mylar Polyester Film^{††}

Properties	Typical values		Test condition*
	Type A	Type T	
Physical			
Ultimate tensile strength, MD, psi	25,000	40,000	25
Yield point, MD, psi	12,000 at 4%	Indeterminate	25
Stress to produce 5% elongation, MD, psi	>14,000	22,000	25
Ultimate elongation, MD, %	120	50	25
Tensile modulus, MD, psi	550,000	800,000	25
Impact strength, kg-cm/in × 10 ⁻³	6.0	6.0	25
Folding endurance (MIT), Hz	14,000		25
Tear strength:			
Propagating (Elmendorf), g/in × 10 ⁻³	15	12	25
Initial (Graves), g/in × 10 ⁻³	900	590	25
lb/in	2000	1300	
Bursting strength (Mullen), psi	66	55	25
Density	1.395	1.377	25
Coefficient of friction, kinetic, film-to-film	0.45	0.38	
Refraction index (Abbé)	1.64 _D 25		25
Area factor, in. ² /(lb)(in × 10 ⁻³)	20,500	20,700	25
Thermal			
Melting point, °F	480-510		
°C	250-265		
Service temperature, °F	-75 to +300		
°C	-60 to +150		
Coefficient of linear expansion, in./in)(°F) × 10 ⁵	1.5		70-120°F
Coefficient of thermal conductivity: Btu-in/(ft ²)(h)(°F)	1.035		25
cal-cm/(cm ²)(s)(°C) × 10 ⁴	3.63		25
Heat scalability	No		
Specific heat, cal/(g)(°C)	0.315		
Shrinkage, %	2-3		30 min at 150
Electrical			
Dielectric strength (0.001 in) at 60 Hz, V/in × 10 ⁻³	7000		25
	5000		150
Dielectric constant:			
At 60 Hz	3.25		25
At 1 kHz	3.1		25
At 1 MHz	3.0		25
At 1000 MHz	2.8		25
At 60 Hz	3.7		150
Dissipation factor:			
At 60 Hz	0.0021		25
At 1 kHz	0.0047		25
At 1 MHz	0.016		25
At 1000 MHz	0.003		25
At 60 Hz	0.0064		150
Volume resistivity, Ω-cm	1 × 10 ¹⁴		25
	1 × 10 ¹³		150
Surface resistivity, Ω			
At 0% RH	1 × 10 ¹²		25
At 100% RH	5 × 10 ¹¹		25
Insulation resistance, MΩ-μF	5000		100
	400		130
	100		150

*In degrees centigrade unless otherwise specified.

TABLE 25.29 Chemical Resistance of Mylar Polyester Film

Agent	Percent of following retained			Exposure ^{††}
	Tensile strength	Elongation	Tear strength	
Acetic acid, glacial	75	107	83	33 days at 20°C
Hydrochloric acid, 18%	82	99	113	18 days at 20°C
Sodium hydroxide, 10%	83	30	33	33 days at 20°C
Ammonium hydroxide, concentrated	0	0	0	3 days at 20°C
Acetone	63	151	67	33 days at 20°C
Hydrocarbon oil	92	88	87	500 h at 100°C
Phenolic resin, GE1678	92	3	73	Baked 168 h at 150°C
Moisture absorption	Less than 0.8%			24 h at 23°C

*By immersion unless otherwise specified.

Fluorocarbon and polyvinyl chloride flexible circuits are generally melt-bonded. However, a number of factors led to the introduction of newer circuit types based on adhesive-bonded constructions. One advantage of adhesive-bonded constructions is the ability to utilize high-quality insulating films such as polyester and polyimide, which are not suitable for melt bonding. Another advantage is the improved dimensional control of the circuit pattern which results when the etched circuit base is not remelted under pressure in the cover-coat lamination step. In the adhesive-bonded flexible circuits it is also standard practice to use the same insulating film as in the base laminate for the adhesive-bonded cover coat applied over the etched circuit.

12. Liquid Cover Coat Constructions in which the cover coat film is formed from solution or from a polymerizable coating eliminate the need for lamination of a preformed, prepunched polymer film over the circuit pattern by the use of high heat and pressure. As a result, a number of cost, freedom of design, reliability, and repairability advantages may be possible.

The cost savings possible with the use of liquid cover-coat techniques represent a major advantage for the liquid fabrication method. Plastic films are relatively expensive com-

TABLE 25.30 Comparison of Properties of Kapton[†] Polyimide Film and Mylar[†] Polyester Film

Property	ASTM test	Temperature, °C	Value	
			Kapton	Mylar
Dielectric strength at 60 Hz, 0.001 in thick, V/in × 10 ⁻³	D 149	25	7000	7000
		150	6000	5000
Dielectric constant at 1 kHz	D 150	25	3.5	3.1
		200	3.0	
Dissipation factor at 1 kHz	D 150	25	0.003	0.0047
		200	0.002	0.01
Volume resistivity, Ω-cm	D 257	25	10 ¹⁶	10 ¹⁶
		200	10 ¹⁴	5 × 10 ¹¹
Surface resistivity at 1 kV, 50% RH, Ω	D 257	25	10 ¹⁶	10 ¹⁶
		25	25,000	23,000
Tensile strength, psi	D 882	200	17,000	7000
		25	14,000	12,000
Yield point, psi		200	9000	1000
		25	13,000	13,000
Stress to produce 5% elongation, psi	D 882	25	13,000	13,000
		25	70	100
Ultimate elongation, %	D 882	200	90	large
		25	70	100
Tensile modulus, psi	D 882	25	430,000	550,000
		200	260,000	50,000
Bursting strength, psi	D 774	25	75	30
Density, g/cm ³		25	1.42	1.4
Coefficient of friction, kinetic, film to film	D 1505	25	0.42	0.45
		25	1.35	1.40
Area factor, ft ² /(lb)(in × 10 ⁻³)		25		

*Trademarks of E. I. du Pont de Nemours & Co., Inc., Wilmington, Del.

TABLE 25-31 Detailed Characteristics of Various Insulations

	TFE plastic	TFE glass cloth	FEP plastic	FEP glass cloth	Kapton polyimide film	CTFE	Epoxy-therm	Polyquarlene	Mylar polyester film	Polyvinylchloride	Polyethylene
Spec. pr. Flammable	2.15 No	2.2 No	2.18 No	2.2 No	1.42 Self-ext.	2.10 No	1.38 Yes	0.905 Shm-barring	1.385 Yes	1.25 Self-ext.	0.93 Yes
Appearance	Translucent	Trans	Clear fish-bone	Trans	Another	White and opaque	Translucent	Clear	Clean	Translucent	Clear
Availability with adhesives	Good ^b	Good ^b	Good ^b	Good ^b	Good	Good	Good	Poor	Good	Good	Poor
Handability in	Good	Poor	Good	Good	Poor	Good	Good	Good	Poor	Good	Good
Chemical resistance	Exc.	Exc.	Exc.	Exc.	Exc.	Exc.	Good	Good	Exc.	Good	Good
Stainlight resistance	Exc.	Exc.	Exc.	Exc.	Exc.	Exc.	Fair	Low	Fair	Fair	Low
Water absorption, %	<0.0124 h	0.00065 h	<0.0124 h	0.180, 30 h	0	0	1.5	0.01	0.624 h	0.10	0.0124 h
Volume resistivity, Ω cm	>10 ¹⁰	10 ¹⁰	>2 × 10 ¹⁰	10 ¹⁰	10 ¹⁰	3.1 × 10 ¹⁰	1 × 10 ¹⁰	10 ¹⁰	10 ¹⁰	10 ¹⁰	10 ¹⁰
Dielectric constant, 10 ⁻¹⁰ Hz	2.2	2.55†	2.1	2.55†	3.5	2.5	2.5	2.0	2.8-3.7	3.6-4.0	2.2
Dissipation factor, 10 ⁻¹⁰ Hz	2 × 10 ⁻⁴	7 × 10 ⁻⁴	2 × 10 ⁻⁴	10 ⁻⁴	3 × 10 ⁻⁴	15 × 10 ⁻⁴	70 × 10 ⁻⁴	2 × 10 ⁻⁴	2 × 10 ⁻⁴	1.1 × 10 ⁻⁴	6 × 10 ⁻⁴
Service temp Min., °C	-200	-70	-200	-70	-200	-70	-20	-55	-60	-30	-20
Max., °C	250	290	290	300	300	125	100	100	105	85	60
Tensile str., psi @ 25°C	3100	20,000†	3000	20,000†	23,000	4,900	3,000	5,700	20,000	3,000	2,000
Modulus of elasticity, psi	0.2067	1.378	0.2067	1.378	1.697	0.31	0.8525	0.9927	1.378	0.2067	0.1378
Thermal expansion coefficient, 10 ⁻⁶ /°C	55	50	50	1.1	11	48	11	61	13	1.378	1.378
Dielectric strength, V/10 ⁻⁴ in.	100	90	20	82	20	82	30	110	27	8000	180
Sample size, x 10 ⁻⁴ in.	600	650, 1600†	2800	650, 1600	7000	1100	900	600	700	8000	585
		0.015	0.005	0.003	0.002	0.0192	0.002	0.125	0.001		0.125

^a Must be treated.
^b Depends on glass cloth.
^c Depends on formulation utilized.

pared with liquid coating materials. The plastic films often must be pre-cut and pre-punched prior to cover-coat lamination. Such operations add substantially to cost in terms of equipment, labor, and loss of material as scrap. Then the cover coat must be laminated over the etched pattern either in a platen press or by roll lamination. That is another expensive step. The heat and pressure involved may be harmful to the base circuit and may result in unacceptable dimensional changes in the circuit. Final inspection for those problems, along with improper registration of covercoat pad openings, often may raise the cost significantly owing to poor yield of fully acceptable circuits.

A second advantage of liquid cover-coat construction is the ability to utilize insulation materials which are not available or not satisfactory in the form of unsupported plastic films. Melt-bonded constructions are limited to thermoplastic films, but if the cover coat is applied in liquid form, thermosetting polymers also can be used. Good electrical insulating elastomers such as silicone, polyurethane, and butyl rubber have not been applied widely in flexible circuits because they would easily stretch and deform in service. Those polymers have proved useful in liquid cover coats when applied to dimensionally stable base circuits.

Flexible circuits made with liquid cover coats can be designed for improved reparability by selective stripping of the cover coat with chemical solvents. Whereas the film used for the etched circuit base must be compatible with etching solutions and solvents used for removal of etching resists, that requirement does not apply to the cover coat. Hence the liquid cover coat may be a lacquer that will dissolve in suitable solvents which will not attack the plastic film of the base circuit. After repairs or new connections are made, the liquid cover coat can be brushed over the exposed area to seal the joints fully.

Enhanced reparability may also result from the use of low-melting "solder-through" cover coats which will melt or vaporize under the heat of a soldering iron to permit attaching components, making new connections, or making repairs without provision of prepunched holes in the cover coat or skiving to expose pads.

Reliability of liquid cover-coated flexible circuits clearly exceeds that of circuits not protected with any kind of cover coat. The liquid coating provides the same protection from moisture, contamination, and shorting between conductors as that given by a laminated film cover coat. In some respects, liquid cover-coat systems can be expected to give reliability superior to film cover coats, but in other respects the films no doubt will continue to excel.

