

DECLARATION OF PAMELA STANSBURY

I, Pamela Stansbury, declare:

1. I am an Administrative Supervisor of the Cornell University Library, and specifically the Original Cataloging Unit, located at Cornell University, Ithaca, New York 14853. I am knowledgeable regarding the policies and procedures of the Library as they relate to the receipt, cataloging, and tracking of books and I have personal knowledge of the facts set forth below. I am familiar with these procedures because I have been employed by the Library since 1982, where I began in Technical Services as a Serials Assistant, then a copy cataloger in the Cataloging Unit in 1984. I have held my current position since the mid-1990s.
2. The Cornell Library holds an original copy of the monograph, *Printed Circuit Board Assembly* / P.J.W. Noble. New York : Halsted Press, 1989.
3. Exhibit A to this Declaration is true and accurate copy of the cover and front matter with library date stamp of *Printed Circuit Board Assembly*, from the Cornell Library collection. Exhibit A also includes an excerpt of pages 16 to 29 of that volume, showing Chapter 2 entitled "Board design for assembly."
4. I have been asked for information about *Printed Circuit Board Assembly* – specifically when this item was first made publicly available by the Cornell Library. From my review of the Library's records and my knowledge of the Library's standard procedures, *Printed Circuit Board Assembly* was cataloged and publicly available at the Engineering Library, in its public stacks, shelved by its Library of Congress classification system number within one or two days of June 29, 1989. A date stamp of "JUN 29 1989" on a preliminary page inside of the front cover of the item indicates that it arrived at the Engineering Library on June 29, 1989 and at the point would be made available to the public.
5. My opinion is based on the date stamp on a preliminary page inside the item's front cover, and my knowledge of the Library's standard procedures relevant to date stamping, cataloging, and shelving items. I have not seen any indication that *Printed Circuit Board Assembly* would have been an exception to the Library's standard practice. The Library maintains a searchable and publicly accessible catalog of its publications. Therefore, as of June 29, 1989 for *Printed Circuit Board Assembly* any member of the public could have located this item by searching our catalog by author, title, or subject matter.

I declare that all statements made herein of my knowledge are true, and that all statements made on information and belief are believed to be true, and that these statements were made with the knowledge that willful false statements and the like so made are punishable by fine or imprisonment, or both, under Section 1001 of Title 18 of the United States Code.

80217140_2

Pamela J. Stansbury (signature)

Pamela J. Stansbury (name printed)

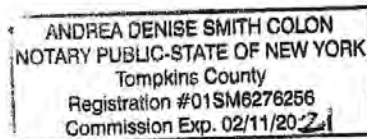
State of New York

County of Tompkins County

Sworn to before me this 12th day of February, 2020

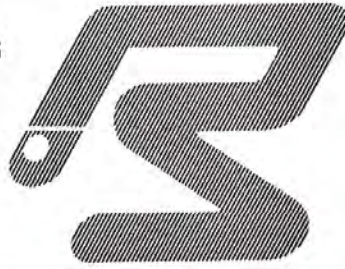
Andrea Smith Colon 2/12/2020

Notary Public



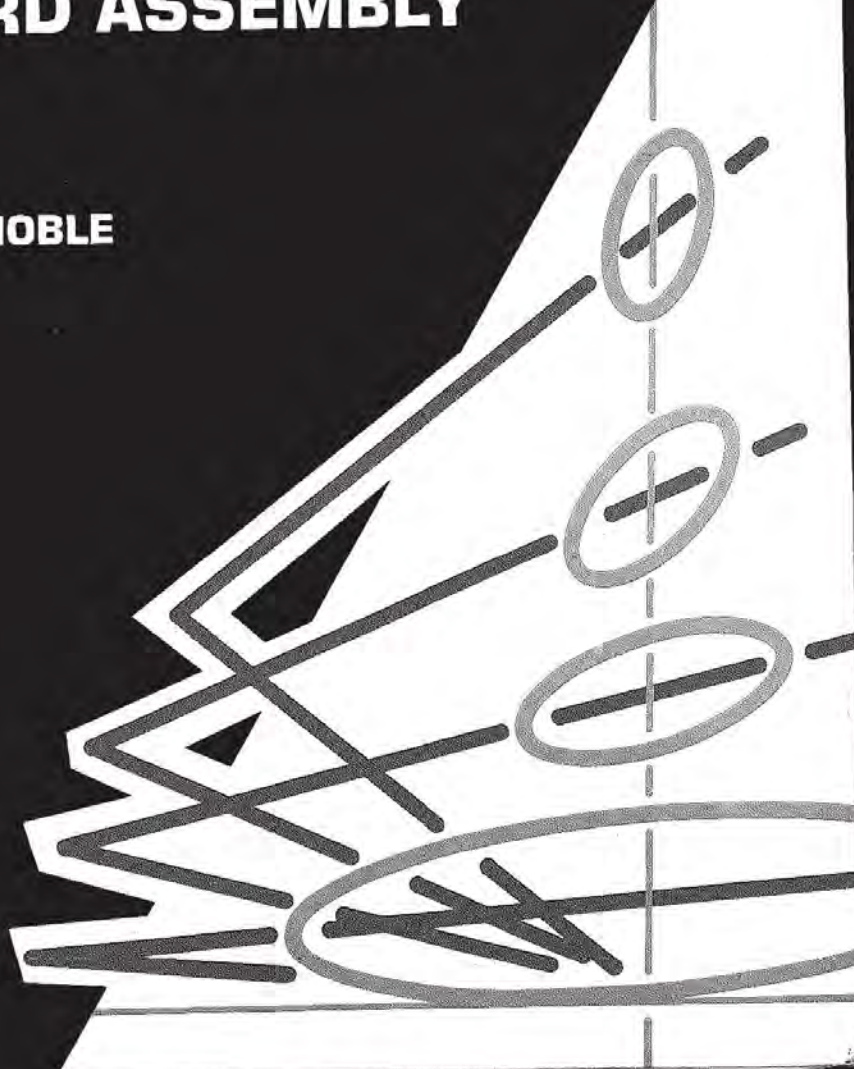
80217140_2

OPEN UNIVERSITY PRESS ROBOTICS SERIES
EDITED BY P.G. DAVEY



PRINTED CIRCUIT BOARD ASSEMBLY

P.J.W. NOBLE



OPEN UNIV
EDITED BY

Printed Circuit Board Assembly
The Complete Works

Preface – P. G. Davey, Meta Machines Ltd

Printed Circuit Board Assembly has been developed from knife and fork techniques (using human hands, fingers and odd mechanical tools, soldering irons, etc) to a very sophisticated fully automated process over the past thirty or forty years. This book describes the modern techniques available to the company or organization wishing to assemble printed circuit boards. These techniques vary from the manual techniques for the very small operation to the fully automatic systems which take the components, the printed circuit board, assemble them under vision control, solder the components in place and produce the finished high quality board. The ground covered includes the programming of the machines, a description of the machines and their operation, the various vision correction systems, the economics of assembly – and references to standards, techniques and the various machines available worldwide. Further material on manual, semi-automatic and robotic assembly, together with recommended approaches to board design and component selection for assembly, planning, staff training and surface mounted components make this a comprehensive book. The book should appeal to the engineer, student, accountant, designer – indeed any member of a team who needs to know more and understand the complexities and economics of printed circuit board assembly.

