SOLDERING HANDBOOK FOR PRINTED CIRCUITS AND SURFACE MOUNTING

SOLDERING HANDBOOK
FOR
PRINTED CIRCUITS
AND
SURFACE MOUNTING

Design, Materials, Processes,
Equipment, Trouble-Shooting,
Quality, Economy, and
Line Management

HOWARD H. MANKO
Manko Associates Inc.
Teaneck, New Jersey

VNR VAN NOSTRAND REINHOLD COMPANY
New York

Library of Congress Catalog Card Number: 85-31449
ISBN: 0442-26423-2

Manufactured in the United States of America

Published by Van Nostrand Reinhold Company Inc.
115 Fifth Avenue
New York, New York 10003

Van Nostrand Reinhold Company Limited
Molly Millars Lane
Wokingham, Berkshire RG11 2PY, England

Van Nostrand Reinhold
480 Latrobe Street
Melbourne, Victoria 3000, Australia

Macmillan of Canada
Division of Gage Publishing Limited
164 Commander Boulevard
Agincourt, Ontario M1S 3C7, Canada

15 14 13 12 11 10 9 8 7 6 5 4 3 2 1

Library of Congress Cataloging-in-Publication Data

Manko, Howard H.
Soldering handbook for printed circuits & surface mounting.

Bibliography: p.
Includes index.
1. Printed circuits—Handbooks, manuals, etc.
2. Solder and soldering—Handbooks, manuals, etc.
I. Title.
TK7868.P7M36 1986 621.381'74 85-31449
ISBN 0-442-26423-2

SOLDERING HANDBOOK
FOR
PRINTED CIRCUITS
AND
SURFACE MOUNTING

As always to Mira

PREFACE

The printed circuit industry has achieved maturity and universal acceptance. No known interconnection technology threatens to render it obsolete in the foreseeable future. It offers two unique advantages that are important for any assembly technology: quality (reliability) and economy.

The mode of component attachment to printed circuit boards, however, is undergoing a radical change. Technical and economic pressures are forcing the industry to convert some or all of its assembly to surface-mounting techniques. We are moving away from the traditional large through-the-hole connection with its mechanical security. It is being replaced by a small surface butt and/or lap joint, sometimes with no added mechanical support to the solder. This change requires a complete reassessment of design, production, and inspection techniques. A major portion of this book is devoted to the changes imposed by surface mounting. This recent development is an extension of the established hybrid (thick- and thin-film) industry. Yet when it is applied to conventional printed circuits, there are major differences.

One must view the printed circuit board as a planar surface designed to provide interconnections between electronic devices. The electronic industry is using them for mass-production techniques to join discrete, integrated, and special components (leaded and leadless). This book applies to all board variations including single-sided, double-sided, multilayer, and flexible circuits.

This work concentrates on assembly, soldering, and cleaning, with emphasis on manufacturing techniques. Unfortunately, space limitations prevent us from dealing with the full intricacies of printed circuit board fabrication. We will, however, touch on those aspects that have a direct impact on the final quality of the solder joint, as well as its inspection and repair: board design and material considerations for reliable soldering, assembly techniques and their impact on joining, and management techniques. For obvious reasons, soldering is closely related to cleaning. Flux removal is only part of the true need for clean assemblies.

Space does not permit an in-depth discussion of solder technology. For a detailed study of this science, the reader is referred to a comprehensive soldering text by the author.* We will, however, include enough description of soldering practices to make this book a self-contained unit. Our main emphasis will be on methods of design and material selection, mass production processing, touchup, and repair techniques.

The quality of the final printed circuit assembly is not neglected. There is a chapter on quality standards and inspection techniques. It takes the reader from incoming inspection through process control to final inspection.

Finally, the book addresses important management issues of cost and control. In light of aggressive international competition, these are crucial problems. The health and future of the electronics industry in general and printed circuit manufacturing in particular depends on our ability to produce high-reliability products at the lowest possible cost.

Howard H. Manko

---

* *Solders and Soldering*, 2nd ed., McGraw-Hill Book Co., New York, 1979.

ACKNOWLEDGMENTS

No one has an internal source of information; we all gain our knowledge from a series of other people: parents, teachers, friends, co-workers, clients. We are enriched by reading and learning from a chain of many previous intellects, each one adding a little of themselves to the synthesis of the human experience.

Thus, it is truly impossible to acknowledge the numerous contributors to the present book, especially since many are not obvious even to the conscious mind. However, the author wishes to thank his colleagues at Alpha Metals, Hollis Automation, and Hexacon Electric for the specific help they have given in the preparation of this book. Special mention should go to Ron Bullewith, who has prepared many of the cross-sections presented herein.

An attempt has been made to represent many diverse pieces of equipment. The author wishes to emphasize that no endorsement of any models or makes is implied in their presentation. He has selected an extensive cross-section of those pictures vendors were kind enough to provide and which seemed appropriate to back up the text.

CONTENTS

1
DESIGN FOR GOOD SOLDERING AND CLEANING

1-1 PRINTED CIRCUIT PROCESSING—A LOOK INTO THE FUTURE

There are a number of basic methods used to produce the printed circuit board. The processes used to generate the image and the impact on soldering are important. Let us review them from the historical vantage point.