Elimination of the high heat and pressure cycle involved in film cover-coat lamination permits closer conductor spacing, does not adversely affect the bond between conductors and the base circuit film, provides better dimensional control, and results in less damage to plated-through holes in two-sided or multilayer constructions. Liquid systems have better ability to fill the space between closely spaced conductors, and laminated films have some tendency to "bridge" those spaces and leave the conductor edges incompletely sealed against wicking of moisture and contaminants. On the other hand, laminated films can be completely free of pinholes and porosity, which are problems common to liquid coatings. Hence, for highest dielectric strength and integrity, laminated films will no doubt prove superior.

13. Properties of the Cured Cover Coat The liquid cover coat is cured after application to the etched circuit base. That converts the liquid into a polymer film with properties which must meet the following general requirements:

1. Good adhesion to base circuit.
2. Sufficient flexibility and flexibility life for the specific application.
3. Provision of environmental protection (against contaminants or moisture) adequate for the application.
4. Provision of electrical properties (dielectric strength, dielectric constant, dissipation factor, or insulation resistance) adequate for the application.
5. Maintenance of all properties over the service temperature range and for the desired life of the circuits.
6. Provision of solder resist properties and resist solder flux cleaning solvents if needed for the specific application.

7. Provision of good appearance. In some cases an opaque coating may be desired.

8. Meeting additional requirements, if any, imposed by customer for specific circuits.

Detailed requirements, of course, vary with the application. Since distinct advantages are often associated with various polymer film insulation systems and since there is usually a trade-off between performance and cost, a range of construction materials for differing flexible circuit applications has evolved. Liquid cover coats also offer a range of performance values, and it appears that no one coating system would be suitable for all types of base circuits.

Experience in the selection of insulation materials for flexible cables and circuits has resulted in a set of design standards and test methods published by the Institute of Printed Circuits. The principal standards on circuits without covercoats or circuits with laminated film covercoats are IPC-FC-220, Specification for Flat Cable, Flat Conductor, Unshielded, and IPC-FC-240A, Specification for Flexible Printed Wiring. IPC-FC-220 classifies cables as to service temperature range on the basis of the insulation material used. Table 25.32 presents the temperature and material classifications given in IPC-FC-220. As in other insulation applications, the expected service temperature is one of the prime determinants of material selection. However, flexibility, dimensional stability, electrical properties, corrosion protection, strength, abrasion resistance, flame resistance, and other properties may be of equal or greater significance in certain applications.

One requirement which is unique to coatings to be used as flexible circuit cover coats is a high degree of flexibility. That factor sets the coatings apart from solder resist coatings or conformal coatings for rigid printed boards. Coatings for rigid printed circuits require only

TABLE 25.32 Flat Cable Type Classification*

Type	Temp. range, °C	Self-exting.	Insulation material(s)
AN	0 to 65	No	Polyvinyl chloride
AS		Yes	Polyvinyl chloride
BN	-20 to 105	No	Polyester (heterogeneous)
BS		Yes	Polyester (heterogeneous) or FEB (C bond)
CI	-40 to 200	Yes	FEP (melt bond) or polyimide FEP (melt bond) or TFE (homogeneous)
DI	-40 to 250	Yes	TFE (homogeneous) or polyimide (polyimide bond)

*SOURCE: IPC-FC-220 Specification for Flat Cable, Flat Conductor, Unshielded.

† FEP is fluorinated ethylene propylene; TEF is polytetrafluoroethylene.

‡ Plated conductors recommended for this temperature.

sufficient flexibility to prevent cracking due to expansion and contraction during thermal cycling. That allows the use of fairly hard, almost brittle resin materials such as alkyd-melamine and rigid epoxy. Obviously, flexible circuit applications require a higher degree of flexibility than is provided by such materials, but not the same degree in all applications. When the circuit is not flexed in service, flexibility must be only sufficient to permit normal handling in assembly. On the other hand, flexible cables that may be rolled or folded repetitively in service at very low temperatures require coatings which will withstand such abuse.

The differing capability of polymer films to meet flexibility requirements at low temperatures is the basis for the lower limit in the service temperature range given in IPC-FC-220 (Table 25.32). Those high-molecular-weight thermoplastic polymer films have a high degree of ductility which is hard to attain in the thermosetting polymers or low-molecular-weight thermoplastic polymers used in liquid coating formulations. For that reason, highly flexibilized or elastomeric formulations have proved most suitable for liquid cover coat applications. Polyurethane, butyl, and silicone elastomers are very satisfactory. The dimensional support is provided by the film used for the circuit base, and in some cases a somewhat heavier film may be chosen to assure adequate strength.

A complete discussion of all performance factors cannot be presented in the space available. The following discussion and test data are intended to illustrate a range of circuit constructions for which the use of one or more types of liquid cover coats appears advantageous.

14. Cover Coats for Polyimide Circuits Polyimide flexible circuits have been accepted widely in recent years for reason of their combination of excellent overall properties and high service temperature capability. In a great many applications, the dimensional stability, toughness, low-temperature flexibility, or flame resistance of these circuits is of greater concern than their heat resistance. While polyimide film itself is capable of operation up to 250°C (IPC-FC-220, type D), the polyimide adhesives and plated copper conductors also used in the construction of cables or circuits involve other heat-resistance limitations. A polyimide-FEP (fluorinated ethylene propylene) melt-bonded construction with laminated film covercoat is a much simpler product and provides satisfactory service up to 200°C (IPC-FC-220, type G).

In order to reduce costs and facilitate manufacturing, polyimide circuits using epoxy adhesives or other heat-resistant adhesives have been developed. The adhesives reduce the maximum service temperature limit of the circuits somewhat, but they can be formulated to maintain the other desirable properties. That type of circuit has proved to be quite suitable for the application of liquid cover coats. Table 25.33 presents test data on four types of liquid cover coats which were developed for use with adhesive-bonded polyimide constructions. The choice among the four for any particular application would be based on the features of greatest importance. The silicone coating provides the best heat resistance; the epoxy system has the best moisture and abrasion resistance; and the polyurethane provides a good combination of low-temperature flexibility and moisture protection. The butyl rubber coating gives excellent low-temperature flexibility and moisture resistance but lacks heat resistance for higher-temperature applications. It might be more useful on polyester circuits. The polyimide film is not the best material for operation under high-humidity conditions, as the data show. Silicone coatings are more moisture-permeable than epoxies or polyurethanes and so do not provide much protection if tested under actual humidity conditions. If, however, the circuit is removed from the humidity chamber and allowed to dry out, the silicone-coated specimens recover faster than the others do.

Liquid cover coats do not appear to be practical for polyimide-FEP melt-bonded constructions because of the difficulty of getting the coating to adhere to the FEP surface. Liquid polyimide and polyamide-imide coatings have been studied for use on polyimide circuits. They present many of the same problems as the polyimide adhesive systems present. In general, the coatings require very high curing temperatures and tend to coil circuits owing to high shrinkage during cure. For that reason, they are not practical at present.

15. Cover Coats for Polyester Circuits Polyester circuits constructed by using adhesive bonding with a polyester adhesive constitute the largest volume of flexible cable and flexible circuit sales. The type of construction covered by IPC-FC-220, type BN, provides

TABLE 25.33 Liquid Cover Coats on Polyimide Circuits

Cover coat material	Low-temp. flexibility (%-in mandrel bend at temp. shown)	Insulation resistance in humidity (IPC-FC-240A comb pattern 96 h @ 35°C, 90% RH, MΩ)	Insulation resistance at elevated temp. (IPC-FC-240A comb pattern, MΩ)			Solderability
			105°C	130°C	155°C	
Silicone RTV	Passes -65°C	100	6,000	400	40	Solder resist
Flexible epoxy	Passes -35°C	200,000	4,500	530	150	Solder resist
Polyurethane	Passes -65°C	1,000	1,800	100	50	Solder resist
Butyl rubber	Passes -65°C	30,000	350	50	20	Not a resist
No cover coat	Passes -65°C	30	1,600	130	20	Solderable

TABLE 25.34 Liquid Cover Coats on Polyester Circuits

Cover coat material	Low-temp. flexibility (% in mandrel bend at temp. shown)	Insulation resistance in humidity (IPC-FC-240A comb pattern 96 h @ 35°C, 90% RH, MΩ)	Insulation resistance at elevated temp. (IPC-FC-240A comb pattern, MΩ)			Solderability
			90°C	105°C	130°C	
Polyester	Passes -65°C	2,500	3,000	1,000	160	Solder through
Acrylic	Passes -20°C	200,000	30,000	30,000	2,700	Solder through
Polyurethane	Passes -65°C	50,000	2,600	1,000	300	Solder through
No cover coat	Passes -65°C	20,000	4,500	1,400	220	Nonsolderable

excellent dimensional stability at very low cost. The service temperature range is generally limited to -20 to 105°C. Another limitation is the difficulty of making a flame-retardant polyester system. Because the circuits are used in very cost competitive applications, the use of a liquid cover coat appears particularly attractive.

The test results for several promising liquid cover coats on polyester circuits are given in Table 25.34. The polyester and acrylic cover coats permit solder connections to be made through the coatings, since the heat of a soldering iron is sufficient to melt the coatings away. That must, of course, be done with considerable care on polyester circuits to avoid damage to the base circuit. Stripping of the acrylic coating is possible by using solvents which do not attack the polyester film.

16. Cover Coats for PVC Circuits IPC-FC-220, types AN and AS, cover flexible cables of polyvinyl chloride fabricated by melt bonding. PVC insulation is even less expensive than polyester, and it is capable of providing good environmental protection within its service temperature range. A major limitation of that construction in complex circuit configurations has been dimensional instability.

The instability problem is important because, in the cover coat lamination step, the temperature and pressure must be sufficient to melt the cover coat, force it between the etched conductors, and fuse it to the base film. Conductor movement or "swimming" can be a serious problem during the operation. The use of a low-temperature-curing liquid cover coat eliminates that step; the base laminate is formed by either melt or adhesive bonding. After the circuit is etched, the cover coat is applied by spraying or screening. Table 25.35 shows test results of one such system which promises adequate performance for many applications at a minimum circuit cost.

TABLE 25.35 Liquid Cover Coat on a Polyvinyl Chloride Circuit

Property and test method	Liquid vinyl cover coat	No cover coat
Low-temperature flexibility (1/8-in mandrel bend test)	Passes -20°C	Passes -20°C
Insulation resistance in humidity (IPC-FC-240A comb pattern, MΩ 96 h @ 35°C, 90% RH)	2500	30,000
Insulation resistance at elevated temperature (IPC-FC-240A comb pattern), MΩ		
70°C	1900	2100
90°C	600	410
105°C	280	200
Flammability	Self-ext.	Self-ext.

