AN INTRODUCTION TO PRINTED CIRCUIT BOARD TECHNOLOGY

by

J. A. SCARLETT

ELECTROCHEMICAL PUBLICATIONS LIMITED
1984

APPLE 1008

TK7872.P65
S286
c.2

ELECTROCHEMICAL PUBLICATIONS LIMITED
8 Barns Street, Ayr, Scotland

©
Electrochemical Publications Ltd.
1984

All Rights Reserved
No part of this book may be reproduced in any form
without written authorisation from the publishers

ISBN 0 901150 16 9

Phototypesetting by Brian Robinson,
North Marston, Bucks
Printed and bound in Great Britain by
M°Corquodale (Scotland) Ltd.

PREFACE

The Electronics industry has been using printed circuit boards for over 30 years; through-hole plated multilayer boards are over 21 years old, and 'new' forms of boards such as flexi-rigids or fine line types mostly date from the late 1960s. Because it has developed from tag strips and other similar pieces of hardware there has been a tendency among electronics engineers to ignore the printed circuit board and it is only in the last few years that many engineers have come to appreciate that the humble, disregarded piece of plastic sheet has developed into one of the dearer and more important components of most electronic systems. Some of the questions asked about printed circuit boards and their design reveal that there are engineers who think that all boards have the same or very similar electrical properties; that they are all made in the same way, and that they can all be assembled or repaired by the same methods.

One printed circuit board may resemble another only to the extent that one transistor resembles another: the size and shape can be different, the base materials can differ, and there may be little resemblance between the AC electrical properties. Unlike most transistors, almost all printed circuit boards are custom made to suit their one specific application.

Because they are custom made, the engineer whose circuit is to be built on a board should know enough about printed circuit technology to ensure that he is able to choose the best type of board for the job, to see that the design is cost effective and that the board can be manufactured without undue difficulty. Also, he should know sufficient about board technology to make any choices of components or circuit configurations in whichever way will be most helpful to the board designer.

There is a considerable amount of literature available about many aspects of boards, mainly in conference and symposia proceedings, and there are now several books. Because the range of board types is large, and their manufacture is highly specialised, most of the available books tend to be large, expensive, and possibly over-detailed for the young engineer or student who is not going to be directly involved in board design or manufacture, or for the senior manager who wants a fairly brief 'update' on the latest technology. Also, many of the books available tend to have a bias towards the section of the industry in which the author is employed. This present book is an effort to provide a brief, unbiased account of all major types of printed circuit boards and the latest methods of manufacture from design through to assembly. Some of the processes described are the subject of Patents.

Much of the material is condensed from the author's larger books, 'Printed Circuit Boards for Microelectronics', 'Transistor-Transistor Logic and its Interconnections'; from the author's chapters of 'The Multilayer Printed Circuit Board Handbook', and from the course notes compiled by the author for the course on printed circuit board design run by the Institute of Circuit Technology

at the University of Oxford's Department for External Studies.

The biggest problem in compiling this book has not been what to write, but what to condense or omit without losing too much clarity. As far as possible technical jargon has been avoided, and any essential technical terms should be explained briefly in the Glossary. The aim has been to produce a book which can be understood by any reader with the average school leaver's knowledge of physics, chemistry and engineering.

In the hope that this book will interest readers sufficiently for them to wish to read more about what is a fascinating area of modern electronics engineering, the references list other books and papers which extend in more detail the topics covered here.

All the illustrations used are by courtesy of STC Exacta Ltd, Selkirk, Scotland, except for Figure 1.8, which is reproduced by the kind permission of Messrs BICC-Vero Electronics Ltd.

I must acknowledge my gratitude to all my colleagues in the industry with whom I have discussed so many points, to Professor D. S. Campbell of the University of Technology, Loughborough for his comments and suggestions, and to Tony Baillie of Exacta for his help in providing the photographs and for his excellent work in re-drawing all the illustrations for this book, either as copies of those in my other books or from original sketches.

January, 1984

John A. Scarlett, F Inst CT
Galashiels

TABLE OF CONTENTS

PREFACE (v)

CHAPTER ONE **Introduction**

1.1 The Printed Circuit Board 1
1.2 History 2
1.3 The PCB in Interconnect Technology 8
1.4 External Connections to Boards 10
1.4.1 Plugs and Sockets 10
1.4.2 Edge Connectors 13
1.4.3 Discrete Wires 14
1.4.4 Back Wiring Panels 16
1.5 The Electrical Importance of the Interconnect 16
1.6 Board Types 18
1.6.1 Single-sided 18
1.6.2 Double-sided, Non Through-plated 18
1.6.3 Flexible 19
1.6.4 Double-sided, Through-hole Plated 19
1.6.5 Flexible, Through-hole Plated 19
1.6.6 Rigidised Flexible 20
1.6.7 Multilayer 20
1.6.8 Flexi-rigid 20
1.6.9 Buried Via Hole 21
1.6.10 Double-double Boards 21
1.6.11 Double-multi Boards 22
1.6.12 Sequential BVH Boards 22
1.6.13 Pillar Plated Multilayers 22
1.6.14 Flush Bonded Boards 23
1.6.15 Stripline or Microwave Bonded Circuit 23

CHAPTER TWO **Mechanical and Electrical Properties**

2.1 Base Materials 24
2.1.1 Cladding 24
2.1.2 Base Laminate 24
2.2 Strength and Stiffness 25
2.3 Stability 26
2.3.1 Dimensional Stability 26
2.3.2 Thermal Stability 27
2.4 Electrical Properties DC 28
2.4.1 Resistance 29
2.4.2 Voltage Clearance 32
2.5 Electrical Properties AC 33
2.5.1 Capacitance 33
2.5.2 Inductance 35
2.5.3 Characteristic Impedance 35

31649

2.6 Effects of AC Properties 35
2.6.1 Line Reflections 35
2.6.2 Surge Currents 42
2.6.3 Cross-talk 43

CHAPTER THREE **Design of Boards**

3.1 Standards 47
3.2 Choice of Type 48
3.2.1 Non Plated Boards 49
3.2.2 Plated-through Hole Boards 49
3.2.3 Multilayer Boards 49
3.2.4 Surface Mounting 50
3.2.5 Flexi-rigids 50
3.2.6 Conclusion on Board Type 50
3.3 Determination of Size 51
3.3.1 Small Systems 51
3.3.2 Large Systems 51
3.3.3 Cost 51
3.3.4 Manufacturing Limits 52
3.3.5 Electrical Function Splits 52
3.3.6 Interconnections 52
3.3.7 Testing 53
3.3.8 Repairs and Costs 53
3.3.9 Design Difficulties 54
3.4 Determination of Feature Size 54
3.4.1 Hole and Pad Size 54
3.4.2 Track Widths and Gaps 55
3.4.3 Tracks Between Package Pins 56
3.5 Package Placement 56
3.6 Power Distribution 59
3.6.1 Power and Earth Planes 59
3.6.2 Decoupling 59
3.6.3 Surface Tracking 59
3.7 Track Routing 64

CHAPTER FOUR **Filmwork Generation**

4.1 Films 67
4.1.1 Care of Films 67
4.1.2 Nomenclature 67
4.1.3 Stability and Size 68
4.1.4 Registration 70
4.2 Film Generation 71
4.2.1 Plotted Films 71
4.2.2 Cut and Strip 72
4.2.3 Manual Artwork 72
4.2.4 Two Colour Artwork 73
4.2.5 Single Pad, Multiple Track Sheet Artworks 73
4.2.6 Hybrid Artworks 73
4.3 Planes and Solder Resist Films 74
4.4 Photo Reduction 75
4.5 Photo Tools 76
4.6 Films and Processing 76

CHAPTER FIVE **Board Manufacture**

5.1 Introduction to Manufacture 78
5.1.1 Laminate Manufacture 78
5.1.2 Planning and Preparation 78
5.2 Subtractive Board Manufacture 81
5.3. Printing 83
5.3.1 Photomechanical Printing 83
5.3.2 Screen Printing 84
5.4 Etching 86
5.4.1 Etchants 86
5.4.2 Etching Machines 87
5.5 Hole Generation 88
5.5.1 Sight Drilling 88
5.5.2 Punching 88
5.5.3 Jig Drilling 89
5.5.4 Numerically Controlled Drilling 90
5.6 Profiling and Edge Treatment 92
5.6.1 Profiling 92
5.6.2 Chamfering 93

CHAPTER SIX **Surface and Through-hole Plating**

6.1 Edge Connector Fingers 94
6.2 Through-hole Plating 95
6.2.1 Drilling and Cleaning 95
6.2.2 Electroless Plating 95
6.2.3 Panel Plating 97
6.2.4 Pattern Plating 100

CHAPTER SEVEN **Basic Multilayer Board Manufacture**

7.1 General Principles 104
7.2 Inner Layer Preparation 104
7.2.1 Tooling 104
7.2.2 Film Preparation 106
7.2.3 Printing and Etching 107
7.2.4 Oxidation 107
7.3 Lay-up 109
7.3.1 Press Plates and Packing 109
7.3.2 Prepreg and Build 110
7.3.3 Layer Numbering 111
7.4 Bonding 111
7.4.1 Presses 111
7.4.2 Press Cycles 112
7.4.3 Bonding Defects 114
7.5 Drilling and Finishing 115
7.5.1 Drilling 115
7.5.2 Smear Removal and Etch Back 115
7.5.3 Printing and Etching 116
7.6 Internal Short Circuits 117
7.6.1 Shorts on One Layer 117
7.6.2 Layer to Layer Shorts 117
7.6.3 Track to Hole Shorts 117

7.6.4 Hole to Hole Shorts 117
7.7 Reliability of Multilayer Boards 118

CHAPTER EIGHT **Manufacturing Variants**

8.1 Additive Processes 119
8.1.1 Additive Copper on Bare Laminate 119
8.1.2 Semi-additive Processing 119
8.1.3 Sensitised Laminates 120
8.2 Multilayer Variants 120
8.2.1 Buried Via Holes 120
8.2.2 Doubled and Sequential Multilayers 120
8.2.3 Added Surface Layers 121
8.2.4 Heat Ladders 121
8.3 Selective Plating 123
8.4 Pillar Plating 126
8.5 Metal Cored Boards 127

CHAPTER NINE **Flexibles and Flexi-rigids**

9.1 Flexible Circuits 129
9.1.1 Base Materials 129
9.1.2 Cover Coat 129
9.1.3 Cover Coat Bonding 131
9.1.4 Back Etching 131
9.1.5 Solder Buttons 132
9.1.6 Roll to Roll Processing 132
9.1.7 Plated-through Hole Flexible Circuits 132
9.2 Rigidised Flexibles 133
9.3 Flexi-rigid Boards 134
9.3.1 Types of Flexi-rigid Board 134
9.3.2 Manufacture of Flexi-rigid Boards 136
9.3.3 Flexi-rigid Applications 137
9.3.4 Designing Flexi-rigids for Maximum Life 138

CHAPTER TEN **Board Finishes**

10.1 Bare Copper 140
10.2 Lacquer 140
10.3 Roller Tinning 141
10.4 Solder Levelling 142
10.5 Tin-lead Plating 143
10.6 Reflow 143
10.7 Gold and Palladium-nickel 145
10.7.1 Gold 145
10.7.2 Palladium-nickel 145
10.8 Solder Resist 146
10.8.1 Types of Solder Resist 146
10.8.2 Effect of Board Finish 147
10.8.3 The Need for Solder Resist 148
10.9 Legends 148

CHAPTER ELEVEN **Inspection and Testing**

11.1 Specifications 150
11.1.1 General Standards 150
11.1.2 Company Standards 151
11.1.3 Concessions 152
11.1.4 Fitness for Use 152
11.2 Inspection 153
11.2.1 Sampling 153
11.2.2 Process Inspection 154
11.2.3 Final Inspection 156
11.3 Electrical Testing 157
11.3.1 Electrical Testing and Visual Inspection 158
11.4 Test Coupons 158
11.5 Microsections 159
11.6 Repairs and Rework 160
11.6.1 Cut Tracks 161
11.6.2 Replacement Tracks 161
11.6.3 Inadequate Clearance 161
11.6.4 Defective Edge Connector Plating 162
11.6.5 Replacement of Edge Connector Fingers 162
11.6.6 Track Repairs on Inner Layers 162
11.6.7 Lifted Pads 162
11.6.8 Missing or Undersize Holes 163
11.6.9 Warped or Out of Flat Boards 163

CHAPTER TWELVE **Assembly and Soldering**

12.1 Component Mounting 164
12.2 Component Insertion 167
12.2.1 Manual Insertion 167
12.2.2 Automatic Insertion 168
12.2.3 Cutting and Crimping of Leads 168
12.3 Soldering 170
12.3.1 Solderability 170
12.3.2 Manual Soldering 172
12.3.3 Drag Soldering 172
12.3.4 Wave Soldering 172
12.3.5 Vapour Phase Soldering 173
12.4 Common Soldering Faults 173
12.4.1 Dry Joints 173
12.4.2 Bridging 174
12.4.3 Crazing 174
12.4.4 Icicling 176
12.4.5 Outgassing 176
12.4.6 Delamination or Blistering 176
12.5 Desoldering Through-plated Holes 177
12.6 Surface Mounting 177
12.6.1 Welding 178
12.6.2 Reflow Soldering 178
12.6.3 Thermal Expansion 179

CHAPTER THIRTEEN **Achievable Densities and Performance**

13.1	Packing Densities	180
13.1.1	Track Densities and Board Size	181
13.2	Performance	182
13.2.1	Electrical Performance	182
13.2.2	Mechanical Performance and Reliability	182
13.3	Size, Performance and Cooling	183
13.4	Alternative Interconnection Techniques	184
13.4.1	Wires	184
13.4.2	Thin Film Circuits	185
13.4.3	Thick Film Circuits	185
13.4.4	Porcelain on Steel	186
13.5	The Future for Printed Circuit Boards	186

GLOSSARY OF PCB TERMS 187

REFERENCES 194

INDEX 197

Chapter 1

INTRODUCTION

1.1 THE PRINTED CIRCUIT BOARD

A printed circuit board is a piece of plastic material on which electronic components can be mounted for mechanical support, and which also electrically interconnects all the components it supports by means of a pattern of metal tracks on its outer surfaces and sometimes on inner layers.

Most printed circuit boards have holes through which the component leads are inserted, and the conducting track pattern has enlarged pads, which are usually but not always circular, around these holes. The component leads are mechanically secured and electrically connected to these pads by soldering. The barrels of the holes may be plated with metal to provide electrically conducting paths from one layer or surface of the board to another.

Some boards carry their components mounted on the surface of the board, with leads which emerge from the sides of the component bodies soldered onto small rectangular pads on the boards, or with the joints made to metallised areas under the body of the component. Multilayer boards which carry such surface mounted components may have their layer to layer connections formed by solid metal pillars within the board.

There are printed circuit boards which carry few or no components, but on which the track pattern forms the working part of the circuit. Such boards can be used as the rotors for low inertia motors, deflection coils for magnetically focussed cathode ray tubes, stators for rotary switches, dipole aerial arrays, mixers, couplers, and so forth for microwave circuits, or for loop aerials for directionally sensitive radio receivers. Printed circuit boards have also been made with conducting patterns etched from medium or high resistivity material, usually overlaid with copper from which a normal interconnect pattern can be etched, to yield matched sets of connected resistors.

Almost all electronic equipment, from large computers and radar systems down to pocket transistor radios, uses printed circuit boards as the main component support and interconnection system. On large systems the individual boards which carry the components are often interconnected by further printed circuit boards carrying sockets into which the component boards can be plugged.

Because all the interconnecting tracks lie on the surfaces of the board and are insulated by the plastic base material of the board, printed circuit interconnections take up less space than ordinary insulated wiring and, because the plastic substrate also supports all the components it interconnects and no other panel or frame is required, printed circuit wiring is usually lighter and smaller than any other form of mounting and interconnection system.

The pattern of tracks on the board can enable the electronic components to be mounted very close together, so the lengths of all tracks are minimised. Although

the electrical signals travel along the tracks at about 60% of the speed of light, modern silicon microelectronic devices work so rapidly that the time taken for the signals to travel along even a few inches of track can be critical. Badly designed boards or the use of the wrong type of board can cause electrical or logic malfunction such as cross-talk, line reflections, or line current surges, any of which can cause a high speed system to fail to function as its designer intended. The defined widths and spacings of the tracks on a printed circuit board make it possible to predict in advance of production the effects of resistance, capacitance and inductance of the tracks.

Printed circuit boards can vary enormously in their complexity, in the way in which they are manufactured, and in the way they must be handled when components are mounted. Every board type must be designed specially to suit its particular application. The costs of boards can vary from a few pence each to hundreds of pounds, and great care and expertise is needed in the selection of the correct type of board for any application.

The interconnection pattern on a printed circuit board is generated from a set of master films, and thus, once the films have been proved correct, all boards made from these films will have substantially the same electrical properties and even the most complex interconnection pattern will be assembled with all the several thousand interconnections correctly made.

In quantity assembly all soldered joints between the component leads and the pads on the boards are made by mass soldering techniques on machines which can be set to ensure that all joints are neatly made and are more reliable than joints made individually with a soldering iron.

The advantages of printed circuit boards over other forms of packaging or wiring can be summarised as:

Predictability; Repeatability; Reliability; Low Cost; Light Weight; Low Volume; Easy Assembly and Servicing; Ease of Cooling; Ability to Use Mass Soldering Techniques.

1.2 HISTORY[1,2,3]

The earliest electronic interconnections tended to be large, often consisting of metal strips or rods mounted on wooden bases. These were superseded by wires, still often quite heavy and connected to screw terminals on larger components. As systems grew more complex and components decreased in size a metal chassis mounting became standard, with larger components bolted to the chassis and smaller components mounted either by connecting their end leads to terminals of larger components, or with their bodies secured to the chassis by metal or plastic P clips. (See Figure 1.1.)

As electronic systems grew in complexity, increasing numbers of small components were needed and mounting them in the wiring became impractical. The most commonly used mountings in the 1940s were tagstrips, strips of plastic material about 10 cm wide with double-ended metal pins along each edge. Axial lead components could be mounted in rows with their leads wrapped and soldered to the inner legs of the tags and interconnections made by wiring harnesses, which could be pre-assembled, soldered to the outer legs. Figure 1.2 shows a short length of such a tagstrip carrying standard resistors with experimental transistors mounted over them. Note that this is not a production board.

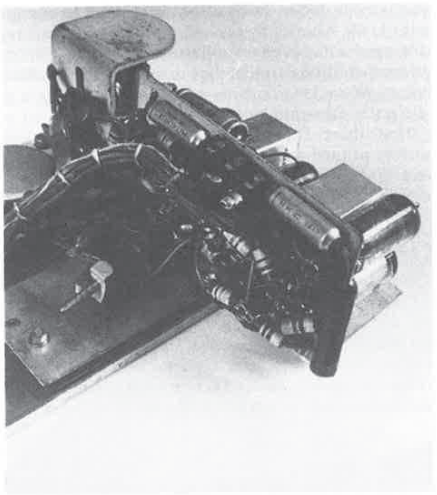


Fig. 1.1 Radio receiver on aluminium chassis with components suspended in the sub-chassis wiring.

As errors could arise in connecting the wires from a harness to the tags, efforts would be made to minimise the external wiring by grouping components so that connections could be made between adjacent tags or by short wires beneath the components on the strip. From such wires it was a logical step to progress to precut copper patterns which could be assembled to the plastic base with the tags. From a tagstrip with pre-made internal connections the move to a conventional printed circuit board is, with hindsight, not a great step.

While the bulk of electronic equipment was being made on metal chassis with tagboards and discrete wiring, there had been developments in other ways of making electrical connections. The use of metal deposited on glass under vacuum

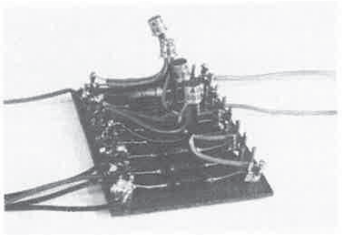


Fig. 1.2 Tagstrip carrying resistors and transistors.

had been known since the 1850s, but had not been used to any great extent. As early as 1925, in the USA, Charles Ducas had submitted a patent application for a method of making conducting lines on an insulating base by electroplating over a conducting paste applied through a template or stencil, and in 1928 Samuel Ryder patented a technique of making coils by painting or spraying a conductive ink pattern onto a dielectric substrate.

In 1940 the first thick film circuits comprising metal conductors and composition resistors printed and fired onto a ceramic base were produced, but such circuits were limited in size by the availability of suitable substrates and could not be considered as offering an alternative interconnection for large systems.

The first practical printed circuit boards in which the functions of the chassis, component mounting base and wiring were all combined into a single unit made by etching the conductor pattern from a copper clad laminate as in Figure 1.3 were made by Dr Paul Eisler in the United Kingdom, in 1942 and patented in February 1943.

One of the two boards in the radio receiver he designed carried an etched aerial loop while the other, which was double-sided, formed the main chassis. The industry as a whole was slow to adopt this new development, partly because of the lack of suitable materials and processing capability, and it was during the

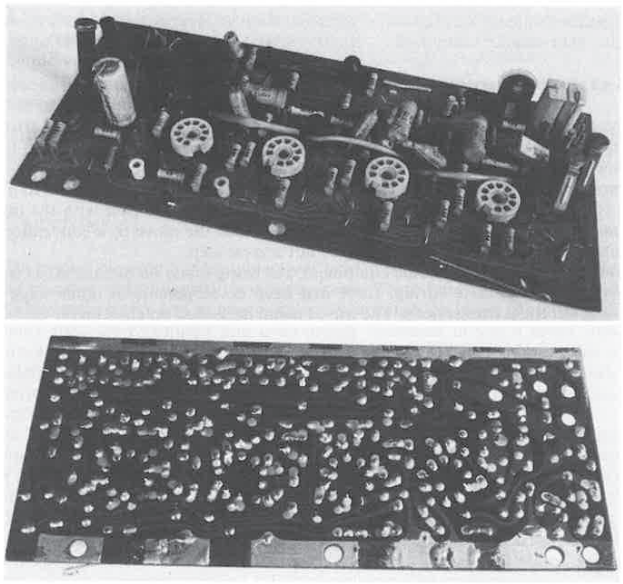


Fig. 1.3 Single-sided board with miniature valve sockets. Note the use of wire links.

1950s, with the change in the industry from thermionic valves to transistors, that phenolic paper or epoxy glass based etched printed circuit boards came into widespread use. Even then the full potential of the boards was not always exploited and often much of the actual interconnecting was carried out by external wiring. Figure 1.4 shows a board on which the resemblance to four rows of tagstrip can be clearly seen, and on which most of the tracks run from the components to the board connector.

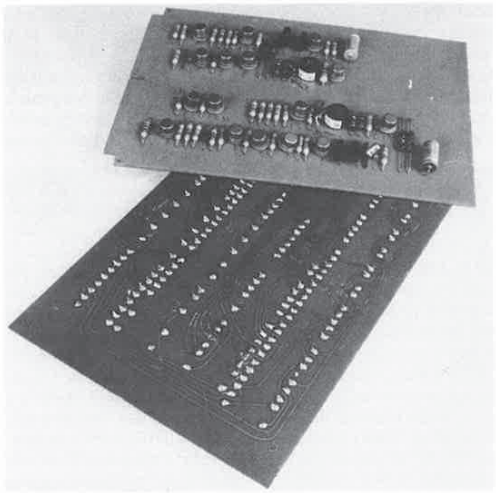


Fig. 1.4 Single-sided board which serves mainly as a component carrier with few actual interconnections.

Complete working circuits could be implemented on boards, but where tracks were required to change sides on a board there were problems. Through-board connections were arranged only with difficulty, usually in the form of discrete wires, eyelets, metal stakes or rivets, and the art in designing boards lay in routing tracks round the ends of other tracks and in using the bodies of components or wire links as bridges to form unavoidable crossovers. Such links can be seen in Figure 1.3.

Pillar plating techniques were tried to make some early multilayer boards. Successive layers of the circuit were built up by electroplating, with intervening layers of dielectric material deposited round the layer to layer plated pillars. This technique, which has similarities to techniques used in making ceramic based thick film circuits, proved too expensive for general use.

The development of the plated through-hole opened the way to fresh board types as in Figure 1.5. Several firms were involved in this work, but it is usually accepted that the 1961 patent, by Hazeltyne in the USA, of the seeded, plated-

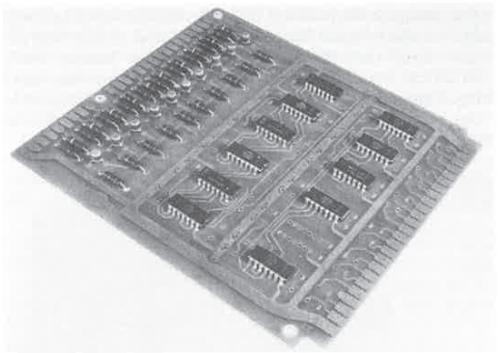


Fig. 1.5 Through-plated hole board of the early 1960s carrying integrated circuits and discrete components.

through hole in a multilayer structure was the start of the new era. The ability to interconnect tracks on two sides or on inner layers of a board wherever necessary made it possible to achieve vastly increased interconnect densities by ordering the tracks on the two sides into orthogonally crossing parallel runs with 'diagonal' connections implemented as Ls with layer changing via holes on the corners. With the barrels of the holes through-plated and no requirement to make a soldered joint on the pad around the holes the size of the pads could be reduced to allow the running of tracks between the pins of a standard dual in-line package.

The ability to run tracks between the pins of DIPs (Dual In-line Packages) instead of having to pass round the ends yielded a highly significant improvement in packing densities, and by the later 1960s it was possible to implement digital

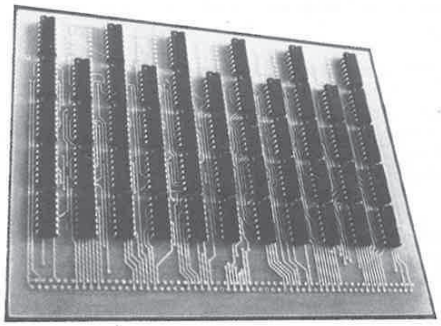


Fig. 1.6 Late 1960s' double-sided board with extensive use of via holes to achieve high packing density.

circuits with about 4 sq cm board area per DIP on double-sided boards. An example of such a board, with packages on 2·03 by 1·4 cm spacing, is shown in Figure 1.6. The underside of the board carries the tracking between the package pins, some of which terminates on the via holes visible on the ends of some of the tracks on the components side.

Through the 1960s improvements in resin chemistry, photo-resists, and in etching and plating capabilities improved the tracking densities achievable. With the use of fine lines, down to 0·05 mm wide, and multilayer techniques it became possible to implement the tracking densities which would be needed by the new generations of very large scale integrated circuits of the later 1970s and 1980s. The use of lines less than 0·2 mm wide made it possible to run two or even three tracks between adjacent pins of a DIP. Figure 1.7 shows a portion of a double-sided board designed in 1973, mainly with two tracks between pins, but with three tracks indicated by the point of the pencil. It can also be noted on Figure 1.7 that some of the via-hole pads are significantly smaller than those for the DIP leads.

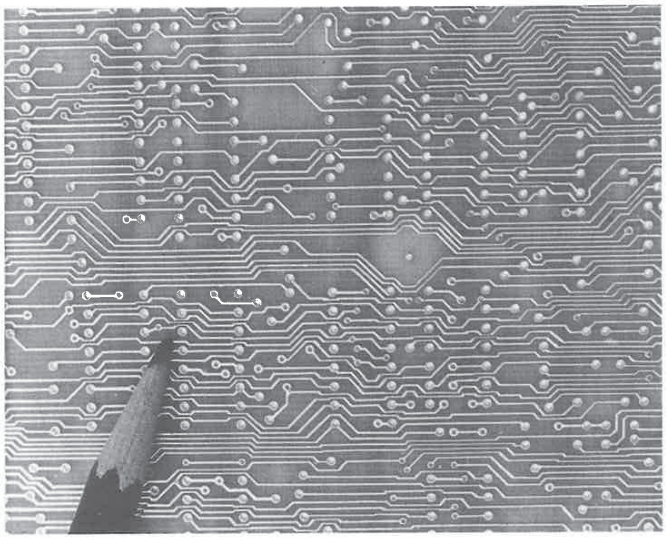


Fig. 1.7 High density tracking on double-sided board.

Sizes of boards used increased through the 1960s and 1970s and their design and manufacture moved from small scale operations which could be carried out by relatively unskilled staff using simple equipment to a high technology industry. In the 1980s printed circuit manufacture demands large capital investments and the design of boards requires a very high level of specialised skill plus considerable experience.

1.3 THE PCB IN INTERCONNECT TECHNOLOGY

Interconnect technology covers all the steps in connecting together the basic electronic components such as resistors, capacitors, inductors, diodes, transistors and so forth to form usable working systems. Such interconnections can be divided into four levels of: micro-connections, mini-connections, ordinary connections, and macro-connections.

Micro-connections are usually defined as being those within the normal packages of conventional components. This includes the lead-out wires of resistors, capacitors, coils, diodes, transformers and so on, and it also includes the steadily growing field of on-chip metallised connections of integrated circuits from the two internal connections of a packaged Darlington pair of transistors to the thousands of connections on a VLSI chip.

Mini-connections are less easy to define, but usually these are accepted as being small connections made between conventional components in a manner which converts them into a single working unit which may often be regarded as a non-repairable item. The clearest examples of mini-connections are those on ceramic or plastic substrate hybrid circuits. Other instances of mini-connections are the thin metal strips used to weld groups of resistors or flat-packs into the so-called cordwood modules. The connections formed on the plastic substrate of a spider bond or TAB lead frame may be regarded as part of the micro-connection level when they are incorporated into a sealed integrated circuit pack, but they can equally well be regarded as mini-connections when they are used to connect to unpackaged chips, when they are clearly a second level connection.

Ordinary connections are usually regarded as covering the two levels of 'conventional printed circuit boards' and 'unit or shelf wiring'. These are the levels at which the individual components or hybrid circuits are interconnected to form units with recognisable working functions or sub-functions, or even complete working systems. Whilst printed circuit boards are the commonest forms of the lower level of ordinary connections, the level also includes discrete insulated wires and cableforms internal to a unit. The other level of unit or shelf wiring includes printed circuit back wiring panels, other back wiring systems such as the AMP 'Termipoint', wire wrapping, and so forth. This higher level of ordinary connections merges into the next level up of macro-connections, which is generally taken as including the interconnection of units or shelves into frames or cabinets and the inter-cabinet cabling necessary on large multi-unit installations such as computer systems or radar stations.

Macro-connections are usually in the form of cables, or waveguide for very high frequencies. Within a unit or rack the macro-connections may be formed from discrete wires tied or laced into a harness, or they may be flat ribbon cables. Inter-unit connections are more commonly made from standard cables, but there is now increasing interest in the use of optical fibres. In some cases, such as vehicle wiring, flexible printed circuits are replacing conventional wiring.

When all the levels of connections are reviewed it can be seen that the first use of printed circuits is to be found at the micro level in the form of spider bonds, with increasing use through the mini-connection field, and almost total domination of the lower level of ordinary wiring.

The ability to predict the electrical properties and the inherent reliability and repeatability of printed wiring coupled with the availability of multilayer panels in sizes up to 0·5 metre square is moving an increasing amount of back wiring onto printed circuit boards, and long flexible circuits are proving to be competitive with conventional wiring in some of the macro-connection areas.

Thus it can be seen that throughout the entire field of electronic interconnections printed circuit boards can play a part. As new developments in semiconductor technology result in increases in the amount of the interconnect on the silicon chip, the effect is to move the area occupied by the macro-connection field. What a few years ago would have been a board carrying hundreds of integrated circuit packs is now a single large package, mounted on a board which carries what used to be the cabinet or rack wiring. Many equipments now contain only one board but as miniaturisation progresses system designers produce ever more complex total systems, and frame wiring and inter-unit cables are still being designed, giving ample scope for future generations of printed circuit boards.

The use of more complex chips with large numbers of input and output pins is raising the level of interest in boards with very fine tracks which can be used as larger substrates than has hitherto been possible with ceramic based thick film circuits for direct chip mounting or for mounting sealed, leadless chip carriers.

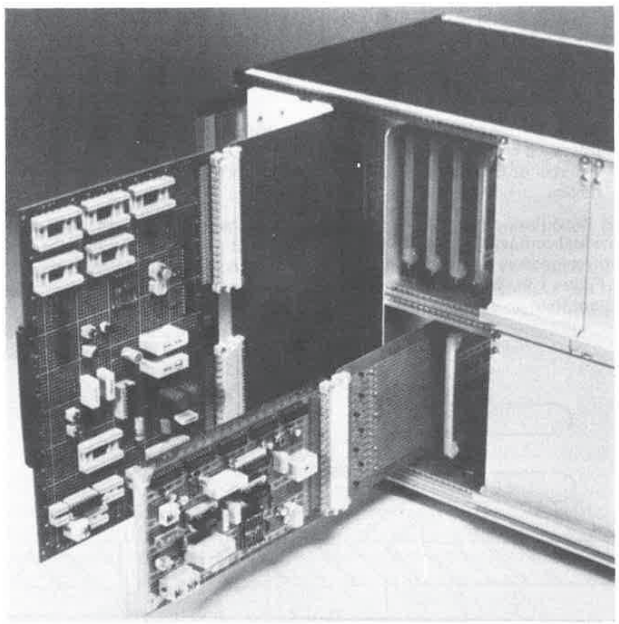


Fig. 1.8 Half and full height boards in a cabinet, plugging into a back wiring frame. Two cards are mounted on extender boards for test access.

1.4 EXTERNAL CONNECTIONS TO BOARDS

Like integrated circuits, printed circuit boards can be used only when external connections are made to them. In large multi-board systems the individual boards are usually arranged in rows in cabinets or frames, as in Figure 1.8. The boards slide in guides mounted on metal rails running across the cabinet. The back edge of each board carries contacts which fit into sockets along the back of the frame or cabinet. On smaller systems, the boards are more usually held by mechanical fastenings in the frame or case and connections are made either by cables with plugs which can fit into sockets placed anywhere on the board, or by wires connected permanently or semi-permanently to the board.

When boards are to be plugged into a rack or frame the connection to the frame wiring may be made through a plug and socket, with the plug attached to the board with its contact pins lying parallel to the board face at the back edge; or contacts may be formed directly on the board to fit into an edge connector socket (see Figure 1.5). Both systems have advantages and disadvantages, and whichever is used the connector must be chosen to fit the proposed type of board.

1.4.1 Plugs and Sockets

Some users of rack-mounted, plug-in boards prefer to make electrical connections through sockets with separate plugs bolted and soldered to the board.

The reasons usually stated include:

(i) The fit of the plug in the socket is guaranteed.
(ii) Damaged contacts can be individually replaced.
(iii) The risk of corrosion affecting the contacts is less than with edge connector fingers.

Of these three reasons, only the first is valid. On most plugs, the backs of the individual contacts are usually bent through a right angle to solder into the board, so to replace any contact the entire plug must first be removed from the board (see Figure 1.9). Whether a discrete plug and socket will be more or less corrosion resistant than an edge connector depends on how both are made.[4]

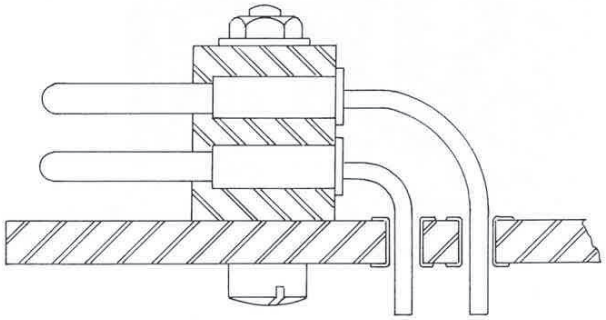


Fig. 1.9 Section through typical PCB connector.

One very real advantage of discrete plugs and sockets is that when large numbers of contacts are to be made plugs and sockets are available which can offer significantly lower insertion and withdrawal forces than is possible with edge connectors whilst maintaining adequate contact pressure.

Most of the types of plug and socket connectors used on the edges of boards have their contacts in two rows as shown in Figure 1.9 and on the board in Figure 1.6, with the pins of such a size that tracks to the lower bank of pins can pass between the pads for the upper row. Some similar connectors are available with three or four rows of pins, and it should be noted that these are unsuitable for use on double-sided boards unless extremely fine tracks are used: they are for use on multilayer boards where tracks can be run to the third and fourth rows of pins on inner layers (see Figure 1.10). It must be borne in mind that, if attempts are made

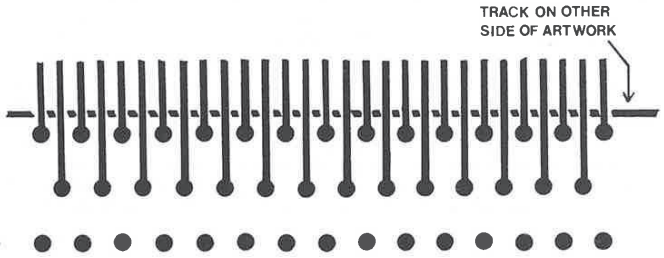


Fig. 1.10 When only one track can pass between pairs of pins on a double-sided board a third row of pins may be unusable.

to run tracks out from the connector on both sides of a double-sided board, there will be no layer left for tracks to cross those from the connector.

These miniaturised three and four row connectors for multilayer boards should not be confused with the four row ribbon cable connectors which have their pins on 5·08 mm pitch in each row, allowing ample space for all necessary tracking on one side of the boards, as shown in Figure 1.11.

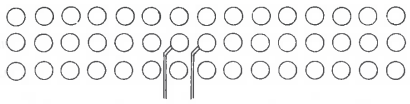


Fig. 1.11a Three row plug and socket connector.

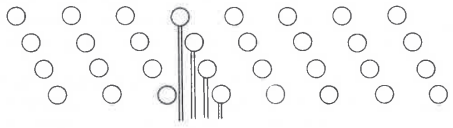


Fig. 1.11b Four row ribbon cable connector pattern.

When a plug and socket connector is being selected to connect a board into a rack, the size of the board should be taken into consideration. Layout on large boards may be made easier if two or more connectors are used to spread the input and output signals along the edge of the board. If the back wiring is to be a printed circuit the use of multiple plugs and sockets can cause alignment problems, and usually assembly fixtures will have to be used during soldering to ensure that the end to end spacing of the sockets matches that of the plugs, and that the sockets are truly in line. There are some types of sockets for use with discrete wire back wiring systems which have a limited degree of float in their mountings, but the stiffness of a large number of wires can counteract this float and make it very difficult to insert the boards. When multiple sockets are being considered, it is likely that edge connectors, which can tolerate a limited degree of misalignment, will be found to be more suitable than plug and socket connectors.

Connections from the board to the rest of the unit on single board equipments are often made by ribbon cables, which can be terminated in plugs with a pin pattern as in Figure 1.11, or in plugs which fit into standard Dual In-Line Package (DIP) sockets. The latter type are particularly useful on digital systems, as the necessary cable connections can be arranged at a suitable point in the circuit without disturbance to a regular package pattern.

Most commonly used types of plugs can be mounted on boards without undue difficulty but one type to be avoided whenever possible is the circular cable connector of the type originally designed for sealed bulkhead mounting, which has pins arranged in concentric circles as shown in Figure 1.12. Connection to the

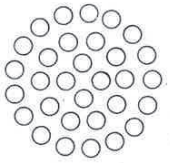


Fig. 1.12 Circular pin pattern of bulkhead connector.

outer row of pins is simple, but usually tracks can be run to the inner rows only on a multilayer board on which pads around the outer rows of holes are omitted on the inner layers. This is thoroughly bad practice as will be explained in later chapters.

When a printed wiring connection must be made to such a plug or socket, it is best made in the form of separate, unbonded flexible layers for each circle of pins, with open centres to allow access to lower layers for soldering and clearance holes for pins in outer rows to which contact is not being made, as shown in Figure 1.13.

Whenever connectors which have mechanical fastenings to boards are used, adequate clearance must be left around the fixing holes to allow for a washer under the nut or screw head. Failure to observe this precaution may result in the washer shorting two tracks together or shorting a track to the equipment frame.

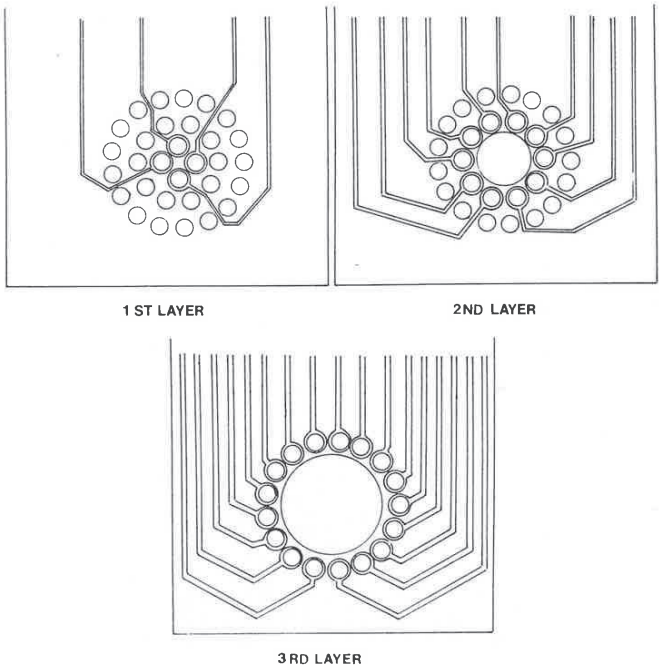


Fig. 1.13 Three layers of flexible circuit for connection to circular bulkhead connector.