There are 100 figures and illustrations, a glossary of terms, a listing of standards, and their sources, and much technical detail for the engineer.

Peter Noble (CEng, CPhys, MIERE, MINSTP, MBIM) is a Technology Consultant.

Available in the USA, Canada and Latin America from Halsted Press, a division of John Wiley & Sons, New York – Toronto



ISBN 0 470 21266 7

Available in the UK and the rest of the world from Open University Press, Milton Keynes



ISBN 0 335 15418 2 (Paper)
ISBN 0 335 15419 0 (Cased)

NOBLE
PRINTED CIRCUIT BOARD ASSEMBLY

PRINT
BOARD

P.J.W. N

TK
7660
P7
H75

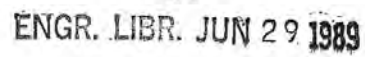
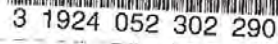
Engr
TK
7868
P7
N75

CORNELL
UNIVERSITY
LIBRARY



THE
WALTER S. CARPENTER, JR.
ENDOWMENT

ENGINEERING LIBRARY



FEB 19 1991	MAR 15 2000
MAY 14 1991	MAY 15 2000
AUG 16 1992	MAR 15 2004
NOV 14 1992	JUN 14 2011
OCT 21 1993	
MAY 14 1994	
JUN 14 1994	
DEC 11 1994	
FEB 17 1995	
APR 10 1997	
APR 25 1997	
MAY 14 1997	
CAYLOR	PRINTED IN U.S.A.

Open University Press Robotics Series

Edited by

P.G. Davey CBE MA MIEE MBCS C.Eng



This series is designed to give undergraduate, graduate and practising engineers access to this fast developing field and provide an understanding of the essentials both of robot design and of the implementation of complete robot systems for CIM and FMS. Individual titles are oriented either towards industrial practice and current experience or towards those areas where research is actively advancing to bring new robot systems and capabilities into production.

The design and overall editorship of the series are due to Peter Davey, Managing Director of Meta Machines Limited, Abingdon; Fellow of St Cross College, Oxford University; and formerly Co-ordinator of the UK Science and Engineering Research Council's Programme in Industrial Robotics.

His wide ranging responsibilities and international involvement in robotics research and development endow the series with unusual quality and authority.

TITLES IN THE SERIES

Industrial Robot Applications	E. Appleton and D.J. Williams
Robotics: An Introduction	D. McCloy and M. Harris
Printed Circuit Board Assembly	P.J.W. Noble
Robots in Assembly	A. Redford and E. Lo
Robot Sensors and Transducers	R. Ruocco

Titles in preparation

Intelligent Robotics	M.H. Lee
----------------------	----------

PCV 57

Printed
assembly
- the comp

P.J.W. Noble

HALSTED PRESS
John Wiley & Sons
New York - Toronto
and
OPEN UNIVERSITY
Milton Keynes

Printed circuit board assembly

- the complete works

P.J.W. Noble

HALSTED PRESS
John Wiley & Sons
New York - Toronto
and
OPEN UNIVERSITY PRESS
Milton Keynes

Open University Press
12 Cofferdge Close
Stony Stratford
Milton Keynes MK11 1BY

First published 1989

Copyright © 1989 P.J.W. Noble

All rights reserved. No part of this publication may be reproduced, stored in a retrieval system or transmitted in any form or by any means, without written permission from the publisher.

British Library Cataloguing in Publication Data

Noble P.J.W.

Printed circuit board assembly.

I. Electronic equipment. Printed circuit boards. Making -- Manuals

I. Title

621.381'74

ISBN 0-335-15419-0

ISBN 0-335-15418-2 Pbk

Published in the U.S.A., Canada and Latin America by
Halsted Press, a Division of John Wiley & Sons, Inc., New York.

Library of Congress Cataloging-in-Publication Data

Noble, P.J.W. (Peter J.W.)

Printed circuit board assembly: the complete works/P.J.W. Noble.

p. cm -- (Open University Press robotics series)

Includes index

ISBN 0-470-21266-7

I. Printed circuits -- Design and construction. I. Title.

II. Series

TK7868.P7N63 1988

621.381'74 -- dc 19

88 -- 15214

CIP

Typeset by Vision Typesetting, Manchester
Printed in Great Britain by the Alden Press, Oxford

Contents

Series editor's preface

Acknowledgements

Chapter 1 Introduction

An outline of the printed circuit board size, production and the assembly process processes.

Chapter 2 Board design

Methods of design from the designer's viewpoint, and series

Chapter 3 Component selection

The basic electrical specifications must be introduced for components and their specifications.

Chapter 4 Planning

Essential in the production of constituent components

Contents

<i>Series editor's preface</i>	vii
<i>Acknowledgements</i>	ix
Chapter 1 Introduction An outline of the printed circuit board assembly process, covering the elements of board size, production quantities, types, etc. as well as giving an introduction to the assembly process from design to finished board and the interactions in the processes.	1
Chapter 2 Board design for assembly Methods of design that will produce a good-quality design from the assembly viewpoint, and series of points that could be considered as 'Do's and 'Don'ts'.	16
Chapter 3 Component specifications and their impact The basic electrical specifications that are desired; also those specifications that must be introduced for good pcb assembly, such as mechanical, etc. Machine specifications and how they may differ from the international component specifications.	30
Chapter 4 Planning the kitting Essential in the process of pcb assembly, the kitting operation is producing the constituent components at the right time, correctly, and ready for assembly.	44

Chapter 5 Standards	58
The minefield of international standards has been met head on. A comprehensive list of all international standards that relate to the assembly process is listed, from component standards through soldering to final testing.	
Chapter 6 Planning the production area	69
The essentials of the production set-up, including automatic, manual, etc., also the automatic machine handling of components.	
Chapter 7 Programming the machine	86
A guide to the various methods of machine programming and the alternatives available.	
Chapter 8 Inspection – is it really necessary?	95
The basic questions are asked, namely: 'Is inspection necessary after ACI?' Arguments are put forward to show that in many instances it is not necessary to inspect at this stage, thus substantial savings can be made.	
Chapter 9 Choosing the best machine	98
Despite this title, no recommendation is made, but a series of guidelines is established to allow the potential user to carry out his own investigation, and eventual decision.	
Chapter 10 Manual, light-guided, semi-automatic and robotic assembly	117
An outline of the four methods of non-automatic assembly, with recommendations as to their use.	
Chapter 11 Training of operations, programming and planning staff	127
An essential part of every assembly plant is correct training for all levels of staff.	
Chapter 12 Cost considerations	133
Aimed at the financial deciders in the plant, as a guide to how to assess the benefits of automation.	
Chapter 13 Mass soldering technology	144
A guide to the various soldering techniques, etc. and a series of guidelines as to how to use the equipment effectively.	
Chapter 14 Surface-mount components	159
Chapter 15 Conclusions	166
Appendix 1 Glossary	168
Appendix 2 Standards	172
Index	202