GLOSSARY

- ACCELERATOR:** A chemical that is used to speed up a reaction or cure, as cobalt naphthenate is used to accelerate the reaction of certain polyester resins. It is often used along with a catalyst, hardener, or curing agent. The term "accelerator" is often used interchangeably with the term "promoter."
- ADDITIVE PROCESS:** A process for obtaining conductive patterns by the selective deposition of conductive material on an insulad base material.
- ADHESIVE:** Broadly, any substance used in promoting and maintaining a bond between two materials.
- AGING:** The change in properties of a material with time under specific conditions.
- AMBIENT TEMPERATURE:** The temperature of the cooling medium, such as gas or liquid, which comes into contact with the heated parts of an apparatus (or the normal temperature of the surrounding environment).
- ANNULAR RING:** The circular strip of conductive material that completely surrounds a hole.
- ARC RESISTANCE:** The time required for an arc to establish a conductive path in a material.
- ARTWORK MASTER:** An accurately scaled configuration used to produce the production master.
- BASE MATERIAL:** The insulating material upon which the printed wiring pattern may be formed.
- BASE MATERIAL THICKNESS:** The thickness of the base material excluding metal foil cladding or material deposited on the surface.
- BLISTERING:** Localized swelling and separation between any of the layers of the base laminate or between the laminate and the metal cladding.
- BONDING LAYER:** An adhesive layer used in bonding other discrete layers during lamination.
- BOND STRENGTH:** The force per unit area required to separate two adjacent layers by a force perpendicular to the board surface, usually refers to the interface between copper and base material.
- B STAGE:** An intermediate stage in the curing of a thermosetting resin. In it a resin can be heated and caused to flow, thereby allowing final curing in the desired shape.
- B-STATE LOT:** The product from a single mix of B-stage ingredients.
- B-STAGE RESIN:** A resin in an intermediate stage of a thermosetting reaction. The material softens when heated and swells when in contact with certain liquids, but it may not entirely fuse or dissolve.
- CAPACITANCE:** The property of a system of conductors and dielectrics which permits the storage of electricity when potential difference exists between the conductors.
- CAPACITIVE COUPLING:** The electrical interaction between two conductors caused by the capacitance between the conductors.
- CATALYST:** A chemical that causes or speeds up the cure of a resin but does not become a chemical part of the final product.
- CHLORINATED HYDROCARBON:** An organic compound having chlorine atoms in its chemical structure. Trichloroethylene, methyl chloroform, and methylene chloride are chlorinated hydrocarbons.
- CIRCUIT:** The interconnection of a number of electrical devices in one or more closed paths to perform a desired electrical or electronic function.
- CLAD:** A condition of the base material, to which a relatively thin layer or sheet of metal foil (cladding) has been bonded on one or both of its sides. The result is called a metal-clad base material.
- COAT:** To cover with a finishing, protecting, or enclosing layer of any compound.

COEFFICIENT OF EXPANSION: The fractional change in dimension of a material with a unit change in temperature.

COLD FLOW: The continuing dimensional change that follows initial instantaneous deformation in a nonrigid material under static load. Also called creep.

COLLIMATION: The degree of parallelism of light rays from a given source. A light source with good collimation produces parallel light rays, whereas a poor light source produces divergent, nonparallel light rays.

COPOLYMER: See *polymer*.

COMPONENT HOLE: A hole used for the attachment and electrical connection of a component termination, including pin or wire, to the printed board.

COMPONENT SIDE: The side of the printed board on which most of the components will be mounted.

COMPOUND: A combination of elements in a stable molecular arrangement.

CONDUCTIVE FOIL: The conductive material that covers one side or both sides of the base material and is intended for forming the conductive pattern.

CONDUCTIVE PATTERN: The configuration or design of the electrically conductive material on the base material.

CONDUCTOR LAYER 1: The first layer having a conductive pattern, of a multilayer board, on or adjacent to the component side of the board.

CONDUCTOR SPACING: The distance between adjacent edges (not centerline to centerline) of conductors on a single layer of a printed board.

CONDUCTOR THICKNESS: The thickness of the copper conductor exclusive of coatings or other metals.

CONDUCTOR WIDTH: The width of the conductor viewed from vertically above, i.e., perpendicularly to the printed board.

CONFORMAL COATING: An insulating protective coating which conforms to the configuration of the object coated and is applied on the completed printed board assembly.

CONNECTOR AREA: The portion of the printed board that is used for providing external (input-output) electrical connections.

CONTACT BONDING ADHESIVE: An adhesive (particularly of the nonvulcanizing natural rubber type) that bonds to itself on contact although solvent evaporation has left it dry to the touch.

CORNER MARKS: The marks at the corners of printed board artwork, the inside edges of which usually locate the borders and establish the contour of the board.

COUPON: One of the patterns of the quality conformance test circuitry area. (See *test coupon*.)

CRAZING: A base material condition in which connected white spots or crosses appear on or below the surface of the base material. They are due to the separation of filers in the glass cloth and connecting weave intersections.

CROSS-LINKING: The forming of chemical links between reactive atoms in the molecular chain of a plastic. It is cross-linking in the thermosetting resins that makes the resins infusible.

CROSS TALK: Undesirable electrical interference caused by the coupling of energy between signal paths.

CRYSTALLINE MELTING POINT: The temperature at which crystalline structure in a material is broken down.

CURE: To change the physical properties of a material (usually from a liquid to a solid) by chemical reaction or by the action of heat and catalysts, alone or in combination, with or without pressure.

CURING AGENT: See *hardener*.

CURING TIME: In the molding of thermosetting plastics, the time in which the material is properly cured.

CURING TEMPERATURE: The temperature at which a material is subjected to curing.

CURRENT-CARRYING CAPACITY: Maximum current which can be carried continuously without causing objectionable degradation of electrical or mechanical properties of the printed board.

DATUM REFERENCE: A defined point, line, or plane used to locate the pattern or layer of a printed board for manufacturing and/or inspection purposes.

DEFINITION: The fidelity of reproduction of the printed board conductive pattern relative to the production master.

DELAMINATION: A separation between any of the layers of the base laminate or between the laminate and the metal cladding originating from or extending to the edges of a hole or edge of the board.

DIELECTRIC CONSTANT: The property of a dielectric which determines the electrostatic energy stored per unit volume for a unit potential gradient.

DIELECTRIC LOSS: Electric energy transformed into heat in a dielectric subjected to a changing electric field.

DIELECTRIC LOSS ANGLE: The difference between 90° and the dielectric phase angle. Also called the dielectric phase difference.

DIELECTRIC LOSS FACTOR: The product of dielectric constant and the tangent of dielectric loss angle for a material.

DIELECTRIC PHASE ANGLE: The angular difference in phase between the sinusoidal alternating potential difference applied to a dielectric and the component of the resulting alternating current having the same period as the potential difference.

DIELECTRIC POWER FACTOR: The cosine of the dielectric phase angle (or sine of the dielectric loss angle).

DIELECTRIC STRENGTH: The voltage that an insulating material can withstand before breakdown occurs, usually expressed as a voltage gradient (such as volts per mil).

DIMENSIONAL STABILITY: Freedom from distortion by such factors as temperature changes, humidity changes, age, handling, and stress.

DISSIPATION FACTOR: The tangent of the loss angle of the insulating material. Also called loss tangent or approximate power factor.

DRILL FACET: The surface formed by the primary and secondary relief angles of a drill tip.

DUMMY: A cathode with a large area used in a low-current-density plating operation for the removal of metallic impurities from solution. The process is called "dummying."

EDGE-BOARD CONTACTS: A series of contacts printed on or near an edge of a printed board and intended for mating with a one-part edge connector.

ELASTOMER: A material which at room temperature stretches under low stress to at least twice its length but snaps back to its original length upon release of the stress. Rubber is a natural elastomer.

ELECTRIC STRENGTH: The maximum potential gradient that a material can withstand without rupture. It is a function of the thickness of the material and the method and conditions of test. Also called dielectric strength or disruptive gradient.

ELECTROLESS PLATING: The controlled autocatalytic reduction of a metal ion on certain catalytic surfaces.

ELEMENT: A substance composed entirely of atoms of the same atomic number, e.g., aluminum or copper.

EMULSION SIDE: The side of the film or glass on which the photographic image is present.

EPOXY SMEAR: Epoxy resin which has been deposited on edges of copper in holes during drilling either as a uniform coating or as scattered patches. It is undesirable because it can electrically isolate the conductive layers from the plated-through-hole interconnections.

ETCHBACK: The controlled removal of all of the components of the base material by a chemical process acting on the sidewalls of plated-through holes to expose additional internal conductor areas.

ETCH FACTOR: The ratio of the depth of etch to lateral etch.

EXOTHERMIC REACTION: A chemical reaction in which heat is given off.

EXOTHERM: A characteristic curve which shows heat of reaction of a resin during cure (temperature) vs. time. The peak exotherm is the maximum temperature on the curve.

FIBER EXPOSURE: A condition in which glass cloth fibers are exposed on machined or abraded areas.

FILLER: A material, usually inert, added to a plastic to reduce cost or modify physical properties.

FILM ADHESIVE: A thin layer of dried adhesive. Also, a class of adhesives provided in dry-film form with or without reinforcing fabric and cured by heat and pressure.

FLEXURAL MODULUS: The ratio, within the elastic limit, of stress to corresponding strain. It is calculated by drawing a tangent to the steepest initial straight-line portion of the load-deformation curve and calculating by the equation $R_E = L^3m/4bd^3$, where E_B is the modulus, L is the span, in inches, b is the width of beam tested, d is the depth of the beam, and m is the slope of the tangent.

FLEXURAL STRENGTH: The strength of a material subjected to bending. It is expressed as the tensile stress of the outermost fibers of a bent test sample at the instant of failure.

FLUOROCARBON: An organic compound having fluorine atoms in its chemical structure, an inclusion that usually lends stability to plastics. Teflon* is a fluorocarbon.

GEL: The soft, rubbery mass that is formed as a thermosetting resin goes from a fluid to an infusible solid. It is an intermediate state in a curing reaction, and a stage in which the resin is mechanically very weak.

GEL POINT: The point at which gelation begins.

GLASS TRANSITION POINT: The temperature at which a material loses properties and becomes a semiliquid.

GLASS TRANSITION TEMPERATURE: The temperature at which epoxy, for example, softens and begins to expand independently of the glass fabric expansion rate.

GLUE-LINE THICKNESS: Thickness of the fully dried adhesive layer.

GRID: An orthogonal network of two sets of parallel lines for positioning features on a printed board.

GROUND PLANE: A conducting surface used as a common reference point for circuit returns, shielding, or heat sinking.

HALOING: A light area around holes or other machined areas on or below the surface of the base laminate.

HARDENER: A chemical added to a thermosetting resin for the purpose of causing curing or hardening. A hardener, such as an amine or acid anhydride for an epoxy resin, is a part of the chemical reaction and a part of the chemical composition of the cured resin. The terms "hardener" and "curing agent" are used interchangeably.

HEAT-DISTORTION POINT: The temperature at which a standard test bar (ASTM D 648) deflects 0.010 in under a stated load of either 66 or 264 psi.

HEAT SEALING: A method of joining plastic films by simultaneous application of heat and pressure to areas in contact. The heat may be supplied conductively or dielectrically.