1.4.2 Edge Connnectors

Edge connector contacts are an apparently simple way of making connections from a printed circuit board to the frame or unit wiring. The plug is formed on the board itself in the form of 'fingers', which are usually gold plated, along the back edge of the board. These fingers mate with sprung contacts in the socket, which should contain some means of locating the board so that the contacts will lie more or less centrally over the corresponding fingers. The main advantages of edge connectors are that they avoid the costs of buying and fitting separate plugs, and if a finger is damaged it can be re-plated or completely replaced on a board without any risk of damaging other contacts.

It has been claimed that edge connector fingers are unduly sensitive to corrosion, but such comments apply only to boards on which the gold plating lies only on top of the fingers. On properly made boards where the gold lies over the edges of the fingers as well as the top there should be no corrosion problems.

Earlier types of edge connector sockets relied for location on a shaped tongue on the edge of the board which engaged with the ends of the moulded connector body. This means that the board profile must be tightly toleranced relative to the positions of the fingers on the board. Over the past fifteen or so years such 'end locating connectors' have generally been superseded by 'key locating' types in which the socket body is open-ended and a pair of opposing contacts are replaced by a key which engages in a slot cut into the board. Such open-ended sockets are usually available in lengths of 15 to 20 cm so the user can cut them to whatever length is required, and most types have front-insertable contacts. On large production runs, the sockets can be assembled with all unused contacts omitted, which can result in substantial savings as the gold plating on unused contacts can represent quite a large sum of money in a sizeable machine.

Another advantage of such open-ended connectors is that the edge of the board does not require special profiling, other than the locating slot, and contacts can be arranged wherever they are required along the whole edge of the board. Most edge connectors used in the 1980s are double-sided on 2·54 mm pitch, but single-sided and wider pitch connectors are available, and may be more appropriate in some low contact density applications. When an edge connector is being selected, it is important to ensure that the pitch matches the proposed board layout standard. Figure 1.14 shows how the use of too fine a pitch connector for the board layout standard can result in some of the pins becoming unusable.

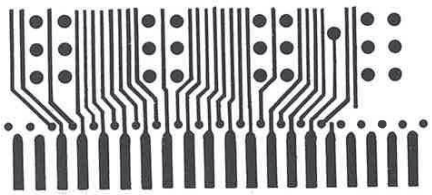


Fig. 1.14 Edge connector finger pitch should be chosen to match the track pitch on the board.

The only real problem in the use of edge connectors lies in the mechanical tolerancing, both on board thickness and on finger position. This tolerancing, which is fully described in a paper 'Making Printed Circuit Boards Fit their Edge Connectors' (Proceedings of the 1973 Internepcon Connector Symposium p. 60–80), should not present any problems to a trained designer. The risks lie in failing to realise that the tolerancing can be critical.[5]

1.4.3 Discrete Wires

The easiest way to attach a discrete wire to a printed circuit board is to pass the end of the wire through a suitable sized hole in the board and solder it in place. When only a few connections are required in a small equipment this can be satisfactory, but in larger equipments it may not be advisable.

As the number of wires involved increases so will the risk of connecting the wrong wire to a given hole. In larger equipments the possibility increases of having to break the wired connections to a board, either for error rectification,

servicing or replacement of a defective board. Any such removal of wires from holes in the board involves a risk of damage to the board, either a pad lifting from the substrate on an unplated board, or irreparable damage to the through-plated barrel on a multilayer board. This risk of damage to the board can be minimised by attaching the wire to a pin which has been soldered into the hole in the board. If a suitable sized pin is used, the wire can be soldered to and desoldered from the pin without any risk of damage to the board. Another advantage of the use of pins is that the risk of attaching a wire to an adjacent via hole instead of to its correct hole is eliminated.

Some analogue circuits contain resistors whose required value can be determined only after the performance of the remainder of the circuit has been established. Any such 'select on test' resistors are best mounted on pins soldered into the board.

Whenever such pins are used to minimise the risk of damage to the board being caused by later manual soldering or desoldering, they should be mounted in places where access for soldering will be as easy as possible. Close spacing of such pins should be avoided, as not only can this make subsequent soldering difficult, it can also lead to thermal damage to the laminate when the board is initially wave soldered.

Generally the pins used will be larger in diameter than typical component leads, and any closely spaced group of pins will represent a mass of metal which can retain enough heat after soldering to cause delamination, crazing or blistering of the base laminate between the pins.

As an alternative to pins, eyelets of suitable diameter can be swaged or pressed into the board and soldered in place to serve as 'non destructible' terminating points for discrete wires. Whilst such a reinforced hole is clearly superior to a plain or through-plated hole as a termination for a wire which will probably have to be desoldered and replaced, it is not as good as a pin for several reasons.

First, it is much more difficult to make and inspect satisfactory soldered joints between the tracks on a board and the swaged over end of an eyelet than it is to a pin; secondly, it is much harder to clear the solder from an eyelet after the wire has been removed, and there is a greater risk of damage to the board; and finally, the use of eyelets involves access to both sides of the board for attaching or replacing wires, whilst the use of pins requires access to only one side.

A third possible attachment point for discrete wires to printed circuit boards is the 'fishbead', a light split pin of flattened tinned copper wire passed through a small ceramic insulator (see Figure 1.15). These fishbeads have the advantages over pins that by opening the ends slightly they are self-retaining in the board

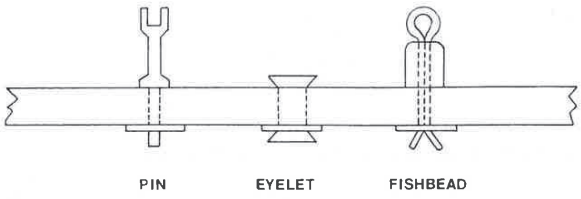


Fig. 1.15 Terminations for discrete wires.

until they are wave soldered, and they have a much lower thermal mass so they can safely be mounted on closer centres than the commonly used types of pins. The ends of 'select on test' components or wires can be passed through the eyes of fishbeads, obviating any need to bend the leads and thus making for quicker and easier assembly.

1.4.4 Back Wiring Panels

When boards are mounted in rows in a rack or case, plugged into sockets, some means of interconnecting the sockets must be provided. On large systems, in which a cabinet some 2 m high may be filled with boards, it may prove cost effective to wire an entire frame with discrete wires soldered, wrapped or clipped to the back pins of the board sockets, but many manufacturers are finding that it is easier and cheaper to use printed circuit back wiring panels in each shelf of boards, or on pairs of shelves. All the advantages of printed circuit boards over discrete wiring apply to back or frame wiring just as they do to the interconnection of components, and there is usually a significant bonus in that a printed circuit back wiring panel adds rigidity to a frame, and can often lead to the elimination of quite expensive metalwork and to substantially reduced assembly costs.

At one time it was thought that the use of printed circuit back wiring panels would cause intolerable electrical interference between what would obviously have to be relatively long parallel groups of tracks, but work carried out by the Marconi Company and others in the 1960s, the results of which are described in the book 'Transistor-Transistor Logic and its Interconnections', demonstrated that in fact well designed back wiring panels result in less electrical interference than random wiring, and their performance can be predicted at the design stage.

If an existing equipment design using discrete back wiring is taken and attempts are made to replace the wires with a printed circuit panel, it is probable that the result will be an expensive multilayer board which requires extra metalwork in the frame to hold it in place. For printed circuit back wiring to be fully effective the electronic system design and the mechanical design of the frame or case must be started with the intention of exploiting the possible benefits from the use of printed back wiring.

One change which may be necessary is to increase the length of the board connectors to give more contact positions for each board. Not all the signals present on the back wiring panel are likely to be required by all boards and, if connectors with only the correct numbers of pins for the most complex board are used, it may prove very difficult to find routes for other signals past the socket for that board. A number of redundant pins on each connector allows more signals to run past each connector position. At the limit, such redundant pins can make it possible to reduce the design of a back wiring panel to a simple 'pin for pin' linkage in which every signal is present on every board socket. Such a panel can be made in the form of a single-sided board, and there is the added bonus to the user of the equipment or the service engineer that any board in the system can be plugged into any vacant socket.

1.5 THE ELECTRICAL IMPORTANCE OF THE INTERCONNECT

In the earlier days of electronics, especially digital electronics, the only significance of the interconnect was that it had to route the signals to the correct

destination. The interconnect was a solid representation of the lines on the circuit diagram which joined together the various active components. In high gain, high frequency analogue circuits some of the interconnections could be recognised as being critical, especially those to grids of thermionic valves or the bases of amplifying transistors, and feed-back loops required special care. Digital signals were relatively slow, and delays along wires which ran from cabinet to cabinet were not significant compared with the delays in the active devices.[1,3]

The advent of the diffused planar silicon integrated circuit, with the possibility of sub-nanosecond devices, changed the situation. Delays down even a few inches of track on a printed circuit board became critical, and designers had to consider all the electrical effects described in Chapter 2 if their circuits were to work as they hoped. Instead of the tracks on a board being solid representations of plain lines on a circuit diagram, they could more properly be shown with symbols representing their resistance, capacitance and inductance if the diagram was to give a true picture of the circuit performance.

The widespread adoption of digital integrated circuits brought another fundamental change to the design of electronic systems. Control of one of the most vital parameters, the switching speed of a device, passed from the equipment designer to the component designer. The equipment designer retained control of the frequency at which a device would be switched, but the actual speed of the transitions was determined by the device type used with only a slowing or degradation of the switching speed possible because of a poorly designed interconnect. Sometimes deliberate attempts were made to slow down high speed devices in order to allow them to perform satisfactorily with an inadequate interconnect. These attempts resulted in short device lives. Typical of this approach were the efforts to slow down TTL devices by applying a high capacitive load to the output. The almost unlimited instantaneous discharge currents of such capacitors were excellent for severely curtailing the life of an output transistor.

Variations in the clock frequency have no effect on line surge currents, ring, and crosstalk, all of which are determined by the switching characteristics of the device. Another popular fallacy which led to the design of inadequate boards is the thought that, since digital logic circuits work by setting either a 'high' or a 'low' level, what happens between the levels can be ignored.

The successful design of high density interconnects for digital circuits demands a good understanding both of what happens in the devices when they change their state, and also of the logical working of the circuit being designed.

An approach which relies on keeping all connections so short that their electrical effects can be ignored will usually prove to be uneconomic, and often leads to severe and unnecessary thermal problems as masses of devices are squeezed into the smallest possible volume. In any system there will be some signal paths which can be allowed to be long without any ill effects, and there will be others which can be run in parallel groups without cross-talk causing any problems. The board designer must be capable of assessing the importance of any track and implementing the board layout so that the lengths of critical tracks are minimised without incurring the unnecessary expense of an excessively high packing density. To give a very simple example, it is clearly absurd to attempt to minimise the length of tracks to a manually operated switch at the expense of making a strobe or clock signal line longer.

It is unfortunate that a well designed board often appears indistinguishable

from a thoroughly badly designed board, and it is not always immediately obvious that, whilst the integrated circuits used place a limit on the fastest possible working of any given system, the actual performance achieved will be determined by the design of the printed circuit boards used. For an electronic system to be competitive, it is essential that the interconnect be designed to yield the best possible electrical performance without unnecessary expense, either in direct manufacture of the interconnect or in any consequent operation such as assembly, testing, cooling, or field servicing. The first step to such competitive design is the choice of the right type of board for the job.

1.6 BOARD TYPES

The printed circuit board industry can offer to the electronic designer a wide range of board types, and on each type the pattern density can vary over a wide range. Generally the following list of board types is in ascending order of cost, but often there is some overlap between a high density board of one type and a low density board of the next higher type. Only the more general applications are covered.

1.6.1 Single-sided (Figure 1.16)

This type has a track pattern on one side only, and is used mainly in transistor radios and other similar domestic equipment. Single-sided boards are often made using phenolic paper base laminate, and the holes may be punched out.

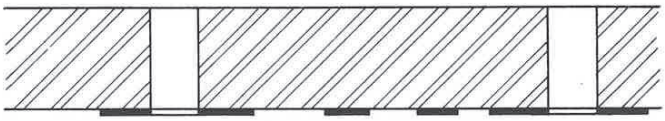


Fig. 1.16 Section through typical single-sided board.

1.6.2 Double-sided, Non Through-plated (Figure 1.17)

These boards are basically the same as single-sided except that they have a track pattern on each side of the board. Their application extends to more complex domestic equipment and a wide range of lower density industrial applications. Double-sided, non through-plated boards may be made on phenolic-paper or epoxy-glass laminates, and holes may be drilled, or punched in one operation with the profile.

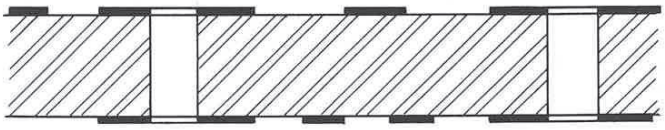


Fig. 1.17 Double-sided, non through-plated board.

1.6.3 Flexible (Figure 1.18)

Flexible circuits are as described in 1.6.1 or 1.6.2, but are made on very much thinner, flexible plastic base materials such as polyester or polyimide. Often the track patterns are covered with a further thin layer of plastic similar to the base laminate, bonded on to seal in the track pattern. Flexible circuits are used in a wide range of applications to replace conventional wiring harnesses. Noteworthy mass applications for flexible circuits include motor vehicle wiring, automatic vending machines, and some types of pocket calculators.

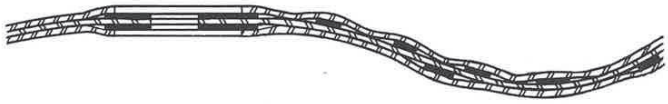


Fig. 1.18 Double-sided flexible circuit.

1.6.4 Double-sided, Through-hole Plated (PTH) (Figure 1.19)

This type looks like that described in 1.6.2, the difference being that the track patterns on the two sides of the board are interconnected by metal plated into the barrels of the holes. The manufacturing processes used are quite different (see Chapters 5 and 6) and the interconnection densities found on PTH boards can vary widely. Their applications are almost unlimited, from sophisticated domestic equipment (Hi-fi) through industrial and commercial equipments of all types to military, telecommunications, and aerospace uses. Most PTH boards are made on epoxy-glass laminates.

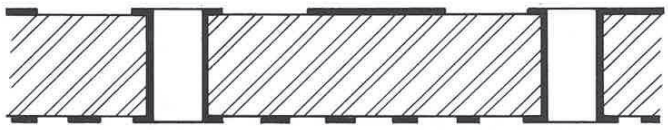


Fig. 1.19 Double-sided PTH board.

1.6.5 Flexible, Through-hole Plated (Figure 1.20)

This is as in 1.6.3, but with the holes through-plated as in 1.6.4, and is found mainly in airborne equipments and other areas where greater reliability or a higher packing density is needed than can be achieved with a plain flexible circuit. They are usually made on polyimide base material.

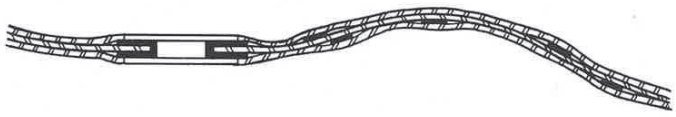


Fig. 1.20 Through-hole plated flexible circuit.

1.6.6 Rigidised Flexible (Figure 1.21)

A rigidised flexible is as described in 1.6.3 or 1.6.5, but with the areas in which components are mounted stiffened by bonding on a rigid plate, either of metal or plastic.

They are used where the primary requirement is for a flexible interconnect and limited numbers of components which must be mounted to be restrained against movement caused by vibration. Note that the stiffening plate plays no part in forming the electrical interconnect, but a metal plate can serve as a heat sink for high dissipation components. High packing densities can be achieved on rigidised flexibles because the thin base laminate permits the use of very small (0·1 mm diameter) via holes which can be drilled, punched or etched, and the components can cover the whole of the rigidised area with no need to allow any space for external connections.

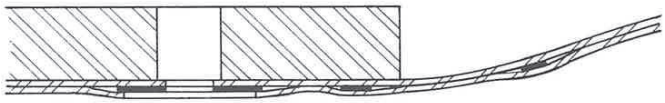


Fig. 1.21 Rigidised single-sided flexible circuit.

1.6.7 Multilayer (MLB) (Figure 1.22)

A multilayer board has three or more layers of tracking, bonded together to form a single unit. Some early types of multilayers did not have electrical connections between the layers, but relied on access holes in the layer or layers on the solder side of the board to allow component leads to be soldered to pads on inner layers. Nowadays, the term MLB can be taken as applying to a board on which access to the inner layer tracking is made by through-plated holes. Multilayer boards are used in the more densely packed industrial, military and aerospace applications.

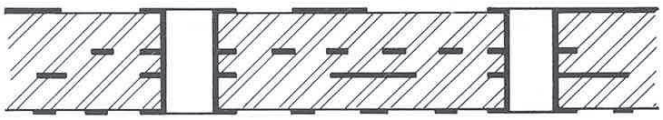


Fig. 1.22 Four layer multilayer board.

1.6.8 Flexi-rigid (Figure 1.23)

A flexi-rigid board is basically a multilayer board on which one or more of the rigid epoxy glass layers is replaced by a layer or layers of tracking on a flexible base laminate, usually polyimide. The flexible layer or layers extends beyond the rigid area or areas, either to form 'hinges' which interconnect two rigid areas, or to form 'flying tails' which can be used to make direct connections to components mounted off the board. Holes in a flexi-rigid are through-plated. The build may vary from a single flexible layer bonded to a rigid area which has tracks only on its opposite face, up to very complex multilayer builds with a number of single or double-sided flexible layers. The applications of flexi-rigids are far too numerous to list, and include such areas as machines where a portion of the electronic components must be carried on a moving part, such as indicator lamps on a door, to ultra high density applications where a board must be folded to fit into a space which would not contain a rigid board. The simpler flexi-rigids offer similar tracking densities to rigidised flexibles, but the multilayer build of a flexi-rigid extends this capability to meet almost any tracking density likely to be needed. The more sophisticated flexi-rigids can be very expensive indeed, but their performance capability can make them cost effective in commercial applications as well as in aerospace equipments.

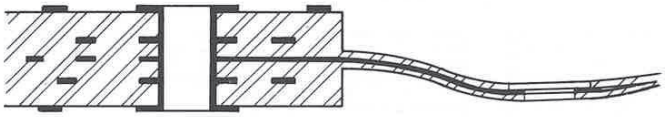


Fig. 1.23 Five layer flexi-rigid with single tail.

1.6.9 Buried Via Hole (BVH) (Figure 1.24)

A buried via-hole board is a multilayer board in which pairs of internal layers have connections made by through-plated holes. These via holes are plated through before the layers are bonded, so the same sites can be used for via holes on more than one pair of layers, yielding a significantly higher tracking density than is possible on a normal MLB where all via holes must pass through all layers. Flexi-rigids may be made with plated-through holes in flying tails, and this is a form of BVH construction, even though the holes are exposed on the finished board.

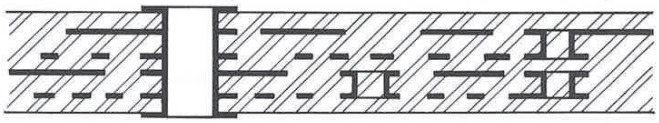


Fig. 1.24 Six layer buried via-hole board.

1.6.10 Double-double Boards (DDB) (Figure 1.25)

A double-double board is a variant of a BVH board which is made by bonding together two thin plated-through hole double-sided boards. A thin double-sided laminate carrying power and earth planes may be included in the middle of a double-double board to give a 6 layer build. The essential difference between a DDB and a BVH board is that on the DDB all via holes access one or other of the outer layers whilst on a BVH board none of the holes plated prior to bonding contacts an outer face.

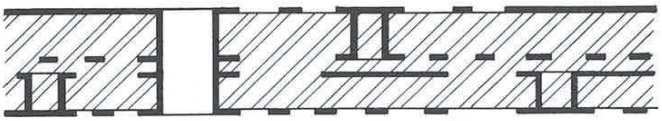


Fig. 1.25 Double-double board with no internal planes.

1.6.11 Double-multi Boards (DMB) (Figure 1.26)

These are basically as double-double boards, except that two multilayer boards with all via holes plated are bonded together.

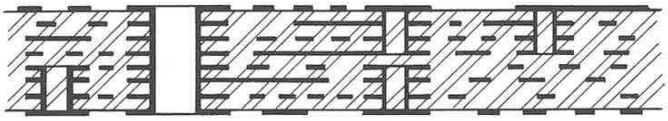


Fig. 1.26 Double multilayer board.

1.6.12 Sequential BVH Boards (Figure 1.27)

The ultimate in multilayer sophistication is the sequential BVH board in which holes between a pair of inner layers are through-plated; a further piece of rigid laminate with an etched pattern on one face and with plated via holes is bonded on, and the composite is then drilled and through-plated with second level via holes. This process can be extended through as many layers as necessary until the final outer layers are bonded on and the component mounting holes are drilled and plated to provide electrical access to all layers. Boards as in Section 1.6.9 onwards are obviously very expensive to make, and their use is generally confined to the most sophisticated areas of the high speed computer and aerospace applications.

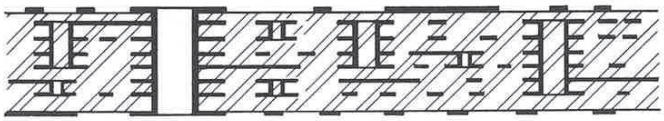


Fig. 1.27 Eight layer sequential multilayer board.

1.6.13 Pillar Plated Multilayers (Figure 1.28)

Pillar plated multilayers are boards in which layer to layer connections are achieved by solid metal pillars instead of by plating onto the walls of holes. These boards have to be built up sequentially, with plated track layers and applied dielectric layers, so they are expensive, but the pillars can be significantly smaller

than can through-plated holes, so high packing densities can be achieved. Plated pillar boards are particularly suitable for mounting miniaturised surface-mounting devices.



Fig. 1.28 Four layer pillar plated board.

1.6.14 Flush Bonded Boards (Figure 1.29)

This type, which breaks the sequence of rising cost with type, is a non plated-through hole board on which the track pattern is sunk into the base laminate to give a flush surface. They are normally used for rotary or linear motion wiping action switches. A variant of this type is the 'bent board', a board as in 1.6.1 or 1.6.2 with a bend permanently formed in it. Bent boards of suitable curvature can be used to form the deflection coils for cathode ray tubes.

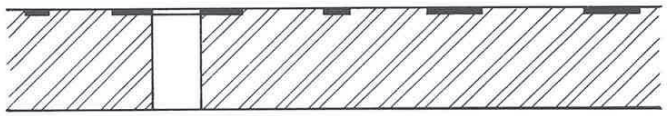


Fig. 1.29 Simple flush bonded board.

1.6.15 Strip Line or Microwave Bonded Circuit (Figure 1.30)

These are highly specialised boards, usually 3-layer build, often made on PTFE base laminate, in which the inner layer carries the signal and the outer layers form earth screens, which may continue as plating over the edges of the board. They are used in place of waveguides or similar techniques as signal mixers, circulators, isolators, aerials and so forth for low power, high frequency applications such as radar or surveillance systems.

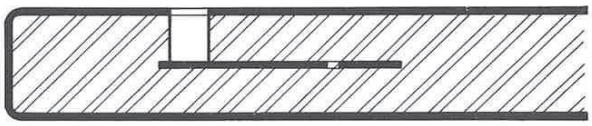


Fig. 1.30 Section through microwave bonded board.

Chapter 2

MECHANICAL AND ELECTRICAL PROPERTIES

2.1 BASE MATERIALS

The mechanical and electrical properties of printed circuit boards are determined by the design of the board and by the type and thickness of laminate used.[1,2,6,7,8]

2.1.1 Cladding

The most commonly used metal cladding is copper, but nickel and nickel alloy clad laminates are available for special purpose boards. The thickness of the metal cladding is often referred to by its weight per square foot, 1 oz copper being the thickness used most in printed circuit board work. This is 0·035 mm thick, usually with a tolerance on thickness of plus or minus 0·002 mm. Whenever calculations which involve copper thickness are being made, it must be remembered that board manufacture usually involves scrubbing or cleaning which will decrease slightly the thickness of the copper, and on through-hole plated boards the copper thickness on the board surface will be increased. Laminates clad with 2, 3 or 4 oz copper are available, as are 0·5 and 0·25 oz claddings, and it is possible to buy laminates with unequal metal thicknesses on the two faces.

The copper foil on most rigid laminates is originally made by electrodeposition onto a large, highly polished metal drum from which it is then stripped, but on some flexible laminates a rolled foil, or rolled annealed foil, is used. The face of the copper which is bonded to the laminate has to be micro-roughened to achieve a good bond, and in most cases it will be oxidised.

2.1.2 Base Laminate

The two main types of base laminate used are phenolic paper and epoxy glass. Phenolic paper based laminates are used mainly for non through-plated boards for the domestic and simpler industrial markets. The main advantages of phenolic paper over epoxy glass in these areas are its lower cost and the ease with which it can be punched. Complex profiles and all required holes can be formed with a single stroke of a large press. The electrical properties of phenolic paper laminates are inferior to those of epoxy glass and its higher moisture absorption makes it unsuitable for most plated-through hole work. Typically, phenolic paper laminates have a dielectric constant of about 5, a breakdown voltage of about 15 kV and a dissipation factor of around 0·04. Also, the mechanical strength of phenolic paper is less than that of epoxy glass. Phenolic paper laminates are distinguishable from epoxy glass by their colour, which is usually a deep purple,

brown or black. Epoxy paper laminates are available, but are used only in special cases where a phenolic resin based laminate would have inadequate properties but the paper base is considered desirable.

Epoxy resin with a chopped strand glass reinforcement can be used where properties such as moisture absorption are critical, and better electrical properties are required than can be achieved with a paper based laminate. Chopped strand glass laminates reinforced with epoxy resin can be punched, and are less expensive than epoxy/woven glass. However, they are not widely used and most of their possible uses are met by epoxy/woven glass laminates which are readily available and have better electrical properties and strength.

Woven glass cloth reinforcement with epoxy resin is used for most through-hole plated printed circuit boards and for multilayers. It offers an excellent combination of mechanical and electrical properties coupled with low moisture absorption. Typically, the dielectric constant is around 5, with a dissipation factor about 0·035 and a DC breakdown voltage of around 40 kV.

Epoxy glass laminates are commonly available in a wide range of thicknesses from 0·1 mm upwards, with 1·6 mm being the most commonly used thickness for double-sided boards. The usual colours are pale, translucent green or brown, but other colours can be purchased to special order.

Rigid laminates for specialised applications are available on base materials such as irradiated polyethylene, PTFE, melamine, triazine, polysulphone, silicone and polyimide resins, some of which may be reinforced.

Flexible laminates are most commonly available on unreinforced base films of polyester or polyimide. Polyester based flexible circuits are cheap and easy to make, but the low softening point of the base material makes them hard to solder to, so it was not until the higher softening temperature polyimide base materials became available that custom designed flexible circuits achieved any real measure of acceptance.

The commonly used laminates are all available in fire retardant forms, usually referred to by the American NEMA (National Electrical Manufacturers Association) designations of FR2 (paper phenolic), FR3 (epoxy paper), FR4 (epoxy woven glass), FR5 (higher working temperature epoxy woven glass, not in common use) and FR6 (polyester chopped strand glass).

No detailed properties are quoted here as they vary slightly from manufacturer to manufacturer and the laminate suppliers' data sheets are the best source of exact information.

2.2 STRENGTH AND STIFFNESS

The mechanical strength of printed circuit boards is seldom, if ever, of critical concern. Boards have the same strength as comparable panels of plain plastic material and they are not commonly used as major structural load-bearing members. The components they carry are generally small and light, and provided the normal rules of good mechanical design are followed it is usually unnecessary to become involved in any calculations on strength.

One point which does have to be considered is the resistance to tearing, cracking and impact damage of corners especially if a mechanical fixing hole is drilled or punched near a corner. Some users place restrictions on the amount of material which must be left between a hole and the edge of the board, such as a minimum allowance of twice the diameter of the hole.

The aspect of mechanical strength which is most likely to be of concern is the stiffness of a board. An epoxy glass based board half a metre long can bow by 2 cm or more in the middle under vibration, and can impose severe acceleration forces on the components it carries. Boards which are mounted in racks plugged into sockets may have their outer edges stiffened by some form of front plate or handle which may be mechanically secured to the frame, but unless special locking board guides are used the top and bottom edges of the board must be free to move in their guides. Sometimes it is assumed that the guides will hold the board sufficiently firmly to prevent any movement, but this is not generally the case. Board guides and the overall size of the board should be designed so that the guides present the connector edge of the board to its socket in such a position that the locating mechanisms between the board and the socket can engage, but the clearance between the board and its guides should be greater than the sum of all tolerances on the trimming of the edges of the board and on the positions of the board guides relative to the sockets. If an inadequate clearance is allowed, the board may jam in the guides as attempts are made to mate the connectors. The tolerances between the connector bodies and the board guides are usually the sum of anything from 3 to 5 or more terms, each of which may be up to plus or minus 0·15 mm. Thus the float of the board in its guides may have to be more than 0·5 mm, which will not restrain the board against vibration.

The effects of vibration on the components carried by the boards must be considered, as also must the effects on the contacts in the sockets which connect to the board.

Boards which carry a mixture of components of different sizes can have their mechanical resonant frequencies changed by moving the components to different positions. If this does not reduce the vibration to acceptable levels the stiffness of the board can be increased by the use of thicker laminate or by bonding, screwing or rivetting stiffening bars to the board. All the normal principles of mechanical engineering apply to printed circuit boards in the same way as they would to any panel.

2.3 STABILITY

There are two major aspects of printed circuit board stability which must be considered. One is the dimensional stability of the base laminate and the other is the effect of heat on the board.[1,8]

2.3.1 Dimensional Stability

Epoxy glass laminates offer better dimensional stability than phenolic paper, but neither material is truly stable. Machining, either profiling or drilling, to exact dimensions is difficult on any plastic material as there is a tendency for the material to compress away from the cutter while the work is in progress and then to 'grow' slightly after the cutting is finished. No printed circuit board can offer closer tolerances than would be achievable on a comparable sized piece of plain plastic material, and for economic manufacture the printed circuit board must be expected to offer less dimensional stability than would plain laminate.

As automatic component insertion is increasingly used in board assembly, it is not uncommon to find tooling holes, which may be almost half a metre apart, toleranced to plus or minus 0·05 mm. Such tolerances may be acceptable in a

precision toolroom, but they are quite unacceptable in mass production of boards made on a base material which may be stable only to 400 parts per million. The mechanical design of boards, and of machines which are designed to assemble or test them, should allow for sensible production tolerances plus material movement. An allowance should also be made for movements which are inevitable during board manufacture (see Chapters 5 to 10). If boards must be made to locate on machines, proper kinematic principles should be adopted and location should be by a hole and slot or slots and not by excessively tightly toleranced hole centres. Board manufacture is eased if holes are drilled in the boards and the beds on which the boards are to be located are fitted with a fixed pin and a pin or pins free to move in a slot or slots.

Material movement during board manufacture occurs because the copper clad laminate is bonded under heat and pressure and when the laminate cools differential shrinkage of the copper and the base laminate causes inbuilt stresses which are released when the surface copper is etched away. This material movement can be reduced by heat treating the sheets of laminate, but it can not be eliminated.

The normal processes of board manufacture involve immersion in liquids, often followed by heating processes. Both of these can and do cause dimensional changes. A designer of cost effective boards should make the most generous allowances possible for material movement, and should never quote tolerances tighter than those applied to mass production of other parts made from similar base materials.

2.3.2 Thermal Stability

Like any other material, printed circuit boards change their size with changes in temperature. The magnitude of the change depends on the resin and the reinforcement, and, in any cases where the size of a board is critical, the appropriate laminate manufacturer's data sheets should be consulted. The resin is the least dimensionally stable component of the laminate, and its movement with changes of temperature is constrained by the reinforcement. Glass cloth reinforced boards are considerably more stable than paper based or chopped strand mat based laminate. On woven laminates the coefficients of expansion vary between the warp and weft directions of the laminate. All laminators of printed circuit board material include coloured markers in the material to indicate which is the weft direction, but these markers are visible only after any copper cladding has been etched off, so care must be taken when cutting stock sheets of laminate into blanks to ensure that the boards will be produced the right way relative to the warp and weft. Thermal instability of assembled boards in the X and Y axes and differential expansion between the boards and the components they carry can be accommodated by ensuring a degree of compliance in the leads of components which are mounted in holes in the board. This is usually achieved by bending the leads as in the case of axial lead resistors or DIPs, or by using thin copper leads with a stand off mounting as with TO18 or TO5 transistors. Large surface mounted components may require special consideration and the use of either a special substrate or of mounting pillars designed to incorporate some flexibility.

The aspect of thermal stability of most concern to the packaging engineer is Z plane expansion, or changes in the thickness of the laminate. Z plane expansion

in board laminates is always significantly greater than X and Y plane expansion because there is no continuity of reinforcement through the thickness of the boards and expansion of the resin is not constrained to the same extent as it is along and across the board. This can cause problems on all types of boards. On single-sided boards the component leads are normally passed through holes in the board, bent over, and soldered to the copper pads. If the board material expands more than the lead, the lead will deform or stretch to make up the difference, and when the board contracts the component will be slightly loose. Then vibration can cause the component leads to push the pads to which they are soldered off the base laminate over a period of time. Phenolic paper based laminates, on which the adhesion of the copper cladding to the base is less than that on an epoxy based board, are particularly liable to suffer from this type of damage. Some manufacturers of airborne equipments specify that all holes must be through-plated to avoid the risk of detached pads.

Through-hole plated boards are not immune from the effects of Z plane expansion. The differential expansion of the base laminate and the copper hole barrels can cause cracking at the 'knee' where the surface copper turns to pass through the hole. This is not usually a hazard on well made double-sided boards of normal thickness, up to about 2 mm, but it is not uncommon after thermal shock testing to find the pads are slightly depressed in their centres. The biggest problems are found on multilayer boards, especially those which have been badly designed and do not have pads round every hole on every layer. Pads on the surface layers of boards are able to 'creep' slightly over the laminate and to 'dish' to accommodate the differential expansion, but pads on inner layers are bonded on both sides and so are less free to deform. Thus the most usual site for thermal damage on a thick multilayer board with a significant number of layers is where the pads on the first layers in from the surface join the barrels of the holes. Some users specify that these first layers in should be 2 oz copper to give added strength at this highest stress point. Layers near the centre of the board are relatively immune to this type of damage.

Omitting the pads so as to avoid the risk of cracking introduces a much greater hazard. If a pad around a hole is omitted, there will be a clear area in the copper pattern for the layer. When the board is bonded, resin from the bonding sheets can flow into this clear area, but the glass reinforcement can not. Thus on the finished board there will be a thin ring of unreinforced resin round the hole. Whilst the Z plane expansion of the reinforced laminate is greater than X and Y plane expansion, it is not as great as that of the unreinforced resin, so the rings of resin round the holes will impose high local stresses on the hole barrels during thermal shock or thermal cycling. Ultimately the barrels will crack at the resin-rich rings. Putting a pad on every layer round every hole ensures that during bonding maximum pressure will be exerted on the board at the hole sites, and there should be no resin-rich zones to cause later reliability hazards.[1,8]

2.4 ELECTRICAL PROPERTIES—DC

In most applications the electrical properties of printed circuit boards can be every bit as important as their mechanical properties, and the characteristics of the tracks should be given as much consideration as their routing. The characteristics of the tracks can be split into the DC properties, which apply at all times, and AC properties which have an effect when there is a change in signal

level. The DC properties to be considered are the resistance of the tracks and their ability to carry given levels of current, the surface resistance and dielectric breakdown through the laminate which govern the allowable voltage differences between tracks.[1,2,8,9,10,11]

2.4.1 Resistance

With the increasing use of microelectronics, most tracks on printed circuit boards carry low level currents between devices which can tolerate substantial variations in signal voltage levels, and the resistance of the tracks is not critical. However, digital logic boards and back wiring panels require power supply tracks which may carry high currents, and on analogue boards the track resistance may have to be controlled.

Track resistance, R, is measured in terms of the copper resistivity, ϱ, and the track dimensions of l (length), w (width), and t (thickness), such that

$$R = \varrho l/wt$$

A commonly used derived term is sheet resistance, which is

$$R_s = \varrho/t$$
$$R = R_s l/w$$

Figure 2.1 shows the typical resistance of copper tracking for half, one, two and three ounce coppers at 25 and 100 degrees C, and Figure 2.2 shows the resistance of plated-through holes in 1·6 mm thick laminate.

In all critical cases the cross-sectional area of the track should be calculated, using worst case values, and the maximum resistance established from the resistivity of the metal. When a small tolerance can be allowed on the resistance, a

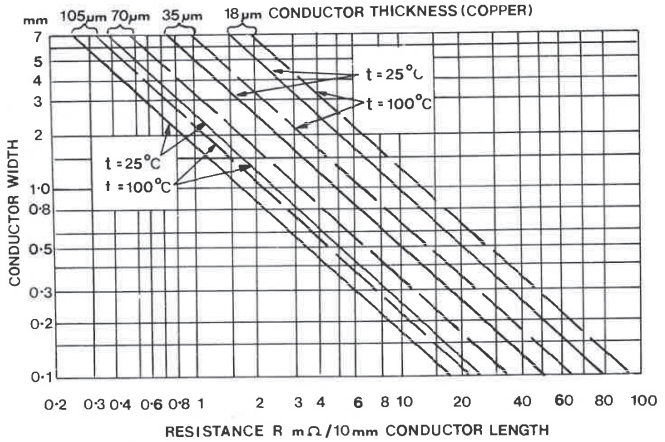


Fig. 2.1 Resistance of copper tracks.

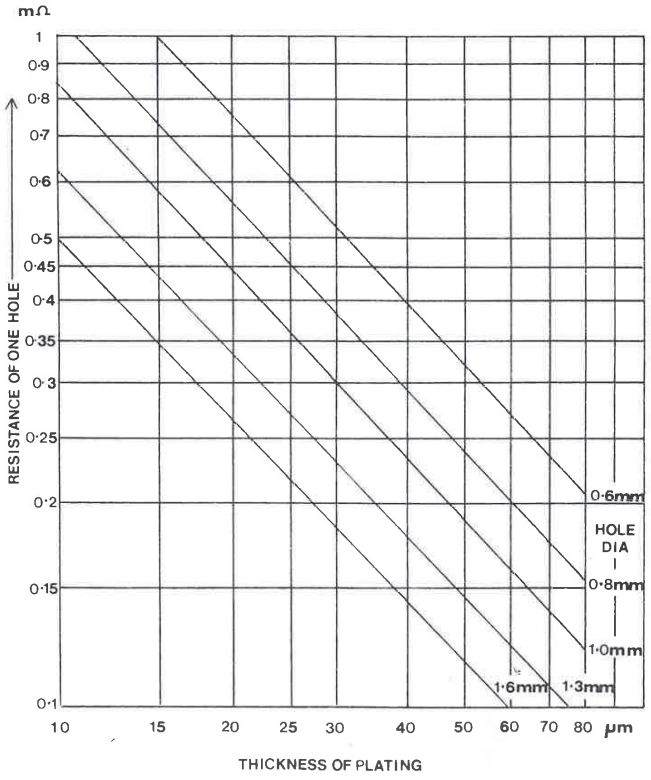


Fig. 2.2 Resistance of through-plated holes in 1·6 mm laminate.

sheet resistance of half a milliohm per square for one ounce copper can be used.

It must be noted that these are theoretical figures based on the resistivity of copper and they do not include allowances for manufacturing tolerances. On boards with plated-up tracks on the surface layers there can be a very wide spread of actual values of resistance caused by the possible variations in thickness of the copper plating and the presence of tin-lead. The tin-lead thickness may be difficult to determine if the board is to be reflowed, and is best regarded as a bonus which is not calculated but will lower the resistance of the tracks.

As an example, one can consider the cross-sectional area of a track nominally 0·3 mm wide on a through-hole plated board made from 1 oz clad laminate with

a minimum of 0·020 mm of copper on the hole walls. The cladding thickness should be between 0·033 and 0·037 mm. Cleaning and scrubbing can widen these limits to 0·031 to 0·037 mm. Plating can add from 0·020 mm to 0·040 mm or more, giving a total thickness of 0·051 mm to 0·078 mm. The track width may be increased by overplating or reduced by over etching, and film errors may vary the width. Limits of 0·2 mm to 0·35 mm are quite reasonable, and yield areas of 0·0102 to 0·0234 mm^2 plus some allowance for tin-lead. At a resistivity of 18 nano-ohms per metre, the resistance of such a track could be from 1·8 ohms per metre length down to well under half that figure.