Series e

Assembly of 'dit
important applic
example paint sp
published inform
process being aut
presented knowl
extremely subtle c
engineering terms
been covered at
depends crucially
process he or she
automation comp

The author of
source of great d
unique value not
understand how

Series editor's preface

Assembly of 'difficult' components onto printed circuit boards is emerging as an important application area for small, fast industrial robots. For other robot tasks – for example paint spraying or arc welding – the applications engineer can rely on a body of published information representing decades of accumulated knowledge about the actual process being automated. But for the process of assembly relatively little systematically presented knowledge exists, mainly because so much manual assembly depends on extremely subtle co-ordination of hand, eye and brain which is hard to represent directly in engineering terms. As for the particular processes of electronic assembly, they have hardly been covered at all in the literature. Yet the design of a good PCB automation system depends crucially on the responsible engineer fully understanding every aspect of the process he or she is automating, whether working for the electronics manufacturer, an automation company, a research laboratory or a machine builder.

The author of this book has had extensive practical experience in all these roles: as a source of great detail on most aspects of the electronic assembly process it will be of unique value not only to the robot specialist but well beyond that to anyone needing to understand how printed circuit boards are manufactured.

P.G. Davey

Acknowledgements

The author is indebted to many companies and individuals from within the pcb assembly industry. The author has been working within and with the industry for many years and over that period of time has made many friends, and feels it true to say no enemies. In that time he has worked for a number of the companies mentioned and he is pleased to say that he has remained on good terms with all of them. It is through the goodwill that has been generated that a high level of co-operation has been received by him from all the participating companies and individuals. In particular the author would like to thank the following companies by name and in alphabetical order:

Ambotech Limited; Amistar; The Automation Group of Companies; Dynapert; Elite Engineering; Prolec Limited; The Plessey Company Ltd; Stickland Electronics Ltd; Surface Electronics Ltd; Teknis Limited; TDK; Universal Instruments Ltd; and Zevatron (Tony Chapman Electronics Ltd), as well as the journal *Electronic Production*.

The author would also like to thank a number of individuals in the industry, particularly two individuals, namely Roy Carr, of Roy Carr Limited who produced the draft for the chapter on soldering, and Bill Mitchell who produced the draft for the chapter on standards and electronic production.

In addition, the author thanks the following individuals from within the industry, not necessarily in any particular order; they are: David Buckley, Peter Fenner, Tim Perfect, Brian Atherton, David Johnson, John Stickland, Adrian Barnes, Andrew Clarke, Peter Ashburner, Doug Every, David Tappin, Gordon McDonnell, Ken Carter, Paul Blundell, Mike Judd, David Bishton, Bernard Parry, and many others who, I am afraid, will not excuse me for apparently forgetting them by name.

Whilst acknowledging the co-operation and help both the individuals and the companies have provided, I would like to say how difficult it has been to compile the

book, in view of the broad subject covered. During the writing of the book, from starting out with the proverbial blank sheet of paper and to some degree a blank mind, it has progressed over a period of months to the present stage, which the author feels at least is a start on a major topic. I hope that in two or three years' time a further book can be produced updating this issue which will incorporate more of the latest technology, particularly the technology on surface-mounted components which in this volume is dealt with in only one chapter, although it is mentioned in many of the others.

Should any individual or company feel that they have been either ignored or left out of the book, the author would like to say that this is not deliberate in any way, shape or form; it is simply that every company known to the author was approached at least three times for information and help by direct letter, telephone, and by sending facsimile requests. The vast majority of companies and individuals replied and the majority of those were most helpful in providing not only photographs and information, but by being totally free in allowing me to reproduce part of their work and texts as examples within the body of the book. Inevitably one or two companies and individuals did not reply or respond in any way and it is those companies and individuals which have to be left out, as it can only be assumed they wish to have no part in the book, regrettable though that may be.

The author would also like to acknowledge the support of his family while writing this book as it requires considerable concentration, and interruptions tended to be met with less than polite response! Finally the author would like to thank Lynn Kilpatrick-Brown for her sterling efforts in typing the manuscript and carrying out a number of subsequent editing functions as the author's mind, concepts and considerations changed; and Mark Noble for his photographs of those machines and equipment, etc. not provided by the manufacturers. The author sincerely hopes that the book at least in part, if not in total will be useful as a guide to the many ways of assembling pcbs and looks forward to receiving comments, criticism (constructive), and help with future editions.

Peter J.W. Noble

Chapter

Introduc

The design, procure printed circuit board manufacturer depend consumer or any company nothing of the requirements cleaning and soldering vast range of choices to anyone involved circuit board assembly assembling printed circuit take them to full au

The implications of commit themselves without full consideration which include components mechanically. That is how it should relate to and assembly.

In most industrialize summed up as follows a huge 60 x 60 cm (6 tendency is to assemble

Chapter 2

Board design for assembly

2.1 Application areas

The application at which the printed circuit board is aimed has a significant effect on its design and layout. The impact of the application runs right through from layout and design to the type of machine chosen for assembly and the soldering techniques used; therefore from the outset it is important to establish which particular type of application the board is headed for in your environment. The basic applications, and considerations, are those of the consumer market, the communications market, the instrumentation market and the military market – and what impact these then have on the space available for the printed circuit board and the standards of assembly required.

Other applications will generally fall within one of the above headings, i.e. control systems would fall into the instrumentation market without detracting from the basic precepts. The different applications have substantially different impacts on design, depending upon the detail consideration of that application. For example, in the consumer field, questions of standards of assembly may be subject to the economics involved on a very simple board with very high volume production and may not require very high standards or indeed a very high reliability of assembly, as it is often cheaper to throw away faulty assemblies than test and re-work. Similarly, the design considerations for these high volume units have got to be considered in the light of the cost of assembly and the cost of the raw materials and components involved, rather than the ultimate reliability of the end product.

Assembly cost in the military application, on the other hand, usually has slight economic import, but very high levels of requirement with regard to the standard of the components, the design, the standard of assembly, and in the administration of assembly, such as traceability. The instrumentation market falls in between these two, in that the standards have to be good, and reliability of the end product high, but cost still plays an

important part. The instrumentation in the market might encounter one heading where it is a requirement for high reliability.

Running through the product, i.e. is it required for personal phones for which we are going to be discussed in Chapter 3.

Before going into the assembly procedure, design before going to the first of these is the design of axial, radial and connectors, transistors and conductors with the component in general automatic assembly component. Although usually a specialist is considering robotic assembly.