HOLE FULL STRENGTH: The force, in pounds, necessary to rupture a plated-through hole or its surface terminal pads when loaded or pulled in the direction of the axis of the hole. The pull is usually applied to a wire soldered in the hole, and the rate of pull is given in inches per minute.

HOT-MELT ADHESIVE: A thermoplastic adhesive compound, usually solid at room temperature, which is heated to liquid state for application.

HYDROCARBON: An organic compound containing only carbon and hydrogen atoms in its chemical structure.

HYDROLYSIS: The chemical decomposition of a substance involving the addition of water.

HYGROSCOPIC: Tending to absorb moisture.

IMPREGNATE: To force resin into every interstice of a part, as of a cloth for laminating.

INHIBITOR: A chemical that is added to a resin to slow down the curing reaction and is normally added to prolong the storage life of a thermosetting resin.

INORGANIC CHEMICALS: Chemicals whose molecular structures are based on other than carbon atoms.

INSULATION RESISTANCE: The electrical resistance of the insulating material between any pair of contacts, conductors, or grounding devices in various combinations.

INTERNAL LAYER: A conductive pattern contained entirely within a multilayer board.

JUMPER: An electrical connection between two points on a printed board added after the printed wiring is fabricated.

LAMINATE VOID: Absence of epoxy resin in any cross-sectional area which should normally contain epoxy resin.

LAND: See *terminal area*.

LANDLESS HOLE: A plated-through hole without a terminal area.

LAYER-TO-LAYER SPACING: The thickness of dielectric material between adjacent layers of conductive circuitry.

LAY-UP: The process of registering and stacking layers of a multilayer board in preparation for the laminating cycle.

LEGEND: A format of lettering or symbols on the printed board, e.g., part number, component locations, or patterns.

*Trademark of E. I. du Pont de Nemours & Company.

MAJOR WEAVE DIRECTION: The continuous-length direction of a roll of woven glass fabric.

MASTER DRAWING: A document that shows the dimensional limits or grid locations applicable to any or all parts of a printed wiring or printed circuit base. It includes the arrangement of conductive or nonconductive patterns or elements, size, type, and location of holes, and any other information necessary to characterize the complete fabricated product.

MEASLING: Discrete white spots or crosses below the surface of the base laminate that reflect a separation of fibers in the glass cloth at the weave intersection.

MICROSTRIP: A type of transmission line configuration which consists of a conductor over a parallel ground plane separated by a dielectric.

MINOR WEAVE DIRECTION: The width direction of a roll of woven glass fabric.

MODULUS OF ELASTICITY: The ratio of stress to strain in a material that is elastically deformed.

MOISTURE RESISTANCE: The ability of a material not to absorb moisture either from air or when immersed in water.

MOUNTING HOLE: A hole used for the mechanical mounting of a printed board or for the mechanical attachment of components to a printed board.

MULTILAYER BOARD: A product consisting of layers of electrical conductors separated from each other by insulating supports and fabricated into a solid mass. Interlayer connections are used to establish continuity between various conductor patterns.

MULTIPLE-IMAGE PRODUCTION MASTER: A production master used to produce two or more products simultaneously.

NAIL HEADING: A flared condition of internal conductors.

NEMA STANDARDS: Property values adopted as standard by the National Electrical Manufacturers Association.

NOBLE ELEMENTS: Elements that either do not oxidize or oxidize with difficulty; examples are gold and platinum.

ORGANIC: Composed of matter originating in plant or animal life or composed of chemicals of hydrocarbon origin, either natural or synthetic.

pH: A measure of the acid or alkaline condition of a solution. A pH of 7 is neutral (distilled water); pH values below 7 are increasingly acid as pH values go toward 0; and pH values above 7 are increasingly alkaline as pH values go toward the maximum value of 14.

PHOTOGRAPHIC REDUCTION DIMENSION: The dimensions (e.g., line or distance between two specified points) on the artwork master to indicate the extent to which the artwork master is to be photographically reduced. The value of the dimension refers to the 1:1 scale and must be specified.

PHOTOMASTER: An accurately scaled copy of the artwork master used in the photofabrication cycle to facilitate photoprocessing steps.

PHOTOPOLYMER: A polymer that changes characteristics when exposed to light of a given frequency.

PINHOLE: Small imperfections which penetrate entirely through the conductor.

PITS: Small imperfections which do not penetrate entirely through the printed circuit.

PLASTICIZER: Material added to resins to make them softer and more flexible when cured.

PLATED-THROUGH HOLE: A hole in which electrical connection is made between internal and external conductive patterns, or both, by the deposition of metal on the wall of the hole.

PLATING VOID: The area of absence of a specific metal from a specific cross-sectional area: (1) When the plated-through hole is viewed as cross-sectioned through the vertical plane, it is a product of the average thickness of the plated metal times the thickness of the board itself as measured from the outermost surfaces of the base copper on external layers. (2) When the plated-through hole is viewed as cross-sectioned through the horizontal plane (annular method), it is the difference between the area of the hole and the area of the outside diameter of the through-hole plating.

POLYMER: A high-molecular-weight compound made up of repeated small chemical units. For practical purposes, a polymer is a plastic. The small chemical unit is called a mer, and when the polymer or mer is cross-linked between different chemical units (e.g., styrene-polyester), the polymer is called a copolymer. A monomer is any single chemical from which the mer or polymer or copolymer is formed.

POLYMERIZE: To unite chemically two or more monomers or polymers of the same kind to form a molecule with higher molecular weight.

POTLIFE: The time during which a liquid resin remains workable as a liquid after catalysts, curing agents, promoters, etc., are added. It is roughly equivalent to gel time.

POWER FACTOR: The cosine of the angle between the applied voltage and the resulting current.

PREPRODUCTION TEST BOARD: A test board (as detailed in IPC-ML-950) the purpose of which is to determine whether, prior to the production of finished boards, the contractor has the capability of producing a multilayer board satisfactorily.

PRESS PLATEN: The flat heated surface of the lamination press used to transmit heat and pressure to lamination fixtures and into the lay-up.

PRINTED WIRING ASSEMBLY DRAWING: A document that shows the printed wiring base, the separately manufactured components which are to be added to the base, and any other information necessary to describe the joining of the parts to perform a specific function.

PRINTED WIRING LAYOUT: A sketch that depicts the printed wiring substrate, the physical size and location of electronic and mechanical components, and the routing of conductors that interconnect the electronic parts in sufficient detail to allow for the preparation of documentation and artwork.

PRODUCTION MASTER: A 1:1 scale pattern used to produce one or more printed wiring or printed circuit products within the accuracy specified on the master drawing.

PROMOTER: A chemical, itself a feeble catalyst, that greatly increases the activity of a given catalyst.

QUALITY CONFORMANCE CIRCUITRY AREA: A test board made as an integral part of the multilayer printed board panel on which electrical and environmental tests may be made for evaluation without destroying the basic board.

RAW MATERIAL PANEL SIZE: A standard panel size related to machine capacities, raw material sheet sizes, final product size, and other factors.

REFRACTIVE INDEX: The ratio of the velocity of light in a vacuum to the velocity in a substance. Also, the ratio of the sine of the angle of incidence to the sine of the angle of refraction.

REGISTER MARK: A mark used to establish the relative position of one or more printed wiring patterns, or portions thereof, with respect to desired locations on the opposite side of the board.

REGISTRATION: The relative position of one or more printed wiring patterns, or portions thereof, with respect to desired locations on a printed wiring base or to another pattern on the opposite side of the base.

RELATIVE HUMIDITY: The ratio of the quantity of water vapor present in the air to the quantity which would saturate the air at the given temperature.

REPAIR: The correction of a printed wiring defect after the completion of board fabrication to render the board as functionally good as a perfect board.

RESIN: High-molecular-weight organic material with no sharp melting point. For current purposes, the terms "resin," "polymer," and "plastic" can be used interchangeably.

RESIST: A protective coating (ink, paint, metallic plating, etc.) used to shield desired portions of the printed conductive pattern from the action of etchant, solder, or plating.

RESISTIVITY: The ability of a material to resist passage of electric current through its bulk or on a surface.

ROCKWELL HARDNESS NUMBER: A number derived from the net increase in depth of an impression as the load on a penetrator is increased from a fixed minimum load to a higher load and then returned to minimum load.

SCHEMATIC DIAGRAM: A drawing which shows, by means of graphic symbols, the electrical interconnections and functions of a specific circuit arrangement.

SHADOWING: Etchback to maximum limit without removal of dielectric material from conductors.

SHORE HARDNESS: A procedure for determining the indentation hardness of a material by means of a durometer.

SINGLE-IMAGE PRODUCTION MASTER: A production master used to produce individual products.

SPECIFIC HEAT: The ratio of the thermal capacity of a material to that of water at 15°C.

STORAGE LIFE: The period of time during which a liquid resin or adhesive can be stored and remain suitable for use. Also called shelf life.

STRAIN: The deformation resulting from a stress. It is measured by the ratio of the change to the total value of the dimension in which the change occurred.

STRESS: The force producing or tending to produce deformation in a body. It is measured by the force applied per unit area.

SUBSTRATE: A material on whose surface an adhesive substance is spread for bonding or coating. Also, any material which provides a supporting surface for other materials used to support printed wiring patterns.

SURFACE RESISTIVITY: The resistance of a material between two opposite sides of a unit square of its surface. It may vary widely with the conditions of measurement.

TERMINAL AREA: A portion of a conductive pattern usually, but not exclusively, used for the connection and/or attachment of components.

TEST COUPON: A sample or test pattern usually made as an integral part of the printed board, on which electrical environmental and microsectioning tests may be made to evaluate board design or process control without destroying the basic board.

TETRA-ETCH*: A nonpyrophoric (will not ignite when exposed to moisture) proprietary etchant.

THERMAL CONDUCTIVITY: The ability of a material to conduct heat; the physical constant for the quantity of heat that passes through a unit cube of a material in a unit of time when the difference in temperatures of two faces is 1°C.

THERMOPLASTIC: A classification of resin that can be readily softened and resoftened by repeated heating.

THERMOSETTING: A classification of resin which cures by chemical reaction when heated and, when cured, cannot be resoftened by heating.

THIEF: An auxiliary cathode so placed as to divert to itself some current from portions of the work which would otherwise receive too high a current density.

THIXOTROPIC: Said of materials that are gel-like at rest but fluid when agitated.

THROWING POWER: The improvement of the coating (usually metal) distribution ratio over the primary current distribution ratio on an electrode (usually a cathode). Of a solution, a measure of the degree of uniformity with which metal is deposited on an irregularly shaped cathode. The term may also be used for anodic processes for which the definition is analogous.

UNDERCUT: The reduction of the cross section of a metal foil conductor caused by the etchant removing metal from under the edge of the resist.

VOLUME RESISTIVITY: The electrical resistance between opposite faces of a 1-cm cube of insulating material, commonly expressed in ohm-centimeters. The recommended test is ASTM D 257-51T. Also called specific insulation.

VULCANIZATION: A chemical reaction in which the physical properties of an elastomer are changed by causing the elastomer to react with sulfur or other cross-linking agent.

WATER ABSORPTION: The ratio of the weight of water absorbed by a material to the weight of the dry material.