The oldest technique used is called *subtractive* and can be outlined as follows. The printed circuit board is made from a copper- clad laminate that is either single- or double-sided. A protective pattern is deposited on the copper surface, and the unwanted copper is etched away hence the name subtractive, which fits this method well. This process can be further divided according to the type of *etch resist* used.

On single sided boards and single layers of a multilayer board, the etch resist may be an organic coat (applied by screening or photographic techniques). After the exposed unwanted copper is etched away, the resist coating is removed. This leaves a bare copper pattern on the boards. This simple technique is also referred to as *print and etch.*

When a plated-through hole is generated, however, a tin-lead plating is used as the etch resist. This is the most common process for all double sided and multilayer boards. These boards end up with a tin-lead plating on the surface of the lands and pads. This solder coat is usually reflowed for technical reasons (see Section 1-24).

A more recent technique develops the metallic pattern on a laminate which is not copper clad. This method is referred to as the *additive* process. All boards made this way end up as bare copper. Coating them with solder is an additional unrelated step to the pattern development.

Today environmental impact is a major issue. The subtractive etching process requires a larger amount of effluent treatment relative to the additive process, which makes it more expensive. If it were not for the substantial investment made in older subtractive equipment and technology, more additive boards would be available. This situation incidentally, gave rise to an intermediate process called the *ultra-thin* or *semi additive* method. Here an extremely thin layer of copper is clad to the surface, protected by an easily removable coating like aluminum. After the top aluminum layer has been stripped, the pattern is build up with more copper. The whole structure is then minimally etched, which removes the thin layer of unwanted copper. A totally bare copper board is thus generated, using the conventional subtractive process.

Several board properties are important for soldering. One is the type of metallic trace exposed to the solder mask (see Section 3-16). The ideal combination is a nonfusible coating like copper under the mask, often referred to as *solder mask over bare copper* (SMOBC). This guarantees that the solder resist will not shift during soldering, preventing flux entrapment, flaking and other problems. We are also interested in the quality of the adhesion of the copper to the laminate, termed the *copper peel strength*. This can be directly related to the joint strength and to thermal damage during soldering like lifted pads and measling (see Section 7-26). Finally, we are concerned with the quality of the laminate surface after etching. This affects the cleaning process and the board's ability to absorb chemicals which tend to lower the surface insulation properties.

It is always difficult to predict future developments, but there are obvious trends. Cheaper laminates, thinner boards, and polymer thick films will help reduce cost. Technical improvements like higher temperature materials, higher insulation resistance, and modified coefficients of expansion are on the horizon. The major foreseeable changes, however, will be in two areas: *fine line* dense boards and *surface mounted devices* (SMD). Both strive to increase package density and reduce the number of total interconnections. This will raise the price of the finished board per square inch while lowering total assembly and device costs.

Fine line circuitry suits low current circuitry because surface insulation resistance becomes the limiting factor. We have the technology to make 2 mil lines on 2 mil centers, but today's consensus regards this as the lower limit. Surface mounting is discussed throughout the book, especially in Chapter 5.

Surface mounting has necessitated additional board materials and construction, with a controlled thermal expansion in the *X* and *Y* directions. These materials must match or at least come close to the *thermal expansion coefficient* (TCE) of the surface mounted devices. Otherwise, the solder joints with their diminished size and strength, cannot withstand large temperature excursions. Increased and modified resin coatings of the laminate offer one solution. Different re- inforcing fibers in the weave or internal layers of stabilizing metals can also modify board properties in this direction. Finally, changes in the metal layers laminated to the board also promise good TCE matching.

1-2 THE IMPACT OF BOARD MATERIAL

The grade of plastic used in the laminate has a great effect on the soldering process. To date, boards made of all the available resin grades can be readily soldered. The grade selected for a particular application usually represents a trade-off between price (economy) and scientific or engineering requirements (quality). Once a material has been selected, it is easy to set soldering parameters compatible with the laminate.

Here are the most important properties to consider during material selection. They affect both soldering and final quality, and are listed in order of importance:

1. *Warpage*. Unfortunately, the flatness of the printed circuit board in all stages of production is not easily maintained. It is normally due to residual internal stresses and/or the effect of additional mechanical operations. The lower cost laminates are more susceptible to warpage that starts in the drilling and etching stage. With multilayer, boards special stabilizing baking operations are needed to relieve the lamination stresses. The soldering process also induces warpage for obvious reasons.

   Since warpage always results in higher production costs, one must carefully weigh the material savings against this unnecessary labor increase. A true cost analysis will often reveal that a better grade laminate is actually less expensive and improves product performance.

   Warpage is also a problem in postsolder processing. Automatic lead cutting and testing are operations that depend on board flatness. The ability to install or plug the board into the equipment is another factor to consider.

   In surface mounting, the mechanical stability of the board becomes more important as compared to through-the-hole bonding. The accuracy of SMD device placement, the reproducibility of joint formation, and stresses under use conditions dictate stable geometries.

   The width of the board, which is a design function, also has an effect on warpage. Board thickness can be related to its ability to support its own weight. If the inherent board properties are ex-