The two most significant effects of track resistance are the heating of the track caused by power loss in the resistance, and the voltage drop along the tracks. Curves of safe current levels for 10 and 20 degree temperature rise are shown in Figure 2.3, and Figure 2.4 shows the voltage drop on boards made from 1 oz clad laminate. Figure 2.4 includes allowances for manufacturing tolerances on the metal thickness. Maximum current densities are usually well under 4,000 A per square cm.

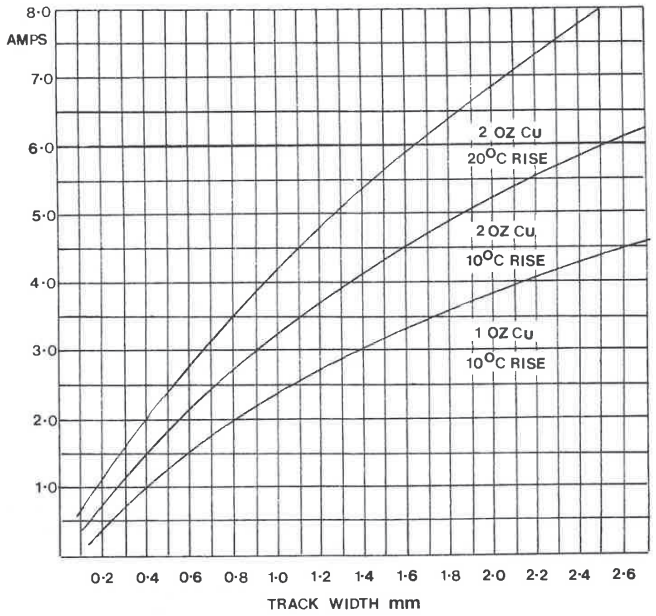


Fig. 2.3 Safe currents in tracks on PCBs.

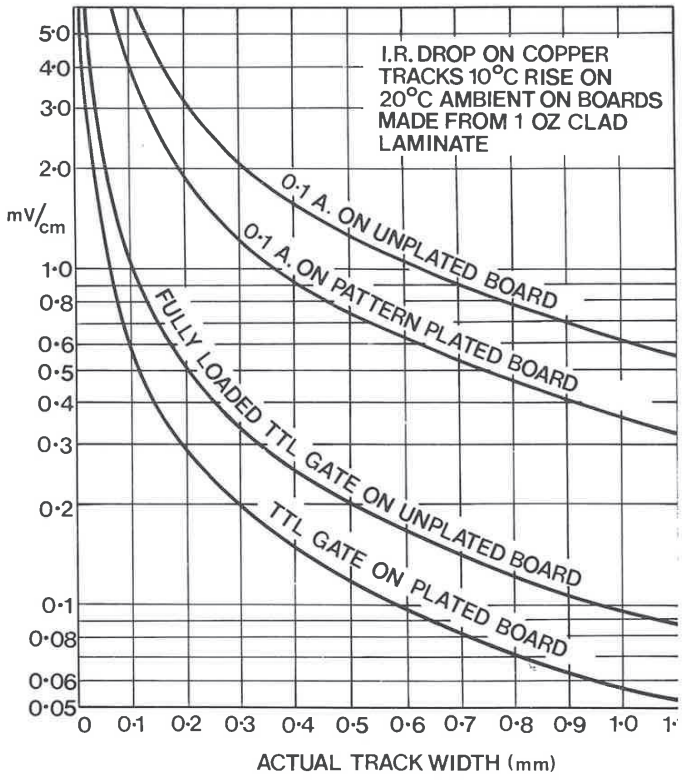


Fig. 2.4 Voltage drop on tracks on PCBs.

2.4.2 Voltage Clearances

The surface resistance of board laminates is high, from about 1000 megohms on phenolic paper to 10,000 megohms on epoxy glass, and the volume resistances are about ten times these values per centimetre. Layer to layer voltage breakdown on multilayer boards is usually tested at 1000 V. The only voltage problems normally met on boards lie in the possibility of breakdown of surface resistance caused by contamination, which depends on the working environment. For normal dry environments free of conductive dust in significant quantities, a spacing of 1 mm per 100 V is usually considered adequate. When AC mains

voltages have to be accommodated on boards, clearances of around 4 mm are normally allowed.

On most digital logic boards the limitations on track spacing are more likely to be for ease of manufacture or because of cross-talk than for voltage clearance. When it is necessary to calculate voltage clearances, an adequate allowance should be made for manufacturing tolerances on track and pad widths and positions, for the possible presence of any metallic particles between tracks, and for the effects of any ionisable surface contamination. Any such ionisable contaminants can conduct currents from track to track, which can cause the growth of metallic whiskers on the board surface. These whiskers can reduce voltage clearances to unacceptable levels, and can grow at quite low voltages over a period of time.

2.5 ELECTRICAL PROPERTIES—AC[1,2,8,9,10,11]

Most tracks on printed circuit boards carry signals which change their level, either continuously on analogue boards or more abruptly on digital boards. The way in which the tracks influence these changing signals is determined by the properties of inductance and capacitance (Figure 2.5) and their combination as a line characteristic impedance. On short interconnections the track characteristics can be ignored; on medium length connections the lumped characteristics of inductance and capacitance have to be considered, whilst long connections must be considered as transmission lines.

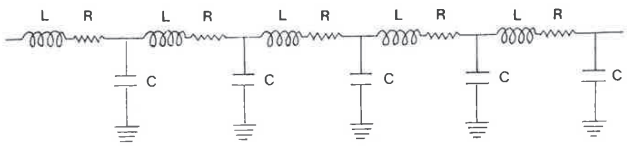


Fig. 2.5 Electrical characteristics of a track on a PCB.

The 'length' of the connections is determined by the highest operating frequency or, in the case of digital boards, by the rise or fall time of the pulses. The higher the frequency or the faster the pulse edge, the shorter the tracks have to be if the lumped characteristics are to be used instead of the characteristic impedance. It must be noted that the 'length' is the effective electrical length, not the measured mechanical length, and the effective electrical length depends on the line characteristics.

In almost all AC calculations on printed circuit boards the resistance is low enough to be ignored and lines can be treated as lossless other than on high Q analogue circuits on which tracks will usually be kept down to the minimum practical length of around 2 mm.

2.5.1 Capacitance

The capacitance of a single track on the opposite side of a board to an earth plane can be calculated from the area of the track, the dielectric constant of the

laminate, and the board thickness. Any such calculations must include allowances for tolerances, so the result is at best an approximation. On practical boards the capacitance of a track is difficult to calculate as the influence of adjacent tracks must be taken into consideration, and often the back of the board will carry other signal tracks and not an earth plane. Useful calculations of capacitance are made more difficult by the fact that, when tracks are on the surface layers of boards, part of the coupling between adjacent tracks is through the air above the board. For very rough estimations, a value of 0·5 pF per cm can be assumed for tracks about 0·3 mm wide on moderate density double-sided digital logic boards. Precise calculations of capacitance are usually necessary only on very high frequency analogue boards or microwave boards which are likely to be multilayers with earth screening layers.

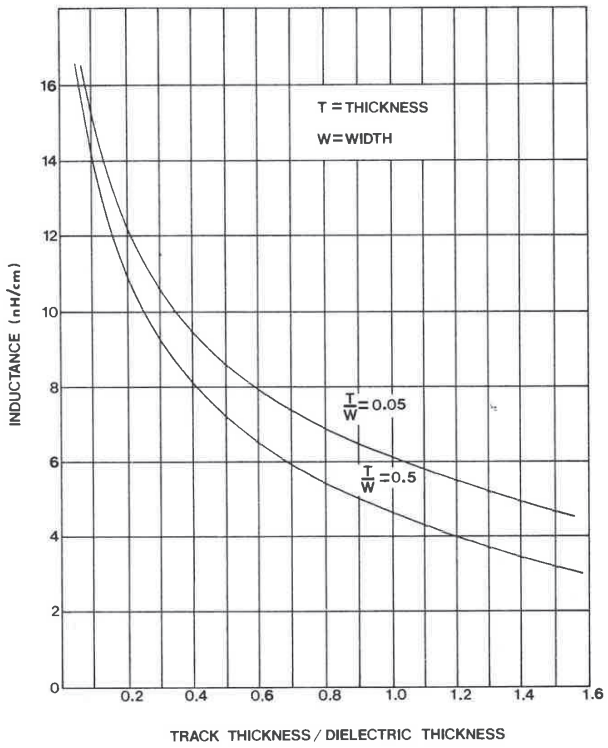


Fig. 2.6 Inductance of tracks on PCBs.

2.5.2 Inductance

The inductance of a track varies inversely with the ratio of thickness to width, and inversely with the ratio of track size to separation from the return current track. On most practical boards the precise value of inductance is not critical and it is sufficient for the designer to appreciate that tracks do have measurable inductance and hence if small value capacitors are mounted on spur tracks of appreciable length the result is more likely to be an oscillatory circuit than a pure capacitance.

On some analogue or line receiver circuits it may be necessary to incorporate filter circuits with controlled values of inductance, often in the form of a spiral track with connection to the centre made by a through-plated hole. The design of such filters is beyond the scope of this book and for most printed circuit board design work the curves of Figure 2.6 can be used to predict inductances.

2.5.3 Characteristic Impedance

The combined effect of the distributed capacitance and inductance of a track makes the track appear as if it was a resistance to earth while it is carrying a changing signal. When the voltage on the track changes, the line capacitance has to charge or discharge through the line inductance, so a current flows in the line during the change, the amplitude of the current being the voltage change divided by the characteristic impedance (the effective equivalent value of the resistance to earth) of the line.

Values of characteristic impedances for tracks on FR4 epoxy glass laminate are shown in Figure 2.7, on which the dots mark the normally used dielectric thicknesses for boards with 2, 4, 6 and 8 layers.

The distributed inductance and capacitance of a line delay the signals it carries as shown in Figure 2.8, and both the characteristic impedance and the line delay are functions of track width, laminate thickness, and dielectric constant. Delay increases as the characteristic impedance is lowered, which happens when the track width is increased or the dielectric thickness is lowered. Thus to maintain a given electrical performance, a decrease in laminate thickness must be compensated by the use of thinner tracks.

2.6 EFFECTS OF AC PROPERTIES

The AC properties of tracks on printed circuit boards can have critical effects on the satisfactory working of the board. It is obvious that in high frequency analogue work the inductance and capacitance of the tracks must be taken into account. Whilst it is not so obvious, it is equally true that the board design can have a significant influence on the working of digital logic circuits. Line reflections can introduce delays into settling times, and cross talk and surge currents can cause spurious switching. It is important that all who are involved in printed circuit board design understand these critical AC effects.

2.6.1 Line Reflections[8,9,10,11]

As the voltage on a line is changed a current must flow in the line and, when this current reaches the end of the line, something must happen. If the end of the

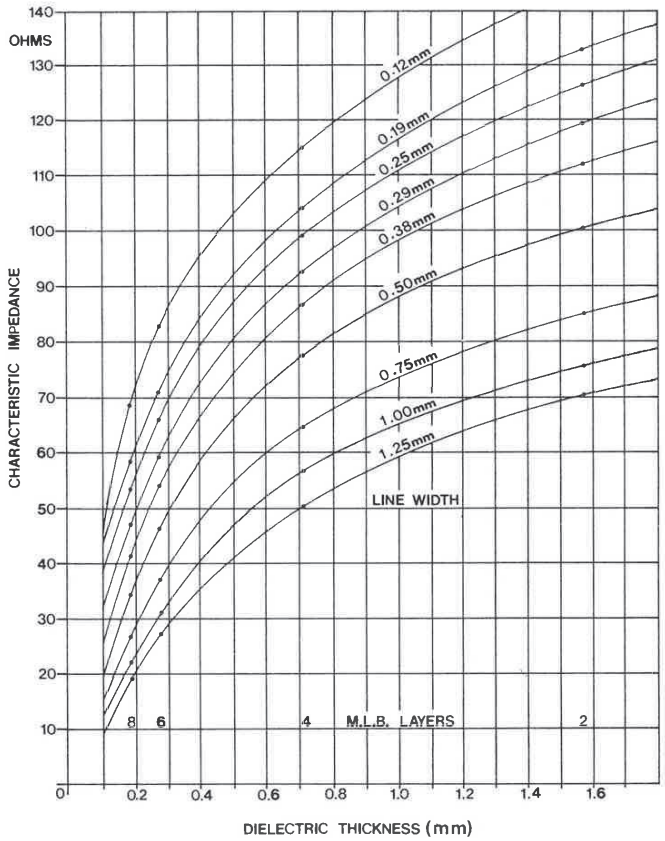


Fig. 2.7 Characteristic impedance of tracks on PCBs.

line is an open circuit, there can be no continuous current flow, so the line current which was established by the charging of the line capacitance must be cancelled by an equal and opposite current, which means the voltage on the open end of the line must double when the initial change reaches the end.

If the line is connected to a resistor to earth whose value is equal to the line impedance, the current flows into the resistor and there will be no reflection.

If the end of the line is a short circuit to earth, then there can be no voltage at

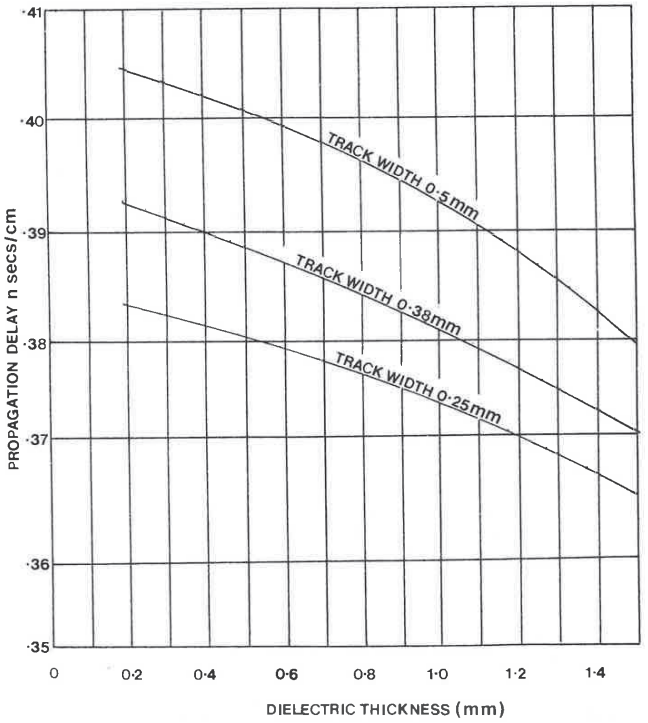


Fig. 2.8 Propagation delay of tracks on PCBs.

the end, so a current must flow to discharge the line capacitance. Any such reflected currents propagate back down the line at the speed shown in the graph in Figure 2.8, until they arrive at the source which caused the initial voltage change. If the impedance of this source matches the line impedance, there will be no further reflections, but if it does not match then further reflections will occur until ultimately the line will settle at a voltage determined by the values of source and load impedances.

Reflection levels are easy to calculate when each end of the line has a resistive termination but calculations become more complex when the terminations are semiconductor devices with non-linear input and output impedances. Then it is easier to use a Jones diagram for graphical solution. For graphical solutions of line reflections on digital circuits, the input and the high and low state output characteristics of the devices are drawn superimposed on voltage and current axes

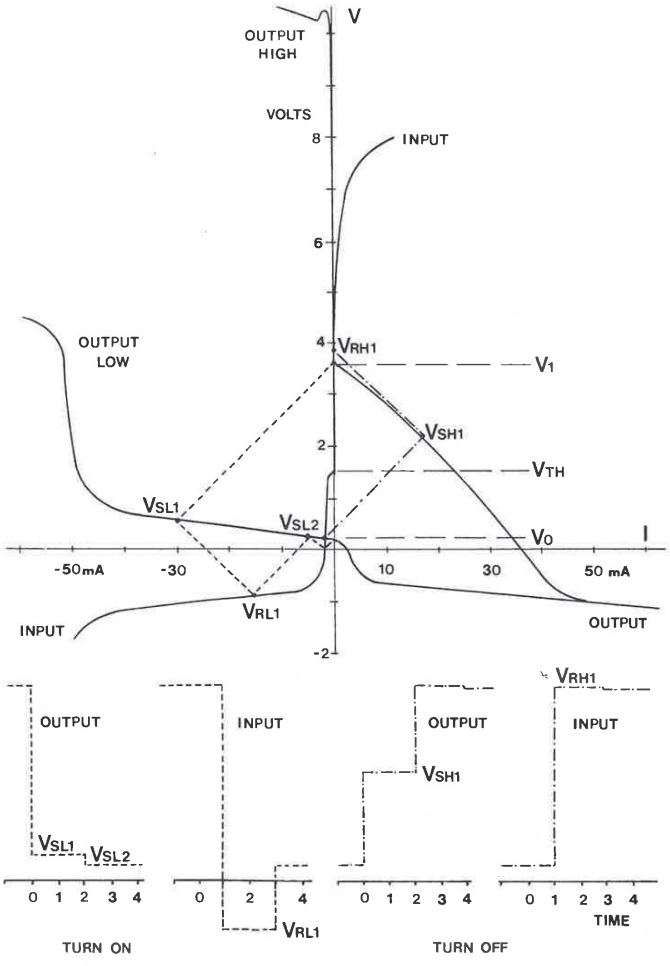


Fig. 2.9 Determination of line reflections on a typical track interconnecting TTL gates.

(see Figure 2.9). If any resistance on the track is ignored, the two steady states of the system will be where the input curve crosses the low and high output curves at V_0 and V_1. The driven device will change its state at V_{TH}. To find what reflections will occur, lines are drawn at slopes which represent the line impedance.

A fairly typical line impedance on a board is 100 ohms, so drawing is eased if the scales are chosen with 1 V equal to 10 mA. Then the track impedance lines for 100 ohms lie at 45 degrees to the axes. During a turn-on transition, the output of the driving device tries to change from V_1 to V_0, but, because it has to drive the line impedance, it actually changes from V_1 to V_{SL1} (the point at which the 100 ohm line crosses the low output curve). After one line delay, the receiving end of the line will move towards this voltage, and its actual level is found by drawing the reversed slope 100 ohm line from V_{SL1} to V_{RL1}. The line mismatch gives rise to a further reflection, which results in a new voltage level V_{SL2} at the driving end of the line after two line delays. The process is repeated, with each end of the line resting for two line delays at its successive level, until the 100 ohm slope lines are close enough to V_0 for any further reflections to be ignored.

Similarly, on turn-off, the 100 ohm line from V_0 gives the first output level as V_{SH1}, then the reverse slope line gives the first receiving end level at V_{RH1}. The four lower diagrams of Figure 2.9 show the voltage levels at sending and receiving ends for the on and off transitions, with a time scale in line delay periods. These sketches show what would be seen on a very long line with no discontinuities.

At first sight graphical solutions by Jones diagrams can seem very complicated, but provided the scales are well chosen and care is taken to ensure that the input and output characteristics are drawn with the current axes in the same sense, they are very easy to use and will yield adequate accuracy for any practical designs. On Figure 2.9, currents to the right of the vertical axis are from sending end to receiving; to the left of the axis the current flow is in to the driving device. With this convention, all sending end levels occur where a positive slope resistance line crosses an output curve, and receiving end levels are where a reverse slope line crosses the input curve.

The TTL characteristics used in Figure 2.9 show that the input is very high impedance relative to the line, whilst the low state output is very low impedance and the reflections during turn-on are limited by the diodes to the substrate formed by the planar diffusion of the integrated circuit. Only the high state output gives a reasonable match to the line impedance likely to be found on a printed circuit board. Figure 2.10 shows the effect of using a line impedance significantly higher than that of the high state output. During turn-off the first sending end level is higher, and the first receiving end level is well above the normal high level. The expanded sketch on the right shows how the input and output impedance curves are not actually superimposed, but lie very close together so the overshoot voltage on the line decays very slowly to the normal level by multiple reflections. This decay of an overshoot can also be visualised as the discharge of the line capacitance through the leakage of the cut-off transistors at each end of the line. The turn-on lines on Figure 2.10 have been drawn starting from the overshoot level of the first reflection instead of from the normal high level to demonstrate that any such overshoot has virtually no effect on turn-on.

Figure 2.11 shows the effect of the use of a low line impedance, and it can be seen that this introduces delays at the switching threshold level and the turn-off edge changes from a sharp edge to a long staircase, which on practical boards appears as a rounding of the pulse edge.

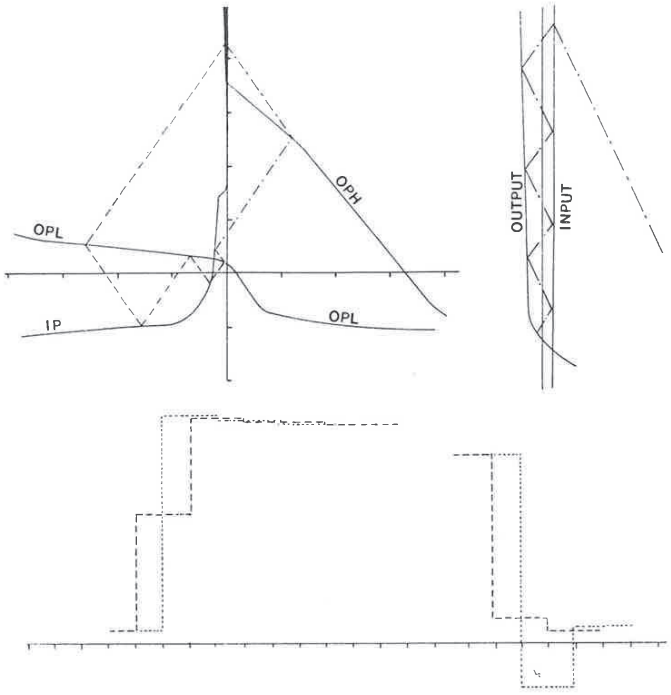


Fig. 2.10 Reflections on a high impedance line.

Figures 2.10 and 2.11 demonstrate clearly that, if the line impedance can not be matched to the high state output impedance of saturated digital logic devices, then it is very much better to use a high impedance line. For satisfactory electrical performance, the use of lines of low impedance should be avoided whenever possible.

While TTL characteristics have been used for these examples, similar principles can be applied with other types of device to determine the optimum line impedance to be used. ECL devices usually require controlled line impedance of 50 ohms; fast versions of DTL and RTL are similar to TTL, and MOS devices are of very high impedance and are sensitive to any track capacitance rather than to line impedance.

Any change of track width or discontinuity such as a via hole in the track will cause some reflections, but in most practical cases on digital logic boards there are so many such discontinuities that any calculations become impossibly

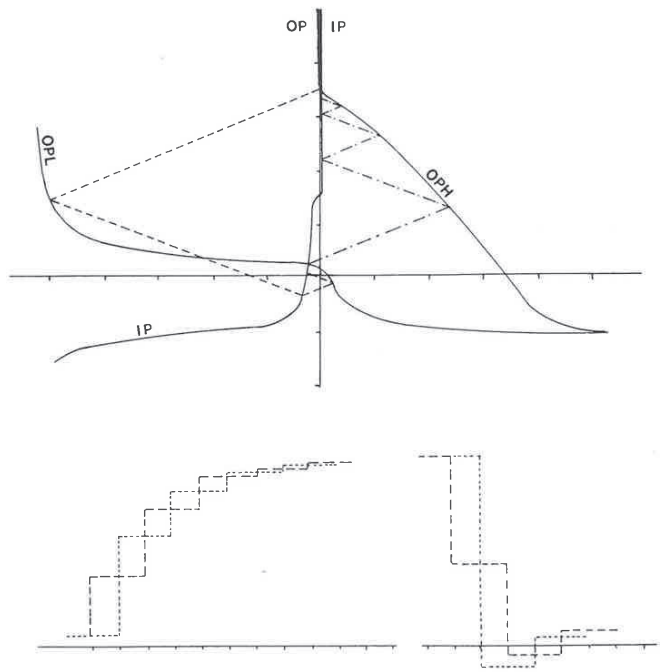


Fig. 2.11 Reflections on a low impedance line.

complex and the overall effect appears as a slight lowering of the line impedance. On very high speed digital circuits the effects of these discontinuities in tracks can be detected, but if reasonable care is taken in the design of the board they will be at a level which can be ignored.

Any branches in a track will also form discontinuities. If the branch is short, it and the device it feeds can be regarded as a local capacitive load on the line, but a long branch will appear in parallel with the main track, halving the effective characteristic impedance beyond the junction. Similarly, it has to be borne in mind that if the device which drives a track is placed in the middle of the run of track, its output will 'see' two equal line impedances in parallel. Some appreciation of such transmission line properties is essential if package placement and track routing are to be carried out properly to produce board designs which will have good electrical performance. It is worth noting that transmission line effects are ignored in most CAD programs for package placement.

It should be noted that any reference to line length is to the effective electrical length of the line, not to its physical length. A line is said to be of critical length when the delay along the line is half the rise time of the signal it carries, and a short track is any track less than one tenth of the critical length. When a change in voltage is applied to a line, the voltage at the driving end changes to a value determined by the ratio of the source and line impedances, then the voltage at the receiving end changes after one line delay period. Any reflection back to the driving end will arrive there after two line delays, and so on, so that on a line of critical length each voltage level of the reflections will be fully established, but will immediately start to change again. On short lines, the full levels will not be reached, whilst on lines of over critical length there will be a pause at each reflection level.

2.6.2 Surge Currents[8,11]

Section 2.6.1 shows how, when a digital logic device changes its state, a current significantly higher than normal will flow in the track connecting the device to those it drives.

The duration of these surge currents depends on the line length, and if the tracks on the board approach critical length it will usually be necessary to provide decoupling capacitors adjacent to the devices which drive the line to provide the instantaneous current requirements.

The surge current in a line must be accompanied by an equal and opposite current in the earth return path, and this return current can cause malfunctioning of the circuit. Devices which are not concerned with the signal being considered may be connected to the earth line along the portion carrying the return surge current, and any such devices will experience a disturbance to their earth level during the surge. This disturbance will affect the earth to input signal levels of the device, and may be sufficient to cause spurious switching.

Any such spurious switching may mistakenly be thought to be caused by cross-talk, since an oscilloscope with its probe connected normally between the earth and signal pins of the affected device will show the disturbance on the signal, which is seen relative to the moved earth level. Unfortunately, all the steps which might be taken to cure cross-talk, other than reducing line length, will tend to increase the surge current, if they have any effect at all. The only sure cures for such surge currents are a complete re-package of the board, a reduction in track lengths, or a raising of the characteristic impedance either by increasing the dielectric thickness or by using narrower tracks. Increasing the width of the earth return track will make a slight improvement, but not *pro rata* to the increase in width since this is an AC effect and it is the characteristic impedance of the earth line which governs the amplitude of the disturbance, not its DC resistance.

Attempts to cure surge current problems on double-sided boards by the use of a multilayer board with internal power and earth planes may not succeed because the increase in line surge current caused by the lowered line impedance and the increased propagation delay is unlikely to be balanced by the improvement in the earth return path. It must be remembered that an earth plane in a digital logic PCB is not a continuous sheet of copper, but is interrupted by package pin and via holes with often only very narrow tracks between the holes. Its effective characteristic impedance to a high speed signal may be well over half that of a surface earth return track. Even if a change from a double-sided board to a multilayer does result in a reduction in the earth return surge voltage, it is unlikely to justify the increased board costs.

Surge current effects should not need to be cured; they should be designed out by the use of short, high impedance lines and a package placement which minimises the risk of earth return interference effects. The use of the highest packing density practicable with short tracks will help to minimise surge current effects, but it may increase the risk of cross-talk.

2.6.3 Cross-talk[1,8,10,11]

When two tracks run parallel on a PCB, a signal on one, the signal line, can be fed into the other, the pick-up line, by capacitive and electro-magnetic coupling. A backward cross-talk signal will be detected at the 'sending' end of the pick-up line and forward cross-talk will be detected at the 'receiving' end. Note that 'sending' and 'receiving' end refer to the ends of the signal line (see Figure 2.12).

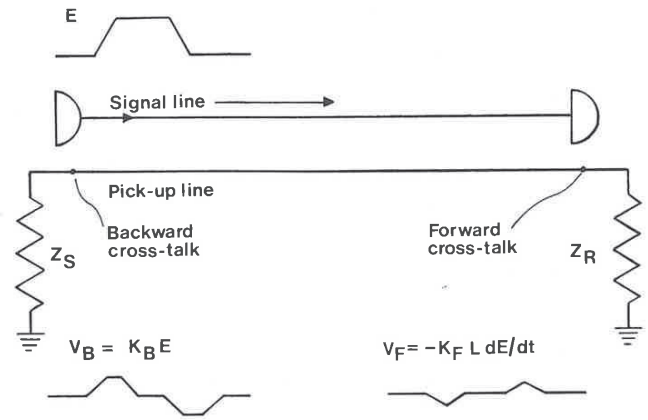


Fig. 2.12 Cross-talk definitions and waveforms.

The maximum amplitude of the back cross-talk is given by the amplitude of the signal causing the cross-talk multiplied by the cross-talk coefficient from Figure 2.13, whilst forward cross-talk is the product of coupled line length, the slope of the driving pulse, and the cross-talk coefficient from Figure 2.14.

Forward cross-talk on practical boards is usually small enough to be ignored, but it should be noted that it does not limit at the critical length.

Values of cross-talk calculated as in Figure 2.12 apply when the pick-up line is terminated with resistors whose value matches the line impedance. If there is a mismatch reflections will occur and, since the input to most digital logic devices can be regarded as an open circuit compared to a typical track impedance, the value of the cross-talk signal will be doubled at a device input.

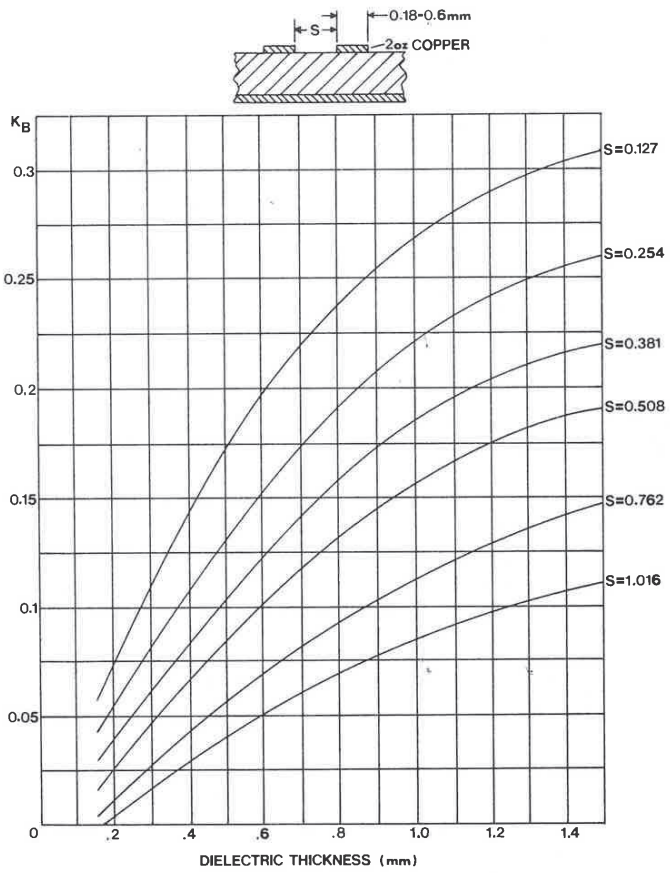


Fig. 2.13 Back cross-talk coefficients.

Back cross-talk occurs as the edge of the driving pulse propagates along the line. When the pulse reaches the end of the line a reflection occurs and the cross-talk signal is cancelled. Thus on lines of over critical length the length of the cross- talk pulse is determined by the line length but on lines of less than critical length the pulse will not rise to its full amplitude before it is cancelled by the

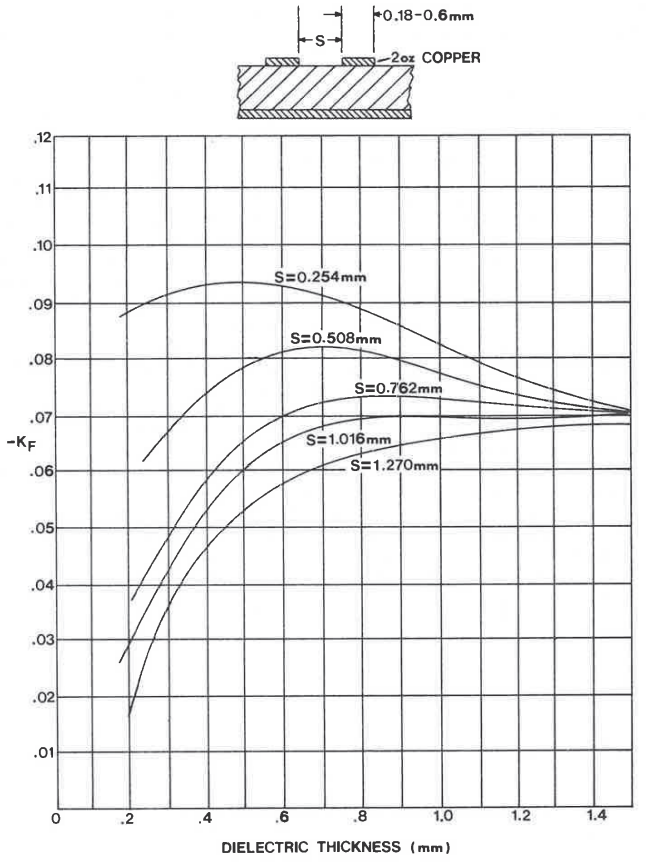


Fig. 2.14 Forward cross-talk coefficients.

reflection. From Figure 2.13 it can be seen that back cross-talk can be decreased by increasing the separation between tracks or by decreasing the dielectric thickness as well as by keeping tracks shorter than the critical length. The simplest control over back cross-talk is to keep the normal track separation at a value which will keep the amplitude of any back cross-talk below the switching

threshold of the logic family being used. For TTL devices on double-sided boards this figure is about 0·8 mm separation. Clearly tracks can be run closer than this for short lengths when necessary, for instance to pass between pairs of pads.

Calculations of cross-talk between a single pair of tracks are complicated by the presence of line reflections on both the signal and pick-up lines. On practical boards they are further complicated by the presence of other tracks, some of which may carry signals which switch at the same time as the signal line. This introduces a further factor, the configuration constant, into any calculations.

The value of the configuration constant can vary between 0·5 when an earth track is run beside the pair of tracks being considered, up to 3 when the pick-up line is surrounded by six or more signal lines carrying signals which switch simultaneously. The cross-talk signal will vary depending on whether adjacent lines are held in the low or the high state, and from which end they are driven. Meaningful calculations are further complicated by the fact that at some stage what was first considered as the pick-up line will carry a signal which will result in cross-talk on what was considered as the signal line.

In any practical system there will be many tracks on which cross-talk is of no significance for most of the time. Any lines which feed strobed or clocked devices will be sensitive only around the time of the clock or strobe signal and if good board designs are to be produced without unnecessary restrictions on track lengths or separations it is essential for the board designer to understand how the circuit is intended to work.

One point which is often not appreciated is that tracks which are normally not connected, such as monitor lines from the middle of a circuit to a connector on the edge of a board, will carry reflecting pulse edges and these transient pulses on 'unused' tracks can cause cross-talk on adjacent 'used' tracks.

Calculations of the line reflections and cross-talk, and values of configuration constants are discussed in more detail than is possible here in the book 'Transistor-Transistor Logic and Its Interconnections.'[11].

Chapter 3

DESIGN OF BOARDS

The design of printed circuit boards is often thought of as comprising package placement and track routing, and there are contract design services and CAD facilities which offer only these functions as a 'complete design service'. In fact the design of a cost effective printed circuit board extends far beyond the actual layout of the board and should be considered in depth as early as possible in the overall design of any new electronics project.

The boards should be considered as the major electrical and mechanical components which they are, and their type, size, feature sizes, and interconnection method should all be considered in relation to the ultimate end product. It is not uncommon to find that board size has been determined either by using the same board as was used on a previous generation of equipments, or by the arbitrary choice of an available standard size. Such decisions are likely to result in equipments which are unnecessarily expensive to design and build, difficult to test and service, and which may well fail to perform as was expected. If electronic systems are to be truly cost effective the interconnect and component mounting must be properly considered in relation to the whole equipment, which means that it must be considered at the earliest possible stage in the overall design cycle. Failure to give adequate early consideration to the interconnect can result in systems designers having to re-think large areas of an equipment so as to be able to contain specific functions within an inadequate board area, and it can also result in the layout of boards being unnecessarily difficult, possibly to the level of having to use expensive multilayer boards where more thought earlier in the design stage could have allowed the use of a cheaper board style.

It is not uncommon to find that plugs and sockets or edge connectors have been selected for an equipment on the basis of reliability of the gold plating or cost, without any consideration of the effect of pin pitch or connector length on the board layout. Similarly, feature sizes such as track widths and spacings might be selected without adequate consideration of their electrical properties, and even sometimes without full thought of their effect on the manufacturing costs. In some larger organisations some of the decisions which affect the design and the cost effectiveness of the boards used may be taken by committees which are not aware of either the type of system on which the boards are to be used or the limitations of board layout and manufacture.

3.1 STANDARDS

Most large companies like to lay down company standards wherever possible, and in many cases the use of a standard board size, type and style can offer benefits which more than outweigh the possible disadvantages of increased difficulties in designing the electrical system to fit the standard boards and in

laying out the boards. However, it should never be assumed that this will be the case, and the overall board design should be given its proper consideration at the start of every new project, and existing standards should be used only if it can be proved that they will be truly cost effective. It must always be borne in mind that standards are usually drawn up after a first equipment of a new type has been designed, and their use on future equipments inevitably means that the standards are unlikely to reflect fully the latest technological developments.

It is not unknown for design work on a large system to be started with the intention of using standards drawn up for a previous generation of equipment, and then, when a substantial amount of design work has been completed, for it to be found that the standards are not sufficiently flexible to accommodate some newer aspect of technology. As an example, in the 1960s several standard board layouts which would not permit the use of DIPs with more than 16 pins were produced, so designers either had to ignore the possibilities of using microprocessor or other large packs, or had to design non-standard boards. Once non-standard boards are introduced into what had been conceived as a totally standard system many of the benefits of the attempts to standardise are usually lost, leaving only the disadvantages of less than optimum electronic design on those parts which were made to fit the standard. It has been said with some degree of truth that excessive standardisation is the enemy of good design.

It must be recognised that there should be some standards laid down for design work, but all such standards ought to be reviewed regularly and should allow the board designer adequate choices so that cost-effective designs can be produced to suit the particular requirements of each type of equipment.

3.2 CHOICE OF TYPE

The choice of type of board is basically simple—one uses the board type which will perform the required function at the lowest cost. What is not quite so simple is establishing the true costs of using a given type of board and determining precisely its functions. The cost of buying or making the board may well be one of the smallest items in their overall cost to the user. The influence of board type on the costs of assembly, testing, and field servicing must also be considered. The required function may include an ability to package the electronics in a given volume. If there are no such limitations on space, then the influence of board type on the equipment size must be considered as part of the costs, and savings made by the use of larger, cheaper, low density boards must be balanced against the costs of a larger frame or case than would be required if more expensive, higher density boards were used. The function must include an ability to interconnect the components carried with minimal degradation of the electrical signals, and it should also include reference to reliability, life expectation, ease of servicing and possibly some purely mechanical aspects. When higher density boards are being considered the means of connection to them should be taken into account both from the cost aspect and the electrical and mechanical performance.

The choice of type may also be influenced by the manufacturing capacity available. As the area of choice moves towards the more sophisticated multilayer types it may be found that there is only a single source available for the boards, or that delivery times will be unacceptably long. It may also be difficult to find adequately skilled design capacity for a large number of sophisticated board types

for an equipment required in a limited time scale. Designers whose experience has covered only simple board types may not produce cost effective designs for sequential multilayers.

The first point to be settled is whether the boards will be plug-in types, carried in a frame as in Figure 1.8, or will form the major chassis of a unit as in Figure 1.3. From this consideration, the mechanical requirements for the boards can be determined, and also the means of making connections to them. On plug-in boards the question of whether to use edge connectors or plugs and sockets must be resolved. Edge connectors may appear cheaper to buy, but will mean special plating, usually gold, on the fingers on the board, which will increase the price of the boards. The costs of assembling multi-pin plugs on the boards must be considered, and so must the possibility of repair work. When connectors are being considered, the size of the boards should also be borne in mind.

3.2.1 Non Plated Boards

When all these inter-related points have been settled, the actual board type can be considered. On small, cheap, single board units which use few components with large numbers of pins, single or double-sided boards made on paper phenolic laminate may be adequate. Requirements for reliability, mechanical strength, or electrical performance may dictate the use of a more expensive laminate, or the use of plated-through holes. Any requirements for flexible circuits or rigidised flexibles will usually be self-evident.