Chapter 3, on various standard components and tolerances applicable ready to be placed consideration has to be given to which in the case of radial in the case of axial components of the leads matches is illustrated in Figure 1. This implies, is simply by types of preform used for relief for the component assembly – more for now becoming obsolete.

When assembling components cropped to length or soldering machine component in the board.

important part. The communication market is probably a cross between military and instrumentation in terms of standards in that a high reliability is expected of product and the cost is not of vital importance. In the days of personal mobile phones and other forms of advanced portable communication it could even be considered that the communication market might encompass the consumer market and the instrumentation market under one heading where in the high volumes required for the consumer application there is also a requirement for high reliability and high standards of product procurement and design.

Running through the above applications are considerations of space of the end product, i.e. is it required to fit any particular environment (and here we might refer back to personal phones for example), and in the case of the military in particular, standards to which we are going to work? Military and other relevant standards are designated and discussed in Chapter 4.

2.2 Basic considerations

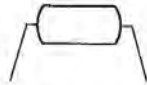
Before going into layout and design considerations one should review the standard assembly procedures and requirements, so that there is an understanding of the basis for design before going into more detail. In essence there are two pcb assembly techniques: the first of these is the use of leaded components, the component falling into the categories of axial, radial and dual in-line (DIL); there are other types of component such as connectors, transformers and various passive devices, as well as high-power semi-conductors with their own particular type of connection. Typical examples of the type of component in general assembly are illustrated in Figure 2.1. For the emphasis on automatic assembly we are really considering only the axial, radial and DIL type of component. Although there are some machines capable of assembling all types, this is usually a specialist area, and therefore will only be referred to in outline, under the section considering robotic assembly techniques in Chapter 10.

Chapter 3, on component specifications and component procurement indicates the various standard component sizes, shapes and dimensions together with a range of tolerances applicable to components both as individual items and packaged in a form ready to be placed on an assembly machine. When assembling leaded components consideration has to be given to the preparation of the component prior to assembly which in the case of all three components requires control over the leads, and particularly, in the case of axial components requires those leads to be bent in such a way that the pitch of the leads matches the pitch of the holes through which the leads should be placed. This is illustrated in Figure 2.2 which shows a typical goal-post preform which, as the name implies, is simply bending the leads at right angles on the correct pitch. There are other types of preform usually employing a joggle in the lead, which acts as a stand-off or strain-relief for the component body. This technique is not generally used for automatic assembly – more for manual assembly and usually manual assembly of a type which is now becoming obsolete.

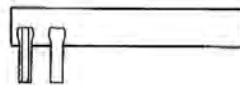
When assembling new components into a board prior to soldering, the leads have to be cropped to length or alternatively, cropped to a specific length, depending on the type of soldering machine or technique to be used. Also, it is usually necessary to retain the component in the board prior to soldering, because the board will need to be transferred



Typical axial component

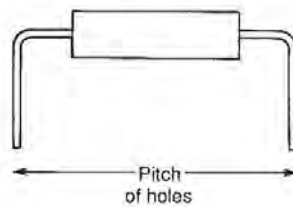


Typical DIP (Dual In-line Package)



Typical radial component

Figure 2.1



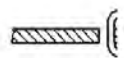
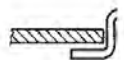
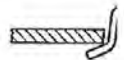
'Goal-post' preform

Figure 2.2

from the assembly machine components, some for component on the board to hold the component and bend the leads so they then not escape. A relief of any track on the board bending the lead also in the soldering phase.

One alternative technique process the board through. In this technique the board place (that is none of the) taken to ensure that the leads are isolated from a high-speed rotating leads. The board then was retaining the component has been and is still to be for the very high effectively a dual sold

The effect both of both illustrated in Figures 2 by different machines. surface of the board at detail here depends on to use, but the normal soldering area of all side surface.



from the assembly machine to a soldering machine and, to increase the security of those components, some form of retention is desirable. Various techniques for holding the component on the board have been tried and are still in use, such as using a plastic coating to hold the component in place. However, the most commonly used technique is to crop and bend the leads simultaneously under the board in such a way that the component can then not escape. A refinement of the lead-bending technique is to bend along the direction of any track on the board, rather than bend the lead over a gap between tracks. By bending the lead along the track this decreases the chance of shorts, etc. during the soldering phase.

One alternative technique for retaining the components is to leave the leads long and process the board through soldering, using what is termed a solder-cut-solder technique. In this technique the board is processed through the soldering machine with long leads in place (that is none of the leads are cropped). Care during the soldering operation has to be taken to ensure that the surplus solder is removed from the leads in such a way that the leads are isolated from one another correctly. After this process the leads are then cut with a high-speed rotating blade just beneath the board to give a low height profile to the cut leads. The board then proceeds over a second soldering system which melts the solder that was retaining the components in place and cover the cut edge of the lead. This technique has been and is still being used successfully by some companies, but the application tends to be for the very high-volume end of the business as the capital investment in what is effectively a dual soldering machine and cropping system in line is substantial.

The effect both of bending the leads and bending the leads along the axis of the tracks is illustrated in Figures 2.3 and 2.4. Figure 2.3 demonstrates various angles of bend achieved by different machines and different techniques, in some cases the lead is bent flat along the surface of the board and in other cases only bent sufficiently to retain the component. The detail here depends on the company's particular choice in assembly techniques it intends to use, but the norm is for the lead to be bent at an angle of around 45° allowing a clean soldering area of all sides of the leads, and ensuring that the lead is wetted throughout its surface.

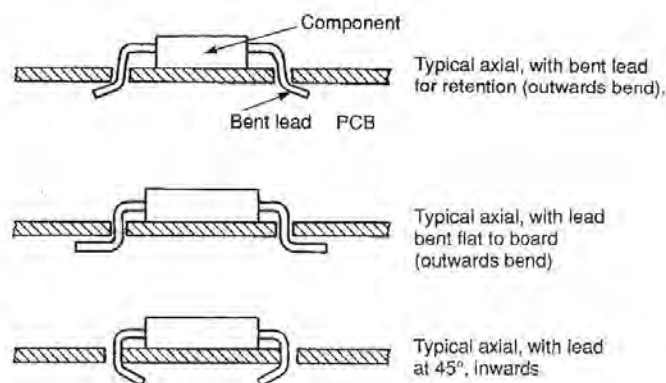


Figure 2.3

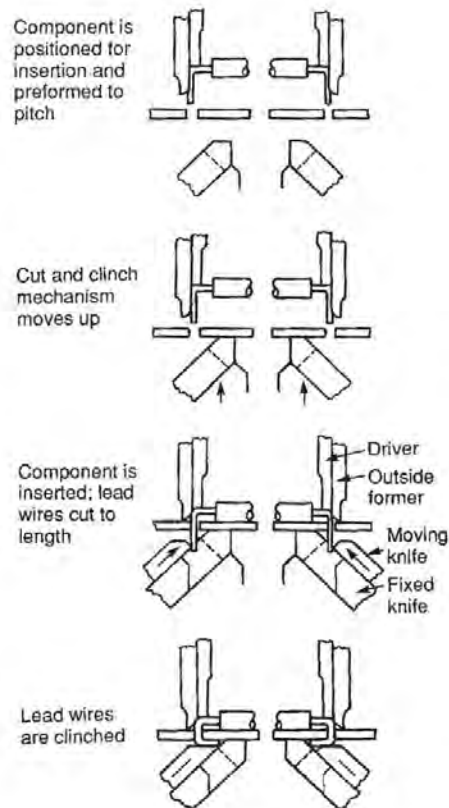


Figure 2.4

The other major category of component type is the leadless, or surface mounted, component which, as the name implies, has no leads for either retention or contact and is mounted on the surface of the printed circuit board and soldered *in situ*. Figure 2.5 illustrates different types of surface-mounted component assembled on to a printed circuit board.