WEAVE EXPOSURE: A condition in which the unbroken woven glass cloth is not uniformly covered by resin.

WEAVE TEXTURE: A surface condition in which the unbroken fibers are completely covered with resin but exhibit the definite weave pattern of the glass cloth.

WETTING: Ability to adhere to a surface immediately upon contact.

WICKING: Migration of copper salts into the glass fibers of the insulating material.

WORKING LIFE: The period of time during which a liquid resin or adhesive, after mixing with catalyst, solvent, or other compounding ingredients, remains usable. See *potlife*.

*Trademark of W. L. Gore and Associates, Inc.

Index

- Acceptability of fabricated circuits, 19-2 to 19-32
- Acceptability criteria:
- annular ring, 19-11, 19-12
 - base material edge, 19-10
 - determination of, 19-3
 - dimensional inspection, 19-23
 - electrical inspection, 19-27
 - environmental inspection, 19-29
 - mechanical inspection, 19-23
 - solderability, 19-24, 19-25
 - test patterns, 19-2, 19-3
 - test specifications, 19-30
 - thermal stress, 19-26
- (See also *specific process*)
- Acids, handling of, 7-36
- Acrylic conformal coating, 12-2, 12-4, 12-5, 12-8
- Additive circuits, 1-10, 1-11, 7-33 to 7-35
- full additive, 7-34
 - laminar preparation, 7-34
 - materials for, 2-19
 - pros and cons of, 7-35
 - semiadditive, 7-34
 - solderability considerations for, 14-13, 14-14
 - swell and etch process, 7-35
- Air pollution, 9-14 to 9-16
- Clean Air Act, 6-28
- Alcohol, 7-36
- Alkaline cleaner, 8-2
- Aluminum in etching process, 8-35
- Ammonium persulfate, 8-17 to 8-20
- waste treatment of, 9-9, 9-10
- Assembly:
- automatic (*see* Automatic assembly)
 - cleaning (*see* Assembly cleaning)
 - components, special, 10-9
 - conformal coatings, 10-11
 - considerations in, 10-5, 10-6
 - cost of, 10-5, 10-6
 - design for, 10-5, 10-6
 - economic justification of techniques of, 10-5
 - inspection of, 10-9, 10-10
 - of leads: clinching, 10-7, 10-8
 - performing, 10-7
- Assembly (*Cont.*):
- learning curve in, 10-6
 - methods analysis for, 10-7
 - performing leads, 10-7
 - presolder considerations in, 10-8, 10-9
 - process of, 10-4, 10-7 to 10-14
 - prototype process, 10-11 to 10-13
 - reference board, 10-4
 - soldering of, 10-9, 10-13
 - test of, 10-11
 - touch-up of, 10-9
 - visual aids for, 10-6
 - volume, large, 10-12, 10-13
 - volume considerations in, 10-6
 - work station, 10-7, 10-12
- Assembly cleaning, 16-2 to 16-20
- chloride determination test, 16-18
 - circuit effects, 16-14 to 16-17
 - ionic contamination, 16-15 to 16-17
 - MIL-P-28809, 16-15
 - contamination, types of, 16-2
 - defects related to, 16-12 to 16-14
 - measling, 16-14
 - white residues, 16-12, 16-13
 - equipment for, 16-8 to 16-12
 - brush cleaning, 16-9
 - vapor cleaning, 16-9 to 16-12
 - wave cleaning, 16-8, 16-9
 - Halphen-Hicks test, 16-17, 16-18
 - ionizable contamination, 16-18 to 16-20
 - Liebmann-Storch test, 16-2, 16-17
 - measurement of cleanliness, 16-2 to 16-4, 16-14 to 16-20
 - ionograph, 16-2, 16-3, 16-12, 16-19, 16-20
 - radioactive flux, 16-12
 - rosin contamination, 16-2, 16-3, 16-17
 - solvent-based cleaning, 16-4 to 16-6, 16-19, 16-20
 - standards for, 16-2, 16-3
 - toxicity, 16-5, 16-6
 - water-based process, 16-6 to 16-8
- Assembly repair, 13-2 to 13-18
- axial-lead components, 13-11, 13-12
 - component replacement, 13-7, 13-11 to 13-15
 - conductors, 13-3 to 13-7

- Assembly repair (*Cont.*):
 contact tabs, 13-10
 cost effectiveness, 13-2
 design for, 13-2
 guidelines for, 13-10
 laminate defects, 13-16, 13-17
 multilayers, 13-17, 13-18
 internal shorts, 13-18
 plated-through holes, 13-11, 13-15
 radial-lead components, 13-11
 rework: conformal coating, 13-2, 13-3
 preparation for, 13-2
 shorts, 13-6
 solder removal, 13-13 to 13-15
 soldering iron concerns, 13-11 to 13-15
 spurs, 13-6
 terminals, 13-7, 13-8
 missing, 13-10, 13-11
 warped boards, 13-15, 13-16
- Automatic assembly:
 axial-lead components, 11-5, 11-8 to 11-15
 component input to machine, 11-10 to 11-15
 component insertion, variations in, 11-8 to 11-10
 component preparation, 11-10 to 11-15
 sequencing, 11-18 to 11-21
 component spacing, 11-5 to 11-8
 dual-center-distance (DCD), 11-4
 variable-center-distance (VCD), 11-4, 11-9, 11-20
 cost savings in, 11-2
 insertable components, general, 11-2, 11-10
 insertion machines, 11-15 to 11-18
 integrated circuits (DIP) for, 11-6, 11-8, 11-10, 11-14, 11-20
 preparation for insertion, 11-17
 variations in, 11-15
 numerical control, 11-18, 11-34
 pantograph, 11-16, 11-17
 radial-lead components, 11-5 to 11-6
 preparation for insertion, 11-15
 variations in, 11-10
 variables: hole parameters, 11-3, 11-6
 layout, 11-2
- B-stage material [*see* Multilayer materials, prepreg (B-stage) materials]
- Brackets, 3-10, 3-12
- Bus bars, 3-4, 3-5
- Camera room, 6-5, 6-6
- Camera techniques:
 camera selection, 6-3, 6-4
 film, 6-4 to 6-7
 inspection, 1-42
 materials, 1-42
- Capacitance between conductors, 1-16, 1-27
- Characteristic impedance, 1-27, 1-28
- Chlorinated solvents, 7-36, 9-21, 9-22
- Chromic acid, 7-36, 8-25 to 8-27
- Chromium waste flow, 9-11, 9-12
- Cleaning (*see* Assembly cleaning)
- Clips, 3-10, 3-12
- Components:
 adjustable, 3-4
 automatic assembly of (*see* Automatic assembly)
- axial-lead, 3-3
 (*See also* Automatic assembly, axial-lead components)
- defective, 13-11
 density of, 1-7
 identification of, 1-37
 lead size, 3-2
 lead spacing, 3-2
 mechanical tolerances of, 3-2
 mounting of, 1-13, 1-14, 3-10 to 3-14
 multiple leads, 3-3, 3-4
 orientation of, 1-35, 1-36
 placement of, 1-34
 radial-lead, 3-3, 3-4
 (*See also* Automatic assembly, radial-lead components)
- replacement of, 1-31, 1-32, 13-11 to 13-15
 selection of, 1-31, 1-32, 3-2
 size and shape of, 3-2
 sockets, 3-4
 thermal considerations, 3-2
- Conductors:
 current-carrying capacity of, 1-22 to 1-27
 location of, 1-22
 pattern integrity, 19-15, 19-16
 repair of, 13-3 to 13-7
 resistance, 1-22 to 1-27
 (*See also* Copper foil, resistance)
- routing, 1-36, 1-37, 3-2
 shapes of, 1-24
 spacing of, 1-24
 temperature rise, 1-22 to 1-27
 thickness of, 1-15, 1-22, 1-23
 width of, 1-15, 1-22, 1-23, 19-12, 19-13
 (*See also* Copper foil)
- Conformal coatings, 1-22 to 1-24, 10-11, 12-2 to 12-9
 acrylic, 12-2, 12-4, 12-5, 12-8
 application of, 12-7, 12-8
 assembly repair through, 13-2, 13-3
 characteristics of, 12-6, 12-7
 chemical components of, 12-6
 contaminants in, 12-3
 curing temperature for, 12-6
 design for, 12-7
 diallyl phthalate (DAP), 12-3
 electrical properties of, 12-6

- Conformal coatings (*Cont.*):
 epoxy, 12-2, 12-3, 12-9
 limitations of, 12-3
 fungus resistance of, 12-6, 12-7
 humidity resistance of, 12-3 to 12-7
 inspection of, 12-8
 polyimide, 12-3
 polyurethane, 12-2, 12-8
 potlife, 12-4
 preparation for assembly, 12-7
 removal of, 13-2, 13-3
 repair of, 12-4, 12-8, 12-9
 safety precautions when applying, 12-9
 selection of, 1-33, 1-34, 12-4
 silicones, 12-3, 12-9
 storage of materials, 12-9
 thermal properties of, 12-6
 thickness of, 12-4
- Connectors: 3-5 to 3-11
 classes of, 3-8, 3-9
 contact type, 3-6
 design checklists for, 3-9
 disadvantages by type, 3-6
 one-part, 3-6, 3-7
 plating for, 22-6
 sheet connector specification, 3-10
 specifications and standards for, 3-8 to 3-10
 termination types, 3-7, 3-8
 two-part, 3-6, 3-7
- Contaminants (*see specific process element*)
- Copper foil:
 additive, 2-22
 color, 2-6, 2-7, 2-13, 2-29, 2-30
 current-carrying capacity of, 1-22 to 1-27
 in etching, 8-31
 in flexible printed wiring, 24-17 to 24-19
 inspection of, 2-2 to 2-3
 peel strength, 2-8 to 2-11, 19-26
 retention test, 2-23
 solder operations, 2-8 to 2-11
 Underwriters Laboratory, 2-10
 resistance, 1-26
 silk-screen ink adherence, 2-7, 2-8
 in soldering design, 14-5
 surface standards for, 2-6
 thickness standards for, 2-6, 2-7
 thin material, 2-14, 8-31
- Copper plating: 7-14 to 7-19, 7-39
 fluoroborate, 7-18, 7-19
 solution analysis, 7-39
 specifications for, 7-19
 plating rates, 7-20
 pyrophosphate, 7-14 to 7-16
 solution analysis, 7-37 to 7-39
 specifications for, 7-14
 sulfate, copper, 7-16 to 7-19
 solution analysis, 7-39
- Costs, relative, for finished boards, 1-5 to 1-9
- DCB (Dynachem photoresist), 6-9, 6-11
- Design of printed wiring, 1-4 to 1-43
 artwork generation, 1-13, 1-38 to 1-43
 for assembly, 10-5, 10-6
 automated techniques in, 1-15, 1-16, 1-38 to 1-41
 board strength, 1-18, 1-19
 bus bar layout, 1-39
 component mounting holes, 1-8, 1-9
 computer-aided design (CAD), 1-30, 1-38, 1-39
 conductor patterns (*see* Conductors) for conformal coatings, 12-7
 cost trade-offs, 1-6, 1-7
 digitizing, 20-26, 20-27
 dimensioning, 1-13, 1-14
 documentation, 1-42, 1-43
 multilayer, 20-25 to 20-29
 drawings, 1-14, 1-15, 1-35 to 1-38, 1-42, 1-43
 electrical factors, 1-26 to 1-28
 environmental factors, 1-28 to 1-33
 shock and vibration, 1-26 to 1-33
 flexible printed wiring (*see* Flexible printed wiring) for fundamental requirements, 1-5
 grid system, 1-15, 1-36 to 1-39, 2-22
 ground planes, 1-39
 holes (*see* Holes)
 index holes, 1-35
 interconnection capabilities, 1-8
 layout, 1-35 to 1-40
 checklist for, 1-39, 1-40
 component considerations for, 3-4
 lead patterns, 1-7
 material (*see* Laminates)
 mechanical factors, 1-17, 1-18
 mounting board, 1-18, 1-28, 1-29, 1-38
 (*See also* Components, mounting of)
 multilayer (*see* Multilayer design)
 multiple patterns, 1-41
 partitioning, 1-19
 photography techniques (*see* Camera techniques) for repair of assembly, 13-2
 shock and vibration, 1-28 to 1-31
 for soldering, 14-4
 specifications, 1-12 to 1-15
 American National Standards Institute (ANSI), 1-14
 Department of Defense, U.S., 1-13, 1-14
 Institute of Printed Circuits (IPC), 1-12, 1-13
 International Electrotechnical Commission (IEC), 1-14, 1-15
 test points, 3-14
 tolerancing, 1-12, 1-14
- DIP (dual in-line package) [*see* Automatic assembly, integrated circuits (DIP) for]