3.2.2 Plated-through Hole Boards

As the required interconnection density increases, the required board type will move up to a plated-through hole double-sided board, with via holes, on epoxy glass laminate.

Such double-sided boards can cover the widest field of applications of all types, and usually offer the best possible combination of electrical performance, cost, and packing density. Any move from double-sided to the significantly more expensive multilayer types requires very careful consideration. It is not always appreciated that any circuit which can be designed can be implemented on a double-sided board laid out with the tracking on X and Y co-ordinates, with a via hole and a change of side where tracks must change direction. The limitations on the universal use of double-sided boards lie in the overall size allowable, the packing density achievable, the track and gap widths to be used, and the sizes of via holes and their pads. Packing densities readily achievable on double-sided boards for standard 16 pin DIPs run from about 7 sq cm to about 3·5 sq cm per pack.

3.2.3 Multilayer Boards

Some analogue circuits require earth screening, and some types of digital integrated circuits require the use of tracks with controlled line impedances. Both of these requirements will usually dictate the use of multilayer boards with power and earth planes, and often a four layer board will be found to be the most cost effective type. Limitations on overall space and on the minimum track and gap widths usable can mean that 6 or 8 layer multilayer boards have to be used on

digital systems. Unless the electrical requirements dictate the use of screening or controlled impedance lines, four layer boards are unlikely to prove cost effective on saturating logic or MOS digital systems. It is sometimes assumed that the use of a pair of internal planes for power and earth, with all signal tracks on the surface layers, will allow a significant increase in packing density. This is fallacious, as is demonstrated in Section 3.6, and the gain in packing density will not be sufficient to compensate for the significant decrease in electrical performance and the increase in cost of the boards.

It is generally well understood that the best electrical performance will be achieved from any system when the interconnection lengths are minimised. What is less widely appreciated is that the lengths referred to are the effective electrical lengths, not the physical lengths, and, while a change from double-sided to multilayer boards can reduce the physical lengths of interconnections, the reduction in line impedance associated with the use of thinner dielectric layers (see Figure 2.7) usually has the effect of a significant increase in the effective electrical length of the interconnections.

3.2.4 Surface Mounting

When surface mounted packages are to be used at high packing densities, the choice will usually be made from multilayer, double multilayer, or pillar plated. Since the high packing density will probably cause cooling problems, the choice may be biased towards the use of a pillar plated type on a metal core.

3.2.5 Flexi-rigids

The choice of a rigid board type is usually a matter of achieving the right balance of packing density, electrical performance, and cost, and the final decision is almost invariably a compromise with one of the three main considerations being sacrificed to some extent. A decision on whether to use a rigid board with external flexible wiring or to integrate the unit into a multilayer flexi-rigid is not so simple. The flexi-rigid can offer lower assembly costs since it eliminates joints between the rigid board and the flexible wiring. It offers better packing densities because there is no need to leave space for connectors; it can increase the reliability of a system because it eliminates many soldered joints or plug-in connectors, and it offers opportunities to fold or bend boards to fit into awkward spaces. Against all these advantages there are the higher initial cost and possible difficulties in design and supply.

3.2.6 Conclusion on Board Type

Without knowledge of all aspects of an equipment's purpose and market, it is impossible to say there is any 'right' type of board. All that can be said in general terms is that for most digital systems the balance of probabilities favours the use of double-sided, plated-through hole boards, and that multilayers may not prove to be as cost effective as is sometimes thought.

One reason for a choice of board type which should be firmly disregarded is the age of the technology. A flexi-rigid board has a vastly greater interconnect capability than the 40 years old non plated, paper phenolic based board, but, if the older board type will meet all requirements at lower cost, then it is the best type to use.

3.3 DETERMINATION OF SIZE

It may seem that determining the size of board to be used for a new equipment is a very simple matter, and a choice can be made from a few available standard sizes, such as the Eurocard or double Eurocard size. Such a decision may prove to be the right one, as standard framing and cases can be used, but it can prove to be quite wrong, and should be taken only after all related aspects have been given full consideration.[1,2,11,12]

In some types of equipment, especially airborne, the choice of board size is limited by the case size to be used for the system. Extensions to an existing range of equipments will generally use the same board size and type as the original unit, but this should not be assumed to be an essential requirement.

3.3.1 Small Systems

Many smaller equipments can be designed with all their components on one or possibly two boards. In such equipments, the board size will usually be dictated by the case size available, which in turn may be governed by the need to incorporate a keyboard or a cathode ray tube display. In such cases the size of the board should be limited by its ability to carry and interconnect the components, and by the ability of manufacturers to produce large boards.

3.3.2 Large Systems

On larger systems which will contain a significant number of boards the choice of size is not quite so simple. For a given type of board and packing density, the whole system will require a defined usable board area. If this area is split down into a large number of small boards, the actual area required will be significantly greater as each board will have some unusable area at the edges which fit into board guides, at the back for the connector, and probably at the front for a handle. The widths required for all these will be substantially the same whatever the board size, so a significantly higher percentage of the actual area of small boards will be lost. On the other hand, if the boards are made very large, the diameters of pads may have to be increased to allow for material movement so the tracking density may have to decrease, but there may be a requirement for an increased track density. Also as boards are made larger there may be increased wastage in cutting the blanks from the sheets of laminates.

3.3.3 Cost

The cost per board will obviously rise as board size is increased, with an increasingly evident step function as boards are designed at sizes which can be made ten per sheet of laminate, then eight or six and so on. As the cost of each board rises with size, so the number of boards required falls, and the cost of all the boards needed for a system rises sharply if the boards are made too small. The cost also rises if the boards are made too large, and on large systems the optimum

board size is usually between 30 and 50 cm on the longer edge, which is usually the connector edge, with an aspect ratio of around 4 to 3.

3.3.4 Manufacturing Limits

As well as the possibility of overall board costs rising if boards are made too large, the limitations of all the manufacturing processes must be considered. Film costs can become appreciable on multilayer boards, and drilling costs may rise if large boards necessitate the use of only alternate heads on multi-spindle machines. Throat depth on drills, printing frames, plating baths or fixtures, conveyor widths on etching or reflowing machines, or the size of a bare board electrical tester can all impose an upper limit on board size. In the assembly shop, an automatic component insertion machine or wave soldering machine may limit one or both dimensions of the board. The risks of bowing and twisting and of vibration effects on very large boards must also be borne in mind.

3.3.5 Electrical Function Splits

Once a likely range of physical sizes has been determined from financial and mechanical aspects, the electrical content of the boards can be considered. Clearly the boards should be large enough to contain complete working sub-sections of the whole equipment. In the earlier stages of the design of any new equipment, it is advisable to add a safety factor to any figures suggested for the number of integrated circuit packages per sub-function. In areas where the design work is not fully integrated, it can be useful to understate both the number of packages per board and the number of connector pins available for signals. The systems designers will discover any such safety allowances sooner or later, and will then make use of the full capability, but any such allowances which can be made in the early stages of a project usually prove justified when the prototypes are tested and modifications and corrections to the original design have to be made. Sometimes such modifications result in a decrease in the board content, but not often.

There can be a tendency when designing large digital systems to split the system into board types based on functions, with as many bits or bytes of the word length as possible on each board. Such a split of a system has many advantages. It can simplify fault detection; it makes it possible to replace whole functions as newer systems are developed, and it can often enable some types of boards to be used in more than one area of a large system.

However, there is a possible alternative, which is to split along the data flow path, so that each board will contain fewer bits or bytes but will carry several different working functions. This data flow type of split can result in a significant decrease in the number of interconnections required and it will also result in the use of a number of identical boards to make up the total required word length. Such use of identical boards can simplify fault finding and it will also decrease the costs of holding a full set of spares for the machine.

3.3.6 Interconnection

The larger the boards can be made, the fewer will be the board to board connections required, so the total cost of connectors will be decreased and the overall reliability of the system will increase. The number of connections to an individual board may have to be increased as the boards are made larger, but experience over the history of integrated circuits has shown that this increase is far from linear with board content and in fact as it becomes possible to package complete functions or sub-functions on boards the number of ways required per board can actually fall. A very rough check on the boards made by Exacta Circuits Ltd in 1981 and 1971 showed that the average 1981 board was two to three times the size of the 1971 boards and, with LSI packages being used extensively, the logic content is undoubtedly very much greater, but the number of edge connector fingers per board had remained roughly constant over the ten years.

3.3.7 Testing

Testing assembled boards must be considered when board size is being selected. It has to be remembered that there are four quite different types of testing to be undertaken. First, prototypes will have to be tested to establish that they work as planned. This testing will have to be carried out over the whole system, and the smaller the boards are made the greater will be the probability of a faulty board to board connection causing problems. Second is the 'Go—No go' testing of production boards. When this is performed on a large, computerised tester, the test cost per system will decrease as the board size is increased, but the complexity and cost of the tester will increase with board size. Thirdly, fault finding during service must be considered. On most large systems this fault finding will probably be limited to locating and replacing a faulty board. Clearly the fewer boards there are in a system the easier it will be to find which has failed, and the greater will be the possibility of including diagnostic logic so that the machine can indicate which board has failed.

Fourthly, there is the detailed testing to locate the actual fault on boards rejected by second or third level testing. The testers used at these levels may give an indication of the nature of the fault, but whether they do or not, the cost of locating most types of fault is unlikely to vary significantly with board size unless the boards are very small. On larger boards the fault and its effects are likely to be contained in one area of the board. If board size influences this type of testing significantly, it is possible that the greater logic content of a larger board could simplify the fault detection. If automatic or semi-automatic testers are used then the cost of such testers can be expected to increase as the board size rises.

Overall, the impact of board size on test costs is that the use of bigger boards decreases the cost of testing, but increases the cost of automatic test equipment.

3.3.8 Repairs and Costs

If large boards are to be used, the assembly facility must include adequate provisions for the replacement of faulty devices, and the boards should be of a type which can be repaired without undue difficulty. If assembled boards can not be easily repaired, then the failure of one device can mean the scrapping of the whole assembly. Under some conditions of service, the cost of a 'throw away item' can be a major factor in determining the size of board to be used.

As larger boards are used, the probability of an assembled board carrying a faulty component and having to be held back for repair will increase. It may be

suggested that this can cause cash flow problems as the value of work held back for repairs will be increased. However, this is an argument of doubtful validity when the boards form part of a large system, as the system can be sold only when all the boards are correct, and if smaller boards are used there will be more good boards held in storage awaiting the completion of repairs to the faulty boards, and overall the value of the goods held is likely to be greater as the smaller boards will involve more connectors.

3.3.9 Design Difficulties

It may be suggested that problems in designing large boards can place a limit on size. This is not the case. If a given system or sub-system can be designed on a number of small boards, then clearly it will be no more difficult to design the same system on one board. The design of the one large board will clearly take longer than that of any of the smaller boards, but the actual work will be no more difficult. Once the designer appreciates that very large circuits usually break down into a number of discrete areas it will be found that any design difficulty barriers to the use of large boards are imaginary. Since the larger board must carry the board to board connections as well as the actual content of the smaller boards, the packing density per unit square may have to be relaxed slightly, but, because connectors and back wiring are eliminated, the overall packing density will be improved.

It should be noted that there are some types of bus-oriented systems in which the interconnections are basically three dimensional, with a number of data, address and control signals which are used in many areas of the system. These systems are not so amenable to design on a single large board, but fit better onto smaller boards plugging into the bussed back wiring.

3.4 DETERMINATION OF FEATURE SIZE

The three main features on a board which must be determined are the pad diameters, track widths, and minimum gaps allowable.[1,2,8]

3.4.1 Hole and Pad Size

The pad sizes used will depend on the hole diameters and on the type and size of the board. Holes must be large enough to allow the component leads to be inserted without undue difficulty, but on plated-through boards they should not be too large if optimum quality soldered joints are to be achieved. An allowance of 0·15 mm to 0·3 mm over the component lead diameter will give the best soldered joints in plated-through holes. The shouldered rectangular leads of DIPs should be mounted in holes large enough to accept the largest possible lead allowable, but not so large as to allow the shoulder to enter the hole. This requires a minimum diameter of 0·7 mm, and most board users specify holes of 0·8 mm diameter after plating, usually with a tolerance of plus or minus 0·05 mm.

Some automatic component insertion machines require the use of holes of significantly larger diameter. When the use of such machines is being considered, the board designer, the production manager, and the quality manager must weigh up whether the savings in assembly costs justify the possible losses in track space

on the boards, the lower quality soldered joints, and the risks of the shoulders of DIP leads entering the holes.

Via holes which do not carry any component leads can be any diameter, subject to limitations of drilling and through-plating, for which a reasonable minimum limit on boards 1·6 mm thick is 0·4 mm diameter. However, unless tracking requirements force the use of minimum diameter via holes, it is better to keep via holes the same size as the smallest component holes, which are usually those for DIPs. The use of a number of different hole sizes on a board increases the manufacturing costs, and the use of minimum sized holes can add significantly to the cost.

A requirement to minimise the number of hole sizes on a board is not strictly compatible with achieving the best possible soldered joints in all holes, and a reasonable compromise is to call for holes in steps of 0·2 mm diameter up to 1·0 mm, then any larger holes as necessary.

On non plated boards, the pads must be large enough to allow adequate adhesion to the laminate, and a figure of a minimum remaining land of 0·25 mm is often used. This means the pads must be 0·5 mm larger in diameter than the holes, plus all manufacturing tolerances. With allowances for tolerances on hole and pad diameters and hole and pad positions, the normally used pad diameter for DIPs in 0·8 mm holes is 1·9 or 2·0 mm.

When the holes are to be plated-through, there is no need to leave a large area of pad either for adhesion or to carry a soldered joint. The pad need only be large enough to contain the drilled hole prior to plating, with allowances for all tolerances. On multilayer boards there will have to be some allowance for material movement, and film and material movements can make the use of small pads undesirable on very large boards. Normally used sizes with 0·8 mm diameter finished size holes are 1·27 mm on small or medium sized double-sided boards; 1·4 mm on larger boards; or 1·52 mm on multilayer boards. If the design of the boards can be restricted to single tracks between pairs of pads on 2·54 mm pitch, then the 1·52 mm pads are best on double-sided boards.

3.4.2 Track Widths and Gaps

The three requirements on track widths are that they should give an acceptable electrical performance, should be easily manufacturable, and should be capable of yielding the required tracking density. Fortunately, on double-sided boards these requirements are compatible. Saturated digital logic circuits usually require a line impedance of over 100 ohms, and tracks of 0·1 mm up to 0·3 mm width can be made, and usually yield adequate packing densities. Clearly within this range of widths the boards will be cheaper and easier to make if the widest track (0·3 mm) is used.

The space between long runs of tracks will usually be dictated by cross-talk requirements but locally it is manufacturing considerations which impose a lower limit. The values for track widths can be used, 0·1 to 0·3 mm, bearing in mind that the smaller gaps will make the boards more difficult to manufacture. When track and gap widths are being considered it must be borne in mind that the difficulty in manufacturing a board, and hence its cost, is a function not only of the minimum feature size used but of the length over which this minimum feature is applied. A gap of 0·1 mm between the side of a track and an isolated pad is not too difficult to achieve, but gaps of 0·1 mm between long runs of tracks can

make a board much more difficult to process. Also, such gaps would probably result in intolerable cross-talk.

3.4.3 Tracks Between Package Pins

The usual situation where track widths and gaps become critical is when signal lines have to pass between the pins of a DIP on 2·54 mm pitch. With pads 1·52 mm diameter, a single track of 0·3 mm can be accommodated, or two at 0·2 mm width. With two such tracks, 1·4 mm pads allow a little more tolerance, and 1·27 mm pads allow the use of 0·25 mm tracks and gaps. The use of 1·27 mm pads can allow three tracks of 0·15 mm width, as shown in Figure 1.7. It can be noted that the board shown in Figure 1.7 used minimum track and gap widths only where three tracks pass between pairs of pads. When deciding track and gap widths, it must be remembered that there will be tolerances on the positions and sizes of pads and tracks, and the sum of pad diameter, all track widths and all gaps should total less than the 2·54 mm nominal pitch.

3.5 PACKAGE PLACEMENT

With board type, shape and size, feature sizes and type of connector determined, layout can start with placement of components. Areas round the edges of most boards will have to be left clear for fastenings, handles, or to fit into board guides and on large boards there may be areas within the board which must be left free of components to take spacers or stiffening webs. There may be restrictions on the placing of two-lead components such as chip capacitors near the edges of boards to avoid risks of their being bent over when assembled boards are handled. The use of automatic component insertion machines may impose limits on the orientation and spacing of components.

Component placement on high frequency analogue circuits usually follows the circuit diagram quite closely, with each component being placed as close as possible to those with which it must be connected. The placement of components on high gain amplifier circuits will usually be carried out along with the routing of most of the tracks to ensure that there can be no risk of any track from an output having to pass near to any input circuit and to ensure that all sensitive components are connected by minimal length tracks.

Digital boards with large numbers of packages need a different treatment for package placement. The standard sizes and shapes of DIPs or chip carriers make it possible to place the packages in regular rows and columns, with spaces between for decoupling capacitors, pull-up resistors and any other discrete components needed for timing circuits. Some portions of digital circuits, such as clock generators, may use significant numbers of discrete components. Such sub-circuits can usually be treated as small, self-contained analogue areas which occupy the space of two or three packages in the regular array.

The spacing of rows and columns of packages should be determined by considering the type of circuit, the track width and gaps, and the likely need to run numbers of tracks across and along the board. It is not necessary to maintain constant spacings between rows or columns. On some circuits, a broader space can be left after every third (or fourth) row to accommodate longer tracks along the board, or possibly to accommodate an address or data bus group of tracks.

When DIPs are used, it is important to mount them the right way round. If the board is to be wave soldered (see Section 12.3.4), it is best to make the board of such a size and shape that it can be soldered with the gold contacts of the board connector along one side, so they project beyond the end of the wave. If boards are soldered with the connector on an end, the connector contacts will have to be masked to ensure that they are not touched by the solder.

When the soldering of a DIP is considered, it is clear that there must be a greater risk of solder bridges being formed from pin to pin if the package is passed lengthwise over the wave than if it goes over sideways. Along the pack the pins are on 2·54 mm pitch, whereas across the closest spacing is 7·62 mm. Solder bridges are much more likely to form along the direction of board travel than across. In fact, if the solder is clean and does not contain an excess of dissolved copper or other metals sideways bridges are very rare. Thus for the best soldering, DIPs should be mounted end on to the board connector.

The tracking should also be considered when the package orientation is determined. Unless the board connector is very lightly used, there will be a substantial number of tracks running out from the connector into the main board area. If the DIPs are placed side on to the connector, all these tracks will have to pass through between the 'fences' caused by the first row of package pins (see left hand sketch in Figure 3.1). If the packs are placed end on to the connector, their pins offer minimal obstruction to tracks from the connector.

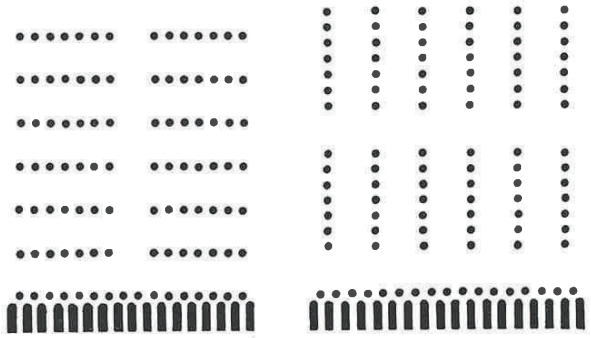


Fig. 3.1 DIPs side on and end on to board connector.

On double-sided boards, or on the outer layers of a multilayer board, it is then logical to run the tracks from the connector on the components side of the board, leaving the solder side for tracks across the packs, which will be running in the direction of wave soldering and so will be unlikely to be bridged with solder.

A further point to be considered on package orientation is cooling. If the board is to be plugged into a vertical back panel and cooled by air passing through between the board guides, as is the case in most large systems, then packages placed end on to the connector will present their longer sides to the air flow and will be more effectively cooled than if they were mounted side on to the connector.

The allocation of circuit elements to package positions on the board has been the subject of considerable study and a number of CAD routines have been derived to help with this aspect of design. Most of these routines work by examining the track layouts which will result from various placements. One common system treats all the required connections as if they were elastic bands interconnecting the pins directly, and moves packages around to minimise stress in the boards, that is, it attempts to minimise track length. These programs give no consideration to natural electrical splits or flow in the circuit, and they often result in very dense tracking in the middle of the board, with a large number of diagonal connections.

Some more recent algorithms work by making a trial placement, then the numbers of tracks which cross the spaces between rows and columns are counted, and packages are moved to equalise these numbers. These systems do not result in the minimised track lengths, but they do yield layouts on which the tracking is more likely to be possible to implement, and, by tending to confine functions to rows or columns, they are more likely to produce an acceptable result electrically. Also, they result in boards which are easier to make as the tracking density is more uniform over the board area.

A few CAD programs take into account the power dissipation of the packages and adjust the placement to avoid 'hot spots' or the risk of high dissipation packages affecting other areas of the circuit. However, to the best of the author's knowledge, no CAD routines perform package placements based on electrical considerations such as ensuring that fast devices which drive moderate length lines are at one end of the line and not in the middle, or that multiple branched tracks are avoided. In fact the 'elastic band' routines may lead to bad layouts in both these respects.

Manual package placement is best achieved by first splitting the circuit into any obvious working groups. Then attempts can be made to allocate such groups to rows or columns of packs, keeping packs which have the highest numbers of tracks to the board connector in the row adjacent to the connector. If a conscious effort is made to place packages in such a way as to yield the most direct straight line connections to other packages, either within the rows or columns, it will usually be found that most circuits can be implemented area by area without too much difficulty, following the logic flow pattern of the circuit diagram.

Most logic circuits contain a number of simple gates which are packaged with 2, 3 or 4 gates to a pack. In some cases board layout can be eased if full use of all such packages is not made. A circuit may call for the use of two 4 input gates, which could mean just one 4 input package. However, it may be found that, with optimum layout of the rest of the circuit, one of these gates is required near one end of the board and the other at the far end, which means running five tracks right across the board if the package use is minimised. When this occurs, the best solution is to use two 4 input packages, strapping together some of the inputs of the 'spare' half packs to use them as 2 or 3 input gates. Any such strapped inputs should be connected to the node with the lowest fan-out.

Some types of circuits appear to lack any main logic flow pattern and, once some obvious 'key' packages are placed, the designer is left with an apparently random tangle of possible flow paths. Such circuits may be placed by tracking from the connector to the key packages, then noting which of the unplaced packs has the greatest number of connections to the partial layout and placing this pack in the nearest available space. This type of working is facilitated if all gates or

packages on the circuit diagram are numbered, and as each pack is placed the numbers of the devices to which it must be connected are written in the circles representing its pads on the layout drawing. The board is thus laid out row by row, with tracks to unplaced packs being extended through the layout as the work proceeds. Package placement by this method is tedious and time consuming, but it does ensure that a usable board will result and there is no risk of the package placement resulting in an untrackable layout.

3.6 POWER DISTRIBUTION

On analogue boards care must be taken in routing power supply and earth tracks to ensure that they can not cause any malfunction, and often a star distribution will have to be used to avoid any risk of interference caused by return currents.

Digital boards pose different problems, as the supply rail and earth may have to make connections to 200 or more devices. How this is achieved can have a major impact on the design of the board.

3.6.1 Power and Earth Planes

If the track density and limits on board size dictate the use of a multilayer board, then the use of a pair of internal planes is an obvious and easy way of distributing power and earth to all packages. However, the inclusion of internal planes in what might otherwise have been a double-sided board is of very dubious benefit.[13]

The inclusion of planes will mean a significantly thinner dielectric below the signal tracks, which, as can be seen in Figure 2.7, will result in a lower line impedance and hence increased propagation delay (Figure 2.8). Line reflections will effectively slow all edges and unless the surface layer tracks are made very thin the electrical performance will be degraded. The cost of the board will be increased significantly, but the gain in packing density over a double-sided board with a well thought out power distribution grid will be small.

It may be suggested that the use of a closely spaced pair of internal planes can eliminate the need for local decoupling capacitors. This is not the case.

3.6.2 Decoupling

When a saturated digital logic device switches, its output current must charge or discharge the line it drives, and the instantaneous surge currents can be up to 30 mA as shown in Figure 2.9. These currents are far higher than the normal device supply currents, and for the smoothest working they have to be supplied by a decoupling capacitor. For this capacitor to be effective, track inductance between the capacitor and the power and earth pins of the switching device must be minimised. Suitable ways of mounting decoupling capacitors between DIPs are shown in Figure 3.2.

3.6.3 Surface Power Tracking

If the power distribution is to be on the surfaces of a double-sided board, five requirements must be met. First, the tracks must be of sufficient width to carry

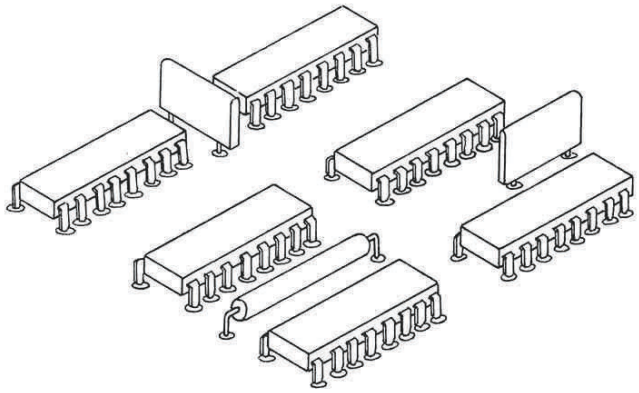


Fig. 3.2 Three ways of mounting decoupling capacitors in an array of DIPs.

the currents involved; secondly, the tracking between package pins and decoupling capacitors must be as short as possible; thirdly, the power tracking must offer minimal interference to signal tracking; fourthly, the individual distribution tracks must be easily routed to the board connector; and, finally, the system should provide the best possible earth return paths between packages.

Most of the commonly used logic packs have their power and earth pins on diagonally opposite corner pins, so there are two logical ways of distributing power and earth—across the ends of the packages on the solder side of the board as shown on the left of Figure 3.3, or beneath the bodies on the components side.

Both of these allow generous track widths, which would usually be 1 mm or over. A 1 mm track has a safe current rating of over 2 A, and on normal size boards will not cause excessive voltage drops. Tracking past the ends of the packs allows the use of tubular or chip capacitors, whereas tracking beneath the bodies is suitable only for use with chip capacitors. If the board is likely to be subjected to vibration, these might have to be bent over to have their bodies glued to the board. Figure 3.4 shows the minimum lengths of tracking between package and

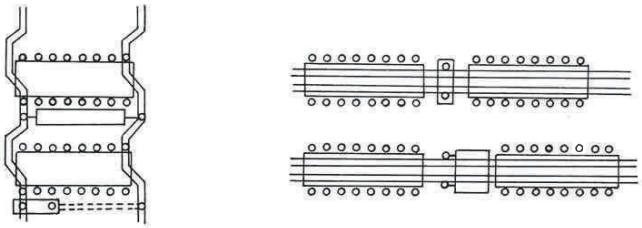


Fig. 3.3 Two ways of routing power tracks to DIPs.

capacitor pins, the right hand sketch showing tracking to a chip capacitor bent over and bonded to the board.

It can be seen from Figure 3.4 that tracking past the ends of the packages will result in the more efficient decoupling, especially if tubular capacitors are used.

Interference with signal tracking is the third point to be considered. One way of measuring this is by the effect of the distribution system on the trackability factors of the package pins. Tracks to another pack can leave a pad in four directions on each side of the board; diagonally up to the left, then up, or down; diagonally to the right and up or down, and similar across tracking. Thus on a double-sided board the 14 signal pins of a 16 pin DIP will have a total trackability factor of 112 as shown in Figure 3.5. Any tracking run past the pins will reduce the factor.

It can be seen from Figure 3.5 that power tracking run past the ends of the packs reduces the trackability factor of the pack to 108, whereas tracking beneath the bodies reduces it to 84.

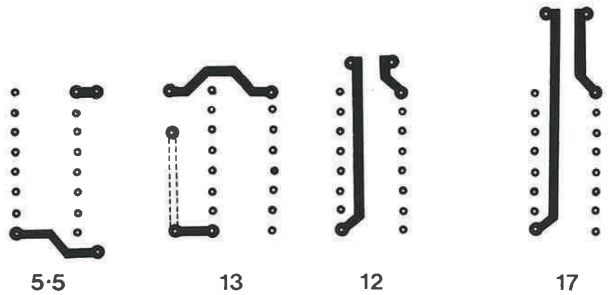


Fig. 3.4 Track length from decoupling capacitors to package pins (in units of 2·54 mm).

Alternatively, the number of signal track routes taken by the power tracking can be considered. If tracks are run on the lines of a 1·27 mm grid, the tracking beneath the bodies will displace five signal tracks whereas that past the ends will displace only two. If the tracks past the ends are bent as shown in Figure 3.5 to run through the pads and the signal tracks between adjacent packs are similarly bent then only one signal track per pack will be displaced. Thus from both considerations it can be seen that tracking past the ends of the packages has the least effect on signal tracks. If it is assumed that five tracks would be run in the spaces between the ends of the packs, then the power distribution past the ends will displace only one track in twelve (5 between the ends of packs plus 7 through between the pins).

Thus the maximum gain in the ease of tracking a board if power and earth are confined to internal planes will be under five per cent (one track more in twelve on one side of the board).

Distribution tracks run past the ends of the packages lend themselves naturally to connection to the board edge connector or plug by running bus bar strips down the side of the board. Power can be on the components side, connected to the

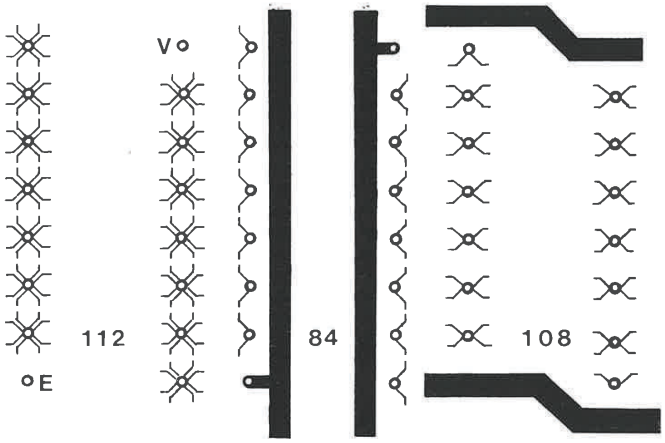


Fig. 3.5 Trackability factors of standard 16 pin DIP.

individual distribution tracks by via holes, with earth on the solder side joining directly to the rails as shown in Figure 3.6.

Figure 3.6 shows two contacts each side of the board used for each rail. The number of contacts used can be adjusted to suit the current loading of the board. In some systems power and earth might be fed in at one end of each board as shown, which will result in a 'superimposed E' distribution system on the boards. Power and earth might be fed in at both ends of the board, in which case a pair of bus bars each end will convert the distribution system to a ladder network.

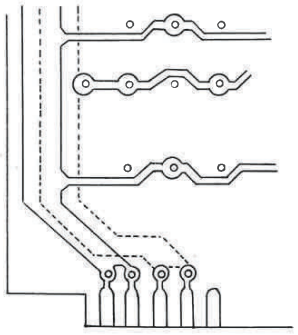


Fig. 3.6 Corner of board viewed from solder side showing power and earth bus bars.

Such a ladder network results in closed loops on both earth and power rails, but on digital logic boards such closed loops are not undesirable, as they are on analogue boards.

If power and earth are fed in at opposite ends of the board, an intermeshed E power distribution system will result as shown in Figure 3.7, while if the supplies are brought in other than at the ends of the board a centre fed system as shown in Figure 3.8 can be used.

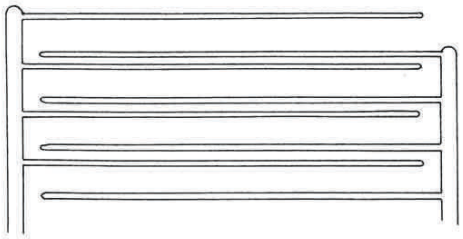


Fig. 3.7 Intermeshed E distribution system.

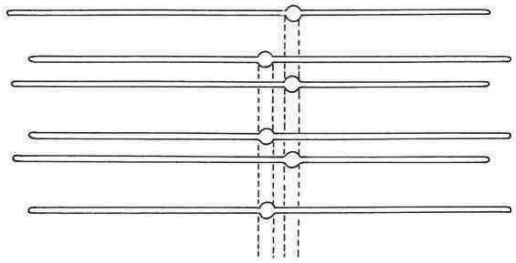


Fig. 3.8 Centre fed distribution system.

On small and medium sized boards with up to about 15 rows of packages a ladder distribution system will provide an adequate earth return path, but E or centre fed systems should have the open ends of the earth rails linked by straps to complete the earth loops. Larger boards may need extra links between the earth rails. The position of these links is not usually critical, and often they can be added in available space after the signal tracking has been routed. On very large, densely packed boards it is usually better to allocate spaces for earth links before routing the signal tracks.

From the above it can be seen that the choice of board connector and its position on the board, the orientation of the packages, the way the board is to be soldered, and the power distribution are all inter-related and all can have a significant impact on the packaging density and the electrical performance achievable. The dangers inherent in considering any of these features in isolation should be clear.

3.7 TRACK ROUTING

The difficulty experienced in routing tracks is usually a function of how well or badly the package placement and allocation of gates to packages have been carried out. The routing of tracks can often be eased if there is freedom to change the package placement as the work proceeds. If good boards are to result it is important to keep in mind the electrical function of the tracks as they are routed and to avoid risks of cross-talk on critical interconnections.[11]

In the earlier stages of track routing the problem lies not in finding a route for a track, but in choosing the route which will least impede the laying of future tracks. As with deciding on a power distribution system, a count of pad trackability factors remaining and of numbers of track routes left open can help. The number of possible routes for a track is not always fully appreciated. There are over a thousand routes for a track from one corner of a matrix of 9 DIPs in rows of 3 to the opposite corner if the packs are reasonably spaced on a double-sided board with tracks on a 1·27 mm grid and up to 3 via holes. If the number of via holes allowed is raised to 6 there are over a million routes possible. Over larger groups of packs the number of routes becomes astronomical. A moderately experienced designer can usually select a fairly good route almost at a glance, but CAD systems need to evaluate all possible routes and so tend to be poor at choosing those which will cause minimum inconvenience to the routing of future tracks.

As layout progresses, it can be necessary to move a track laid earlier to open up a route. On larger boards there can be several alternative re-routings possible and it may be necessary to change five or six tracks to clear a given path. In the earlier stages of a double-sided layout it is usually advisable to use via holes whenever a

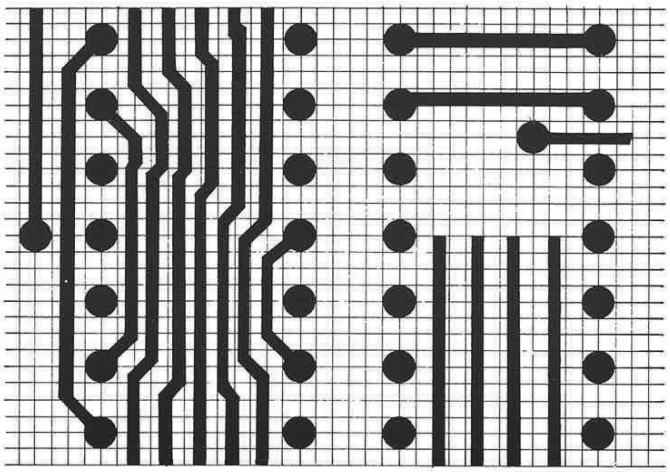


Fig. 3.9 Advantages of the use of off-grid tracks.

track needs to move over to another grid line, but as the work progresses and it becomes clearer what spaces must be left for future tracks it can be possible to use pairs of bends instead of changing sides.[1]

Track routing is simplified if all tracks are run on the lines of a layout grid which matches the pin spacing of the packages used. When TTL devices in DIPs are used, cross-talk should be kept to acceptable levels if the tracks are kept to the lines of a 1·27 mm grid. However, departures from the true grid can improve clearances and can allow extra tracks to be accommodated as shown in Figure 3.9.

With 0·33 mm tracks and gaps up to 9 tracks can be run beneath the bodies of DIPs instead of the 5 possible on the grid lines, giving a significant increase in the ability to complete a given layout. A finer grid such as 0·63 mm can be used, but this tends to result in groups of close tracks and then wider gaps, which increases the risks of cross-talk and can make the boards more difficult to manufacture. Some designers use alternate lines of a 0·432 mm grid which allows on-grid tracks to be centred between DIP pins and gives a reasonable tracking density without too much risk of cross-talk.

The correct use of bends to allow tracks to move over to an adjacent grid line can make a significant difference to the tracking density achievable. Some CAD programs for track routing allow the use of only right angle bends. With modern manufacturing methods these do not pose any real hazard as traps for etchant or for points where the copper might start lifting, but nevertheless they are better not used if they can be avoided. A bend at 45 degrees eliminates any possibility of such hazards, and as can be seen in Figure 3.10 angled bends save space along the run when a group of tracks are to be moved over.

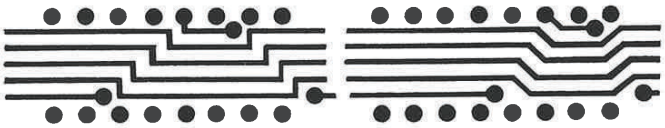


Fig. 3.10 Use of angled bends to improve track density.

The saving on a single group of bends may not seem very great, but over a medium sized board the use of angled bends can mean the difference between accommodating all the tracks on a double-sided board and having to use a multilayer. Another apparently small point which can have a significant impact is the way tracks enter package pads. It is obvious and logical to allow all tracks to enter their pads radially but sometimes significantly higher tracking densities can be achieved if the tracks are allowed to run tangentially to the pads as shown on the right of Figure 3.11.

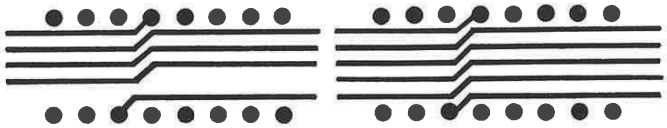


Fig. 3.11 Radial and tangential tracks to pads.

It can be seen on the left of Figure 3.11 that, when the tracks enter the pads radially, there is space for only three tracks to run through beneath the DIP, whereas the small change to tangential entry enables a fourth track to be run through.

Component mounting multilayer boards are usually best tracked in the same way as double-sided boards, with the tracks run on pairs of layers connected by via holes where a track has to change direction. Considerable care is needed in placing the via holes so they will offer minimal possible interference to tracking on other pairs of layers. Generally the best location for via holes is on the same grid lines as the package pins so that tracks running past the package have to angle in to the via holes. On ordinary multilayer boards the capability of X-Y co-ordinate tracking is limited by the via holes which have to pass through all layers, and it may prove better to use buried via-hole or double-double multilayers on which there is no such interference with tracking on other layers. The alternative is to minimise the use of via holes by greater use of angled tracking, but unless this is very carefully planned it can soon be found that a required route is blocked on all layers, and then often the only practical solution is the use of more layers.

Multilayer back wiring panels are more usually routed direct from pin to pin without the use of via holes.

It may appear that the design of a multilayer board should be easier than that of a densely tracked double-sided. This is not the case. It may be easier to use a very large number of layers with rather random tracking, but such a board can not really be said to be designed. Proper design of a multilayer board calls for balancing the amounts of copper on each layer, keeping the track distribution as uniform as possible over the layers, and minimising the number of layers. This is never as easy as designing a dense double-sided board.

Computer programs for track routing have existed since the early 1960s and, together with package placement programs, can produce board layouts faster than they can be achieved manually. However, CAD systems often yield less efficient layouts and a lower packing density than would a skilled manual designer. Most CAD systems impose limitations on the layouts which can be handled and usually points such as the optimisation of electrical parameters, distribution of power, thermal management, and balancing of multilayer layouts have to be handled by skilled manual intervention on interactive systems.

The wide variations possible in requirements and the continual updating of programs make it essential for any would-be users of CAD for printed circuit boards to evaluate the available systems against their foreseeable future demands, not overlooking the possibilities of linking into computer aided manufacture and testing (CADMAT).

Chapter 4

FILMWORK GENERATION

4.1 FILMS

The pattern on most printed circuit boards is printed directly or indirectly from films, and on all but the coarsest of patterns the success of printed circuit board manufacture depends on the quality of the film set and the care with which it is handled.

4.1.1 Care of Films

The set of films for any board should be regarded as precision phototools and they should be handled and stored with care. Films should always be stored flat, either in boxes or packets, fully supported over their entire area, and with sheets of interleaving paper between each pair of films to avoid any risk of damage to the emulsion. The storage racks should be in a controlled temperature room, and, while close control over humidity is not essential, it should be regarded as desirable.