Surface-mounted components are retained on the board prior to soldering in a number of ways; however, all the methods effectively boil down to the use of a glue, in order to create adhesion between the component and printed circuit board. The glue can be deposited in a number of ways either in one operation by using screen printing techniques, or depositing on the board just prior to the application of the component, and the author is aware of one technique for coating the component prior to placing it on the board. A second class of adhesion prior to soldering is the use of solder paste which acts as a glue to hold the component in place, and then by heating techniques is converted to solder to



form the contact between various constraints at leads and the adhesion

With printed circuit possible to achieve a creates the printed a backing. This, howe form of printed circ circuit either $\times 2$ or $\times 3$ be minor at this stage form these errors can for assembly. The us commonplace. The techniques which int the intention of this t indicate the end re manufacturer can be form of CAD system

This is a system w

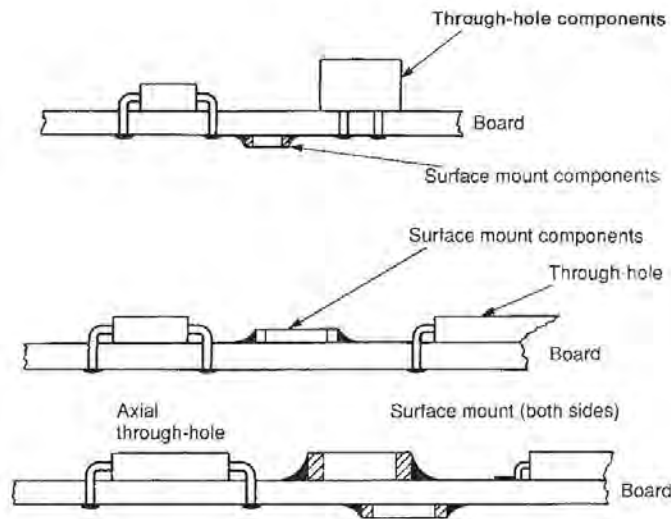


Figure 2.5

form the contact between the component and the track. When designing a board these various constraints and requirements should be borne in mind, particularly the bending of leads and the adhesion of components to the pcb.

2.3 Layout techniques

With printed circuit boards, whether they be single-layer or multi-layer, it is always possible to achieve a layout using manual methods, i.e. the designer or draughtsman creates the printed circuit board interconnection pattern, using tapes and translucent backing. This, however, has a number of significant disadvantages which apply to any form of printed circuit board application. Even when using a grid and laying-out the circuit either $\times 2$ or $\times 4$ it inevitably leads to mechanical and positional errors. These may be minor at this stage, but when the eventual printed circuit board is produced in its final form these errors can multiply, particularly if one is considering step and repeat methods for assembly. The use of computer-aided design (CAD) systems is becoming much more commonplace. The latest forms of CAD systems have allowed for development of techniques which introduce a substantial element of self-checking into the layout. It is not the intention of this book to discuss the merits of different types of CAD system, merely to indicate the end result that the CAD system should produce in order that the manufacturer can have the best product for his use. Thus I have chosen to describe one form of CAD system which is used widely throughout the printed circuit board industry.

This is a system which is to a large degree self-checking, in that the final printed circuit

board layout is guaranteed to follow the block diagram or circuit diagram that the designer produces; it does not however necessarily produce point-to-point full wiring layout. The system actually produces a complete bird's-nest interconnect based on the logic diagram. This is then manipulated by the operator or programmer until a satisfactory printed circuit board layout is achieved within the mechanical constraints required by the system. The final result is a manipulation of the bird's-nest – but the human operator is vital in placing components correctly for best electronic and interconnection effect. The final result is also guaranteed to follow the original block/circuit diagram.

The mechanical layout of printed circuit boards and the choice of grid, etc. should take account of the processes following after the manufacture of the bare board. After the bare board is made the processes may include bare board testing, for those companies who test their boards prior to assembly. It includes automatic component insertion or possibly manual or semi-automatic insertion (or indeed in some companies a combination of all three), and then finally it includes the cleaning, soldering, and testing of the board assembly.

In many cases the printed circuit board shape and layout is not controlled by the designer of the electronics of the board, but by the mechanical constraints of the final product. In the majority of cases the board will be rectangular but in some applications boards can be peculiar shapes to fit handsets, to fit into mechanical devices, and may also include a flexible printed circuit board. In all these cases early consideration has to be given to the *assembly* of the printed circuit board. Occasionally it is not possible, due to constraints of space, to satisfy all the needs of the assembly operation, but it should be borne in mind that cost can rise dramatically when difficulties are encountered during assembly, created by poor printed circuit board layout.

First we should consider the choice of grid. The norm in the industry for many years, indeed decades, has been a grid of 0.1 in., and this for virtually any application is ideal. It is suitable for bare board testing using multi-probe apparatus, it is suitable for automatic component insertion, and subsequently suitable for the handling through cleaning and soldering, and final testing which again may include a vacuum-based, multi-probe system. The fundamental 0.1 in. has not been chosen by accident for printed circuit board grids; it is a circumstance forced on the board designer by the designer of the integrated circuit package or transistor package which, despite metrication in practically every other industry, is still on a 0.1 in. grid (dual in-line packages of 0.1 in grid being the best known example). Thus it is sensible to choose a 0.1 in. grid. However, it should be borne in mind that this is not necessarily possible with some high-density boards. If it is necessary to decrease the grid spacing it is again sensible to do so in sub multiples of 0.1, automatic component insertion either to 0.05 in. or 0.025 in., but the restrictions on (ACI) machines and test systems should be considered.

The first of these is for the multi-probe test systems which have been designed to cope with a 0.1 in. grid. Probes, and their holders in vacuum fixtures, have been designed specifically with that restriction in mind. It is of course possible to get probes operating on smaller grids but the cost will rise rapidly and the reliability of the test system will decrease due to the more demanding engineering requirements. A way round the test problem is of course to use test points on a 0.1 in. grid, with via holes set off-grid, then the restrictions on the testing and automatic component insertion largely disappear.