Drilling:

- backup material, 5-10, 5-11, 22-9 to 22-12
 - deburring, 5-12, 22-10
 - drill life, 22-10
 - drill shape, 5-8 to 5-11, 22-9 to 22-11
 - drill sharpening, 22-10
 - drill sharpness, 5-8, 22-10
 - drill spindle pressure feet, 22-11
 - feeds and speeds, 5-11, 5-12, 22-9
 - hole cleaning, 22-12
 - for multilayers, 22-8 to 22-12
 - plating problems, 8-5, 8-6
- Dynachem photoresist (DCR), 6-9, 6-11

EIA (Electronic Industries Association)

- solderability test, 14-7
- Electroless copper, 7-3 to 7-9, 22-12, 22-13
- formulations, typical, 7-5, 7-6
 - line steps, 7-3, 7-4
 - multilayer problems, 7-7, 22-12, 22-13
 - properties of, 21-9
 - thickness of, 7-4
 - troubleshooting, 7-7, 7-8
- Electroless plating (*see* Electroless copper, Plating)
- Engineering (*see* Design of printed wiring)
- Environmental Protection Agency (EPA), 9-2

Epoxy (*see specific applications*)

Epoxy formulas, 23-2

Epoxy laminate (*see specific material property*)

Epoxy smear, 19-8

Etch factor, 8-37

Etchback, 19-21 to 19-23, 22-12

Etching, 8-2 to 8-45

- post-etching steps, 8-2
- pre-etch cleaning, 8-2
- problems during, 8-8, 8-9
- resist problems, 8-4 to 8-11
- solutions (*see* Etching solutions)
- surface contamination, 8-8
- undercut, 8-8, 19-20

Etching area, 8-43

Etching equipment, selection of, 8-43

Etching facility, 8-43

Etching solutions, 8-11 to 8-27

- alkaline ammonia, 8-11 to 8-15
- ammonium persulfate (*see* Ammonium persulfate)
- chromic-sulfuric acid, 8-25 to 8-27
 - solder effect on, 8-26, 8-27
- cupric chloride, 8-14 to 8-17
- ferric chloride, 8-20 to 8-25
- general considerations, 8-2
- laminates, effect on, 8-27 to 8-31
- metals, effect upon, 8-31 to 8-37
- persulfates, 8-17 to 8-20
- solder, effect on, 8-27 to 8-31

Etching techniques, 8-37 to 8-43

- automatic, 8-42, 8-43
 - bubble etching, 8-38
 - immersion etching, 8-39 to 8-43
 - splash etching, 8-38, 8-39
- Eyelets, 1-26, 19-8 to 19-11

Fabrication (*see specific headings*)FEP (*see* Flexible printed wiring materials)

Ferric chloride, 8-20 to 8-25

Film, 6-4 to 6-7

diaz imaging, 6-5

polyimide, 25-16

Flexible printed wiring, 24-3 to 24-34

- access windows, 24-16
 - classifications of, 24-5 to 24-12
 - double-sided, 24-7, 24-8
 - multilayer, 24-8 to 24-13
 - single-sided, 24-6, 24-7
 - copper circuit, 24-31
 - conductor spacing, 24-17
 - conductors, 24-17 to 24-19
 - cover coat, 24-31
 - design of, 24-13 to 24-34
 - edge distance, 24-17
 - electrical parameters, 24-18 to 24-29
 - layout of, 24-13 to 24-17
 - mechanical, 24-29 to 24-34
 - tolerances for, 24-16, 24-17
 - disadvantages of, 24-5
 - electrical parameters, 24-25 to 24-29
 - attenuation, 24-29, 24-30
 - capacitance, 24-25, 24-26
 - cross-talk, 24-28 to 24-30
 - time delay, 24-27, 24-28
 - feasibility of, 24-13
 - hybrid, 24-8 to 24-12
 - interconnection, 24-13
 - mechanical design, 24-29 to 24-34
 - microstrip, 24-19, 24-24, 24-25
 - reliability of, 24-5
 - repairability of, 24-5
 - strip-line, 24-25 to 24-28
 - transmission line, 24-22 to 24-25
 - characteristic impedance, 24-24
- Flexible printed wiring materials:
- base dielectrics, 25-2 to 25-15
 - comparison of: film mechanical properties, 25-27, 25-28
 - general properties, 25-39
 - thermal properties, 25-37 to 25-39
- cover coats, 25-37 to 25-44
- bonding, 25-37 to 25-39
 - cost of, 25-43 to 25-44
 - cured properties, 25-41 to 25-43
 - laminating, 25-37 to 25-39
 - liquid coats, 25-39 to 25-41
 - for polyester material, 25-43, 25-44

Flexible printed wiring materials, cover coats (*Cont.*):

- for polyimide material, 25-43
- properties of, 25-37 to 25-44
- for PVC material, 25-44
- repairability of, 13-2, 13-3, 25-41
- flat cable, characteristics of, 25-42
- Fluorocarbons, 25-2 to 25-9
- Kapton (polyimide), 25-16 to 25-26
- Nomex (nylon), 25-8 to 25-11
- nylon (polyamides), 25-9 to 25-15
- polyesters, 25-37
- properties of, 25-38
- polypambanic acids (PPA), 25-26 to 25-36

Thermoplastic materials, 25-2

Fluoride, waste treatment of, 9-11

Fluoroboric acid, 7-36

Fluorocarbons, 26-2 to 26-8

Flush printed wiring, 19-22, 19-23

G-10 (*see* Laminates)

Gold in soldering system, 14-5

(*See also specific process element*)

Gold plating, 7-28 to 7-31

- etch-resistant qualities, 8-7, 8-32
- immersion, 7-32, 7-33
- recovery, 7-30, 9-20, 9-21
- as solder impurity, 15-6
- solution analysis, 7-43

Grid system, 1-36, 20-22

Hardware, 3-1 to 3-39

(*See also specific components*)

Heat sinks, 3-14

Holes, 1-25, 1-26

hole-to-lead ratio, 14-4

roughness of, 19-8

size of, 1-26

specifications, 19-4

Hydrochloric acid, 7-36

Hydrofluoric acid, 7-36

Hydrogen peroxide, 7-36

IEC (International Electrochemical Commission) solderability test, 14-8

Image transfer, 6-2 to 6-31

camera (*see* Camera techniques)

cleaning, board surface, 6-7 to 6-9

electroless copper effects, 7-3, 7-4

in multilayer, 22-2, 22-3

photoresist (*see* Photoresists)screen printing (*see* Screen printing)silk-screen (*see* Resists; Screen printing)

vapor degreasing, 6-7, 6-8

Immersion plating, 7-32, 7-33

- Inspection operations, 19-2 to 19-4
- Institute for Packaging and Circuit Interconnection (Institute of Printed Circuits) IPC, 1-12 to 1-14
- solderability tests, 14-6 to 14-8
- Interconnection (*see specific type of printed wiring*)
- Ion exchange (water), 9-19

Ketone, 7-35, 7-36

Kodak photoresists, 6-9 to 6-11

Kovar, solder design of, 14-5

Laminate machining, 5-3 to 5-15

blanking, 5-6, 5-7

life of tool, 5-6, 5-7

drilling, 5-8, 20-12

backup materials, 5-10 to 5-12

feeds, 5-11, 5-12

geometry of drill, 5-8, 5-11

locating system, 5-9

location of holes, 5-12

machine design (NC), 5-9, 5-10

machine selection, 5-9

nomenclature, 5-8

numerical control (NC), 5-9

positioning systems, 5-10

problems, 5-11, 5-12

speeds, 5-11, 5-12

piercing (*see* Punching holes, *below*)

punching holes, 5-4 to 5-6

routing, 5-12 to 5-18

double-pin method, 5-17

feed rate, 5-14 to 5-16

multiple-spindle machines, 5-13, 5-14

no-internal-pin method, 5-17, 5-18

numerical control, 5-14, 5-15

single-pin method, 5-17

spindle speeds, 5-15

sawing, 5-7

shearing, 5-7

Laminates:

additive process material, 2-19

amino, 8-28

B-stage, 2-2, 2-3

[*See also* Multilayer materials, prepreg (B-stage) materials]

base material tests, 2-12 to 2-27

blistering, 2-13

color standardization, 2-6, 2-29

conditioning, 2-12 to 2-20

copper foil characteristics (*see* Copper foil)

crazing, 2-13, 13-17, 19-6, 19-7

defect definitions, 2-12, 2-13

degree of cure, tests for, 2-15

delamination, 2-12, 2-13, 2-31, 2-32, 19-7

designations, material, 2-17, 2-18

Laminates (Cont.):