It is common practice to store all films for a type of board in one envelope or box. Whenever this is done, great care must be taken to ensure that an edge of one film can never be scraped over the emulsion surface of another film as they are put in the box or packet. Ideally films should never be slid into a box. The whole box should be taken out of the rack and laid on a clean table. Films should then be taken out of the box and laid flat on a clean surface until the required film is reached. The proper handling of precision films requires a considerable area of easily cleaned flat surfaces, and the manager of any department handling films should not allow these clean areas to be used for any purposes other than the temporary storage of films or work on them.

Whenever a film has to be handled, it is most important that it is picked up and put down in such a way that it can not be creased. A film carelessly laid over a box on a table can overhang the box on two edges, and the edges of the film will curve down under their own weight. Where the two curves meet a small, almost imperceptible, crease can be formed. Such a crease can not be removed, and it can make it almost impossible to take a good print from the film.

4.1.2 Nomenclature

All films should be clearly labelled. The film pattern should include spaces for a board number, layer number, title and so forth, but in addition each print should have a small sticky label fixed on one corner, well clear of the working area, stating the date the print was made and the type of film. Type refers to whether the film is a positive or negative, whether it is emulsion up or emulsion

down, and an indication as to the generation of the film; whether it is a print directly off a master negative, or whether it is a copy of a copy of a copy.

A positive film is any film on which the areas which will be metal, solder resist, or legend ink on the finished board appear in black. On a negative, these areas will be clear.

An emulsion down film is one which when laid against the finished product in registration will have its emulsion surface against the face of the product; an emulsion up film has the emulsion on the side away from the product.

Whether a film is positive or negative can be determined, as can the emulsion side, but the generation and age of a film can not be found other than by reference to a label attached when the film was printed. All this essential information can be conveyed by a single letter and the date. The lettering can be arbitrarily chosen, for example, A for an 'off camera' emulsion down negative; B for an 'off plotter' emulsion up positive; C for an emulsion up negative, which could be a print from an A master on reversal film or it could be an 'off plotter' earth plane film; D for a master emulsion down positive, which could be a 'reverse plotted' film. E would then be a second generation copy of an A film, and so on. An A film could be contact printed to yield an F film, whilst a contact from B would have to be an 'E'.

4.1.3 Stability and Size

Film is an unstable material. It changes its dimension with changes of temperature or humidity. It is therefore important that precision films are handled and stored in controlled environments and not subjected to any unnecessary heat. One frequent cause of problems with precision films is heating and drying by the lightboxes generally used to illuminate films for 'spotting' and other work. When precision films are to be handled, the working faces of the light boxes should be indirectly illuminated, or triple glazed boxes with a flow of controlled temperature air below the working face should be used. Some precision measuring machines are very bad in this respect, as the optical sighting head is illuminated by a very powerful light which can, in time, distort a film to the point at which a blister is raised.

The need for accurate films, especially for multilayer boards, makes it imperative to have some simple way of checking the size of a film before it is used. Even with care in storage and handling, films can change size, and it is not unknown for one or more films out of a set to move by over 0·1 mm between the new film set being checked for accuracy and the films being used, possibly a month later. Most companies use co-ordinate measuring machines with optical heads to check the sizes of films. Unfortunately, such measurements are subject to operator error, and, although the digital read-out may be to 0·001 mm, the actual overall accuracy is usually little better than 0·05 mm. For precision boards, more accurate measurement is necessary, and all films should be provided with size marks of a type which will facilitate rapid checking.

The size gauging marks must be of a type which is not affected by any growth in the black areas of a film caused by overexposure or overdevelopment. The most satisfactory is either a chequer board cross made up of pairs of rows of alternate dark and clear squares as in Figure 4.1, or a segmented circle. The central crossing lines of such marks can be aligned over crossed hair lines on a gauge and, if the gauge lines are filled in a colour, such as red or a mid blue,

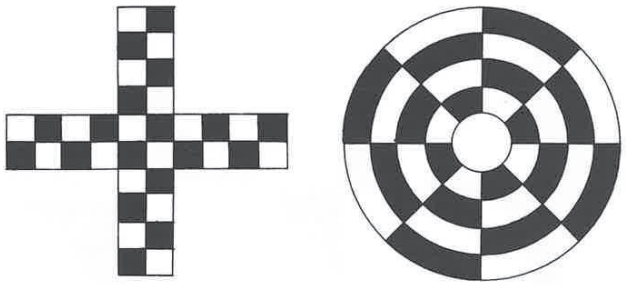


Fig. 4.1 Cruciform and segmented circle gauge marks.

which can be clearly distinguished from the black of the film, any errors in alignment can be detected almost at a glance (see Figure 4.2). The touching corners of the black segments give immediate warning of any exposure or

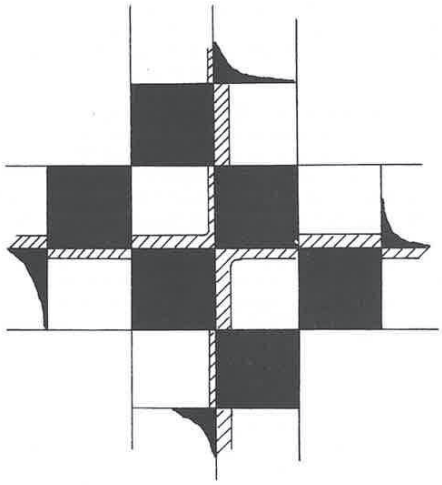


Fig. 4.2 Gauge mark laid over grid lines and misaligned horizontally.

development faults as shown in Figure 4.3, but if the change in feature size can be tolerated the markers can still be aligned accurately to the gauge lines despite the loss of accuracy of individual edges. Any alignment to gauge lines should be performed over at least six segments of the pattern.

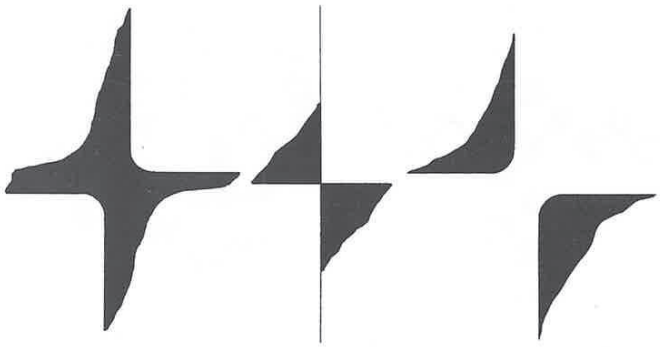


Fig. 4.3 Effect of over-exposure, correct exposure and print from over-exposed negative.

Suitable gauges can be diamond scribed or etched into glass bars with lines about 0·1 mm wide. Since most PCBs are made to imperial grid to match the 0·1 inch standard component pitch it is desirable to keep the gauge marks spaced in inches. If the gauges are made with crosses at 0, 2, 6, 14, 24 and 30 inches, most sizes of film likely to be used can be gauged by choosing the appropriate pair of marks.

Squareness of a film can be controlled by using four marks forming a square. No expensive equipment is needed to check squareness. A negative film is laid in register over its positive, and is then rotated 90 and 180 degrees. If the four markers align after the rotations, then the film is square.

Gauge marks should be placed on the films so they will be printed in the scrap borders of the manufactured panels, where they can be used to check for material movement during processing. The use of gauge marks means that all size measurements are performed with the film emulsion down on the gauge, so parallax errors can not arise. The actual measurement will be in terms of 'Units of departure from gauge distance', measured with a small area calibrated magnifier or optical micrometer. Operator error can have only second order effects on the accuracy of measurement.

4.1.4 Registration

Manufacture of good, precision boards with all holes well centred in their pads is possible only if all pads on all sheets of film in the set are in register. Registration of films, and the regularity of spacing of rows of pads for DIPs, is very simply checked by laying a negative of one layer over the positives for all other layers in turn. If the film set includes a 'pads only' outer layer, or an untracked 'drill chart film', then the negative of this film should be used for the check.

The negative is taped onto a light box, and any area of illuminated glass beyond the edges of the negative is blanked off. Each positive in the set is then aligned over the negative. With the room lights dimmed, an error of 0·01 mm in

registration will be glaringly obvious. Uniformity of rows of holes is checked by moving the positive one hole pitch along over the negative, straightness of the rows is checked by rotating the positive through 180 degrees, and on most boards squareness of the pattern can be checked by a 90 degree rotation.

These tests are extremely sensitive, and unless skilled staff are employed there can be a risk of usable films being rejected. With the use of proper gauge marks and a grid gauge, a film set for a multilayer board can be checked for size, registration and squareness in a very few minutes, the only equipment needed being a cold top lightbox, calibrated magnifier and master size gauge. Such checks are far more accurate than can be made on any co-ordinate measuring machine and take only a fraction of the time.

4.2 FILM GENERATION

Production PCB films can be generated by reduction from oversize master art works, or directly at one to one scale on a photo-plotter.[14,15]

4.2.1 Plotted Films

The usual way of generating one to one films is by photo-plotting on a machine which moves a beam of light over a table on which the film is mounted. Pads and other standard features are printed by a flash of light through specially made masters, and tracks are 'drawn' by moving a beam of light of controlled width and intensity over the film. The master symbols for pads and track apertures are usually carried mounted round the edge of a disc which generally mounts up to 64 such masters.

Selection of the masters, and the travel of the plotting head over the table is usually controlled by NC tapes, but plotters can be linked on-line to a computer which designs the boards. Control of the intensity of the light, and the generation of radii or angled tracks are usually performed by a computer built into the plotter, the control tape merely specifying the width, start and finish of each track, together with any bend points.

Modern plotters are very accurate machines, and usually the table or head position can be controlled to within better than 0·02 mm. However, that is not the only possible source of error. The symbols for pads and tracks may not be perfectly centred on the symbol disc; the disc must have some freedom to move, and the film is unstable. Checks as described in Section 4.1.4 should be carried out on all films, and the results may come as something of a shock to any who think that the introduction of a plotter will be a cure for all film errors. For real accuracy, it may be necessary to plot at scale 2 to 1 or even 4 to 1 and then photo-reduce to obtain the master films.

When boards are designed on a CAD system, photo-plotting is clearly the ideal output. However, when boards are designed manually the benefit of plotting becomes questionable. An accurate master drawing on grid base material must be prepared for each layer of the board, with all line widths, pad sizes and so forth clearly defined, usually by the use of contrasting colours. The drawing must show clearly the exact point on the grid at which each track must bend. These drawings are then digitised on a co-ordinate measuring machine to produce the plotter drive tape, usually after some computing has been undertaken. Most digitisers have software routines to cover repetitive patterns, matrices of pads, and similar

features, and some are able to calculate the correct spacings for groups of tracks passing between pairs of pads or other obstructions.

Digitiser and plotter systems are being continuously improved, but most available systems impose some limitations on the design of boards.

4.2.2 Cut and Strip

For very accurate artworks, cut and strip master films may be used. In this process, a two layer film is mounted on the table of a co-ordinatograph. The lower layer of the film is transparent, covered with a thin, translucent red layer. A knife blade is mounted on a carriage which is free to move on the cross arm of the co-ordinatograph, and is set so it will cut through the red layer of the film but not the base. The cross arm and head are moved, either manually using vernier scale readouts or under NC control, so the knife blade cuts all required edges of the pattern. The unwanted red areas are then removed to leave the red layer defining the pattern.

Special heads are available on manual co-ordinatographs to cut lines at angles and to cut circles. With care, a manual machine can yield work accurate to 0·03 mm. Cut and strip artworks are most suitable for special boards such as rotary switches or microwave products which require relatively few, large areas of metal, or for pad masters for hybrid artworks.

4.2.3 Manual Artwork

The term 'manual artwork' has come to be associated with relatively crude, inaccurate films. This need not be true, as the accuracy achievable depends on the care and methods used, and in fact it is possible to produce more accurate films manually, or rather by 'hybrid' methods, than by photo-plotting at one to one.

The earliest manual artworks were usually made by painting indian ink on white card, normally at two to one or four to one scale. Such artworks clearly had very limited accuracy and were difficult to correct or to alter for any post design modifications. The board shown in Figure 1.3 was probably made from a painted master.

Painted masters were superseded by taped masters. These are made on a base film, to which cut-out symbols in a black crepe material are stuck to represent pads, and tracks are formed from lengths of black tape, stuck down and cut to length. The work is usually carried out on a lightbox. Such taped artworks are easily altered, and, on a blue grid base, which will not reproduce on the photographic film, they can be made with a reasonable level of accuracy. A good operator can stick down all pads on a board to within 0·2 mm of the grid intersections, which at scale 4 to 1 is quite adequate for many double-sided boards. The board shown in Figure 1.4 would be from such a hand-taped artwork.

The die-cut symbols for pads are not easy to stick down accurately, and they can become distorted, so for precision work they were superseded by rub-down dry film transfer pads. With care, these can be applied with an accuracy of greater than 0·1 mm to the grid intersections, and with the aid of a magnifier, to 0·03 mm or better.

One of the biggest problems with manual artwork is ensuring layer to layer registration of all pads.

4.2.4 Two Colour Artworks

It is obvious that the use of one set of artwork pads for both sides of a double-sided board should guarantee side to side registration. A method of making double-sided artworks which achieved some popularity was to stick down black pads, then to use two different colours of translucent tapes, usually red and blue, for the tracks on the two sides of the board. During reduction the track patterns are separated by the use of colour filters. This simple system suffers from several major disadvantages. If both red and blue tapes are stuck on the same side of the base film, then there is a well known law which ensures that any alteration will be to a tape which has the greatest number of the other colour of tapes laid over it. If the tapes are stuck on both sides of the film, then the artworks are difficult to store and handle without risk of damage. Since blue tapes have to be photographed, a blue grid base film can not be used, and photography of the two different colours means different types of film must be used. It is generally accepted that, for accurate work, all films used in a set for a board must be taken from the same batch and processed identically, so the two colour system can never be relied on for real accuracy. Finally, it can not be applied to multilayers.

4.2.5 Single Pad, Multiple Track Sheet Artworks

All the disadvantages of two colour artworks can be overcome by making up a 'pads only' master artwork on blue grid film. This sheet will actually carry all pads, corner marks, reduction gauges, and any other features common to all layers of the board. The sheet is dowel punched, and separate sheets of film which carry only the track pattern for a given layer are located over it, and the sheets are photographed dowelled together to produce the master films.

This system can be used for multilayer boards as well as double-sided, and it guarantees layer to layer accuracy. Its only disadvantage is that the camera used for reduction must be fitted with a vacuum copy-holder, and dry film transfer pads must be used. If there is any separation between the two films in the copy-holder the air gap can act as a lens and distort the size of features on the film further from the camera lens. Also, the camera operator must be perfectly familiar with what is required, and must take care to photograph all films the right way round.

4.2.6 Hybrid Artworks

The simplest way to achieve accurate registration without any risks of mistakes during reduction is to generate a master pad pattern, print this to a master negative, and then run off as many contact prints as there are track layers in the board, plus one extra for a drilling key. Individual layers are then taped onto the prints. Since each layer will carry its size reduction gauge, and all layers should be checked for size after reduction, any film movement which occurs during taping can have no effect on the overall accuracy or registration. The only film error on the masters which can have an adverse effect is major emulsion slip during processing.

The master pad pattern may be plotted, or for the most accurate work it should be made with dry film transfer pads. The master can then be checked against its negative as described in Section 4.1.4, and any errors corrected, using a good area

of the negative as a guide to re-lay misplaced pads. By successive checks and corrections, accuracies of better than 0·02 mm can be achieved, and, once a really good master negative has been made, it can be used as an aid in the generation of other masters for similar boards.

When it is found necessary to add pads to an artwork which is being modified, a misplaced negative can be used as a guide to lay the extra pads, provided all pads are on a 0·100, 0·050 or 0·025 inch grid.

Hand taping is quite adequate on two to one or four to one artworks provided all bends are properly made with cut lengths of tape and a full overlap, and the tape is not laid under too much tension. Centering the tape between pairs of pads is quite easy, and when clearance is required between a track and a single feature a scrap length of tape of an appropriate width can be used as a guide. The boards shown in Figures 1.6 and 1.7 were both made from hand taped hybrid artworks, and at the time it was considered that the board of Figure 1.7 was too complex to digitise and plot, and the requirement for accuracy was such that, even if the track pattern could be plotted, it would have to be done overscale and reduced, as the manual artwork was reduced.

4.3 PLANES AND SOLDER RESIST FILMS

Power and earth planes and solder resist films can cause problems since they consist of clear areas on a solid background instead of a pattern on a clear base. The simplest way to deal with these is to generate the main pattern of pads as negatives, from which contact-printed positives can be mounted on normal artwork base films. The negative masters can be plotted or made with manually placed pads.

Solder resist patterns pose few problems, as they are simply oversize pads, which leave clear areas in the negative.

Earth and power planes should be made up with an annular clearance between a normal sized pad around every hole and the body of the plane. This clearance ring is bridged by a track, usually about 0·5 mm wide, whenever a connection is required from a hole to the plane.

When films are plotted, such planes will require special treatment. One way is to use the plotter to 'draw' all the annuli as thin lines, usually 0·3 or 0·4 mm wide, either drawing complete circles or drawing the partial circles to leave the bridging tracks. This method is accurate, and with a good plotter system the computer will do all the necessary calculations, but it is very wasteful of plotter time.

A more economical method is to plot a normal sheet of pads with short lengths of track to form all bridges necessary, and to plot as a negative the main body of the plane, using flashed down pad symbols to define the outer edges of all the annuli. Both films should be plotted on pre-punched films on a dowel bar secured to the table of the plotter. If the plotter can not be fitted with dowels, then the plotted films will have to be located accurately on a punch and provided with tooling holes. The normal sized pad sheet is then contacted to a negative, using pre-punched film on a dowel bar so that the tooling holes are transferred. This negative is then contact-printed onto another dowelled film, which is not developed.

After the negative of the pad pattern has been exposed, it is replaced on the dowel bar with the plot of the over-size pads which forms the negative of the

outer edges of the annuli, and this is exposed, and the film is then developed normally to leave the body of the plane and the pads and bridges in black.

This double exposure technique saves time, but it usually results in eccentric annuli. The two different sized pad symbols on the plotter seldom align absolutely perfectly, and there is always some risk of distortion or slight misalignment in double printing on dowels. With care, the errors should be acceptable for board production, provided the pads are in adequate registration with other layers of the board. So long as the minimum allowable width of the annular clearance is not infringed, eccentricity of about 0·05 mm in the outer edge will not have any adverse effect on the working of the board.

Planes can be made manually by rubbing down dry film transfer annuli on a clear film laid over the master pad pattern. Borders for the plane are added in tape, negatives of the size markers are stuck down, and the film is then contact-printed to produce a positive. Rubbing down the annuli in registration is made easier if the centres of the annuli are about 0·04 mm larger than the pads on the track layers. This allows a slight tolerance in their position. Bridges from the pads to the plane are taped in on the positive.

Visual checking of a finished board or film set is aided if all the bridges on the earth plane are put over the lower half of the annuli when the board is viewed as it would normally be held, and the power plane bridges cross the upper halves of the annuli. The bridging tracks should be placed such that they connect their pads to the largest possible area of the plane and not to an area which is connected to the main body of the plane only by thin tracks between pads.

4.4 PHOTO-REDUCTION

Photo-reduction of artworks or plotted films for precision boards is basically a simple process, but it must be performed on a first class camera, and the operator must know how to use and check the camera properly. Good process cameras are calibrated to yield any desired reduction, and are usually computer controlled to give the correct illumination and exposure. Precision reductions should never be specified as 'Reduce 2 to 1', or as a percentage, but should be specified as 'Reduce to gauge dimension of X·000 inches'. The operator should check all artworks of a set to gauge sorting them to piles of 'Nominal', 'One thou up', 'Two thou down', and so on. The camera is then set to the appropriate reductions to give the required size. Naturally the camera should be in a properly controlled environment and should be on a vibration free mounting, preferably on a solid concrete floor. Since much of the associated work is dark room work, some users prefer to install cameras and plotters in windowless basements.

If the artwork was not generated in the same controlled environment as the camera, it should be left for at least 24 hours in the camera room to stabilise, preferably with each layer open on separate shelves. Artworks for transport to a camera should always be packed flat, with separating sheets between layers.

The accuracy of reduction and freedom from distortion of a camera is best checked by reducing a square chequer board pattern of known uniformity. The reduced film should be checked positive against negative, with steps and rotations to find any errors in squareness and linearity. Various settings can be checked for accuracy of reductions by aligning the reduced film over a portion of the master and checking for exact coincidence of the edges of the squares.

4.5 PHOTOTOOLS

Off camera negatives or plotted positives should not be used for production work, but should be kept as masters, with the work carried out on accurate copies. Often board production will involve the addition of special borders to the films, and test patterns which the board designer has not included on the artwork. Clearly the closer the contact between the departments which generate the films and manufacture the boards the easier it will be to produce original masters which are usable for production.

Small boards are usually made in multiples on a production blank, and the films will have to be stepped and repeated, either on an S and R printer or by multiple photo-plotting.

Whilst in general master films are not used other than for taking contact prints, in a facility which has a plotter available and programmed to include all borders, tooling marks, test coupons, and any step and repeat, then clearly the most accurate results will be achieved by the use of duplicate direct plots for production.

Any contact prints taken for production should always be treated as a set, and prints for all layers of the board should be run off from the same batch of film. All printing should be done emulsion to emulsion, under a good vacuum. Obviously all films must be kept dust free, and all work should be handled under clean room conditions with 5 micron air filtration, and all operators wearing protective clothing.

4.6 FILMS AND PROCESSING

The lack of stability of films can be overcome to some extent by plotting or photographing precision work onto glass masters. These are less temperature sensitive than film, but they are expensive, difficult to store, heavy to handle, and may be broken. Also, they can not be developed in automatic machines.

Suppliers of films should be consulted about the correct emulsion type and the correct chemicals to use for precision work. It is sometimes assumed that since PCB films are patterns of black on a clear base they can be over-exposed and over-developed. This is a very dangerous assumption since any such over processing can result in significant growth in size of the black areas, and edge definition will be lost.

The films used in production for printing onto boards are often made of a diazo film, in a translucent red or orange which is opaque to the ultra-violet light used to expose the photo-resists, but through which the alignment of a film on a drilled blank can be checked.

If a film has changed its size after it was made and checked, and the change is considered too great for satisfactory processing of the board, the best course of action is to reproduce the film on an accurate process camera set just off one to one. When this cannot be done, attempts can be made to restore the film to size by changing its temperature and, when a check to gauge indicates that it has moved to just beyond its true size, it is contact printed, the new film is carefully checked for size and then contact printed to produce a duplicate of the original at the correct size.

It sometimes happens that a film set for a board is made without allowance for manufacturing tolerances to produce a board to the specified tolerances on

feature sizes, and the board manufacturer has to produce working films with over-width features. This is usually achieved by contact printing the films with a sheet of clear film between the emulsion faces, possibly by taking the negative with the film the wrong way round in the printing frame so the light has to pass through the base film to reach the emulsion. When this is carried out on a contact printing frame with a uniform diffused light source, the light will refract through the film and spread the image. This growth in the black areas of the film can be aided by deliberate over-exposure, but a careful check must be made on the sharpness of the edges on the finished films.

If the changes in feature size required are larger than about 0·05 mm it is better to use a step and repeat printer set to take multiple prints on sine cosine co-ordinates round a small circle. Five such steps will produce adequately circular pads. Process cameras with moving lenses which will produce the same effect are available, but these are very expensive, uncommon machines.

Convergence is a photographic effect which may cause some problems when precise films for microwave work are being reduced. If a pattern includes a very narrow gap between two large metal areas, or a very fine, isolated line, the width of the very narrow feature may be distorted after reduction because the closeness of the edges has caused the light to spread. In one such instance, a gap between metal areas which should have been 0·05 mm wide measured 0·08 mm after a four to one reduction. This effect can be overcome by carrying out a two stage reduction. The intermediate negative has the gap, which shows as a black line, over width, then convergence on the second reduction causes the gap to close to the correct width on the final reduction.

Lasers are not as yet widely used in film work for PCBs, but it is possible that in the future lasers may take over some precision work from conventional photo-plotters. One possible area of application of lasers is in rectification work.

Poor quality films or films which have changed their size could be scanned, and the output from the scanner could be operated on by a computer which would drive a plotter or laser film generator to produce a true size film with all pads correctly on grid and all tracks properly positioned with the required clearances.

It is not impossible that in the future the use of films might be superseded by direct high speed laser printing onto the sensitised board surfaces.

Chapter 5

BOARD MANUFACTURE

5.1 INTRODUCTION TO MANUFACTURE[1,2,8,47]

Printed circuit boards can be manufactured with relatively simple equipment, but for economical production of high precision boards a large investment in capital equipment is required. Some of the large electronic systems companies have their own in-house PCB facilities, but others prefer to use the services of specialist board manufacturers, where the purchase of new equipment and the costs of acquiring new skills can be spread over a number of customers. At the beginning of the 1980s, there are some 400 to 600 specialist PCB factories in the UK with many thousands more in the rest of the world. In the UK the top half dozen employ several hundred people each and have plant costing millions of pounds. About a further dozen firms employ around fifty people, with good facilities, and the remainder are small firms mostly with limited equipment. Some of these small firms can offer individual expert attention and a fast turn-around, but few have BS or other approvals.

5.1.1 Laminate Manufacture

Most board manufacturers buy in sheets of laminate from the specialist laminate suppliers, rather than making their own. Paper phenolic laminates are made by pressing together sheets of resin impregnated paper with the copper foil outer layers and are always supplied fully cured. Epoxy glass laminates may be supplied fully cured, or as partially cured sheets of 'prepreg' in which the resin impregnated glass cloth has been dried to a B-stage partial cure.

Epoxy glass prepreg is produced by drawing the woven glass base, which is usually coated with Silane to promote a good bond to the resin, from the large rolls in which it is supplied from the weavers, through a trough containing the epoxy resin and a solvent. The impregnated cloth is passed through a drying oven in which the solvents are evaporated. The dried prepreg is then cut into sheets, either for sale for use in multilayer board manufacture, or for stacking and bonding with outer sheets of copper into fully cured clad laminates. The control of resin and solvent contents in the prepreg is critical, and the weave and weight of the glass cloth is chosen to yield the desired mechanical properties in the finished laminate. Clad laminates are usually supplied to the board manufacturers in sheets four feet by three feet, or in similar metric sizes.

5.1.2 Planning and Preparation

The manufacture of any type of board starts with planning the job, ensuring that the necessary laminates and prepreg are available; that accurate working

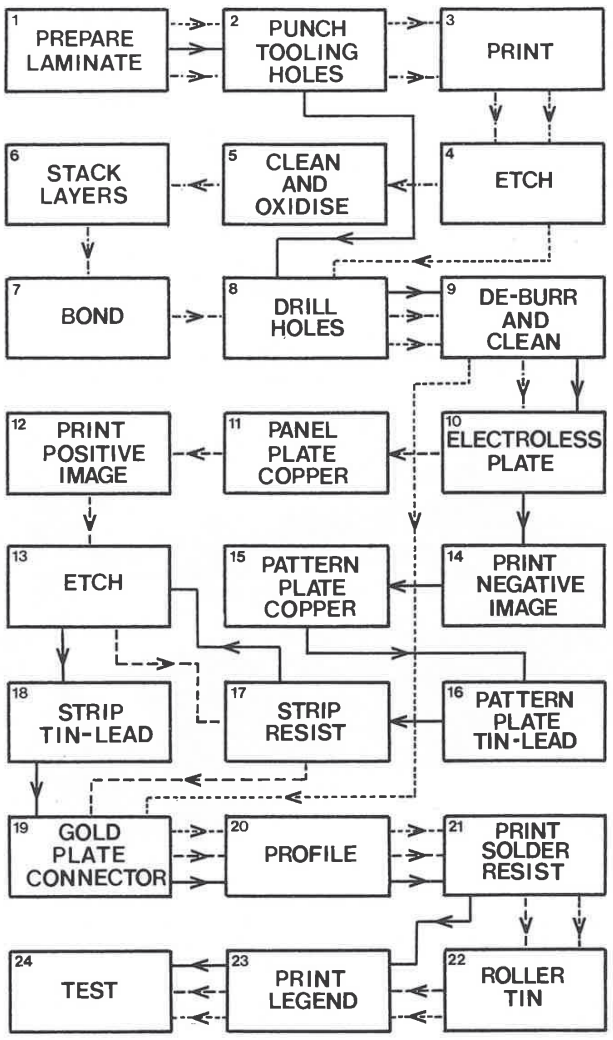


Fig. 5.1 Flow chart of rigid board manufacture.

films have been made, and that any control tapes required by NC machines have been supplied or can be made. Planning must also include the commercial aspects of ensuring that plant capacity is free at the appropriate times, and that quoted delivery dates can be met.

The basic stages of rigid board manufacture are shown in simplified form in Figure 5.1. There are many possible variants in manufacture in specific plants, and the flows described here might not be followed exactly. Some of the stages, such as gold plating edge connector fingers, or the application of solder resist, may not be required on some types of board.

The manufacture of non-plated boards follows a different path from that of through-hole plated boards, and usually the two paths are combined in the manufacture of multilayer boards.

All types start with the preparation of the laminate. The first operation is to cut the large sheets down to suitable sized panels for production. Phenolic paper laminates can be cut on a circular saw, either with a high speed steel or an inserted tooth cutter, but the highly abrasive dust from cutting epoxy glass would very rapidly blunt such saw blades, so either a diamond studded disc cutter is used, or more commonly the material is cut on a large guillotine as shown in Figure 5.2.

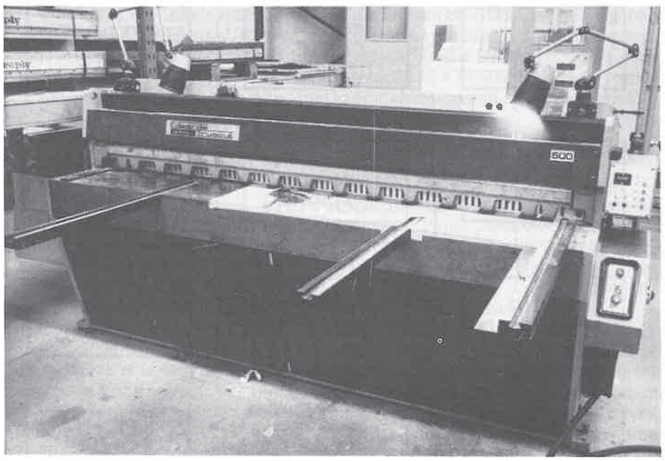


Fig. 5.2 Guillotine for cutting sheets of laminate.

The cutting of laminate on a guillotine may appear to be a very simple process, but the laminate can be damaged if the guillotine is not fitted with adequate means of catching the pieces as they are cut and if the blades are not correctly sharpened and set. Thin laminates, especially flexibles, are easily kinked or creased and if they do develop a crease through not being supported properly as they are cut, then it will be almost impossible to make a good print of the pattern required on the surface and the piece of laminate will have to be scrapped. If the

blades are not set correctly, thin laminates may be bent over instead of being cut, or the bond between the copper and the substrate may be weakened near the edge. The guillotine should be provided with a clamp to hold the sheet of laminate firmly as it is cut so that the shearing action of the blade can not swing the sheet round, resulting in a bowed edge.

The cut panels of laminate are usually punched with tooling holes, and for through-hole plated boards the panels are normally pinned into stacks of 3 or 4 panels as the holes are punched, on machines such as that shown in Figure 5.3.

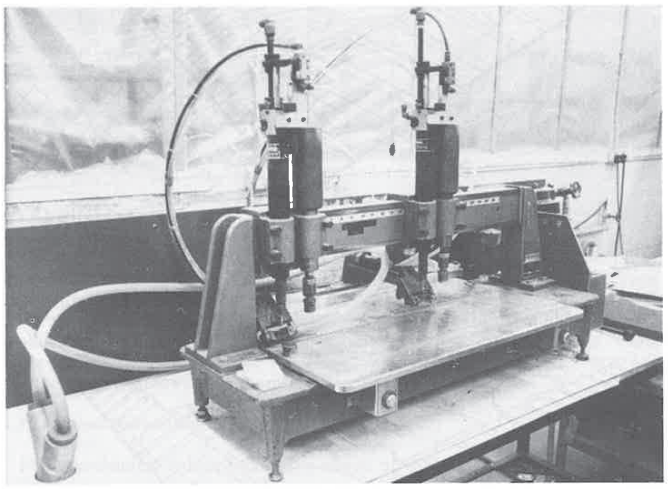


Fig. 5.3 Piercing and pinning machine.

5.2 SUBTRACTIVE BOARD MANUFACTURE

Non-plated boards are produced by a simple 'print and etch' process, following the upper fine dashed line on Figure 5.1. After the panels have been cut and any tooling holes punched they are thoroughly cleaned and then coated with an etch resist to define the required track pattern as shown in Figure 5.4.

For simple laboratory prototypes with bold patterns the resist can take the form of cellulose or oil based paint applied freehand over suitable marks to define the positions of the holes. For more precise patterns than can be hand painted a photographically developed resist is generally used, and in quantity production the resist is usually screen printed onto the blanks.

The copper which is not protected by the resist is etched away to leave the required pattern, the resist is cleaned off, and the board appears as shown in Figure 5.5.

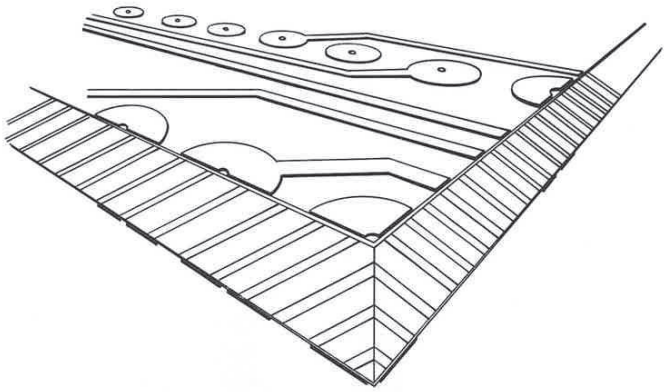


Fig. 5.4 Board blank coated with exposed resist.

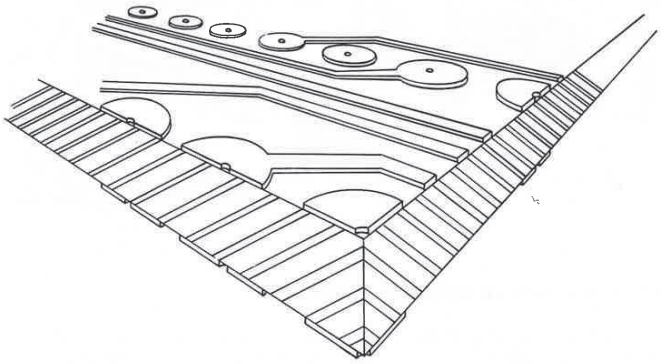


Fig. 5.5 Board blank after etching and resist removal.

The holes are then drilled or punched, going direct from step 4 to step 8 on Figure 5.1 and completing the basic board manufacture to the stage shown in Figure 5.6. After removing any burrs from the drilling, and cleaning the track surfaces, any required finishing stages, such as tinning the pads or the application of solder resist or legend, are carried out as described in Chapter 10. Depending on the size of the boards and the type of plant the boards may be cut or punched to profile before or after these stages.

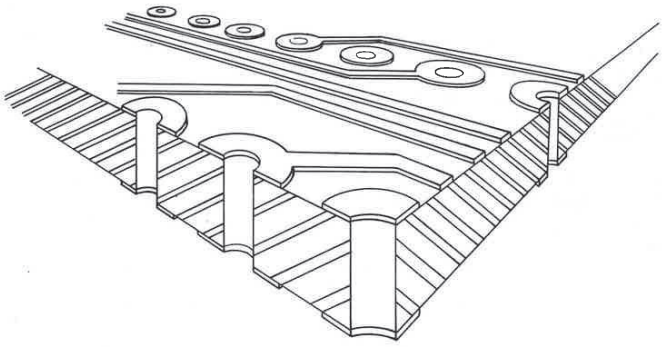


Fig. 5.6 Etched blank drilled or punched.

5.3 PRINTING

The processes of printing a pattern in some form of resist are fundamental to all current forms of PCB manufacture. The first obvious division is between screen printing and the various forms of photomechanical printing. Generally it is taken that for printing track patterns screen printing is more suitable for large production batches of boards with relatively coarse track patterns while photographic methods are better for small batches and for very precise work, but may cost more on long runs than screen printing.

5.3.1 Photomechanical Printing[16,17,18,19,20]

Photomechanical printing covers a range of different types of resists and different exposure systems. The first major division is between wet resists and dry films.

Wet resists are applied to the panel in the form of a liquid which is allowed to dry before exposing. If uniform results are to be achieved it is important that the applied film of liquid dries to a uniform thickness. In the earlier days of PCB manufacture this was usually achieved by spinning the blank at high speed; placing a drop of resist in the centre of the spinning blank, and allowing it to centrifuge off to the required uniform thickness. The major alternative to this was dipping the blank into a tank with a slow controlled withdrawal rate which allowed the liquid to drain off to the correct thickness. Improvements in the formulation of resists allowed roller coaters and later curtain coaters to be used to apply them to the required thicknesses.

Generally liquid resists dry to leave a very thin, tough film with good adherence to clean copper. Roller or curtain coated liquid resist can be applied over uneven surfaces, such as are found when selective plating is used.

For most purposes in PCB manufacture liquid resists have been superseded by dry film resists. These are plastic films which polymerise or modify their structure under U-V light. They are usually supplied between two thin plastic protective

sheets, in rolls, and are applied to the cleaned faces of the board blanks in roll laminators which remove the protective layer from the board face of the film. After exposure the outer protective foil is removed and the dry film resist is developed.

For the application of both types of resist it is essential that the surface of the board be clean and readily wettable[17].

Both liquid and dry film resists are available in positive and negative working versions. A positive resist is hardened by exposure to ultra-violet light, and the unexposed resist is removed during development, whilst a negative resist softens in the exposed areas and the exposed resist is removed by the developer.

Resists must be capable of withstanding attack by plating solutions and etchants, but must be capable of being removed by solvents which will not attack the epoxy or phenolic resins of the base laminates. Solvents based on methylene chloride are commonly used. The resist should dissolve cleanly in the solvent without leaving any flakes which could be trapped under overhanging plated edges.

Photomechanical resists are exposed to the master films in vacuum frames, usually with both sides of the blank exposed simultaneously. For good line definition it is important to ensure that the films have their emulsion on the side against the resist. That is, the films must be emulsion down prints. It is also important to ensure that there is no dust on the surfaces of the blanks or the films, and that the vacuum in the printing frame holds the films close against the surface of the resist. The printing area should be operated under strict clean-room conditions and all personnel should wear protective clothing. Line and gap widths on modern printed circuit boards do not allow easy spotting-in of any printing defects.

5.3.2 Screen Printing[2,8,21]

In the screen printing process the resist, in the form of an ink, is squeegeed over a screen mesh which has been coated with an emulsion or film which closes the meshes of the screen where the ink is not required on the board.

The screen, which may be of stainless steel, polyester, or a metal coated plastic thread, is stretched across a strong metal frame. This frame is held in the upper frame of the printing machine, which is pivotted so the blanks can be placed on the machine bed and located either by tooling pins or by edge guides as shown in Figure 5.7.

The frames hold the screen clear of the face of the blank and parallel to it. The ink is spread over the face of the screen, and the action of the squeegee drives the ink through the open holes in the mesh, pressing the screen down so that the ink makes contact with the face of the blank. After the squeegee has passed, the screen springs back clear of the blank, leaving on the blank the ink which had been in the holes. The discrete particles of ink flow together to yield a uniform coverage of the blank over the open areas of the screen.

Screen printing can be a very accurate process, and it is ideally suited to long production runs provided that the screens are properly made and set up. The material and size of the mesh play a critical role in determining the long term stability of the screen, the definition achievable, and the amount of ink deposited. A thick thread, coarse mesh screen will have the greatest resistance to stretching and to deformation of the pattern under the repeated pressure of the

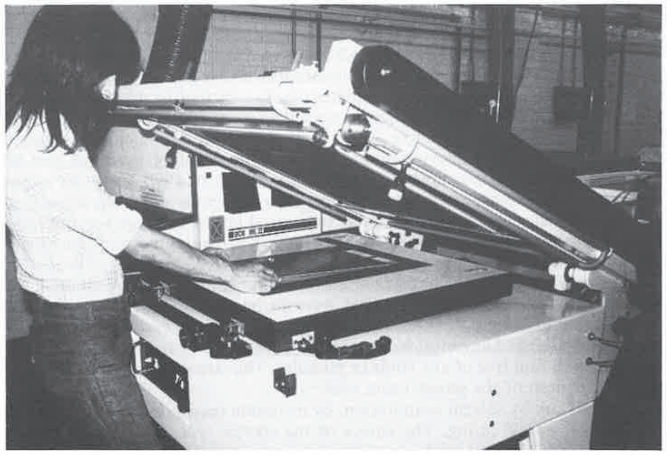


Fig. 5.7 Screen printing frame open with blank being located by edge guides.

squeegee. It will deposit a thick film of ink, but will have limited edge definition because of the thickness of the threads. A finer screen will yield better definition but will deposit a thinner layer of ink. The long-term stability depends on setting and holding the correct tension.