However, this does systems to sense the h currently in use the separated only by a 0. to insert through the which, and therefore again the designers h around where the hol a 0.05 in. grid then becomes impossible machines with built-majority of printed o mind that via holes o cause potential probl ACI and therefore r

There are of course type of problem to th use vision systems us holes – as the machin that the holes for AC vision systems, and t board. It is certain possible for insertio with reliable insertio are the target for an

Depending upon t for holding the boa mended by Univers exactly the methods Most manufacturers form of clamping n limitations of the va

2.4.1 Board opening

Board openings, or v located within the i begins at machine z bushing, and proce

When designing v circuit board as muc surface extending be

However, this does introduce problems with some ACI machines which use vision systems to sense the holes prior to the insertion of the component. In all the vision systems currently in use there is no aspect of pattern recognition; thus two holes appearing separated only by a 0.05 in. grid may confuse the system, and the result is either an attempt to insert through the via hole, or, alternatively a failure of the system to see which hole is which, and therefore the machine stops and awaits operator intervention. However, once again the designers have attempted to get round this problem by allowing for a box search around where the hole is expected to be. If the target hole has two holes either side of it on a 0.05 in. grid then it is almost certain that the vision system will fail and thus ACI becomes impossible without operator intervention. It is recommended that for ACI machines with built-in vision systems which greatly enhance their usefulness to the vast majority of printed circuit board assemblers, then circuit design layout should bear in mind that via holes or other holes close to holes for insertion of leads of components can cause potential problems. It would be sensible to check which machine one has in mind for ACI and therefore relate the design restrictions back to that machine.

There are of course some machines that do not use vision systems; this presents another type of problem to the printed circuit board manufacturer. Those machines which do not use vision systems usually have a very tight specification on the location of the insertion holes—as the machine is operating blind, this is an obvious consequence. It also can imply that the holes for ACI need to be larger in diameter than for manual insertion, or with vision systems, and this can have an effect on both skyline and the finished quality of the board. It is certainly recommended that designers adhere to a 0.1 in. grid wherever possible for insertion machines, and locate via holes and other holes that might interfere with reliable insertion at places where they do not interfere optically with the holes which are the target for any vision system.

Depending upon the type of machine to be specified one also has to consider the system for holding the board in position. Reproduced below are the considerations recommended by Universal Instruments extracted from their design guidelines which shows exactly the methods and systems that they use for holding down the printed circuit board. Most manufacturers have similar systems employing two holes for location and some form of clamping mechanism and the designer of the board should be aware of the limitations of the various systems, before proceeding.

2.4 Workboard holder considerations

2.4.1 Board openings

Board openings, or windows, must be positioned so that all components to be inserted are located within the insertable area of the insertion machine. Maximum insertable area begins at machine zero, located a defined distance from the lower left table reference bushing, and proceeds in a positive *X* and *Y* direction from that point.

When designing workboard holder openings, it is desirable to support the printed circuit board as much as possible with at least 0.125 in. (3.18 mm) or workboard holder surface extending beneath the board at all support areas. Supporting the board at all four

corners is preferable; larger boards may require additional support near the middle of long sides.

A certain amount of edge clearance is required to make room for cut and clinch operations when inserting components close to the edge of the board. Where manual board loading and unloading is used, additional 'finger room' should be provided at the right and left sides of the windows to allow easy loading and unloading of the printed circuit board.

The acceptable distance between the edge of the board opening and an insertion varies with the type of component, machine, insertion head, and cut and clinch unit. In each chapter of these design guidelines, there is a paragraph on uninsertable area for the specific type of component involved. Refer to the chapter which applies to the specific component type to be inserted to determine its uninsertable area. In general, printed circuit designs requiring components inserted close to locating references and board edges may result in loss of automatic insertion of these components due to board support requirements.

2.4.2 Locating methods

Locating hole, 'V' notches, slots or other accurate surfaces may be used for locating the printed circuit board on the workboard holder for insertion. Regardless of the method used, it is required that two locations be utilized, and that the distance between them be the maximum permitted by the length of the board. The same locating references from which all the holes are drilled or punched should be employed for this purpose.

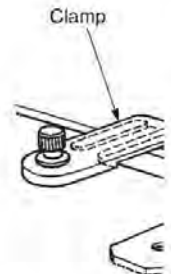
Pins through two locating holes are the most common and preferred locating method; they may be mounted directly into the workboard holder or into inserts which are mounted on to the workboard holder. Inserts allow insertions to be made closer to the locating references than do workboard holder-mounted locating techniques, as well as to provide necessary support in a relatively limited support area (see Figure 2.6).

2.4.3 Hold-down devices

Hold-down devices are used to maintain board position during table movement and component insertion. The most common of these are clamp assemblies which must be positioned manually, during board load and unload or spring stops which use spring-loaded plungers to apply pressure to the edges of the board to hold it in position. Both are shown in Figure 2.6. Hold-down devices are mounted at any place where the board is supported and, most advantageously, at opposite sides of the printed circuit board near the locating pins.

2.4.4 Rotary workboard holder design

Rotary workboard holders usually have one, two, or four board openings. These locations are usually arranged in a symmetrical pattern about the centre of the workboard holder to allow component insertion in both 0° and 90° axial positions (Figure 2.7). For manually loaded machines, 'finger room' for board loading and unloading should be provided in both the *X* and *Y* axes if the workboard holder is to be rotated.



Workboard holder

Removable stop (4)

When designing a si layout which will be

Workboard holder hardware

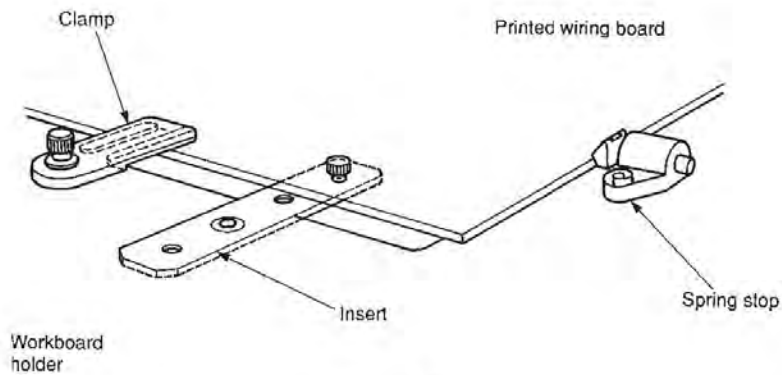


Figure 2.6

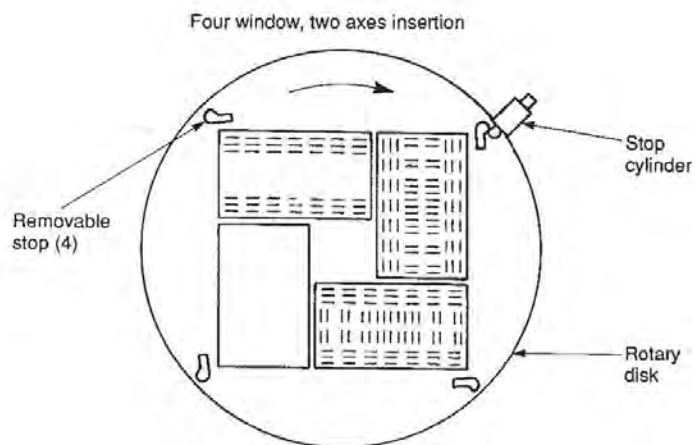


Figure 2.7

2.5 Layout considerations

When designing a single printed circuit board it is clearly a great advantage to create a layout which will be optimum for both the final product and for rapid reliable insertion

and high percentage test passes first time. There are a number of features in the layout of the board which should be borne in mind irrespective of the machine. Below are highlighted a number of the areas which designers should be aware of:

- 1 Ensuring that any holes used for reference, either for bare-board testing, final testing or ACI are located accurately with respect to the holes used for component insertion. These reference holes are used throughout the assembly process, and therefore it is sensible to locate them on the same grid and drill at the same time, so that registration between machine and printed circuit board is held to the best tolerance possible.
- 2 In the case of soldering, cleaning, and ACI machines it is sensible not to place components near the edge of the board, as this can cause problems with fingers on machines contacting the component. With ACI machines the bend mechanism beneath the board and the placement mechanism can interfere with the platen which of necessity must hold the board in place, and in the case of large components they can come outside the limits of the printed circuit board which makes for an untidy appearance apart from possibly creating mechanical problems in use. It is wise to leave perhaps 0.2 in. around the edge of any board.
- 3 From the point of view of the circuit operation it is wise not to route tracks under components or route tracks in such a way that testing an operation on a circuit can cause problems. This, of course, relates to the electronic design of the circuit rather than its mechanical usefulness and assembly. But bearing in mind that automatic component insertion machines place components without reference to any of the tracks, it is possible to place components over tracks and then find on final test a fault because a track has open-circuited or short-circuited underneath a component and thus cannot be seen. This, of course, applies only to those boards which are not tested before assembly. A way to circumvent this problem is to provide test pads on the board so that track continuity can be tested from one side of the component to the other; but this seems an unnecessary addition if it can be avoided altogether.
- 4 Check that the hole sizes are appropriate to the lead diameter and the assembly methods being chosen. Depending upon the type of ACI machine, a different tolerance may be required between the lead diameter and the hole diameter. Some machines can operate down to an annulus of 0.04 in. and others require rather more; thus it would be wise to establish which type of machine is to be used for assembly before committing to the hole diameter. It is not necessarily sensible to increase hole size simply for automatic assembly; doing so has the knock-on effect that bend mechanisms which rely on bending the lead against the side of the hole become less efficient.
- 5 Check component sizes and the capability of the machinery to handle those components. In many cases a component can be second sourced by one of a different mechanical size or shape; therefore once again check back from the ACI requirement to the circuit design and layout in order to choose optimum mechanical configurations for the component. Examples of this include avoiding the use of 0.4 in. pitch DIL components (which are in any case rare, and therefore many ACI machines are unable to handle them). Often an equivalent in 0.6 in. or 0.3 in. pitch is available.

Similarly with doliered in radia pitch; if this is th these transistors TO 92 are often which inevitably

The use of du ACI machine h available to the c to put down virt single in-lines th insertion - an e

A table is atta without referring back to his At components. Th height of the cc machines have i

- 6 When consideri requirements, al one should bea soldering functi One should ther not to create in: layout of the scri component by it the board is ass service, where th component fron of components l components for component ider some form of co the board. The c and its location does form a vita the screen why t component ider
- 7 Depending upo solder side of t considered in co effect the solderi earthplane may local condition: Although this is design stage.

Similarly with transistors – three-legged transistors are often available banded in radial form which gives the three leads an in-line configuration on 0.1 in. pitch; if this is the case then radial assembly machines will often be able to insert these transistors automatically. The traditional layout of transistor leads such as the TO 92 are often un-insertable by ACI machines other than very specialist machines, which inevitably will increase cost.

The use of dual in-lines for resistor packs also eliminates the need to consider an ACI machine handling single in-line resistors packs – which is another option available to the designer. If an ACI machine handles dual in-lines then it is possible to put down virtually all types of resistor pack. If an ACI machine does not handle single in-lines then the penalty may be that these require manual or semi-automatic insertion – an extra process.

A table is attached showing the various alternatives and preferred arrangements without referring to any particular ACI machine. What the user will have to do is go back to his ACI machine specification and then create a 'preferred list' of components. The other mechanical considerations with components include the height of the component above the board, and also the lead diameter, as most machines have a maximum lead diameter capability.

- 6 When considering the layout of the board one also has to consider solder-mask requirements, although this is dealt with in Chapter 13. When designing the layout one should bear in mind that the solder-mask not only has an impact on the soldering function, but also on the test function, and the bare-board test function. One should therefore be careful in considering solder-mask design and layout so as not to create insulation areas over test pads for example! In addition there is the layout of the screen ident for the components, and it is wise if possible to identify the component by its number outside the area which the component will occupy when the board is assembled. This serves a number of purposes. One of these is in field service, where the field service engineer may need to be able to refer to a particular component from the circuit diagram or vice versa. Another is for the identification of components for inspection and test purposes, and of course the identification of components for rework should this be required. It is nearly always vital to include component idents on the board design because most ACI machines will require some form of component ident in order to digitize that component and locate it on the board. The component ident often forms the link between the component type and its location on the board. Although ACI machines will operate without this it does form a vital link when checking boards or when machines stop and indicate on the screen why they have stopped. The operator can then rapidly refer back to the component ident on the board and see why the machine has stopped.
- 7 Depending upon the application it may be necessary to have an earthplane on the solder side of the board. The design and layout of this earthplane should be considered in conjunction with the soldering operation, as a large earthplane could effect the soldering operation such that adjacent components downstream from the earthplane may not be soldered properly due to a change in local temperature, or local conditions, caused by a sudden excess requirement for solder and flux. Although this is dealt with under the soldering section it is wise to consider it at the design stage.

The manufacture of bare printed circuit boards is carried out in several ways depending on the required end result. It is not the purpose of this book to go into that manufacturing technique – suffice it to say that in the Appendices there is further information available on screen printing and companies related to screen printing and processing of printed circuit boards, as well as reference to the printed circuit board association and all the standards.

When considering the various points made above one should add the tolerances within the production of printed circuit boards which might help the designer to understand some of the problems involved with, for example, hole position. If one refers to the specifications of ACI machines, hole position tolerance for these machines is anything from 0.002 in. to 0.02 in. – a rather staggering variation of 10 to 1. Those machines with vision correction systems are clearly much more likely to allow a larger tolerance than those without. Factually, the production of printed circuit boards for most uses will result in the positional tolerance of a component hole being held within ± 0.003 – 0.004 in. from its true position.