- dielectric breakdown, 2-25
- dielectric constant, 2-25, 2-26
- dielectric strength, 2-25
- dimensional stability, 2-14, 2-15, 2-33
- dissipation factor, 2-25, 2-26
- epoxy, etching concerns, 8-28, 8-29
- evaluation of, 2-5 to 2-17
- flame-retardant, 2-13, 2-14
- flammability of, 2-13, 2-14, 2-25
- flexural strength, 2-20, 2-21
- fungus resistance, 2-23, 2-24
- grades, standard materials, 2-17 to 2-20
- high-frequency materials, 2-19, 2-22, 2-23
- insulation resistance, 2-15, 2-16
- laminating, 2-2 to 2-5
- machinability, 2-8, 2-30
 - punched test specimen rating, 2-8, 2-9
 - (See also Laminate machining)
- measling, 2-13, 2-31, 2-32
- mechanical properties of, 2-20 to 2-23
- peel strength, 19-26
 - (See also Copper foil)
- phenolic, 8-29, 8-30
- polyimide, 8-30
- quality control, 2-5, 2-6, 2-27, 2-28
- silicone, 8-30
- silk-screen adherence, 2-7, 2-8, 2-27, 2-29
- solder float test, 19-26
- solder problems, 2-32, 2-33
- solder resistance, 2-11, 2-12
- surface problems (clad and unclad), 2-28, 2-29
- surface standard (copper), 2-6, 19-4, 19-5
- Teflon, 2-17 to 2-19, 8-30
- temperature, maximum operating, 2-22
- thermal conductivity, 2-23
- thermal expansion, 2-23
- thickness of base material, 2-20
- traceability, 2-27, 2-28
- treater (coater), 2-2
- troubleshooting of laminate problems, 2-27 to 2-33
- volume resistance, 2-17
- warp and twist in, 2-13, 2-20, 2-21, 2-31, 19-14, 19-15
- water absorption, 2-23
- weave exposure, 2-13, 2-31, 2-32
- weight of base material, 2-20

Laminating, 22-4, 23-1 to 23-19

B-stage [see Multilayer materials, prepreg (B-stage) materials]

bond, requirements for, 23-13

circuit preparation, 23-12

cycles, comparison of, 23-12 to 23-16

fixtures, 22-8, 23-6 to 23-8

maintenance of, 23-10

flexible circuitry, 23-6

Laminating (Cont.):

- gel time, 23-10
 - glass cloth, 23-2, 23-3
 - adhesion, 23-3
 - filament surface, 23-3
 - weave, 23-3, 23-10
 - laminate, copper-clad, 23-4
 - laminate preparation, 23-10, 23-11
 - lay-up, 23-11, 23-12
 - platen: flatness check, 23-9
 - heating check, 23-9, 23-10
 - prepreg:
 - characteristics of, 23-3, 23-10, 23-11
 - high-flow, 23-4
 - low-flow, 23-3
 - resin-flow, 23-4
 - press cycles, 23-13
 - press equipment, 23-7 to 23-10
 - loading equipment, 23-10
 - pressure transducer, 23-9
 - thermocouple welder, 23-8
 - press output, 23-12
 - press-type: compared, 23-8
 - flow test, 23-18, 23-19
 - pressure cycle, 23-13 to 23-15
 - cold start, 23-13 to 23-15
 - hot start, 23-13 to 23-15
 - single-stage, 23-14, 23-15
 - two-stage, 23-13, 23-14
 - registration pin, 23-6, 23-10
 - tack time, 23-10
 - troubleshooting laminating problems, 23-16 to 23-18
 - vacuum lamination, 23-15
- Layout (see Design of printed wiring layout)

Machining laminate (see Laminate machining)

Make-or-buy question (printed wiring boards), 4-3, 4-4

Materials:

- laminated (see Laminates)
- nail-heading, 5-12, 22-9
- soldering (see Presolder considerations; material systems)

Materials review board (MRB), 19-3

Measling, 16-12, 16-14

Measling, 2-13, 16-14, 16-15

Microsectioning, 7-43, 7-44, 19-17 to 19-20

Microstrip, 24-24, 24-25

Mounting, board, 3-10 to 3-13

(See also Design of printed wiring, mounting, board)

Multilayer boards, repair of, 13-17, 13-18

Multilayer design, 20-3 to 20-29

advantages and disadvantages of, 20-5, 20-6

applications of, 20-4

Multilayer design (Cont.):

- artwork, 20-26, 20-28
 - registration, 20-27, 20-28
- automated systems, 20-14
- capacitance, 20-19, 20-20, 20-21
- clearance holes, 20-6, 20-9 to 20-11
 - comparing plated hole and clearance hole, 20-11
- conductor spacing, 20-19
- conformal coatings, 20-6, 20-7
 - (See also Conformal coatings)
- corner marks, 20-28, 20-29
- cost of, 20-18
 - relative, 20-6, 20-7
- density, packaging, 20-6, 20-7
- design cycle, 20-14
- design process, 20-7 to 20-9
 - artwork, 20-7, 20-8
- digitizing, 20-26, 20-27
- documentation, 20-25, 20-29
 - dimensions, 20-25, 20-26
 - hole designation, 20-25
 - master drawing, 20-25
- drilling, 20-12
- electrical design, 20-18, 20-21
- fabrication technology, 20-6 to 20-8
- general consideration, 20-12 to 20-17
 - design checklist, 20-13
 - hole size, 20-18
 - interconnection features, 20-11
 - interconnections, 20-9, 20-24
 - landless holes, 20-23, 20-24
 - layout, 20-7, 20-20 to 20-27
 - layout details, 20-7
 - markings, 20-29
 - mechanical design, 20-17, 20-18
 - multilayer types, 20-8
 - externally connected, 20-8, 20-9
 - internally connected, 20-8
 - need for multilayer, 20-4, 20-5
 - plated-through-holes connection, 20-9
 - reduction targets, 20-29
 - registration, 19-8, 19-22, 22-2 to 22-7
 - specifications for, 20-12
 - terminal area, 20-18
 - test coupon, 20-29
 - thickness of board, 20-17, 20-18
 - tolerances, 20-14, 20-16, 20-21
 - tooling, 20-28, 20-29
- Multilayer fabrication, 2-6, 22-1 to 22-16
 - additive circuit, 7-66
 - drilling, 20-12, 22-8 to 22-12
 - drilling problems, 22-9, 22-10
 - epoxy smear, 19-8, 22-9 to 22-12
 - etchback, 19-21, 22-5
 - fixtures, 22-8
 - identification, 22-4
 - inspection, 22-14 to 22-16
 - laminating (see Laminating)
 - lay-up, 22-4

Multilayer fabrication (Cont.):

- outer circuit preparation, 22-3
- plating problems, 22-12 to 22-14
- post-laminate bake, 22-4
- process control, 22-15, 22-16
- process outline, 22-1 to 22-6
- registration, 22-2, 22-3, 22-6, 22-7
- tooling, 20-28
- Multilayer layout (see Multilayer design)
- Multilayer materials:
 - C-stage (cured laminate), 22-1
 - clad material: color standardization, 2-6, 2-7, 21-3
 - designations, 2-5, 2-6
 - electrical properties, 2-8
 - concerns, special, 21-6 to 21-8
 - dimensional stability, 21-4
 - thermomechanical analyzer, 21-4
 - electrical testing, 21-3, 21-4, 21-8
 - glass cloth, 21-4
 - glass transition, 21-4
 - laminate manufacture, 21-2
 - moisture absorption, 21-8
 - peel strength, 21-3, 21-7
 - polyimide, 23-4, 23-5
 - polyphenylene oxide (PPO), 23-5
 - prepreg (B-stage) materials:
 - characteristics of, 21-4, 21-5, 23-10, 23-11
 - epoxy properties of, 21-9
 - evaluation of, 21-2
 - flow test, 23-18, 23-19
 - handling, 23-11
 - high-flow, 23-4, 23-14
 - inspection of, visual, 21-5, 21-6
 - low-flow, 23-3, 23-4
 - MIL-C-5536A, designations for, 23-4
 - no-flow, 23-3
 - properties of, 21-8, 21-9
 - resin-flow, 21-5
 - resin starvation, 22-7, 22-8
 - specification of, 21-2 to 21-5
 - storage of, 21-10
 - tack time, 21-5
 - testing, 21-3, 21-4, 22-7, 22-8
 - thickness of, cured, 21-5
 - volatile content of, 21-4, 21-5
 - solder resistance, 21-3
 - specifications, 21-2 to 21-6
 - storage of, 23-11
 - surface standards, 21-2, 21-3
 - Teflon in multilayers, 23-5, 23-6
 - thickness of, 21-2, 21-5
 - tolerance, 21-7
- Mylar (see Flexible printed wiring materials, polyesters)
- Nail heading, 5-12, 22-9
- Nickel Watts, 7-41

Nickel plating, 7-22 to 7-25
 Nickel resist, 8-35
 Nickel sulfamate, 7-23, 7-41
 Numerical control (NC):
 assembly, 11-18
 drilling, 5-9
 Nylon (polymides), 25-9 to 25-15

Offset printing (resist), 6-30, 6-31
 Oil in soldering, 15-16

Packaging density (*see* Printed wiring)
 Photofacility, 6-5, 6-6
 Photography (*see* Camera techniques)
 Photoresists, 6-2, 6-9 to 6-19
 alkali-soluble, 6-2, 6-31
 baking, after developing, 6-19
 DCR, 6-9, 6-11
 developing, 6-18, 6-19
 dyeing, 6-19
 dry film, 6-2, 6-12 to 6-14, 8-9, 8-31
 water-soluble, 6-14
 in etching, 8-4 to 8-6
 exposure, 6-17, 6-18
 facility, 6-19
 liquid, 6-9 to 6-12
 airless spraying, 6-16, 6-17
 application of, 6-14 to 6-17
 board preparation, 6-17
 coving, 6-18
 dip coating, 6-15
 electrostatic spraying, 6-16
 negative-acting, 6-9 to 6-11
 positive-acting, 6-11, 6-12
 roller coating, 6-16
 spray, 6-16, 6-17
 whirl (spin), 6-15, 6-16
 work area for, 6-18
 in multilayer, 22-2, 22-3
 plating effects, 7-12, 7-13
 removal, 6-19
 ultraviolet (UV) curable, 6-2, 6-27, 6-28
 Plated-through-holes connection (*see* *specific process*)
 Plating:
 additive (*see* Additive circuits)
 adhesion, 19-23 to 19-25
 chemical safety, 7-35 to 7-37
 copper electroplating (*see* Copper plating)
 electroless copper (*see* Electroless copper)
 flowchart, 7-3
 gold (*see* Gold plating)
 image transfer effects, 7-11 to 7-13
 microsectioning, 7-43, 7-44
 panel plating, 7-5, 7-7, 7-9, 7-11
 pattern plating, 7-9, 7-10

Plating (*Cont.*):
 photoresist, 7-12
 process control, 7-37 to 7-43
 process evaluation, 7-13, 7-14
 screen resist problems, 7-11
 solder (tin-lead) (*see* Solder plating)
 thickness, measurement of, 19-16 to 19-20
 (*See also specific metal plating*)
 Pollution control:
 air pollution, 9-14 to 9-16
 consultants, 9-3 to 9-6
 disposal, contract for, 9-12, 9-13
 equipment costs, 9-14
 etching solution (*see specific etching process*)
 incineration, 9-13
 landfill, 9-13
 recycling, 9-17 to 9-23
 regulatory agencies, 9-16, 9-17
 safety, 9-13
 water pollution, 9-2 to 9-6
 Prepreg [*see* Multilayer materials, prepreg (B-stage) materials]
 Presolder considerations, 14-4
 material systems, 14-4, 14-5
 metallic surfaces, 14-5
 specifications, 14-4
 sulfur contamination, 14-5
 Printed wiring (*see* Design of printed wiring; Flexible printed wiring; Flexible printed wiring materials)
 Protective coating (*see* Conformal coatings)
 Prototype kits, 4-2
 Purchasing, 4-1 to 4-13
 cost factors, 4-9, 4-10
 delivery schedules, 4-10
 design trade-offs, 4-7
 documentation, 4-2
 master drawing, 4-12
 MIL-D-1000, 4-11 to 4-13
 guidelines, 4-3 to 4-5
 limited quantities, 4-2
 make-or-buy question, 4-3, 4-4
 multilayer specifications, 4-3
 multilayer trade-offs, 4-6, 4-7
 pricing, 4-9, 4-10
 problems, 4-8
 procurement practices, 4-2
 purchase orders, 4-4, 4-5
 specifications, 4-5, 4-6
 vendors: relations with user, 4-6
 selection of, 4-3