The actual stencil which carries the pattern is in the form of a photo-sensitive emulsion. This may be applied to the screen as a liquid coating which can be exposed and developed on the mesh after it has dried. Stencils made from such liquid emulsions are called direct stencils. They offer the advantages of high stability and durability since the image is formed right through the screen.

Indirect stencils consist of a photo-sensitive emulsion on a backing sheet of clear polyester film. They are exposed in normal printing frames, the unexposed emulsion is washed off the backing film, the emulsion is then applied to the screen and when it is dry the backing film is removed. They are easier to expose than direct stencils, but are not as stable. Direct/indirect stencils are similar to indirect stencils but the emulsion is applied to the screen prior to exposing and developing.

The definition achievable from a given screen depends, among other factors, on the angle at which the pattern is set on the screen. The emulsion will bridge over the holes in the mesh to give angled edges to the pattern but the edge will be interrupted by the threads of the mesh. If the pattern is laid with its major lines parallel to the threads edge definition could be in error by the diameter of the thread. At small angles a straight edge tends to break down to a long sawtooth pattern as successive threads are crossed and this effect diminishes as the angle of the pattern to the mesh approaches 22 degrees.

The screen must be properly set in the machine, with the right clearance

between the screen and the workpiece. If this clearance is too great excessive distortion will occur under the pressure of the squeegee, while if it is too small the screen will not pull cleanly away from the deposited ink and flooding may occur. Some screen printing machines incorporate a mechanism to lift the end of the frame after the squeegee has completed about a third of its travel towards the hinge, thus increasing the clearance between the screen and the printed ink.

The stiffness of the squeegee blade, and the depth at which it is set are important factors in achieving good prints. Screen printing machines can be operated manually, but most of the machines used in PCB work are semi-automatic with mechanically driven squeegees and cam operated lift-off to increase clearance. Blanks are manually positioned in the frame. A few firms use fully automatic printers in which the workpieces are automatically fed in and positioned.

The inks used in screen printing PCBs play a critical role in determining the ultimate quality of the product. They have to be resistant to commonly used etchants and to plating and cleaning solutions, but must be easy to remove after etching or plating. They must be capable of wetting the board surface and must dry to leave a film free of any voids or pinholes. The viscosity must be adjustable to suit the mesh of the screen being used.

Inks may dry by solvent evaporation, by oxidation on exposure to air, by oven drying, or by U-V curing. The choice of the correct type of ink to be used is important, and must be made carefully. Screen printing is a highly skilled trade, and a trained operator can produce excellent results on boards with features down to about 0·15 mm. However, if adequately skilled staff are not available, boards which might have been screen printed at lower cost may have to be printed photo-mechanically.

5.4 ETCHING[22,23,24]

Copper is a fairly easy metal to etch as it is attacked by a number of solutions. However, in practical PCB work the choice is limited by the type of etch resist used, which may be a film or ink, or it may be a metal as described in Chapter 6, and by the need to control the effluent to avoid environmental pollution. Also the etching must be kept under control. The unwanted copper must be removed without too much of the sides of tracks being etched away from under the resist. This sideways attack by the etchant is to some extent inevitable, but it must be controlled. The ratio of sideways attack to the thickness of metal being removed is called the etch factor, and can be taken as a measure of the quality of the etching. On boards with fairly coarse track patterns the filmwork can be made with all features over width by an appropriate etch allowance, but on precision boards this can leave inadequate gaps on the films so the etch factor must be kept down.

5.4.1 Etchants

The classic etchant in PCB work is ferric chloride, which offers users the advantages of speed, versatility, and low cost. The rate of etching increases with temperature, and ferric chloride is normally used at 40°C. Undercut can be controlled by the addition of thiourea and by using spray etchers which limit the direction of attack. The two major disadvantages of ferric chloride as an etchant

for PCBs are that it can not be continuously regenerated and it attacks the metals commonly used as resists in through-hole plated work.

Ammonium persulphate superseded ferric chloride for etching through-hole plated boards, and is still in use in many plants. In others it has been superseded in turn by ammoniacal chlorites. One possible problem in the use of such ammoniacal etchants is the emission of fumes and the plant needs to be well ventilated.

Chrome/sulphuric acid can be used as an effective etchant for copper, but poses problems in effluent disposal. Interest is now being concentrated on systems which use a continuously regeneratable etchant. Cupric chloride, which can be regenerated by the action of a mixture of hydrogen peroxide and hydrochloric acid, etches at a constant rate and yields a residue of powdered copper which can be re-used in plating baths. Another regeneration system uses hydrogen peroxide and sulphuric acid and yields crystals of copper sulphate. Commercially available versions of regeneration systems are expensive to install but can offer significant savings over a period as their by-products are usable in the plating shop and they avoid risks of under-etching caused by the use of 'spent' etchant.

5.4.2 Etching Machines

At the simplest level a laboratory prototype board with a hand painted resist can be etched in a beaker of ferric chloride, or, for larger boards, a photographic developing dish. Hand agitation will ensure an adequate etch. Bath or tank etching on a commercial scale was replaced by spray etching which gives improved control of undercutting.

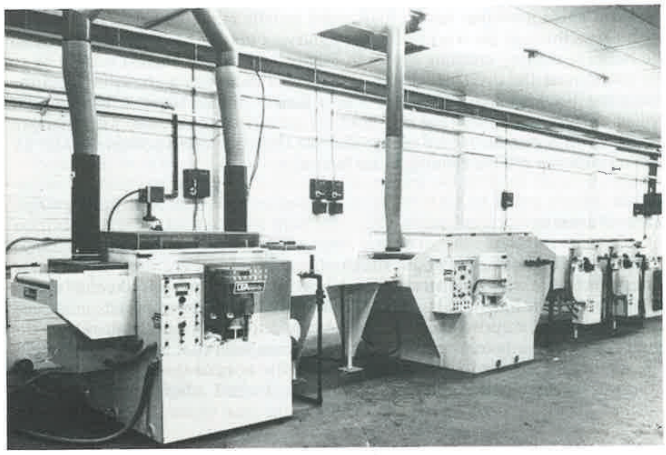


Fig. 5.8 Conveyorised etcher, ink wash-off and rinse.

The early batch spray etchers consisted of a vertical pipe with nozzles, in the centre of a square or hexagonal chamber. Boards are placed round the sides of the chamber, etchant, usually ferric chloride, is pumped through the nozzles, and the central pipe is rotated to spray the etchant over the boards. These 'whirl' etchers can still be found in use in plants making single-sided boards, but for the bulk of PCB work horizontal conveyorised etchers with multiple rows of jets which spray both sides of the blank are commonly used. Such conveyorised etchers are usually supplied with a common conveyor to a resist stripping spray or spray and brush machine and appropriate spray rinse stages as shown in Figure 5.8.

5.5 HOLE GENERATION[25,26,27]

5.5.1 Sight Drilling

The holes in non through-plated boards can be produced quite adequately by sight drilling on a high speed bench mounted pillar drill. The pads on such boards usually have a small open centre and with a little care these can be used to guide the tip of the drill. In laboratories or other areas where only small batches of boards are produced there is little point in investing in more sophisticated equipment. If epoxy glass boards are to be drilled it is advisable to fit a simple dust extraction system to the machine. Ordinary high speed drills can be used, but they must not be expected to have a long life, especially in smaller sizes.

Blanks for through-hole plated boards or bonded multilayers can be sight drilled by printing a coloured pad pattern in photo-resist on one face. As there is no guidance for the drill tip this type of work would not normally be done on a pillar drill, but on one of the special machines designed for PCB work such as the Posalux. These machines have a high speed spindle mounted below the table so the bit rises through the workpiece. A magnifying optical head similar to that on the NC tape making machine shown in Figure 5.11 can be aligned accurately to the position of the drill, which can not be operated until a clamp to hold the board has been lowered. Stacks of 3 or 4 boards pinned together can be sight drilled on such machines. The bits used are usually solid carbide, with enlarged shanks to fit a standard sized collet, and the drills are often powered by an air turbine with the spindle running in air bearings.

5.5.2 Punching

When large quantities of non through-plated boards are required the holes are often punched in a single operation on a heavy duty power press like that shown in Figure 5.9. The individual tools for such punches must be accurately made and can be expensive, but two stage tools can be made to blank out the profile on the same stroke as the hole piercing and on long runs such tooling is cost effective. The quality of punched holes is not suitable for normal through-hole plating, although special plating processes have been devised which can be used on punched holes.

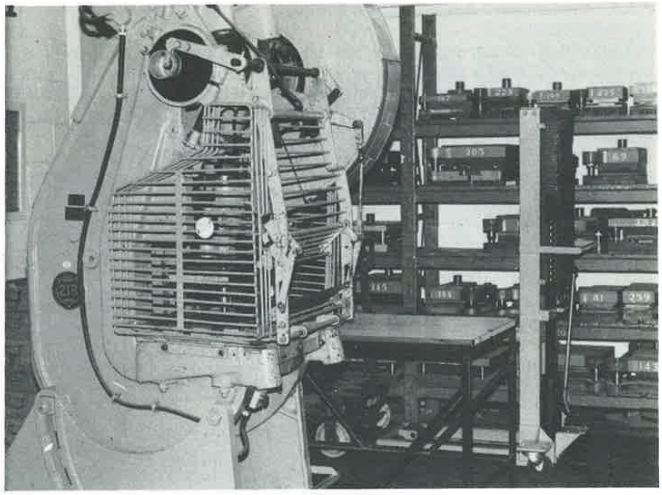


Fig. 5.9 Piercing and blanking press.

5.5.3 Jig Drilling

Sight drilling is practicable only on very small batches. Until the widespread adoption of numerically controlled machines the standard method for drilling production batches of boards was jig drilling on a Posalux or similar machine. The jig is a plate, usually of acrylic resin, fitted with dowel pins to locate the stack of boards to be drilled. The upper face of the jig is drilled with conical dimples to define the hole positions, this drilling being carried out with the aid of the optical sighting head. The jig is then pinned on top of the stack of board blanks, and the optical head is replaced on the machine with a stylus head which has a movable stylus which drops down into the dimples on the jig. The machine is interlocked so the drill can rise only when the stylus is properly engaged in a dimple. With a carefully made jig and the stylus properly set in true alignment with the bit such jig drilling can be accurate to within 0·05 mm.

On plated-through hole boards where the films are manually aligned to the holes some degree of misalignment between the stylus and the drill can be tolerated, and may even go undetected, if the work is always kept the same way round on the table, but if the work is allowed to rotate as it is moved from hole position to hole position such misalignment will have a serious effect.

5.5.4 Numerically Controlled (NC) Drilling

The need for greater capacity without large numbers of skilled operators led to the widespread adoption of NC drilling in the PCB industry. Some boards may have 5 or 6 thousand holes, often in repetitive patterns, and drilling such boards accurately in large quantities would clearly place an intolerable strain on any jig drilling facility.

One major advantage of NC or CNC drilling is that drill tapes are often available as a by-product of computer aided design, although when board manufacture is contracted out there can be problems over tape format, step and repeat work, and the provision of test coupons and tooling holes.

When drill tapes are not available from a CAD system they are made on an optical head digitiser coupled to a tape punch as shown in Figures 5.10 and 5.11. Modern drills do not require instructions for every individual hole, but can drill regular patterns from the location of a master hole and instructions on the size and number of steps required.

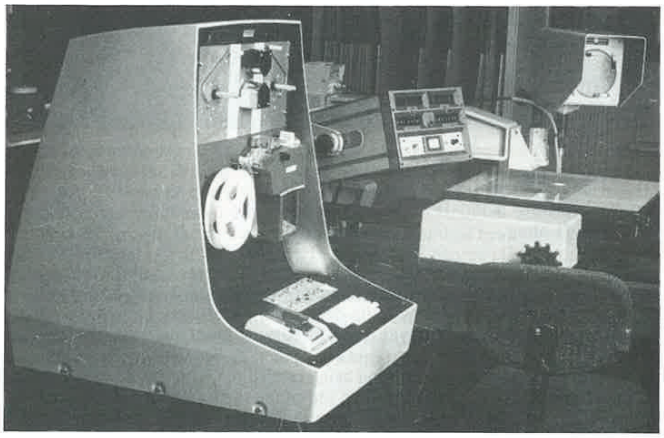


Fig. 5.10 NC drill tape making equipment.

When a board has been designed with all holes on a regular grid the digitising of the hole pattern can be further simplified by using a grid round-off facility which automatically instructs the drill to work to the nearest grid intersection to the measured point.

The most commonly used NC drills have 5 heads, of which the outer four are normally used. When the boards to be drilled are larger than the spacing between the heads, the two outer and the centre head are used, and even larger boards can reduce the capacity to only one outer and a centre spindle. Figure 5.12 shows the arrangement of such a machine. The illustration actually shows a router, but the

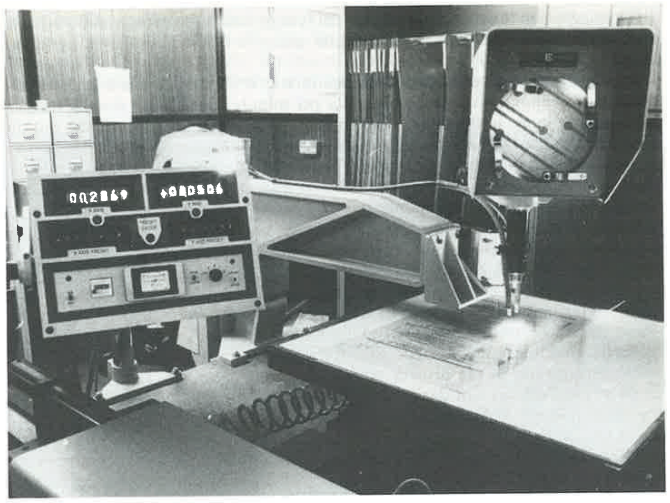


Fig. 5.11 Close-up of optical head and read-out.

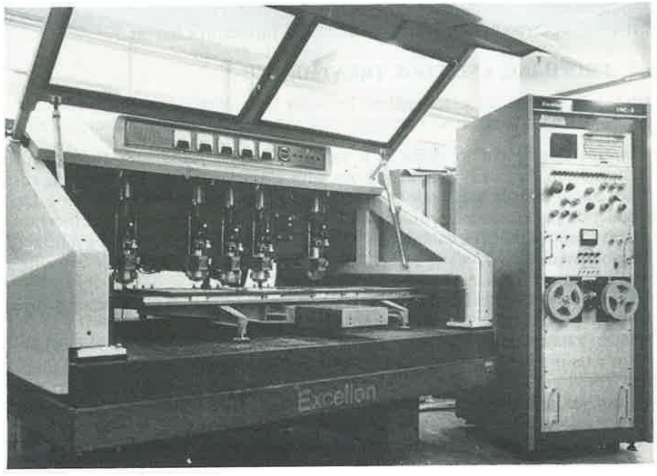


Fig. 5.12 Multi-head NC routing machine.

differences are in the bearings and control system, and it is virtually impossible to distinguish between a drill, a drill/router and a router without looking at the label.

The machines are equipped with automatic drill change facilities and are capable of drilling up to about 120 holes per minute. Boards are usually stacked 3 or 4 high, pinned to the table. The tables are positioned very accurately, but the actual accuracy of drilling depends on the size of the hole, the number of boards in the stack, and the feed and hit rates. The drills can be deflected by the weave of the base laminate, and the bottom board of a stack may be up to 0·05 mm less accurately drilled than the top board.

Stacks of boards are made up with backing sheet, often of an aluminium clad non-woven base laminate, either side to minimise the formation of entry and exit burrs. The drills, which are solid carbide, are run at speeds up to 80,000 rpm, at feed rates between 0·05 and 0·2 mm per revolution. The usable life of drills before re-grinding varies from plant to plant, from 2000 to about 5000 holes. Some firms use only new drills on multilayer boards, re-ground drills being restricted to double-sided boards, and non-plated work after a second or third re-grind. Sharpening of the drills is critical, and it is important to ensure that the lands of the drills are not damaged in any way. At high feed rates the quality of the holes, and hence the quality of any plating to be deposited therein, depends as much on the sides of the drill as it does on the tip.

Positional tolerances of NC drilled holes usually lie in the range of plus or minus 0·05 mm to 0·07 mm on 3 high stacks of boards. Tighter tolerances can be achieved by cutting down the stack height and the hit rate, but these add to the cost of drilling and the designer of cost-effective boards should avoid any need for tight tolerances. The actual number of holes drilled has less influence on the cost of drilling than the number of different hole sizes used. The quality of drilling and its influence on through-hole plating is discussed further in Chapter 7.

5.6 PROFILING AND EDGE TREATMENT[8]

5.6.1 Profiling

Boards can be cut to shape in a variety of ways. As already mentioned in Section 5.5.2 they can be punched out with a single stroke of a blanking tool. When small boards are made with a number stepped and repeated on a large production panel, special profiling tools may be made which return the individual boards to the panel after punching so they can be easily handled during automatic component insertion, wave soldering and testing, after which they are simply snapped out of the panel.

Rectangular boards can be guillotined to shape or cut on circular saws, using diamond studded disc cutters for epoxy glass laminates.

Most epoxy glass boards are cut to shape by routing. This can be done with the aid of a template pinned to a stack of boards, the template being pushed against a high speed cutter in a fixed position spindle (see Figure 5.13). Such freehand routing has largely been superseded by NC routing, and some firms have found it more economical to profile simple boards on an NC router than to buy punch tools. The versatility of NC drill/routers makes their use attractive even in fairly small facilities.

The accuracy of NC routed edges relative to datum tooling holes within the board outline can be improved by making a first cut 0·2 mm outside the required edge followed by a final cut. Routing leaves radii on any internal corners and the designer should always specify the widest tolerance possible on any such radii to leave the manufacturer a free choice of cutter diameter. A specification of zero to 3 mm radius is suitable.

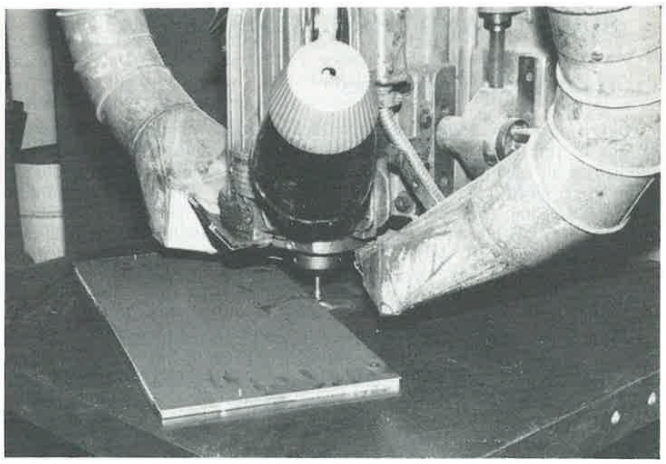


Fig. 5.13 High speed template router.

Any polarising or locating slots required in the board can be put in as the profile is routed and the diameter of cutter used may be dictated by the width of such slots.

5.6.2 Chamfering

A chamfer is often specified on edge-connector edges of boards. Such chamfers can be cut with pairs of rotating disc cutters; on an angled bed linishing machine; or by drawing the board through a vee shaped carbide cutter. The basic process of putting a chamfer on a board is not difficult, but meeting tight tolerances on the depth of a chamfer can pose problems, especially if the drawings call for the end of the chamfer to be at a tighter tolerance to the features on the board than the tolerance on the edge to which the chamfer is to be applied.

Surface finishes for boards are discussed in Chapter 10.

Chapter 6

SURFACE AND THROUGH-HOLE PLATING

6.1 EDGE CONNECTOR FINGERS[5]

While copper is an excellent metal for tracks and pads on printed circuit boards its lack of resistance to surface corrosion means it must be overplated with another metal, usually gold, to form reliable, low resistance, connector fingers. Some users specify that connectors shall be plated with sufficient gold to guarantee freedom from pores through which the base copper can corrode, a thickness of 0·005 mm often being used, while others specify thinner gold over an underplate of nickel. With modern plating methods, which include features such as the use of pulsed currents or the use of ultrasonic agitation to dislodge hydrogen as it is formed, adequate pore-free gold can be plated at 2·5 μm thickness.

Gold is an extremely difficult metal to etch, which makes it suitable for use as a mask for etching copper. Connector fingers can be formed by plating the gold direct onto the sheet copper of the blank, using a negative mask of resist to define the shape of the fingers. This resist is cleaned off, the track pattern is defined with a positive resist overlapping the gold, and the blank is etched. The above simple process leaves the sides of the fingers with unprotected copper which can corrode sufficiently to form a severe reliability hazard in adverse environments.

This failing of simply made connector fingers led to the use of edge connectors being banned in some equipments, most notably those for American military use. However, it is possible to make thoroughly reliable fingers by etching the pattern first then plating so the gold is applied over the sides of the fingers as well as on the faces.

Electroplating gold on the etched pattern of fingers requires some means of conducting the plating current to the individual fingers. This can be achieved by clamping the board above the fingers between compliant conducting bars so the current is conducted down the tracks to the contact fingers.



Fig. 6.1 Edge connector fingers connected to plating bar in scrap border of production panel.

This method of plating ensures that only the fingers which are actually used are plated with gold, which may result in a substantial economy. However, corrosion on unused fingers can be a reliability hazard, so it is more usual to plate the fingers with current supplied through a plating bar on the scrap material beyond

the board edge, with each finger joined to the bar by a track about 0·2 mm wide as shown in Figure 6.1. The plating bar is positioned such that the routing cutter removes only the narrow connecting tracks, and the gold on the plating bar can be recovered.

6.2 THROUGH-HOLE PLATING

Electroplating copper through the holes in a board requires the presence of metal over the surfaces of the board to conduct the plating current to all holes. To achieve this the order of the stages of production is changed from that used for plain boards, and after cutting and punching for tooling holes the panels go direct to drilling as shown by the solid line on Figure 5.1. After de-burring and cleaning the blanks go through an electroless plate which deposits a thin layer of copper through the holes and over the surface of the copper cladding on the laminate. The holes can then be electroplated and the track patterns printed and etched.

After electroless plating (step 10 on Figure 5.1) there are two major variants in the method of manufacture. The boards may be electroplated with copper over the whole of their surfaces and through the holes (the heavy dashed line on Figure 5.1 from step 10 through steps 11 to 13 and then to step 17), or the surfaces may be plated only in the areas which will form the pads and tracks on the finished board. This process, known as pattern plating, is shown by the solid line on Figure 5.1, from step 10 through 14 to 17 and then back to 13. Steps 18 and 19, stripping tin-lead from the connector fingers and gold plating, are omitted if gold plated edge connector fingers are not required. It must be noted that Figure 5.1 is greatly simplified and shows only one possible way of making panel or pattern plated boards. Variants of the processes are dealt with in more detail in later sections of this chapter.

Panel plating involves etching through the original copper cladding and the plated copper layer so it is not so suitable for fine track patterns as pattern plating.

6.2.1 Drilling and Cleaning[16,25,26]

For through-hole plating to be successful the holes must be cleanly drilled and they should not be too small relative to the thickness of the board. Holes down to one third of the laminate thickness are generally acceptable for plating, but the ratio does depend on the hole diameter and with larger holes a 5 to 1 ratio can be plated without difficulty. Any burrs left after drilling are removed by fine linishing and the panels are then usually scrubbed and put through a fine vapour blast to remove any debris from the holes. They are next given a mild etch and chemical cleaning to ensure that the electroless copper will adhere to copper cladding and to the walls of the holes. Figure 6.2 shows a section of a drilled blank.

6.2.2 Electroless Plating[28,29]

Electroless deposition of copper is a process wherein copper ions in aqueous solution are reduced to the metallic state by a suitable reducing agent which is simultaneously oxidised in the course of the reaction. The solution is carefully

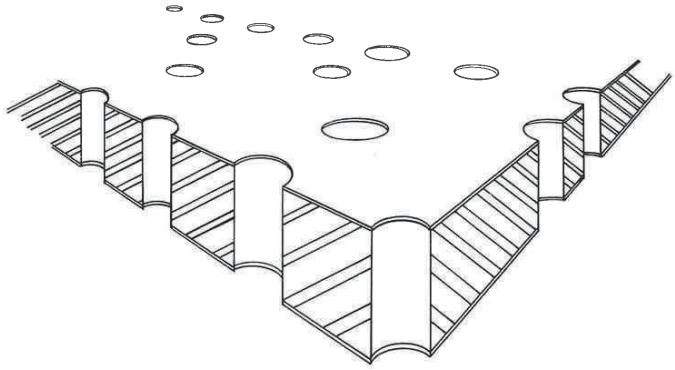


Fig. 6.2 Drilled blank for through-plated hole board.

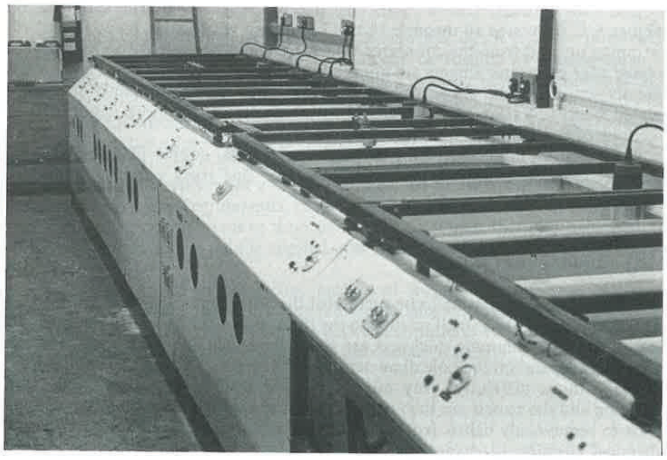


Fig. 6.3 Small manual electroless copper line.

formulated and conditions are chosen so that random reduction does not occur in the bulk of the solution but only on specially prepared catalytic surfaces. Copper itself is a catalyst for the particular reducing system employed so that the reaction proceeds autocatalytically, the thickness of the copper deposit increasing with time.

The normal practice for printed circuit boards is to immerse the boards first in a solution of stannous chloride and palladium chloride in hydrochloric acid. This solution is adsorbed onto the epoxy glass surfaces of the holes and on the copper clad faces of the board. Immersion in an accelerator then removes unwanted tin compounds, and leaves the palladium, or more correctly the hydrogen associated with the palladium, to act as the catalyst to start the copper deposition. The baths containing the electroless solutions are heated to increase the deposition rate. The boards are usually mechanically agitated to ensure a flow of solution through the holes. Small lines like that shown in Figure 6.3 can be operated manually with only the agitation mechanised, but full scale production justifies the use of automated conveyors to move the racks of boards from tank to tank. Solutions are continuously monitored and active chemicals are replenished as they are used.

Cleanliness is important in electroless plating because any particles which fall into the electroless bath can initiate deposition, as can any drag-over from the palladium-tin stage. Plating jigs or frames must be kept clean and free from growths of copper or palladium which might become detached and contaminate a bath.

Over the years since boards were first made with plated-through holes there have been significant improvements in electroless plating solutions, and these improvements can be expected to continue. Considerable work has been carried out on the replacement of palladium seeding solutions with copper based solutions, and on electroless baths which deposit more copper on the exposed epoxy resin in the holes than on the surface copper of clad laminate. Electroless copper is used mainly as a base for electroplated coppers, but high build electroless copper can be used to deposit the full thickness of copper required for tracks in situations where the supply of plating current is difficult to arrange. Various such fully additive processes have been available for some dozen or more years but they have shown no advantages over the conventional processing, either in line width achievable or in cost and despite considerable publicity the processes have not been widely adopted.

6.2.3 Panel Plating

Once the electroless copper has been deposited on the walls of the holes to obtain continuity over all surfaces the boards can be electroplated with copper (Figure 6.4). This can be undertaken in small baths or on a large automated line as shown in Figure 6.5. An acid copper sulphate solution with brighteners and other additives to improve throwing power was used at first, but in the later 1960s was superseded by pyrophosphate coppers. Continual improvements in the commercially available solutions have led to a return to the use of acid sulphate copper.

The important features of a copper plating bath for printed circuit boards are that it should yield a uniform fine-grained, ductile copper and that it should have good throwing power to deposit the maximum thickness of copper on the walls of the holes (see Figure 6.4). Agitation of the boards during electroplating maintains a flow of solution through the holes. The plating jigs are usually carried on a pair of oscillating rails driven by a crank on a geared electric motor. A small amount of play in the bearings of the connecting link can give the jigs a jerk at the end of their travel which helps to dislodge any small bubbles of hydrogen which might form in the holes.

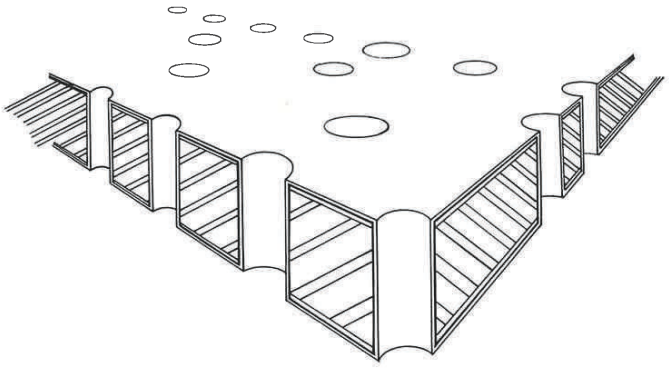


Fig. 6.4 Drilled blank after panel plating.

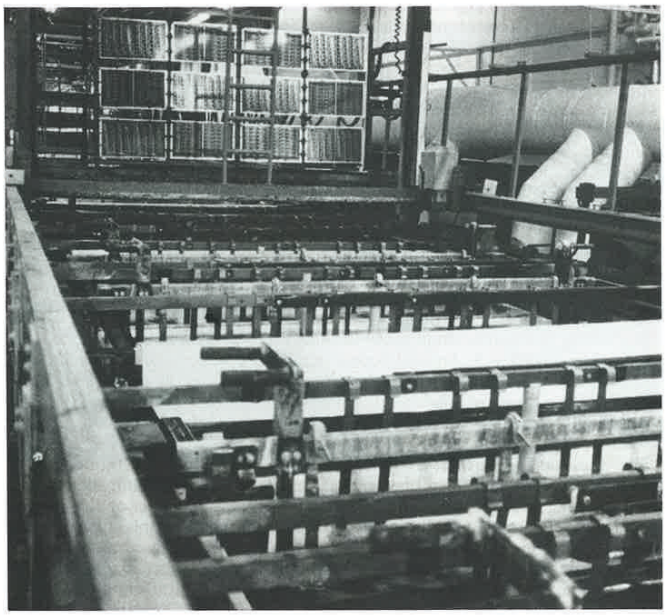


Fig. 6.5 Large automatic electroplating line.

A well controlled bath yields a reasonably uniform thickness of copper over the faces of the board, with a slightly lower thickness down the holes. The thickness, both on the surfaces and through the holes, will increase a little towards the edges of the board.

Processing the boards after panel plating with copper can be carried out in a variety of ways depending on the finish required on tracks and pads. For a plain copper finish the boards can be coated with a dry film resist which is exposed and developed to leave a positive image with the resist tenting over the holes as shown in Figure 6.6. The boards are then etched, the tented resist protecting the barrels of the holes from attack (see Figure 6.7). For this process to be satisfactory the

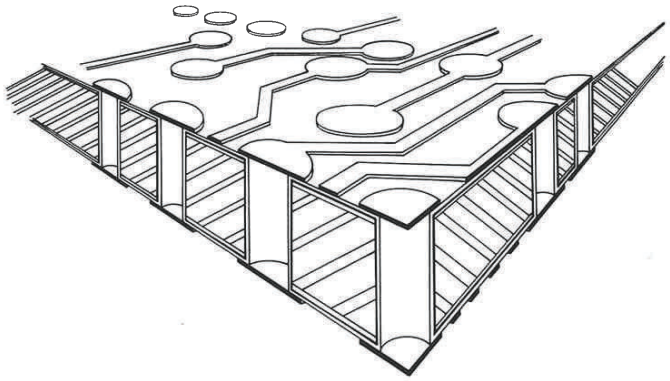


Fig. 6.6 Plated blank with resist tented over holes.

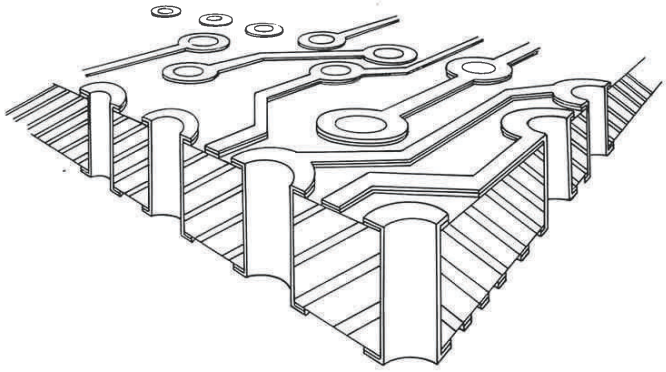


Fig. 6.7 Panel plated copper board after etching.

board must be designed with pads large enough to ensure an adequate land round the hole for the resist to adhere and form a secure cover, allowing for worst case drilling and film registration tolerances.

When the pads are small relative to the hole sizes, panel plated boards are better etched using tin-lead plating as the etching mask as described in the next section.

Panel plating is a wasteful process because most of the copper plated onto the board is etched off again. Plating is easy to control, but the double thickness of copper to be etched can cause undercut problems and panel plating is not generally used for fine line double-sided boards.

6.2.4 Pattern Plating

The essential feature of pattern plating is that only the tracks and pads are overplated with copper when the barrels of the holes are plated so that only the original copper cladding has to be etched away from the clear areas.

In pattern plating the resist is applied directly after electroless plating (see Figure 6.8). Usually a dry film resist is used as its thickness leaves sidewalls which confine the plating and prevent growth of feature sizes.

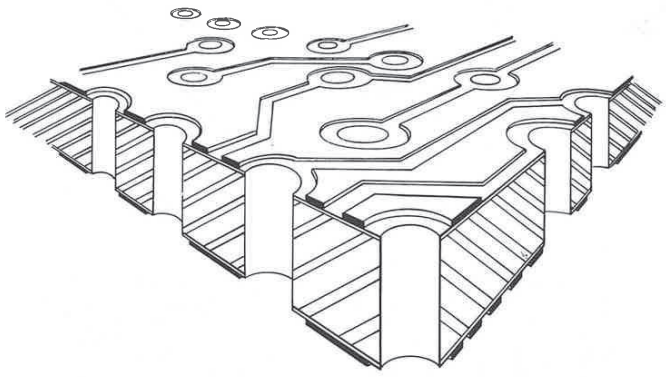


Fig. 6.8 Resist applied and developed after electroless copper deposition.

The panels are chemically clean after the deposition of electroless copper, but all traces of moisture must be removed. This is best achieved by immersion in a liquid fluorocarbon such as Freon* or Arklone** which evaporates to leave a clean dry surface which can be totally wetted by the resist. There should be no opportunity for any dust or other contaminant to settle on the surface of the laminate between its withdrawal from the freon tank and the application of the resist. Resist application should be planned as the last stage of electroless plating

* Trademark DuPont
** Trademark ICI

rather than as the first stage of printing and any 'hold time' in the production schedule should occur after the resist has been applied.

Prior to the adoption of the high build autocatalytic electroless coppers it was customary to give the panels a flash of electroplated copper, (usually referred to as 'flash plating' or occasionally as a 'strike plating'), to protect the very thin electroless copper deposit before applying the resist, but with modern solutions this is not necessary.

The resist is exposed and developed to leave a negative image. That is, the resist is left on the areas which will be clear of copper on the finished boards. After inspection and any touching up necessary (or possible) the boards are electroplated (see Figure 6.9). When most boards had gaps around 0·3 to 0·5 mm wide any printing defects or pinholes in the resist could be spotted in with a fine brush or pen, but on boards with tracks or gaps down to 0·15 or 0·10 mm such spotting in is too difficult and quality control must be concentrated on getting the print right the first time.

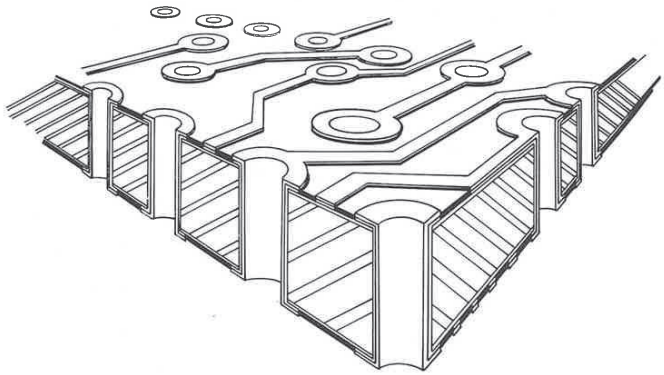


Fig. 6.9 Copper and tin-lead plated in areas clear of resist.

Copper plating is carried out in the same way as for panel plating, except that the actual area to be plated must be measured beforehand, usually at the planning or the film preparation stage, and the plating current is adjusted to suit the area in order that the optimum current density of the solution is met. The scrap borders of blanks for pattern plating are provided with 'robber bars'—areas of copper which divert some of the plating current and prevent burning or over-plating of features near the edges of the board.

When a board is to be panel plated the quality and thickness of the plating are not affected by the pattern of tracking, but on pattern plated boards there is a close relationship. For the best results the pads and tracking should be as uniformly distributed as possible over the surfaces of the board. Isolated tracks or pads should be avoided or, if the design is such that there must be a feature on its own, should be surrounded by an area of cross hatching to balance the plating current. Any area of more than a few square cm which carries no tracking can be

filled with cross hatching of perhaps 0·5 mm tracking on about 4 mm pitch. Abrupt changes of track or gap widths in the pattern, such as running a very narrow track close beside a broad power rail, should be avoided. One allowable exception to the general uniformity of tracking is the running of broad power or earth bus bars up the edges of the board, where they will help the external robber bars.

After copper plating the board is plated with tin-lead. On an automated line like that shown in Figure 6.5 boards go straight into the tin-lead bath after rinsing following copper plating so there can be no contamination of the copper and excellent solderability of the finished board is assured. Like the copper bath, the tin-lead solution has brighteners and other additives to ensure uniformity of plating and good throwing power. The anodes are contained in cloth bags to prevent any shedding of particles into the bath. Tin-lead is very susceptible to burning, and avoidance of isolated features in the design is important.

Tin-lead may be pattern plated onto a blank which has been panel plated in cases where the use of a tented dry film resist for etching is undesirable or impractical. After tin-lead plating the resist is stripped off (Figure 6.10) and the original copper cladding is etched (see Figure 6.11) with the tin-lead serving as the mask for the etchant.

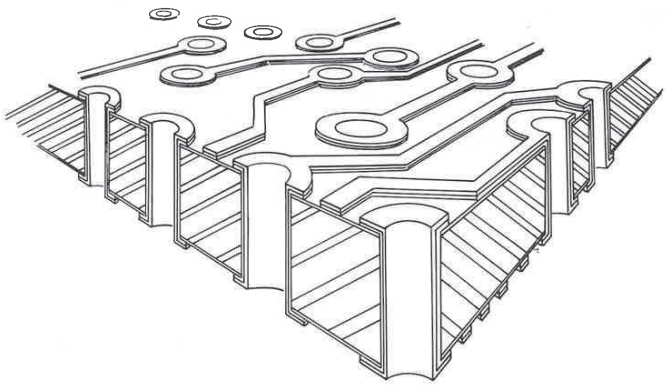


Fig. 6.10 Resist cleaned off pattern plated board.

Metals other than tin-lead can be used to protect the copper during etching, but, since most boards are ultimately soldered, tin-lead is an obvious choice. If gold plated edge connector fingers are required on a pattern plated board the tin-lead is chemically stripped off the fingers with adhesive tape masks protecting the remainder of the tracking in the area adjacent to the connector.

Variants of these processes are discussed in Chapter 9.

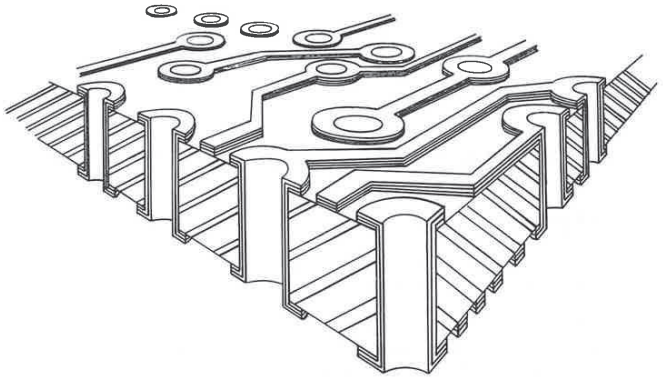


Fig. 6.11 Pattern plated board after etching.