Although this is the norm there are many areas in which this becomes difficult or impossible. Bearing in mind that the datum hole should be drilled at the same time as the component holes and that there are probably two datum holes in order to give the printed circuit board orthogonal stability, then the tolerances begin to add up. The reasons for the tolerance are: drill wander when the boards are drilled in a stack, the drill can wander off its normal axis by the time the drill has reached the bottom of the stack of boards; also changing drill sizes from one to another can add a further tolerance. Always assuming that the drilling machine is perfect (and despite one's enthusiasm to believe all machines are perfect, that it not the case), the drilling machine table and the machine spindle both have operating tolerances, which can in many cases be as much as 0.002 in. from where they are expected to be. Thus the general result from a normally specified high-quality printed circuit board is in the region of 0.003 to 0.004 in.

This is always assuming that the board has been designed using CAD and has been programmed under grid control. If the board has been laid out manually and then digitized without reference to grid, of course these errors can rapidly escalate. For ACI applications where the requirement is to control the hole accurately without the use of vision systems it is clear that this drilling tolerance has to be added to the ACI machine tolerance so that the 0.004 in. may now become 0.006 in. and thus the tolerance of lead diameter to hole diameter starts to escalate. Once the hole diameter escalates, then the pad diameter (to give good soldering) also has to increase, and this might have a knock-on effect in reducing the component density on a board.

Referring back to the question of pad diameter relative to hole diameter, one also has to bear in mind that there are tolerances on the film and production of the masters for printed circuit boards and these have to be allowed for. Most CAD Systems will allow for what might be described as 'normal' pads and therefore the designer using CAD will have no problem. It is only when the design requirement compresses the component density so that pad diameter, grid size, etc. comes under pressure that the limitations of the assembly process must be borne in mind when considering the design.

In order to clarify some of the aspects of printed circuit board manufacture a table is included in order to illustrate the various parts of the process.

When one considers surface mount technology then the problem becomes multiplied in



I

terms of errors but surface mount technology via holes. However, surface mount (sm) placement insertion are such that as low as 0.01 in. T four and ten times 0.025 in. and, with possible, there are mixed technology relate to sm as we

Rather than go into the issue, the reader machines and clearly produce an interference with the design.

The need to use for such applications for economic reasons have been emerging slowed down rather

Figure 2.8 shows illustrates virtually connectors, and a

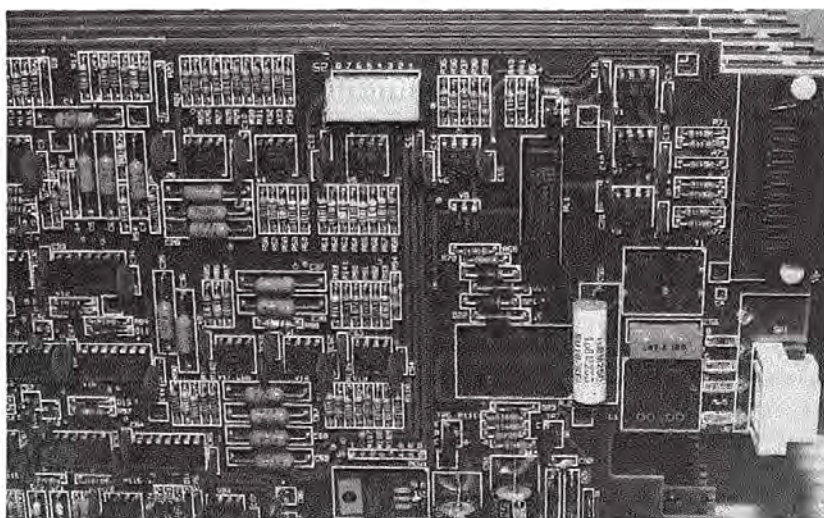


Figure 2.8 Part of a typical printed circuit board

terms of errors but of course the requirement for hole tolerances disappears as a truly surface mount technology (smt) product has no requirement for holes at all, other than via holes. However, with the advent of mixed technology (that is insertion and surface mount (sm) placement) then we run into a situation where the tolerance requirements for insertion are such that have been described above, but the placement grids can come down as low as 0.01 in. The pitch of grids used in surface mount products is typically between four and ten times finer. Typically, lead centres, instead of being 0.1 in., are 0.05 or 0.025 in. and, with the enthusiasm of component manufacturers to diversify as much as possible, there are other standards as well. It is therefore necessary for the designer of mixed technology products to be aware of the limitations of all assembly machines as they relate to sm as well as insertion.

Rather than go into all the details at this stage which would tend to confuse and cloud the issue, the reader is referred to the chapter covering automatic component insertion machines and clarify that situation, then to Chapter 14 on sm technology and then produce an intermediate technology specification for his requirement before setting out with the design.

The need to use sm components is being driven by the need to make high-density boards for such applications as communications, military and space and so on, and not so much for economic reasons. At the time of preparation of the data for this book, statements have been emerging from some companies indicating that the advance into smt has indeed slowed down rather than accelerated because of the problems of testing sm boards.

Figure 2.8 shows a typical well-designed pcb during the process of assembly. It illustrates virtually all types of component—axial, radial, DIL, connectors, switches, edge connectors, and a device with a heatsink.

Printed Circuit Board Assembly

The Complete Works

Preface – P. G. Davey, Meta Machines Ltd

Printed Circuit Board Assembly has been developed from knife and fork techniques (using human hands, fingers and odd mechanical tools, soldering irons, etc) to a very sophisticated fully automated process over the past thirty or forty years. This book describes the modern techniques available to the company or organization wishing to assemble printed circuit boards. These techniques vary from the manual techniques for the very small operation to the fully automatic systems which take the components, the printed circuit board, assemble them under vision control, solder the components in place and produce the finished high quality board. The ground covered includes the programming of the machines, a description of the machines and their operation, the various vision correction systems, the economics of assembly – and references to standards, techniques and the various machines available worldwide. Further material on manual, semi-automatic and robotic assembly, together with recommended approaches to board design and component selection for assembly, planning, staff training and surface mounted components make this a comprehensive book. The book should appeal to the engineer, student, accountant, designer – indeed any member of a team who needs to know more and understand the complexities and economics of printed circuit board assembly.

There are 100 figures and illustrations, a glossary of terms, a listing of standards, and their sources, and much technical detail for the engineer.

Peter Noble (CEng, CPhys, MIERE, MINSTP, MBIM) is a Technology Consultant.

Available in the USA, Canada and Latin America from Halsted Press, a division of John Wiley & Sons, New York – Toronto



ISBN 0 470 21266 7

Available in the UK and the rest of the world from Open University Press, Milton Keynes



ISBN 0 335 15418 2 (Paper)
ISBN 0 335 15419 0 (Cased)