Quality assurance:
 calibrations intervals, 18-7, 18-8
 characteristics for levels, 18-6
 defects, causes of, 18-10 to 18-12

Quality assurance (*Cont.*):
 Hi-rel (high reliability), 18-6, 18-7
 inspection, 18-13 to 18-14
 levels of quality, 18-6
 reliability, 18-13
 sampling, 18-4, 18-8 to 18-10
 incoming, 18-8, 18-9
 in process, 18-9
 sequential grouping, 18-9
 survey checklist, 18-7
 test coupons, 18-4, 18-5, 19-2, 19-3, 20-29, 22-15
 testing: checklist, 18-13, 18-14
 circuitry, 18-4
 conditions, 18-7
 costs, 18-6
 final, 18-5, 18-6
 multilayer, 18-5, 18-9
 preproduction, 18-3
 production checklist, 18-7, 18-13, 18-14
 traceability, 18-12
 visual inspection, 18-10

Reference designations, 1-14, 1-37
 Registration, 19-8, 19-10, 19-22, 20-27, 20-28
 Resist equipment (*see specific resist type*)
 Resists:
 alkali-soluble, 8-3, 8-4
 asphalt-based, 8-3
 cellulose-based, 8-3
 cold-top enamels, 8-6
 electroplated, 8-6 to 8-9
 etching problems, 8-5, 8-6
 hole plugging, 8-4
 inks, 6-26, 6-27
 alkali-soluble, 6-26, 8-3, 8-4
 solvent-soluble, 6-26
 offset printing, 6-3, 6-30, 6-31
 oil base, 8-2, 8-3
 plated etch resists, 8-6 to 8-9
 printing problems, 6-28 to 6-30
 screened on, 6-3, 6-26 to 6-30
 solder resist, 6-2
 strippers, 8-2
 ultraviolet-cured, 6-2, 6-27, 6-28, 8-9
 vinyl-based, 8-3, 8-4
 wet etchable, 6-28
 (*See also* Image transfer; Photoresists)
 Reverse osmosis (RO), 9-19
 Rhodium plating, 7-31

Screen printing, 6-19 to 6-31
 area layout, 6-20
 cleaning (screen), 6-25
 fabrics, 6-24, 6-25

Screen printing (*Cont.*):
 frames, 6-23, 6-24
 hand, 6-24, 6-29
 hole plugging, 6-31
 inks, 6-26, 6-27
 (*See also* Resists)
 machine printing, 6-29
 materials, 6-21, 6-22
 plating effects, 7-11
 plating problems, 7-11, 7-12
 resist problems, 6-28, 6-29
 resist removal, 8-9 to 8-11
 stencil, 6-20 to 6-26
 Scrubbing, mechanical, 7-5
 Silicone, conformal coating, 12-3
 Silk-screen (*see* Resists; Screen printing)
 Silver:
 etch-resist qualities, 8-8
 etch-resistant, 8-36, 8-37
 migration, 14-5
 solder design for, 14-5
 Solder blanket, 15-12
 Solder blowholes, 17-8, 17-9
 Solder cold joint, 17-8, 17-9
 Solder contamination, 15-6
 Solder creams, 15-22 to 15-24
 Solder defects, 17-3 to 17-9
 Solder dross, 15-11, 15-12
 Solder equipment, 15-8 to 15-11, 15-17 to 15-19
 automatic systems, 15-17, 15-18
 solder pot, 15-10 to 15-17
 wave solder (*see* Wave soldering), 15-12 to 15-16
 Solder flux, 15-6 to 15-9
 activators, 15-7
 application of, 15-8, 15-9
 categories of, 15-7
 design of, 15-7, 15-8
 quality control, 17-1 to 17-9
 rosin-based, 15-7, 15-8
 residues, 16-2 to 16-6
 (*See also* Assembly cleaning)
 water-soluble, 15-8
 water-white rosin, 15-7
 Solder materials:
 alloys, 15-2 to 15-6
 composition of, 15-3, 15-4, 19-25, 19-26
 eutectic point, 15-2
 purity of, 15-3
 contaminants, common, 15-3 to 15-6
 impurities in, 15-3 to 15-6
 (*See also* Presolder considerations)
 Solder plating, 7-19 to 7-22
 etch-resist qualities, 8-6, 8-7
 in etching, 8-31, 8-32
 Solder preforms, 15-19 to 15-22
 Solder resists, 6-26 to 6-29, 14-12

- Solderability, 14-4 to 14-14
 acceptance criteria, 19-24, 19-25
 additive process, 14-13, 14-14
 plating effects, 14-12, 14-13
 poor solderability, causes of, 14-14, 14-15
 precleaners, 14-14
 preserving solderability (coatings), 14-8 to 14-10
 quality control, 17-1 to 17-9
 restoration of, 14-14, 14-15
 test methods: dip testing, 14-7, 14-8
 EJA-RS 17814, 14-7, 14-8
 globule test, 14-8
 IPC edge dip test, 14-14, 14-15
 IPC wave solder test, 14-17 to 14-19
 testing, 14-6 to 14-12
 tin-lead surfaces, 14-5, 14-10 to 14-12, 14-15
 wetting and, 14-5, 14-6
- Soldering:
 design for, 14-4
 with oil, 15-16
 quality control in, 17-2 to 17-9
 defective joints, 17-3 to 17-9
 defects, cause of, 17-3 to 17-9
 fluxes, 17-2, 17-3
 process control, 17-2, 17-3
 solder pot, 17-3
- Soldering irons, 15-8, 15-9
- Solvent-based cleaning (*see* Assembly cleaning)
- Specifications, list of, 19-21
 ANSI C8s.88, 3-8
 ANSI Y14.15, 1-14
 ANSI Y32.16, 1-14
 Federal Specification QQ-5-571, 15-2
 IEC-321, 1-14
 IEC-326, 1-15
 IPC-CM-770, 1-13
 IPC-D-300, 1-13
 IPC-D-310, 1-13
 IPC-D-390, 1-13
 IPC-ML-975, 20-12
 MIL-C-21097, 3-8
 MIL-C-55302, 3-8
 MIL-D-1000, 1-14, 4-11
 MIL-G-55636, 20-12
 MIL-I-46058, 1-14
 MIL-P-13949E, 2-5, 4-3
 MIL-P-55617, 20-12
 MIL-P-55640, 20-12
 MIL-P-81728, 7-19, 7-20
 MIL-STD-100, 1-14
 MIL-STD-275, 1-14
 MIL-STD-1495, 1-14, 20-2
 USASI Y14.5-1966, 1-14
- Specifying purchased boards (*see* Purchasing)
- Stainless steel, etch-resist, 8-35
- Stamped circuits (die-cut), 1-10
- Swell and etch, 7-34
- Terminal area, 1-15, 1-22
- Termination (*see* Connectors)
- Test points, 3-14
- Testing:
 design consideration, 1-19
 quality assurance (*see* Quality assurance, testing)
- Thermomechanical analyzer (TMA), 21-4
- Through-hole connections, 1-26
- Tin immersion, 7-33, 14-5, 14-14
- Tin-lead plating, 7-19 to 7-22
- Tin-lead surface, 14-5
 fused (reflow), 14-8 to 14-12
 problems, 14-11, 14-12
- Tin-nickel plating, 7-25, 7-26
 etch-resist qualities, 8-8
 gold-plated, 8-32
 solder-plated, 8-32
- Tin plate, 8-32 to 8-35
- Tin plating, 7-26 to 7-28
 etch-resist qualities, 8-7, 8-8
 specifications, 7-27
- Traceability, 18-12
- Transmission line, 24-22 to 24-25
- Ultraviolet (UV) curable resist, 8-9
- Undercut, 8-37
- Vapor cleaning, 16-9 to 16-12
- Vapor degreasing, 6-7, 6-8, 16-10
- Vibration, 1-28 to 1-32, 3-10 to 3-14
- Warp, cause of, 1-20
 (*See also* Laminates)
- Water-dip lacquers, 14-9, 14-10
- Water pollution, 9-2 to 9-6
 chemical waste, 9-2, 9-3, 9-6 to 9-12
 copper recovery, 9-23
 costs of treatment, 9-14
 effluent requirements, 9-3
 gold recovery, 9-20, 9-21
 palladium recovery, 9-23
 plating wastes, 7-13
 recovery, 9-17
 recycling, 9-17 to 9-23
 toxic materials, 9-2
 (*See also* Pollution control)
- Water treatment, 9-2, 9-17 to 9-23
- Wave soldering, 15-12 to 15-16
 bidirectional wave, 15-13, 15-14
 conveyor, 15-7
 design of wave, 15-12, 15-13
 lambda wave, 15-15, 15-16
 surface tension, 15-14, 15-15
 unidirectional wave, 15-13
- Wetting and solderability, 14-5, 14-6
- White residues, 16-12, 16-13

The second edition includes all-new discussions of such vital topics as:

- dry film photoresists
- repair of circuit boards
- acceptance and rejection criteria for circuits (with examples and illustrations)
- conformal coatings
- cleanliness testing
- new processes for plating, etching, and image transfer
- computers and other design aids
- waste treatment and environmental protection procedures and equipment.

These and all other aspects of the subject are covered by specialists who have been selected for their manufacturing experience as well as for their technical knowledge. Much of the material in the book has already been tested for clarity and ease of use in private seminars and in presentations before professional groups. A comprehensive glossary is included at the end of the book.

About the Editor



CLYDE F. COOMBS, Jr. is currently Calculator Group Manufacturing Manager at the Hewlett-Packard Company in Palo Alto, California. He was in charge of the development and installation of Hewlett-Packard's production processes for printed circuits and core memories, and was also an operating and Quality Control manager at various levels. He was the organizer and six-term president of the California Circuits Association, an influential trade association that has grown with the electronics industry in that state.

PRINTED CIRCUITS
HANDBOOK

Just a sampling of the vast range of specific problems this Handbook shows you how to solve...

- * How to design a cost-effective printed circuit board without compromising the material, hardware, or process
- * How to define and correct problems in one area that are affecting processes in other areas
- * How to make up, maintain, and correct chemical solutions for printed circuitry
- * How to dispose of environmentally dangerous chemicals acceptably
- * How to deal with environmental protection agencies
- * How to handle design and production problems in single-sided, double-sided, multilayer, flexible, additive, and hybrid circuits.

second
edition

McGRAW-HILL BOOK COMPANY

Serving the Need for Knowledge

1221 Avenue of the Americas, New York, N.Y. 10020

0-07-012608-9