Chapter 7

BASIC MULTILAYER BOARD MANUFACTURE

7.1 GENERAL PRINCIPLES[8]

Multilayer board manufacture combines the elements of both non plated and through-hole plated board manufacture, as shown by the chain dotted line on Figure 5.1. Pieces of thin laminate are cut, cleaned, punched with tooling holes, then printed and etched. The pieces of laminate which will form the outer layers are etched on their inner faces only, the copper on the outer faces being left intact. The etched layers are cleaned and the copper is oxidised. They are then stacked on tooling pins with intermediate layers of prepreg and bonded under heat and pressure.

After bonding, the boards are drilled, and thereafter they follow the same nominal process steps as through-hole plated double-sided boards, with either pattern or panel plating and appropriate printing and etching. While the basic process steps appear the same and can be described by the same box on a simplified flow chart like Figure 5.1 there are many points where multilayer manufacture differs from that of conventional boards. A plant designed and built to make multilayers can be used to produce plain or plated-through hole double-sided boards, but the addition of a bonding press will not convert a double-sided plant to a good multilayer facility.

7.2 INNER LAYER PREPARATION

7.2.1 Tooling

When double-sided boards are manufactured there is always a possibility of adjusting the printing to suit the drilling, or *vice versa*, if it is found that inadequate tooling is resulting in misregistration. The usual cost of any such errors in registration is that the print will have to be cleaned off and re-done. Some manufacturers do not rely on tooling to achieve registration on double-sided boards, but use it merely to maintain accuracy which has initially been achieved by 'eyeballing' the printing film or screen to the drilled blank. There are no such safeguards in multilayer manufacture. If the tooling is not right the separate layers will not register and the result will be expensive scrap.

As well as ensuring layer to layer registration the tooling must be able to accommodate the material movement of the inner layers which occurs during bonding. The system must be easy to use, should make it impossible to stack layers inverted or rotated, and should be applicable to any size of blank.

Six possible tooling systems are shown in Figure 7.1 and the merits of each can be examined.

A, B and C are along one edge of the panel, so can be punched with a fairly

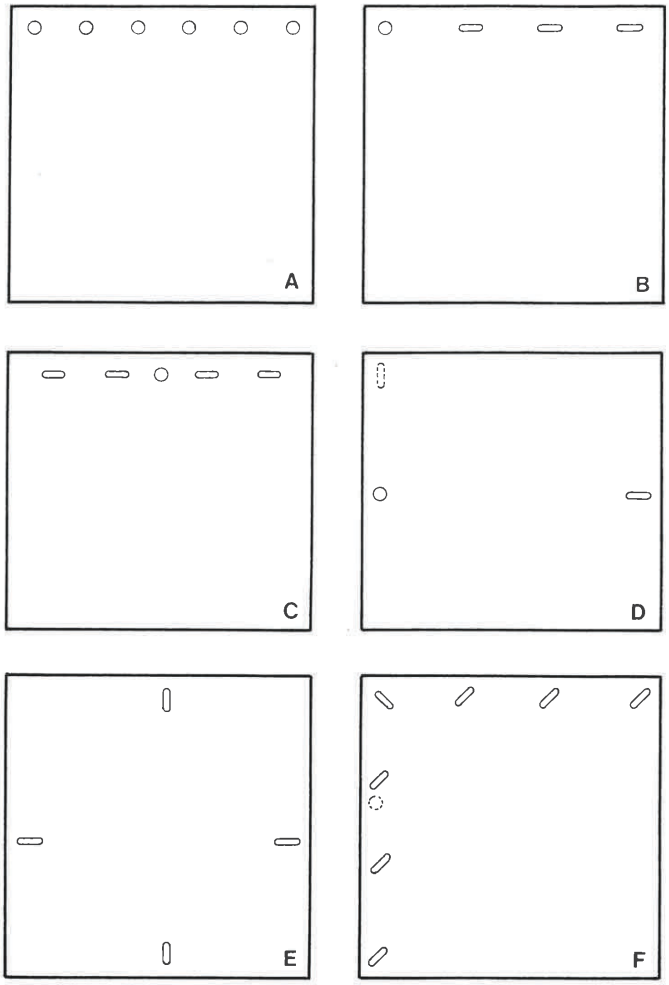


Fig. 7.1 Multilayer board tooling systems.

simple tool. D and E require a much larger punch tool, with movable heads to accommodate various panel sizes, and F is independent of board size but requires a large tool with a facility to select only one pair of heads and the 'master' head in the corner.

System A has no provision for material movement and it permits a layer to be reversed, so it can be regarded as unsuitable for multilayer work. System B can accommodate material movement in one axis, but as the master datum hole is right at one end of the board the full effect of any movement will be observed at the other end. It allows reversal of a layer, so it too can be discounted. C copes with material movement in one axis, with the datum hole in the middle of one side, and, provided the slots are spaced on different centres either side of the pin, it prevents any reversal of a layer. As it requires only a simple punch, and can be applied to any size of blank this is a system which is usable for multilayer work.

System D accommodates material movement but it requires a large punch tool capable of spanning half a blank or film and the tool should be equipped with multiple alternative positions for the slot if more than one blank size is to be handled. A basic single hole and slot system will not prevent layer reversal, but it can be equipped with an extra slot as shown dotted. It should be noted that if true kinematic principles are to be observed this slot must be on the same centreline as the datum pin. With this second slot system D offers the same performance as system C, but at the expense of a much more elaborate punch.

System E accommodates material movement about a datum in the centre of the panel, and provided the two sets of slots are slightly asymmetric it prevents layer reversal or rotation. It requires large, complex tooling, with either multiple slot positions or preferably a different tool for each blank size used. When the range of panel sizes to be used is limited this is an excellent system.

System F also gives a central datum, and as it is applied to only two sides of the panel a single tool can be used for any size. However, it may prove difficult to stop operators trying to use more than three pins, or to persuade them that it is essential to use an opposing pair. Reversal of a layer is possible if only the basic locating pins are used, but it can be prevented by the use of an extra pin in an oversize hole as shown dotted.

All systems except D and E can have any number of slots necessary to fit various panel sizes, and all except F can use more than the two or three essential pins provided the slots are truly in line.

Aligning and punching films for systems D and E can be difficult, and these systems are best used when filmwork can be generated relative to pre-punched tooling holes.

It can be seen that no system is ideal. For an in-house plant handling only one or two blank sizes and with full control over filmwork generation system E will give the best results, but for a general purpose jobbing shop C offers the fewest disadvantages, with F as a possible alternative.

7.2.2 Film Preparation

The films for multilayer board production must be punched with tooling holes to match the holes in the laminate. Layer to layer registration is best ensured by using specially made upward acting punches, on which a master negative for the film set can be aligned, taped down and punched. The positives for each layer can then be aligned accurately to the negative and punched in turn. This gives a last chance to check that the films provided are of adequate accuracy for board production.

If the films were not made in full collaboration with the board manufacturing facility it is likely that they will require a considerable amount of extra work. Size gauges, process monitor gauges and special test coupons may have to be added, as will special borders. Outer layers will usually require the addition of a cross-hatched border or robber bars, and it may be necessary to add layer numbers in the border if they were not included in the board area.

The scrap margins of inner layers are usually left with a border of copper a few cm wide, which contains the tooling holes and any test coupons required. This border helps to minimise material movement of the thin laminate during etching, and it also prevents excessive escape of resin from the prepreg during bonding. The border must be broken at intervals by gas vent slots which will allow entrapped air to escape when the panel is bonded. When standard panel sizes are used for manufacture it may be possible to add all the extra features by double exposure printing on dowel bars, with only the inner edge of the added borders having to be adjusted to suit the size of board being made.

7.2.3 Printing and Etching

Printing of multilayer inner layers is the same as for any other type of board except that it must be done on tooling pins to guarantee layer to layer registration. The pattern can be screen printed or a dry film or liquid photo-resist can be used. When a dry film resist is used the tooling holes should be punched after the resist has been laminated to the blank to avoid any risk of damage to the holes which could occur if the resist has to be removed from over holes punched before the resist is applied.

Etching is chemically the same as for other types of boards, but conveyorised machines made for use with double-sided boards may not produce good results on the very thin laminates sometimes necessary for inner layers. The thin material can be taped to a 1·6 mm thick leader board to ensure that it can follow the conveyor properly, but it is better to use a machine designed and built to handle thin or flexible laminates.

The use of a separate dedicated etch line for inner layers as shown in Figure 7.2 allows optimum mechanical handling of the thin material and it can also allow the use of a different etchant from that used in the line which etches the outer layers after tin-lead plating.

7.2.4 Oxidation

The surface of the copper-clad laminate after removal of the etch resist is not in the best condition for obtaining a strong bond to the epoxy resin of the prepreg. A suitable surface condition for bonding is achieved by oxidising the copper. The traditional method is by immersing the etched panel in a hot caustic bath, which produces a layer of black oxide. The thickness of the layer depends on the duration of immersion, and over oxidising results in a loose, powdery deposit which can be rubbed off the surface of the copper and yields low bond strengths. The immersion process does not lend itself readily to full automation with conveyorised etchers so the industry is turning to proprietary solutions which



Fig. 7.2 Inner layer etching line.

produce a bronze coloured oxide. These solutions are intended for use in conveyorised spray machines and they result in a more stable film with better adhesion to the copper, better abrasion resistance and a more even finish than is possible with the black oxide process. The growth of the bronze oxides is exponential with time so the process is easier to control, and higher bond strengths are achieved.

7.3 LAY UP

7.3.1 Press Plates and Packing

The boards are laid up on steel press plates which carry the tooling pins and help to ensure that the heat from the press plattens is uniformly distributed over the face of the boards. Depending on their thickness, the press plates can be allowed to overhang the plattens very slightly to bond maximum size boards. The press plates must be flat and their faces must be truly parallel and free of any dents or burrs. Any such defects in the faces of the press plates will be pressed into the outer surfaces of the finished boards. Aluminium press plates can be used. They are excellent as heat distributors and are easier to handle than steel plates but they are more liable to suffer surface damage and the tooling holes may soon wear.

The press plates are protected from the liquid resin by a release sheet of Tedlar or similar PTFE based film as shown in Figure 7.3. Depending on the type of board to be bonded, sheets of Kraft paper, blotting paper, or a sheet of a conformal material may be used between the press plate and the release sheet. Conformal sheet is a plastic film which softens at the bonding temperature and distributes the pressure uniformly over the whole area of the board.

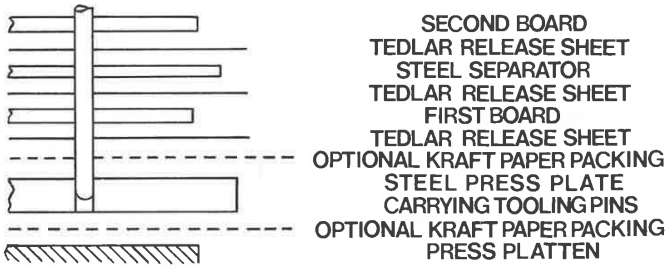


Fig. 7.3 Lay-up for bonding.

Kraft paper may also be used between the press plates and the plattens to control the rate of heating of the stack and to give better heat distribution.

The components of the first board are laid up in turn on the release sheet, followed by a second release sheet on top of which a second board can be laid. Some firms lay up successive boards with only single release sheets between them, but the use of double release sheets and a steel separator plate as shown in Figure 7.3 ensures more uniform distribution of heat and pressure to the inner boards of a stack. Up to five 1·6 mm thick boards can be bonded between a pair of press plates.

The lay-up work must be carried out in a clean environment as any dust included in the board could result in an internal layer to layer short circuit. Care must be taken to avoid any creases in prepreg, release or packing sheets as they are laid on the tooling pins.

Few multilayer boards have an absolutely uniform distribution of copper over all layers, and there will be inevitable 'low spots' where few of the layers have any copper. When a single board is bonded the resin from the more densely tracked areas can flow out into these low spots and equalise the pressure, but if several boards are being bonded in one stack without steel separator plates the pressure on the inner boards may not be uniform and voids may result in the low spots on one or more boards.

The resin content and flow rate of the prepreg used must be considered when deciding whether or not to use steel separator plates.

Conformal sheet is mainly used when a particular job requires the use of a low flow or no flow prepreg. Typical examples are the bonding of access hole multilayers and the bonding of copper or aluminium heat ladders to otherwise finished boards.

7.3.2 Prepreg and Build[8]

The build of a multilayer board can be varied to suit the electrical and mechanical requirements to be met. Figure 7.4 shows three possible builds for a four layer board.

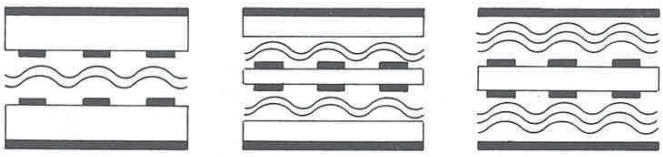


Fig. 7.4 Possible four layer builds.

The simplest build is two pieces of double-sided clad laminate with prepreg in the centre. It may not always be possible to achieve the desired electrical properties with this simple build, and in some cases it may be better to etch the two centre layers on the faces of a thin piece of laminate and use two bonds to single-sided outer laminates. If ordinary commercial single-clad laminate is used for the outer layers the unclad faces will need special treatment to ensure a good bond. Either specially made 'split laminate' which does not have a glazed surface should be used, or double-clad material with the copper etched off one face. A third optional build is to use a thin double-sided core and then build the rest of the board with an appropriate number of sheets of prepreg each side out to layers of oxidised copper foil as shown on the right of Figure 7.4.

These solid cored builds are most likely to be used when the highest possible capacitance is required between internal planes. For a good bond in a simple build, the minimum thickness of prepreg required would be 0·2 mm, but if a solid core is used it can be as thin as 0·05 mm.

Similar principles apply to the build of boards with more layers. A six layer board can be made from three pieces of rigid laminate with two bonds, or from four pieces of which two are single-sided or foil on prepreg.

When edge connectors are to be used the build of the board may have to be

adjusted to ensure that the thickness lies within the tolerance zone required by the connector. Calculations of finished thickness can be complicated by the copper thicknesses. A few isolated tracks on an inner layer will embed themselves in the prepreg, whereas power or earth planes will not, and must have their full thickness allowed. Most practical cases lie between these extremes and the designer must use his experience to determine the allowances to be made for the thickness of tracking. On most boards the area under the connector fingers is likely to be free of any tracking, which simplifies the calculations.

Prepreg is commonly available in thicknesses of 0·06 mm and 0·1 mm, with the latter being the generally preferred standard. There is always a possibility of there being a pinhole defect in a piece of prepreg, and any such defect could be the cause of an inter-layer short circuit, so at least two sheets of prepreg should always be specified in each bond.

With normal flow prepreg, the thickness of prepreg used should be twice the total thickness of copper on the faces being bonded to ensure that sufficient resin is present to flow between all copper features.

7.3.3 Layer Numbering

A well chosen layer numbering system can be a help to all involved in planning and manufacturing multilayer boards.

Checking finished boards is facilitated if each layer carries its number in a clear area in such a position that when the finished board is held up to a light all the layer numbers can be read in sequence. The generation of a film set with all emulsions on the correct side for printing is made easier if the layers are numbered such that all layers which are on the top of their piece of laminate are given odd numbers and those on the underside even. This leaves gaps in the sequence of numbers when builds such as the two right hand sketches of Figure 7.4 are used. Both of these would be numbered 1, 3, 4, 6. Confusion and fears of missing films can be avoided by adding X to the number for each single-sided piece of laminate, so the three laminates used in the centre of Figure 7.4 would be 1X; 34; X6. Consistent use of such a numbering scheme would immediately warn all concerned with the board build that the designer did not want the simplest 12; 34 build.

7.4 BONDING[30,31,32]

7.4.1 Presses

Multilayer boards are bonded by applying heat while the board is pressed, usually in an hydraulic press. Cycle times are long, so multi-daylight presses are generally used in all but development or prototype facilities.

A stack of press plates and five boards with separators is quite heavy, so some form of roller conveyor and loading lift is usually fitted, as shown in Figures 7.5 and 7.6.

The press plattens can be heated electrically, or by hot oil or steam, and they are usually cooled by water circulated through the plattens. It is essential that the faces of the plattens are flat and smooth and that they are truly parallel to one another at all points in their travel.

Electrically heated presses are the easiest to install provided there is sufficient

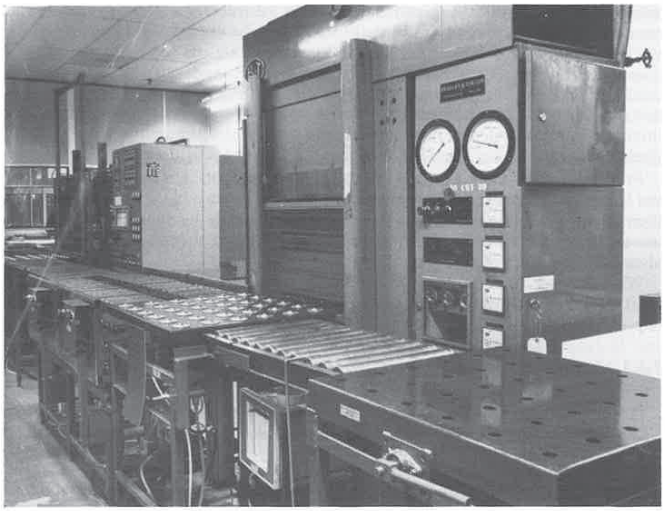


Fig. 7.5 Electrically heated multilayer presses.

power available on the site, but faster and more uniform heating is possible with steam or oil heated presses. These require substantial extra facilities, such as a boiler, and their pipework requires careful maintenance. A failure in a steam pipe can be very dangerous, and overall oil heated presses are generally regarded as the best for multilayer board work.

7.4.2 Press Cycles[8]

The press cycle used to bond a board depends on the resin content of the prepreg, its flow rate and gel time, the number of boards being bonded and the number of layers in each board, and also on the nature of the track patterns on the inner layers. Samples of the prepreg used for a given job should be tested before the boards are laid up, and the cycle times adjusted accordingly. The properties of prepreg change with storage and age, and test results measured when the material was bought may not be applicable.

Some manufacturers use a simple press cycle, starting with the press plattens cold, applying the bonding pressure of between 250 and 350 psi, and raising the temperature of the boards to about 180 degrees C. This heating may take up to about 40 minutes, depending on the type of press. During this time the B-staged resin in the prepreg melts as the temperature rises. The liquid resin flows and forces out any air from the bonds, then it begins to cure at the elevated temperature. The heating time should be adjusted so the resin has time to flow

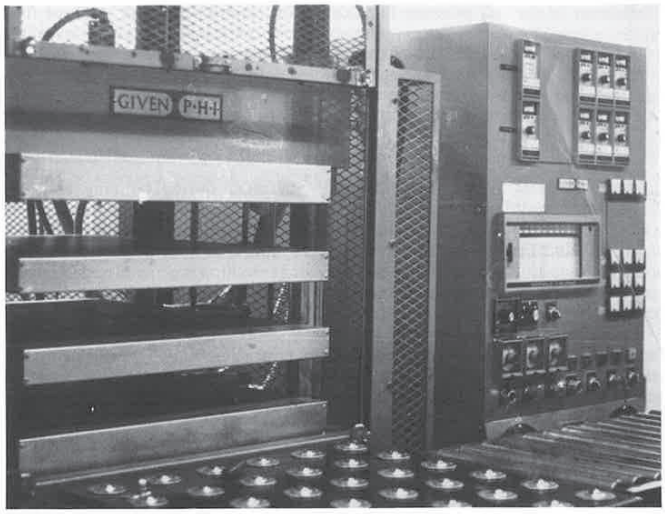


Fig. 7.6 Three daylight press open for loading.

and expel the air before it starts to cure. Curing takes about a further 40 minutes, for which time the temperature and pressure are held constant, then the press is allowed to cool to below the glass transition temperature or softening point of the resin before the press is opened. This cooling usually takes some 20 minutes.

A cold start cycle of this type can tolerate some variation in flow rates and gel times without significant effect on the bond strength, but it is time consuming, so for bulk production a hot start cycle may be used. This starts with the press plattens at bonding temperature, and cuts some 20 minutes off the cycle time, but at the expense of requiring closer control over gel time.

A further considerable time saving can be made by the use of an unheated transfer press. At the end of the cure time the hot press is opened, the stacks of boards are slid across into the transfer press which is then brought up to pressure, and the boards are allowed to cool. The use of a hot start cycle and a transfer press gives a significant increase in effective press capacity and substantial savings in the costs of heating and cooling the main press. The momentary release of pressure as the boards are transferred from press to press is of no significance since the epoxy bonds are fully formed by the time of the transfer, and the only reason for keeping the pressure on during cooling is to prevent the boards from warping badly while the resin is above its glass transition temperature.

Another possible variant in the press cycle is to start the cycle with a relatively low 'kiss pressure' which is sufficient to bring all layers into intimate contact and to allow air to escape as the resin starts to melt and flow. Full bonding pressure is applied at the time the resin is fully liquid but before the curing starts. The time of

application of the full pressure is quite critical, and when a full stack of five boards is being bonded between each pair of press plates the time lag between the heating of the outer boards and the centre board can make the use of a kiss pressure cycle of doubtful benefit or even dangerous if the full pressure is applied too long after cure has started in the outer boards.

7.4.3 Bonding Defects[33]

Most laminates suffer from material movement during bonding and the designer of a multilayer board should allow for this together with the movement which occurs on etching the laminate. Some of the movement during bonding is reversible, and the boards recover after a few hours, or after a 'post bake', but some is permanent. An allowance of 500 parts per million is generally adequate to cover this. If the material movement is excessive or random, the use of extra tooling pins for bonding can help to hold it stable.

In addition to this gross material movement there can be small local movements of individual features, usually known as 'swimming'. These occur when the flow of resin pushes against a track or pad and is able to move it on its base laminate. It is usually found as sideways movement of a track, but lengthwise movements can occur and in extreme cases tracks can be broken by the pressure of the resin. The usual cure is an adjustment to the press cycle or a change in the type of prepreg. The use of conformal sheet can cause swimming, and the substitution of ordinary blotting paper for the conformal sheet may effect a cure without weakening the bonds.

Warping, bowing and twisting are fairly common defects in multilayer boards. Their occurrence can be minimised by ensuring that the boards are designed with a build which is symmetrical about the centre of the boards and that all layers carry substantially equal amounts of uniformly distributed tracking. If the design is well balanced, then any warping may be caused by poor quality laminate or by incorrect cooling after bonding.

Voids can occur in lightly tracked areas, and in extreme cases it may be necessary to include extra pieces of prepreg to fill in gaps in the tracking pattern. These small extra pieces of prepreg can be tacked on to a full sheet by a touch with a hot soldering iron. In less extreme cases the use of a prepreg with a higher resin content, or an adjustment to the press cycle should ensure filling of the voids.

Excessive bonding pressure or the use of a wrong press cycle for the properties of the prepreg can result in too much of the resin being squeezed out of the bonds, leaving resin starvation and weak or non-existent bonds.

Inadequate bond strength or delamination after bonding may be caused by over oxidising with black oxide, by the use of old prepreg or material with too low a resin content, by a press cycle which causes premature cure of the resin, or by contamination of the laminates or prepreg during their preparation or during lay-up. The use of warped press plates which do not make uniform contact with the press plattens can also result in defective bonds.

Air inclusion can cause local voids beside tracks and other features, and may cause subsequent delamination of a board. It is difficult to guarantee that the flow of resin from the prepreg will sweep out all the air which is contained in the etched areas of a copper pattern, especially on earth or power planes. Air is highly compressible and if the edges of a layer are virtually sealed down by the full bonding pressure then small pockets of air can remain trapped and compressed in any closed areas of the pattern.

The use of a kiss pressure can help minimise air inclusion, and if a kiss pressure is not used the cycle should allow the longest possible flow time before curing starts. In cases where it is essential that all air be excluded it may be necessary to bond under a low vacuum, with the stack and the press plates sealed in a PTFE bag.

Surface defects may be caused by burred or dented press plates or by creased packing or release sheets. Their cure is obvious.

7.5 DRILLING AND FINISHING

7.5.1 Drilling[25,26]

Drilling of multilayer boards follows normal practice, but the holes must be of good quality. A first off blank should be checked to ensure that material movement during bonding has not distorted the inner layer patterns to the point at which holes break out of their pads. This check can be made by etching the outer layer copper off a first panel, or by clearing the copper from test pads in the corners of the blank. It may be found necessary to introduce a zero offset on an NC drill to compensate for material movement.

It is most important that the drills be properly sharp and their flutes and lands free of defects or chips. Any land defect on a drill can cause it to smear resin, which is melted as the drill cuts, over the walls of the holes covering the edges of the copper inner layers. This resin smear may occur even with the best of drills, but every possible precaution should be taken to minimise its occurrence. Much of the resin smeared over hole walls is picked up by the drill as it stops its downward travel with its tip in the lower backing sheet of the stack. The use of a thick copper foil backing or of copper foil about 0·15 mm thick over a plastic sheet pre-drilled with slightly oversize holes can eliminate this resin pick up, but is unlikely to be a total cure for all resin smear. Air blast cooling of the drill tips as they emerge from the stacks can also help to reduce resin smear, but it also reduces the hit rate and increases the cost of drilling.

Copper foils which have been used as lower backing sheets can be pressed and bagged and used as anodes for plating baths.

7.5.2 Smear Removal and Etch-back[27,34]

It is important for all resin smear to be removed from the holes before electroless plating, and some multilayer manufacturers apply an etch-back which removes the glass from the walls of the holes as well as the resin surface.

Epoxy resin can be removed by concentrated sulphuric acid. This process requires careful control, as the acid must be thoroughly rinsed from the holes without being allowed to re-deposit sulphonamated resin over the walls of the holes. As the rinse dilutes the acid, it deposits a fine film of resin, and without full control smear removal can become smear redistribution. Other chemicals such as chromic acid can be used for smear removal, but all present problems, either of process control or in effluent disposal. Plasma etching[34] is being increasingly used to remove resin smear, especially from polyimide based boards made with acrylic bonding sheets which are not attacked by chromic or sulphuric acid (see Figure 7.7).

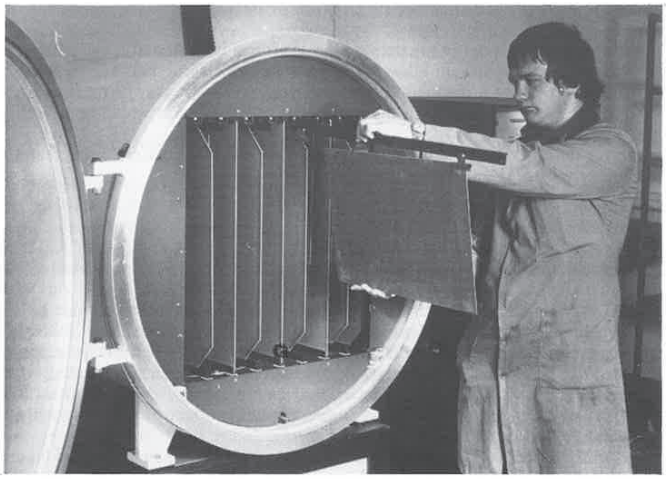


Fig. 7.7 Loading boards into plasma etch chamber.

Full etch-back is usually achieved with a mixture of hydrofluoric acid and sulphuric or chromic acid. When etch-back is used care has to be taken not to remove too much material to leave significant projections of copper. Any such projections lead to irregular through-hole plating, possibly with voids caused by ion depletion.

After smear removal, any etch-back, and a thorough rinse multilayer boards are electroless plated in the same way as double-sided boards.

7.5.3 Printing and Etching

The remainder of the processing of multilayer boards is the same as for double-sided boards. The only point to be noted is that there may be problems in printing fine track patterns on the outer layers. If they are on fairly thin laminate, the inner layer patterns may 'telegraph' through to the outer layers on bonding, especially if conformal sheet is used. The departure from flat is not large, but it can be sufficient to prevent effective contact between the printing film and the face of the board, with consequent loss of definition.

On boards with eight or more layers it may be better to use pads only patterns for the outer layers. If the boards are properly designed, with pads round all holes on all layers, all the pad sites will be at the same height and will be flat, and any irregularities in the surfaces will be in the form of depressions between the pads. Thus a good pads only pattern can be printed where a pad and track pattern could cause difficulties.

7.6 INTERNAL SHORT CIRCUITS[35]

7.6.1 Shorts on one Layer

Track to track or track to pad short circuits on any one layer are almost invariably the result of defects in printing or etching. The usual causes are scratches on films which print as a hair line joining two features, improper development of the resist, or inadequate control of the strength of the etchant.

7.6.2 Layer to Layer Shorts

Short circuits between layers are most often the result of attempts to use insufficient prepreg. The use of a single sheet of prepreg which may contain a pinhole defect is an obvious hazard. A pair of isolated tracks on opposing layers which cross one another at right angles can cause a very high local pressure during bonding, which could lead to a short when the prepreg is inadequate.

The other major cause of layer to layer shorts is the inclusion of metallic particles either on inner layers or in the prepreg. This is almost always the result of poor housekeeping—allowing dust and dirt to accumulate in the preparation, lay-up and bonding areas.

7.6.3 Track to Hole Shorts

Track to hole or hole to plane shorts may be caused by printing or etching defects, but they are most often the result of the designer failing to put pads round all holes on all layers. If a pad is omitted there is a risk that air inclusion during bonding could leave a small void beside the track or the edge of the plane, into which the drilled hole could break. The electroless plating solution can then fill the void, causing a short circuit. It is possible for such short circuits not to form immediately, and for the board to pass an electrical continuity and short test after its manufacture, but to fail later.

It is sometimes thought that the omission of pads will allow more tracks to be run between holes on an inner layer. In fact the reverse is true. The diameter of the pads need be only 0·1 mm larger than the circle within which tolerances allow the hole to be, and the gap between the pad and an adjacent copper edge need be only 0·1 or 0·15 mm, whereas the smallest distance between the edge of a copper feature and any machined edge, either the edge of a board or a drilled hole, which can be regarded as safe is usually considered to be about 0·5 mm. If the boards are bonded under vacuum the distance between a track and the edge of a hole might be reduced to 0·3 mm, but this is still greater than the space which need be left when a pad is used.

7.6.4 Hole to Hole Shorts[36]

There is a mechanism by which electrolytic action can cause short circuits to form along the interfaces between the strands of glass cloth and the resin in a board. This process requires a high potential difference between two holes for it to occur in any reasonable time, and for the majority of multilayer boards it can safely be ignored.

7.7 RELIABILITY OF MULTILAYER BOARDS[35,37]

Properly designed and produced multilayer boards can be very reliable, but mishandling can cause failures. Component replacement must be undertaken properly, and, to avoid any risks of delamination, multilayer boards should be baked dry before they are wave soldered. Adequate pre-heat should always be used when wave soldering multilayer boards.

At one time it was considered that to achieve reliable joins between pads on internal layers and the barrels of the through-plated holes, 2 ounce copper should be used for the inner layers. Modern techniques and better understanding of features such as resin smear make this unnecessary, and 1 ounce copper is normally used for most inner layers.

Thermal expansion in the Z axis can be a cause of failure. The epoxy glass base of the board expands at a greater rate than the copper of the hole barrels and after thermal shock testing some dishing of the outer layer pads may be noticeable. This expansion of the resin stresses the joins between the hole barrels and the pads, the stress being greatest on the layers nearest to the outside of the board. To help to contain this stress safely, some designers prefer to specify 2 ounce copper for the layers next to the surface on boards with 8 or more layers.

The omission of pads on inner layers can form a serious reliability hazard. If a pad is omitted the space around the hole where the pad should have been will be filled by pure resin which has flowed out of the prepreg during bonding. This thin layer of unreinforced resin expands at a greater rate than the glass reinforced material, and so imposes a high local stress on the hole barrel, which is liable to fail after repeated thermal cycling through the military temperature range. Such failures have been observed after the single thermal shock of wave soldering.

Boards which fail either after manufacture or in service or which require modification to accommodate design changes can be repaired as described in Section 11.6. Some designers of multilayer boards have included redundant tracking and via holes which can be used to replace failed tracks or to accommodate design changes. Most such attempts are only partially successful because the spare tracks can introduce cross-talk problems and there can be no guarantee that if an extra track is needed there will be one in the right place.

Chapter 8

MANUFACTURING VARIANTS

8.1 ADDITIVE PROCESSES[1,2,38,39]

Chapters 5, 6 and 7 described the basic methods of rigid board manufacture used generally throughout the industry. There are many variations of these basic processes, some of which were introduced as alternatives to existing methods and some of which were developed to do specific jobs which are beyond the capability of normal processing. This chapter describes a selection of the variants most likely to be encountered, or to be of practical use.

8.1.1 Additive Copper on Bare Laminate

A double-sided board can be made from a sheet of bare laminate, with all the copper track pattern and through-hole plating in electrolessly deposited copper. Ordinary laminate can not be used, as a specially roughened surface is necessary to obtain good adhesion of the copper. Plain laminate can be coated with a 'butter coat', which is then chemically treated to roughen the surface. A negative resist pattern is applied, and the boards are immersed in a high build electroless copper solution. One major advantage of this process is that badly unbalanced areas can be plated without any risk of 'burnt' plating.

A variant of this process is to deposit a normal thickness of electroless copper on the treated laminate, then apply the negative resist pattern and pattern plate the board in the normal way. When the desired thickness of copper has been plated the resist can be stripped and the electroless copper which provided a current path to all features is etched. Because there is only a very thin layer of electroless copper, no mask is required over the tracks and pads from which only the surface will be etched.

8.1.2 Semi-additive Processing[39]

Changing from panel plating to pattern plating made a big difference in the fineness of tracks and gaps possible as it cut down the amount of etching and so reduced undercutting. A further reduction is possible by using thinner claddings on the laminate than the normally used 1 ounce copper. Half and quarter ounce clad laminates are available, but below these weights it is too difficult to handle the copper foil when the laminate is being laid up. This difficulty has been overcome by backing 0·005 mm copper foil with a heavier layer of aluminium. The boards are drilled with the aluminium in place, which helps to keep down burrs in the copper, then the aluminium is etched off in a caustic solution, leaving a clean copper surface ready for electroless plating. The boards are pattern plated normally and can be etched without a resist. Care must be taken in etching to ensure that all copper is removed from the minute pits which keyed it to the

laminate. It is claimed that this semi-additive process results in boards with better adhesion of the copper to the base than is possible with the electroplated additive process, but can be used to the same limits of track and gap widths.

It is questionable whether the reduction in the copper thickness to 0·005 mm offers a significant reduction in line and gap widths over those achievable with dry film resist and pattern plating on half ounce copper, using a properly controlled regenerative etchant in a good spray machine.

The terms 'additive' and 'subtractive' for board processing had a clear meaning when the major alternatives were panel plating or electroless deposition on unclad laminate, but they lose their meaning when efforts are made to differentiate between pattern plating on different copper thicknesses. Strictly speaking, pattern plating on any thickness is a semi-additive process, and any process which involves etching is semi-subtractive.

8.1.3 Sensitised Laminates

A variation of the fully additive process is the use of a laminate which contains seeding chemicals for electroless plating, which are activated by exposure to ultra-violet light. The exposure of the board through a film or by scanning activates the required pattern and the board can be electrolessly plated without any need for a resist.

8.2 MULTILAYER VARIANTS[2,8]

8.2.1 Buried Via Holes

The manufacture of buried via-hole multilayer boards starts with the inner layers being made as double-sided through-plated hole boards, with only the via holes drilled. They can be made by panel plating with a dry film resist tenting over the holes, or by normal pattern plating followed by tin-lead stripping. Special plating fixtures may be needed to hold thin laminates flat against agitation in the baths.

The build might have to be changed from that of a plain multilayer to include extra sheets of prepreg in each bond, and a prepreg with a higher resin content may be needed to ensure the presence of sufficient free resin to fill the via holes.

8.2.2 Doubled and Sequential Multilayers[40]

Double-double multilayers are handled similarly to BVH types except that the outer face of each half is left unetched until after the final bonding and drilling. The via holes must be filled with resin, so either a high resin content prepreg should be used, or the halves of the board can be coated with resin, squeegeed over from the inner face. The resin fills all via holes, and it also fills the spaces between the tracks which will be deeper than on a normal multilayer because of the plating.

After bonding, the outer faces of the board are best linished before drilling and the final plating. A smooth outer surface is essential if a good track print is to be achieved, and the outer layers of copper will be thicker than on a normal board because of the metal added when plating through the via holes.

Similar principles apply to double multilayers, on which the inner layers go through a full multilayer cycle prior to final bonding.

Sequential multilayers are handled in the same way as doubled types, taking care to add each new layer to the opposite side of the panel from its predecessor. After each layer is bonded on, the outer faces must be left unetched to provide the requisite current path for plating and if layers are not added to alternate sides one side will receive another coating of copper at every stage. Linishing the surfaces after bonding will help to keep down the build up of copper.

8.2.3 Added Surface Layers

Boards with added surface layers are a special form of sequential multilayers in which surface layers, usually carrying a pad only pattern, are added to a double-sided or multilayer board which carries the complete interconnection pattern.

The added layers may be formed of copper foil on normal prepreg, or they may be electroless copper on a film of dielectric, or on a layer of dielectric material which has been applied as a liquid by curtain coating, roller coating, screen printing or by immersion, and subsequently cured.

Surface layers may be added to provide mechanical protection to a fine track pattern; to aid soldering in circumstances where the use of a normal solder resist is considered inadequate; to achieve specific electrical properties for tracks on the layer covered by the surface coating, or to achieve specific mechanical properties for the surface pads, such as matching the thermal expansion of surface mounted leadless devices. Extra surface layers may also be used to save board area when surface mounted devices are used.

The pads necessary for mounting surface mounted devices occupy a much larger area of board relative to the size of the devices than do the holes for leaded devices and a substantial saving in board area can often be made by the use of an added surface layer which allows the pads to extend over the track pattern on what would otherwise have been the surface layer.

Leaded devices, whether surface or through-hole mounted in conventional boards, can withstand the differential thermal expansion between the body of the device and the board which carries them because of the compliance in their leads. Leadless devices such as chip carriers have no such compliance and, if joint failure on large devices is to be avoided, either the thermal expansion of the board must be matched to that of the device or the surface mounting pads must be carried on an elastomeric outer layer or the whole board may be made of suitably compliant material.

Suitable compliance in the mounting can be achieved by the use of acrylonitrile butadiene or polyimide added outer layers, with mounting pads connected to the track pattern either by through-plated holes or by plated pillars.

8.2.4 Heat Ladders

Semiconductor devices closely packed on printed circuit boards can generate a substantial amount of heat, which must be removed from the equipment if catastrophic failure of the active devices is to be avoided. Dual In-line Packages can be air cooled at densities up to about 3 square cm per pack but above this density or where air cooling is impractical conduction cooling may have to be used. This usually involves bonding a heat ladder of copper or soft aluminium as

in Figures 8.1 and 8.2 to the face of the board so the active devices can be mounted with their bodies in firm contact with the metal. The heat ladders, which are usually up to 1 mm thick, can be punched or etched out of sheet metal. Aluminium heat ladders are normally anodised before bonding.

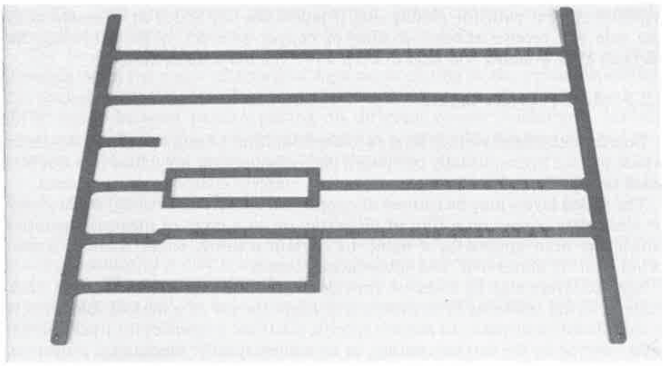


Fig. 8.1 Simple etched metal heat ladder.

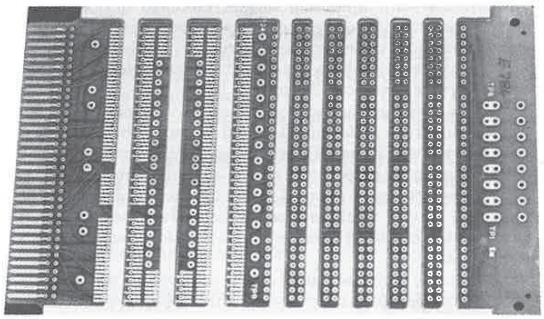


Fig. 8.2 Board with heat ladder bonded in place.

The ladders are bonded to the board with a low flow or no flow prepreg, pre-punched or knife-cut to shape. The holes in the prepreg through which the component leads pass are made about 0·4 mm smaller than the corresponding holes in the metal, so the prepreg will be just proud of the metal all round to avoid any risk of a short circuit between the metal and the pads or tracking on the surface of the board. The board, prepreg and heat ladder are provided with locating holes which ensure alignment during bonding.

Heat ladders of this type must not be confused with heat sinks, whose function is to increase the radiating surface area of a device and thus achieve more effective heat transfer to the surrounding air. The ladders add very little to the effective radiating area of the mounted device and they serve to conduct the heat from the device to the metal frame of the unit or to a 'cold wall'. Boards carrying heat ladders are usually mounted in special board guides which can be moved to clamp firmly on to overhanging edges of the ladder to achieve the best possible thermal transfer to the frame.

The metal used for heat ladders must be soft enough to be able to absorb the stresses introduced by differential expansion against the board material without its distorting the board significantly. Where a substantial heat ladder is necessary and the risk of such distortion can not be eliminated, it may be better to make the board of a material such as polyimide which can absorb the stresses and follow the expansion or contraction of the heat ladder.

8.3 SELECTIVE PLATING

Cases can arise in which part of a printed circuit board has to be plated, but it is undesirable for other parts to be plated to the same thickness or with the same metal. Possible examples are gold plated switch contacts on a rigid board, or the pads around through-plated holes on a flexible circuit. Selective plating of such areas is a form of pattern plating, performed with a negatively patterned resist masking off the areas to be left unplated before the board is etched. Selective plating after etching is possible in cases such as edge-connector fingers where electrical contact to the features to be plated can be made either by compliant clamps or by subsequently removable tracks. In cases such as selectively plated pads on the ends of tracks such contact is too difficult to arrange after the board has been etched.

The actual selective plating presents no problems. The problems arise in defining the pattern for the subsequent etching or other plating operations.

Once the selectively plated area has been plated, the surface of the panel is no longer flat and successful use of dry film resist may be difficult. Unless the plating is very thin it may be impossible to get a dry film resist to bond down to give adequate protection for plating or etching up to the edge of the plated area, as shown in Figure 8.3.

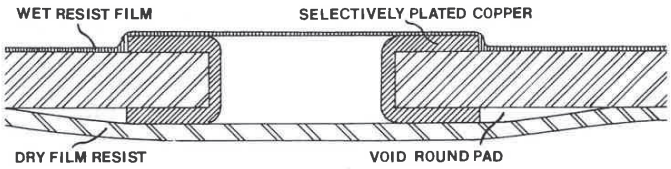


Fig. 8.3 Dry film and wet resists over plated edges.

A liquid resist can be applied to provide full protection up to the plated edge, but the presence of the step can make it hard to achieve good printing definition as the emulsion of the film will not be in contact with the resist surface at the step. The spread of light in normal contact printing at the point where the film is not in

contact with the resist will result in 'necking' after etching. If suitable equipment is available, this defect can be overcome by printing with the film out of contact with the resist, using collimated light, or by some form of beam scanning. However, the special coaters for liquid resists or a collimated light source may not be available, but there are other ways of overcoming the problem.

To obtain plated pads but no plating on the tracks, a negative working dry film resist can be used. It is exposed with the pad pattern, developed, then exposed with a negative of the track pattern (or the entire track and pad pattern), but not developed. The pad areas are clear of resist after the first development so the holes and pads can be copper and tin-lead plated, taking care that the board is not exposed to any source of U-V light during the plating. The latent pattern from the second (track) exposure is then developed to leave resist over the tracks, and the board can be etched with the pads protected by their tin-lead plating and the tracks covered with the resist.

If the plating areas are lit with fluorescent tubes the second exposure may have to be carried out after the pads have been plated. Provided the thickness of metal selectively plated is roughly equal to the thickness of the dry film resist this can be achieved without too much risk of necking.

Whenever two or more printing operations have to be undertaken there will always be some risk of misalignment, and the design of the selectively plated features should allow for the effects of misregistration. Plain copper tracks running right up to copper and tin-lead plated pads will present no problems, but often the design may call for the tin-lead to extend a short distance down the track to ensure that the edge of the tin-lead lies under a solder resist or other protective coating. Then it is advisable to use a tear-drop shaped pad so that any misregistration or necking will lie on the broadened end of the track as shown in Figure 8.4b.

When a single resist can not be used for the two prints there are various ways in which the selective plating can be achieved. If the selective plating is being used to keep the tracks with a bare copper finish, then the job may be panel plated, with only the tin-lead selectively plated as in 'a' of Figure 8.4. The second print involves a liquid resist and the tear-drop end of the track can safely accommodate the necking caused by the light spreading where the film rides up over the edge of the tin-lead plating. When the plating is done selectively to keep down the thickness of copper on the tracks, a three mask process can be used. The first mask defines undersize pads, which are plated with copper, then a full size pad mask defines the tin-lead. The difference between the two plated pad sizes should be sufficient to allow the film to make good contact with the surface of the resist at the printed edge. The third mask covers the tracks for etching as shown in 'b' of Figure 8.4. The necking with this method will be the same as with 'a'.

If the pad diameters are too small to allow for this three mask process, then the same mask will have to be used for the copper and tin-lead plating, and possible necking at the end of the tin-lead plating will have to be accommodated by a wider end to the track as in 'c'. A further way of achieving the selective finish is to pattern plate the whole panel normally, then strip off the coating of tin-lead from the areas where it is not required, using a tented resist mask to protect the pads and holes.

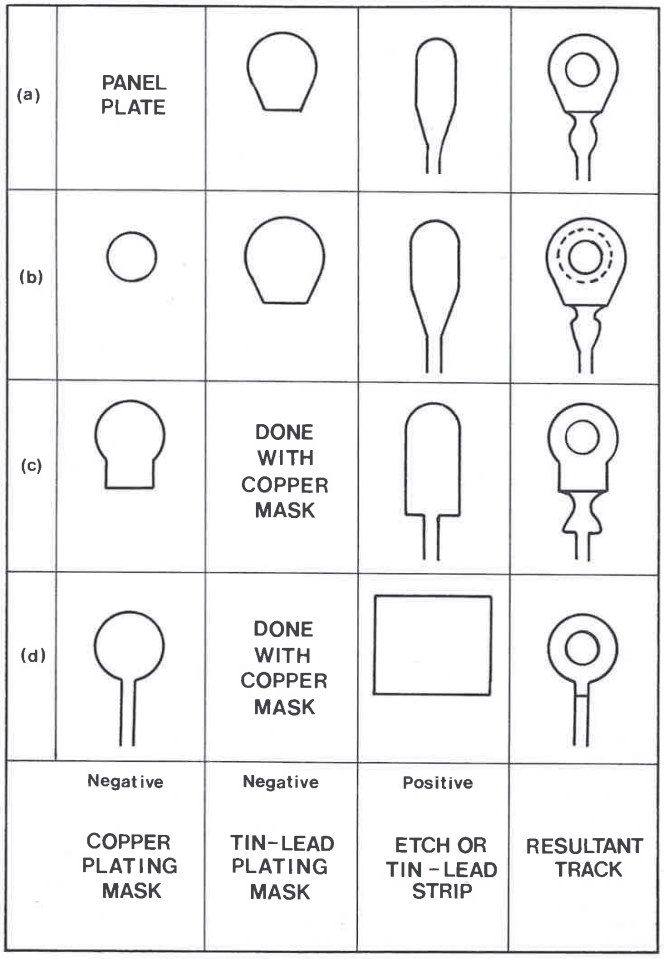


Fig. 8.4 Selective plating masks.

8.4 PILLAR PLATING[41]

Pillar plating is not so much a variant of normal printed circuit board manufacture as an alternative method of making a board with through-connections. As with normal manufacture there are several ways in which pillar plated boards can be made. One process starts by coating a sheet of stainless steel with a resist in which a negative pattern of the first track layer is formed. Copper is plated onto the exposed steel, forming the tracks, and a second layer of resist is applied. This is exposed and developed as a negative image of the pillars between the first and second layers, and the pillars are plated. Unless very thin dielectric layers are required it may be necessary to use several coatings of resist to build up the pillars to slightly more than the required height. The resist is then cleaned off and the dielectric is applied, either as a liquid, curtain coated and cured, or in the form of prepreg, usually pre-punched to pass over the pillars. Prepreg is cured with conformal sheet under sufficient pressure to ensure a good bond to the rest of the board.

When the dielectric has cured, it is linished or ground flat, and the exposed faces of the pillars are chemically cleaned to ensure there is no resin smeared over them by the grinding. More resist is applied and the second track layer is pattern plated, with the current being fed in through the steel base and the pillars. The process is repeated with further pillars, dielectric, curing, linishing and track layers pattern plated until the board is complete. Finally it is removed from the steel plating base to which the adhesion of the copper is poor, but adequate for supplying the plating currents.

Pillar plated boards can be built onto insulating bases or onto the face of a conventional PCB by using high build electroless deposition instead of electroplating. This process is suitable for connecting surface mounting pads on added outer layers to a multilayer or fine line double-sided board.

Panel plating can be used instead of pattern plating, with first the pillars and then the track pattern being etched from a thick layer of plated copper.

A further variation of the process is to apply the dielectric as pre-punched or pre-drilled cured sheets, bonded with a no-flow prepreg after the track pattern has been etched or pattern plated, then to plate up the pillars through the holes in the dielectric. This method may be preferred when a thicker dielectric layer is needed.

A more recent development of building the pillars into the dielectric applies the dielectric as a continuous sheet, into which holes are cut by a carbon dioxide laser. The energy from the laser removes the dielectric material with ease, but has little effect on the copper so there is an automatic control over hole depth. Such use of a laser makes possible the formation of holes and pillars of under 0·1 mm diameter, thus allowing very high tracking densities since no pads are required on the tracking layers to surround the through-connection points. Plating the pillars into blind holes requires very careful control to avoid entrapment of bubbles or ion depletion in the holes.

All forms of pillar plated boards are made sequentially and thus the process is time consuming and it is unlikely that pillar plated boards will ever be as cheap as conventional boards. However, they can be made at tracking densities which are unlikely to be equalled by drilling and through-plating and at these ultra-high densities they must be regarded as being the most cost-effective packaging form.

In the immediate future it is likely that dielectric materials which are photosensitive and U-V curable, or which can be cured by collimated U-V light through an out of contact mask will be used instead of plating resists to form the pillars. The dielectrics could be applied by roller or curtain coating or possibly as dry films.

8.5 METAL CORED BOARDS

The possibilities of combining a metal heat ladder as described in Section 8.2.4 with the interconnect in the form of a metal cored board have attracted some interest since the later 1960s, but the thermal efficiency of such a metal core is less than that of a ladder in direct contact with the body of an active device. The presence of the metal core usually results in high capacitance, low line impedance tracks, which slow down the working of the system and, in high speed systems, add to the heat generated because of the high surge currents associated with the low line impedances. The use of a thick dielectric layer to minimise capacitance decreases the thermal efficiency and the would-be user of a metal cored system must make the choice between good thermal performance or good electrical performance.

A thin metal core can be made by normal multilayer techniques as an over-thickness earth plane, using 3 or 4 ounce clad laminate, and possibly plating on further copper. Generally, though, it is better to use a separate metal core about which a double-sided or multilayer board can be built.

The core can be aluminium, copper, or any other metal considered suitable. Weight for weight, aluminium is the most effective core. The metal is drilled with oversize holes for all component and via holes, then it is insulated. Aluminium can be heavily anodised, or all metals can be coated with epoxy resin and the holes completely filled, except for tooling holes which must be kept clear to be able to print the circuit pattern and drill the final holes concentrically with those in the core. For multilayer boards it is better to fill the holes with resin.

For a double-sided board the insulated core is drilled if a resin coating was used, then it is electrolessly plated with copper and thereafter treated in the same way as an ordinary board. Multilayer board etched layers are bonded to the core and then treated normally.

Clearly the holes in the core must be oversize by more than the total possible tolerances on the two drilling operations plus the minimum thickness of insulant to be left round the barrels of the holes. With conventional drilling and tooling this can amount to some 0·3 mm of insulant, which will be acting as a thermal insulator as well as electrical. Every possible step should be taken to minimise the drilling tolerances and hence the thickness of the insulant in the holes since the major cooling path from the semiconductor chips is conduction down the package leads and through the solder joint to the walls of the holes. The active devices will usually be mounted clear of the board surface to avoid risks of moisture traps, so the dielectric between the metal core and the track layers can be made thick enough to give electrical properties comparable to those of a normal 8 or more layer multilayer without too much loss of thermal efficiency.

When surface mounted devices are to be used, a cored board should be built with only the surface mounting pads on one face of the core, and all the track layers on the other side.

As well as being less efficient either electrically, thermally, or both, than ordinary boards with heat ladders, metal cored boards have a further disadvantage in that they are more difficult to solder to, and require considerable pre-heat to achieve good joints. Replacement of devices without damage to the board or to other joints can be very difficult.

Chapter 9

FLEXIBLES AND FLEXI-RIGIDS

9.1 FLEXIBLE CIRCUITS[42,43]

Flexible circuits are printed circuits made on thin base materials, usually of unreinforced polymeric materials. Their ability to bend allows them to be used as interconnects in place of wiring harnesses, and in places where connections are needed to a moving part of a machine. Flexible circuits have many advantages over conventional wiring harnesses, the two most important of which are that once the layout has been proven it is almost impossible to make a wrong connection when a flexible circuit is soldered into place, and their thinness allows them to be used in locations where the thickness of even a ribbon cable could not be accommodated.

Basically, flexible circuits are made in the same ways as rigid boards and, like rigid boards, the holes can be plain or through-plated.

9.1.1 Base Material

Earlier flexible circuits were made on polyester base film, which gave excellent flexibility and its plastic memory could be used to form retractiles, long strip circuits folded in half then coiled, so the ends could be pulled apart and when released the circuit coiled itself up again. The centre fold gave the tracks a non-inductive bifilar coil and they made excellent connections to slide-out drawer type units. The only major disadvantage of polyester based flexible circuits is that the base material melts at below normal soldering temperatures, which makes them difficult to install and service without risk of damage, and it makes it difficult to bond them to other laminates. Various special forms of connectors were devised with tongues which pierced the insulation and made pressure contact with the copper tracks, but polyester based circuits were largely superseded by polyimide flexibles which can safely withstand soldering and bonding temperatures. Other base materials such as epoxy resins have been used in some special cases.

9.1.2 Cover Coat

Flexible circuits are usually protected by a cover coat of a similar material to the base film which is bonded on after the circuits have been etched. The cover coat serves four distinct purposes. It protects the circuit against corrosion and contamination; it affords protection against mechanical damage which might occur if a moving flexible circuit touches another part of an equipment; it puts the copper of a single-sided flexible circuit on the neutral bending axis, and it helps to anchor the terminal pads to the base film.

An unclad flexible circuit which is repeatedly bent will impose stresses on the copper, which must lie either on the outside or on the inside of the bend. The

addition of a cover coat of equal thickness to the base material balances the build and significantly increases the flexing life of the circuit. Cover coats on double-sided flexible circuits help to increase the flexing life, but since on a double-sided circuit the copper layers can not lie on the neutral axis, continued flexing will lead to cracking of the tracks even with cover coats bonded on.

Copper has adequate adhesion to the polyimide base to form reliable tracks, but this adhesion is reduced at soldering temperatures and unless the pads are given some mechanical support or reinforcement there is a risk of a flexible circuit failing because the pads have lifted. Figure 9.1 shows three ways in which the cover coat can be used to help to hold the pads in place.

The simplest way is to use a pad larger than the hole in the cover coat (Figure 9.1A). The pad must be of sufficient size to ensure that its edge will be securely trapped under the cover coat with worst case misregistration of the etched base and the bonded-on cover coat. This may mean the use of unacceptably large pads, and since the edge of the hole in the cover coat is bonded only to the face of the copper pad there is a potential weakness if the circuit is flexed right up to the soldered joints. The best anchorage is obtained when the pads are provided with three 'ears' which will lie under the cover coat as shown in Figure 9.1B. Then the cover coat can be securely bonded to the base film between the ears, and the pads need be only the normal size appropriate to the hole diameter. In cases such as two row connectors where tracks must pass between fairly closely spaced pads a single 'ear' opposite an enlarged entry for the track has to suffice (Figure 9.1C).

Flexible solder resists which can be screen printed onto flexible circuits are available, but the mechanical protection they can afford is limited and they can play no part in anchoring down pads or in putting the copper tracks on the neutral axis during bending so they can not be used as an alternative to a cover coat in many applications.

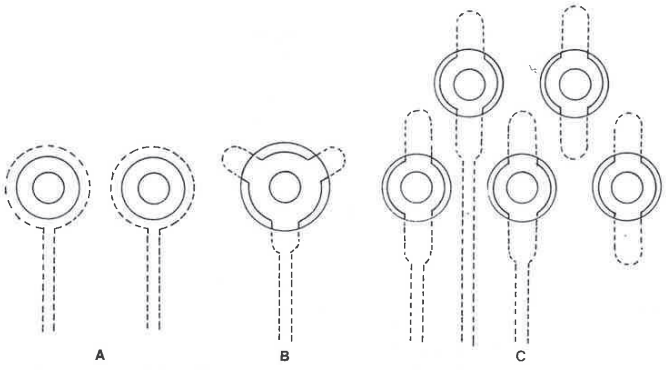


Fig. 9.1 Ways in which a cover coat can anchor pads.

9.1.3 Cover Coat Bonding

Cover coat bonding is similar to normal multilayer bonding, except that the cover coat is usually supplied with the low flow adhesive layer already in place. For a good bond close up to the edges of the tracks it is usually necessary to use conformal sheet or several layers of blotting paper in the press plates. Care must be taken when using conformal sheet to ensure that side pressures do not break any of the tracks and better results may be obtained with less compliant paper packing and bonding under vacuum.

The apertures in the cover coat to expose the pads can be made by drilling, punching, knife cutting, laser cutting or in some cases by etching. Polyimide films can be etched by hot caustic solutions, normally using pre-etched patterns in copper cladding as a mask.

Flexible circuits are usually profiled after the cover coat has been bonded, by knife cutting or by steel rule tooling. They can be routed to shape, but the rotating cutter tends to leave slight feathering on the edges.

9.1.4 Back Etching

The design of a non-plated flexible circuit may require components to be soldered on both sides. A single-sided circuit to carry components on both sides can be made to be assembled with normal soldered joints by starting with a double-sided flexible laminate. The track pattern is etched on one side, and the other side is etched with holes the size of the clearance holes in the cover coat where components are to be reverse mounted. Where components are to be reverse mounted, the cover coat is drilled or punched with holes which are larger than the holes in the base circuit only by the bonding and registration tolerances of the cover coat. After the cover coat has been bonded in place the base plastic is etched away from behind the pads for the reverse mounted components, using the etched copper as the mask. When the base material has been cleared from the backs of the pads, the copper on the upper side of the laminate is etched off, leaving the single-sided flexible circuit with pads accessible from alternate sides as shown in Figure 9.2. The base can also be cut away with a laser.

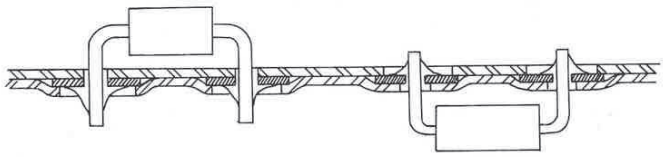


Fig. 9.2 Components mounted both sides of a non-plated flexible circuit.

Back etching for component pads is most easily carried out if the design and manufacture can be arranged such that the drilling of the component holes is undertaken after all etching. Then the cover coat does not need to be pre-drilled for the reverse mounted components.

9.1.5 Solder Buttons

The base layers of most flexible circuits are thin enough to allow joints to be made between pads on opposite sides of the base, or between pads on separate flexibles, by solder which bridges across the thickness of the dielectric as shown in Figure 9.3, where the pad on the lower side of the base has no hole in its centre, and the upper pad and the base dielectric are etched to expose a suitable area of the back of the copper on the underside.

Solder button joints on a double-sided flexible can be made by wave soldering, but joints between separate flexible circuits or between a flexible and a rigid circuit will usually have to be hand soldered unless a reliable means of holding the two circuits close together can be used during mass soldering.

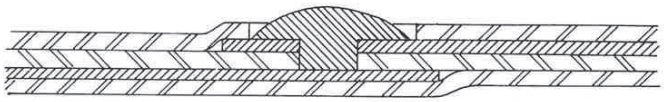


Fig. 9.3 Solder button joint in a flexible circuit.

9.1.6 Roll to Roll Processing

Flexible circuits can be made in bulk by roll to roll processing, in which the material is fed through the various processes up to final cutting to profile in continuous lengths, wound in rolls on drums as it leaves each process. When large numbers of flexible circuits are required, the capital investments in plant necessary for roll to roll work can be justified. It is possible that computerised control of operations, coupled with high speed scanning methods of printing or sensitising, conveyor etching, vacuum hot roll laminating of cover coats, laser generation of holes, and laser cutting to profile may make it economic to use roll to roll or continuous processing of varied mixes of small batches of flexible circuits. With automatic scanning to detect defective etching or hole generation, coupled with feedback to order instant re-makes, such continuous processing of flexibles offers the promise of the total automation of circuit production.

9.1.7 Plated-through Hole Flexible Circuits

Flexible circuits can have holes plated-through in the same ways as rigid circuits, with the advantages that unusual shaped holes can be punched or etched, and the thin base materials allow the use of much smaller holes than are normally practicable on rigid boards. The only difference between the processing of flexible circuits and rigid boards is that special plating fixtures are necessary to hold the flexible laminate against agitation in the plating baths or the pressure of spray rinses.

Circuits which are to undergo continuous flexing or which have to be bent round a tight radius will usually have their holes and pads selectively plated as described in Section 8.3 so as to minimise the copper thickness of the tracks and thus maximise the life during flexing.

Selective plating to achieve minimum metal thickness on the tracks may also be necessary to ensure a good bond between the cover coat and the base over areas of closely spaced tracking. The cover coat adhesive is very low flow material, and an extra plated layer of copper on the tracks may make it very difficult to get the cover coat to conform over narrow gaps, and the use of conformal sheet during bonding may introduce side pressures which can rupture narrow tracks.

9.2 RIGIDISED FLEXIBLES

Flexible circuits, with or without plated-through holes, can be used to replace cable harnesses in a wide variety of uses from aerospace systems to electronic games and vehicle wiring. In some systems a conversion from a wiring harness to a flexible circuit can leave small numbers of components uneconomically mounted mechanically, possibly on a single-sided printed circuit board to which the flexible circuit is connected by pins, solder button joints, or by reflow soldered joints on fingers similar to those of an edge connector.

These joints and the separate PCB can be eliminated and a secure mounting for the components can be provided by extending the design of the flexible circuit to incorporate the connections to the components and by locally rigidising the component mounting area of the flexible circuit. Such rigidising is usually achieved by bonding the completed flexible circuit to a plate of either a suitable plastic material or metal, which has clearance holes for all the component leads and suitable mechanical fixing holes. The rigidising plate lies on the components side of the flexible circuit so that after assembly and soldering the bonding agent serves only to exclude moisture from the area between the circuit and the rigidising plate. When this function is not critical, a low cost, room temperature adhesive such as PVA or an impact adhesive can be used to stick the plate to the circuit. In critical areas an acrylic adhesive, as used to bond the cover coat to the base, is normally used to bond on the rigidising plate.

Metal plates can be used to conduct the heat away from high dissipation components, and if the high capacitance of a metal plate bonded over the tracks and separated only by a thin cover coat is considered undesirable, a composite plate of metal and plastic can be used.

Because flexible circuits allow the use of very small holes for via-hole side to side connections, and can use square holes contained within the width of a 0·15 mm track, very high density interconnects can be achieved on just two layers. A double-sided flexible circuit with an insulated metal rigidising plate can be used to mount and interconnect high density components as an alternative to a multilayer board with a bonded on heat ladder, with the added facility that the flexible circuit can be extended to form part or all of the interconnect to other parts of the unit.

Rigidised flexible techniques can also be used to incorporate the interconnect directly into the main metal case of a unit, possibly eliminating the need for other forms of mechanical mountings and for separate heat sinks.

Combinations of back-etched flexibles and rigidising plates can be used to mount VLSI circuits in chip carriers or even as naked chips. The flexible circuit can be back etched to leave inverse beam leads to bond to the pads of the chip carriers or bare chip, as is done in 'spider bonded' or Tape Automated Bonded chips, and the chips or chip carriers can be mounted directly on the metal rigidising plate. High dissipation semiconductor chips could be inverse beam

mounted in flexible circuits with the assembly immersed in a suitable inert liquid for direct cooling of the chips.

9.3 FLEXI-RIGID BOARDS[44,45]

The simpler flexi-rigid boards are similar in appearance and application to rigidised flexibles, the difference being that flexi-rigids are fully bonded structures with holes plated through both the rigidising plate or layers and the flexible layers after they have been bonded.

9.3.1 Types of Flexi-rigid Board

There are two main types of flexi-rigid board, the 'flying tail' board and the 'hinged' board. A flying tail board is one in which a single rigid, component carrying area has one or more flexible extensions which connect to components mounted on the case of the unit or otherwise non co-planar with the rigid portion, or to components which must be capable of moving relative to the rigid area of the board.

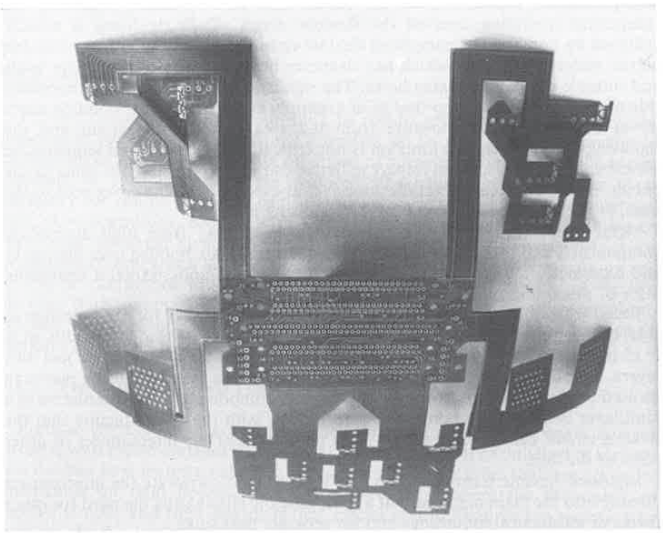


Fig. 9.4 Complex flying tail board with two layers of flexible tails and partial stiffening of some areas by extra cover coat.

A hinged board comprises two or more rigid areas which carry components and are permanently interconnected by one or more integral flexible hinges. In practice there can be blurring of the distinction between the types as a flying tail

board may have the ends of the tails rigidised, and a folding board may also incorporate some tails with plain flexible ends. Flexi-rigids can also be classified by use as 'continuously flexed' and 'bend on installation only'.

The flexible layer or layers run right through the whole board so that tracks can often be run direct from component pin to component pin on the one layer, and within the bonded areas of the boards the flexible layers are treated as part of the normal interconnect. The holes in the flexible tails may be plain or through-plated, and as they can be cut by means other than drilling they can be any shape necessary to fit over component or connector pins.

Extra cover coat layers can be applied to some areas of flexible tails or hinges to give local partial stiffening to ensure that the tails bend where required, and to minimise stresses at points where the flexible layer connects to the components or at the edges of bonded rigid areas.

Flexi-rigids can vary in complexity from double-sided boards on which one outer layer is flexible polyimide, to multilayer structures with several separate flexible layers, built up as buried via hole or sequential boards (Figure 9.4).

Figures 9.5 and 9.6 show a hinged board flat and folded.

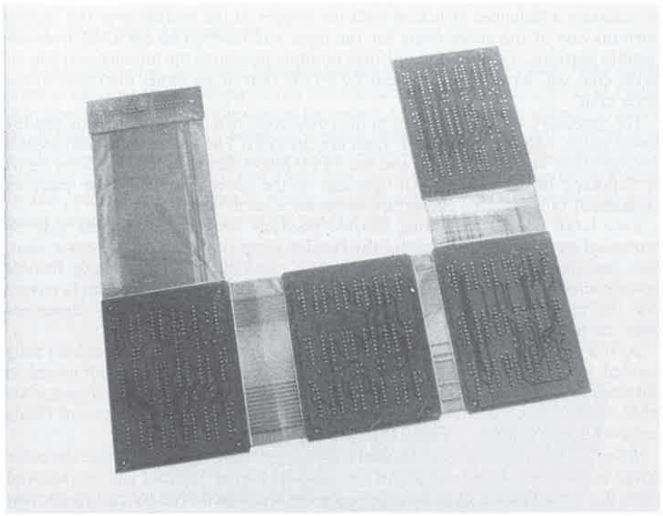


Fig. 9.5 Folding board with four main rigid areas.

Rigid portions of flexi-rigids are usually made from epoxy-glass laminates, but some types which have to work at elevated temperatures may use an all polyimide build. Normal prepreg is used to bond epoxy-glass layers, but an acrylic bonding sheet is normally used for bonding to polyimide. With possibly differently cut-out cover coats on each side of each flexible layer, and specially cut-out adhesive

layers, flexi-rigid boards can require large numbers of film or drawing sheets. The example in Figure 9.4 had more than 20 separate films.

9.3.2 Manufacture of Flexi-rigid Boards

Flexi-rigid boards are made by normal printed circuit board printing, etching, bonding and drilling techniques but with added complications to ensure that cover coat is properly bonded over the flexible areas whilst the flexible areas are not bonded to scrap areas of rigid layers. If there are plated-through holes in the ends of flexible tails these will be treated as buried via holes. Some partial profiling of flexible and inner layers is also necessary before the boards are bonded.

The flexible layers may have to be made from double-sided clad laminate if one tail requires tracks on its upper face and another area requires tracks on its lower face. At this point it should be noted that the bending of a flexi-rigid may make it impossible to define a 'components' and a 'solder' side for the whole board; different areas may carry their components on opposite sides. The use of a double-sided flexible layer with only one side used in the flexible areas means that to maintain a balanced structure with the copper in the middle over the flexing portions one of the cover coats for the layer will have to be cut away over the flexible portions. To maintain uniform bonding pressures the missing portions of cover coat will have to be replaced by PTFE film of an equal thickness to the cover coat.

The necessary apertures are cut in the cover coat, often by knife cutting against a steel guide, and pieces of PTFE sheet are cut to fit. The apertures usually extend into the scrap border material, and the PTFE pieces can be held in place by strips of Sellotape in the margin. All openings in the cover coat to expose pads on unrigidised tails are also cut before the cover coat is bonded.

Very large boards with long flexible portions between rigid areas may be processed by printing and etching the flexible layer, bonding on the cover coat, then building up the rigid areas separately, with the interconnecting flexible portion emerging from one edge of each stack. Great care must be taken to ensure that during processing these interface edges are sealed against any ingress of swarf or moisture.

Such a seal is not easy to achieve, so most flexi-rigid boards are made in fully bonded packages in which the flexible tails or hinges are totally contained in unbonded 'pockets' inside the bonded outline. Bonding of only the appropriate areas is ensured by cutting apertures in the low flow bonding sheets and filling them with PTFE inserts in the same way as for the cover coats.

When the boards have been bonded, drilled, through-hole plated, and the outer layers etched they are profiled and the unwanted rigid material can be removed from the flexible areas. On inner rigid layers this is achieved by cutting narrow grooves along the line of the flexible to rigid interface before bonding. When the final profile cuts are made, they cross the ends of these grooves, freeing the unwanted material. During bonding, all air has been squeezed out from between the rigid layers, the PTFE sheets and the flexible layer so it may not always be easy to remove the pieces of rigid material. It can be quite difficult if the grooves were cut too narrow and the bonding agent has filled them and stuck to the scrap material. Whenever possible the flexible areas should be shaped to leave the best possible access to their edges to ease the removal of the pieces of rigid laminate.

Outer layers can not be grooved right through as that would allow plating and etching fluids to enter the stack. Either a stopped depth groove is cut from their inner faces or the outer rigid layer is made up as a composite with one grooved piece of laminate and a very thin ungrooved cladding layer. On completion of the board the thin outer piece is easily cut through with a knife without risk of damaging the flexible layer beneath.

The polyimide base of the flexible layers is usually very thin, often only 0·05 mm, and has poor dimensional stability on etching and bonding. This lack of stability, which can result in movements of well over 0·1 per cent, is usually countered by leaving broad unetched copper borders all round flexible layer patterns, preferably on both sides of the laminate, in the scrap margins. Movement during bonding can be reduced by the use of multiple tooling pins. The lack of stability can also be partially overcome by bonding single-sided flexible material to the back of pieces of 0·1 mm or thicker single-sided epoxy glass laminate prior to printing and etching the inner layers. Naturally, areas which are to be flexible on the finished board are left unbonded and filled with PTFE sheet, and the rigid laminate is grooved to ease later removal of the unwanted portions.

9.3.3 Flexi-rigid Applications

Applications for flexi-rigid boards appear to be almost unlimited. They can be used to eliminate back wiring panels and connectors from small units as in Figure 9.6; they can replace complex cable harnesses, often with major savings in space, as in Figure 9.4; they can be used to allow large boards to fit into awkward or angled spaces; they can form connections between fixed and moving portions of a machine or between the body of a unit and lamps or switches mounted on a door; in fact, almost any equipment which contains a printed circuit board and some wires connected to it is a potential application for a flexi-rigid board.

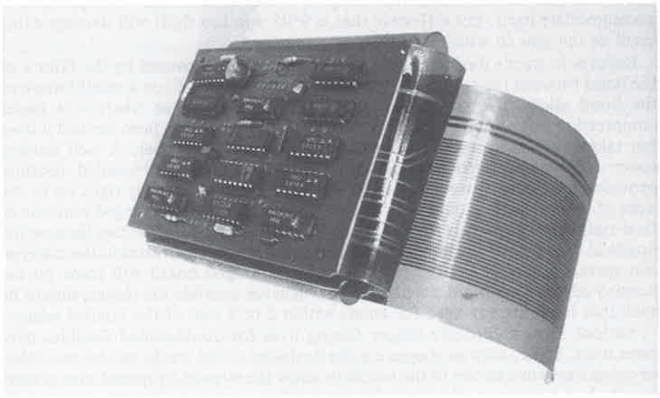


Fig. 9.6 Assembled and folded flexi-rigid board.

Cost may be a limiting factor, but while flexi-rigids are expensive to make they often offer benefits which are difficult to identify and quantify. In several instances known to the author flexi-rigids have proved their worth when engineers have been able to over-rule accountants and have been able to give the more expensive technology a fair trial in service. Probably the worst thing to do in a trial is to make a cost saving compromise using a simple flexi-rigid board to replace only some of the wiring. Even a small simple flexi-rigid can be expensive to design and make and if it has to be supplemented by discrete wiring many of the real benefits possible may be lost.

9.3.4 Designing Flexi-rigids for Maximum Life

Flexible portions made with 1 oz copper on a 0·05 mm base with 0·05 mm cover coat can withstand very large numbers of bends through plus and minus 180 degrees without failure, but double-sided flexibles have very limited lives when bent round small radii. Tracks can crack after only 20 or 30 full bends on a 2 mm radius, so double-sided flexibles should be used only for 'bend on installation' boards. Whenever possible such flexi-rigids should be supplied for assembly either with a rigid bar left uncut beside the flexible part or with one of the outer layers uncut. This prevents every person who handles the finished board bending it a few times to see how flexible it is.

When a hinged board design requires more tracks to cross a hinge than can be accommodated on one layer, it is better to use two separate single-sided hinges than one double-sided. The two hinges must be made long enough to allow space for the flexible which will be on the inside of the bend to form a kink between one of the rigid areas and its bend.

Hinges or tails should always be designed over length so that no possible combination of tolerances or movements of the interconnected parts can result in any strain on the flexible portion, or the formation of excessively sharp bends. An overlength flexible can always adopt a 'hump back bridge' configuration to accommodate itself, but a flexible that is 0·05 mm too short will damage either itself or the pins to which it is soldered.

Failures in tracks during flexing appear to be mainly caused by the failure of the bond between the copper track and the base laminate. Even a small failure of the bond allows the copper to kink away from the base when it is under compression on the inner side of a bend, and once a kink has been formed it does not take many more cycles of bending to fracture the copper. A well bonded cover coat can help to prolong the life of tracks on double-sided flexibles provided the bond between the cover coat and the base extends right up to the sides of the tracks. It should be noted that track failures in the hinged portions of flexi-rigids are more likely than they are on long, unrigidised flexibles because the bonds at each end of the hinge limit the length over which stresses in the material can spread. Some of the bending stresses in a hinged board will come on the bonded edges of the rigid portions, and whenever possible the design should be such that it avoids any need for bends within 2 or 3 mm of the bonded edges.

Various ideas to procure longer flexing lives for double-sided flexibles have been tried. Some, such as staggering the positions of the tracks on the two sides, or using sine wave curves in the tracks to allow the stresses to spread over greater length, had a measure of success, but gave a lower track density than could have been achieved on one layer with straight tracks.

Whenever possible tracks along portions of the flexible which will undergo bending should be run straight and the edges of the portions should be straight and parallel. Even with a cover coat bonded on, the polyimide material is not difficult to tear, and if a tear does start it is unlikely that it will be stopped by a 1 oz copper track. The best precaution against tearing is to design the flexible such that there are no points along its edge where any stress concentration could start a tear. Any change in width or incut should be avoided whenever possible. On most designs a radius will have to be allowed for at the point where the flexible portion joins a rigid area, but this radius should be in the portion of the flexible which will not undergo any bending.

If a change in width or an angled portion is essential to be able to fit the flexible into its case, then the area round the bend in the edge should be reinforced with an extra piece of cover coat bonded on locally.

For maximum flexing life the copper tracks should be as thin as is practicable. Ordinary 1 oz copper appears to have an excellent life provided it is properly sandwiched between polyimide layers of equal thickness, and some boards on which the copper of the tracks has been plated up in through-plating holes in the ends of tails have given satisfactory service for several years. However, it is an obvious fact that the thicker the copper is made the greater will be the risk of failure, so whenever plated-through holes are to be used in flexible tails the holes and pads should be selectively plated. As well as increasing the risk of a track failure during flexing, any plating on the tracks will make it more difficult to bond the cover coat down over them without voids beside the tracks.

Chapter 10

BOARD FINISHES

10.1 BARE COPPER

Many non-plated printed circuits have been used in their 'as etched' condition, with bare copper tracks. Provided they are assembled and soldered very soon after they are made, satisfactory joints can be formed. Sometimes this means the use of an activated flux which can leave corrosive residues, and the tracks corrode fairly rapidly in use even if a non-activated flux is used or if all flux residues are cleaned off. There are many non-critical applications where such corrosion can be tolerated, and, since the cheapest of all finishes must be no finish, the use of bare copper should not be dismissed automatically. In prototype or laboratory work the use of bare copper boards can mean the saving of time and money, but to realise these savings, soldering should be done virtually as soon as the boards have been etched and the resist is cleaned off.

For the great majority of printed circuit board uses the rapid loss of solderability of bare copper pads and the longer term corrosion of unprotected tracks are unacceptable and some form of surface finish is usually regarded as being essential.

10.2 LACQUER[2]

Since the earlier 1960s a number of lacquer finishes for boards have been tried. Most are based on compounds which degrade at soldering temperatures or are dissolved by fluxes at elevated temperatures. Applied over the bare, clean copper of a newly made board these lacquers are intended to preserve the solderability of the pads until the board is assembled, and to offer long term protection to the copper tracks.

In some instances these lacquers have been successful, but in others they have not lived up to their claims. It is probable that in some cases the fault lay not in the lacquer but in the treatment of the copper prior to the application of the lacquer, which resulted in the pads being of poor solderability. In other cases it appeared that the lacquer had pinholes through which corrosion of the copper occurred. Possibly the fault lay in attempts to apply an inadequate thickness, or in a surface treatment of the copper which resulted in its not being adequately wetted by the lacquer. In the worst cases, the lacquer could be removed by the flux in the first few days after its application, but it then appeared to over-cure and to defy any reasonable attempts to clear it from the pads.

In all cases known to the author, the availability of alternative finishes at comparable prices led to the use of solderable lacquer finishes being abandoned with little or no effort to discover the causes of or cures for poor performances. Since flux compatible wire insulation has been in use for some fifteen or more

years there can be little doubt that a satisfactory lacquer finish for boards should be achievable.

One apparent disadvantage of lacquers is that they offer the user no guidance as to the solderability of the copper they cover, whereas most tinned finishes will show up any cleaning or plating defects in the copper which might cause problems in soldering.

The flux compatible lacquers should not be confused with a 'liquid wrapping', a curable or solvent evaporating liquid applied to a newly made board which dries to form a tough protective coat which can be peeled off immediately before use to expose clean, solderable copper pads. Like the lacquer finishes, liquid wrapping is in use in a few special cases, but generally the electronics industry has preferred to use other finishes for protection and solderability.

10.3 ROLLER TINNING[1,2,46]

The commonest finish for non-plated boards is roller tinning in which the etched and cleaned board is lightly coated with flux and passed between a pair of rollers, the lower of which runs in a bath of molten solder. This roller transfers a thin coating of tin-lead to all exposed copper surfaces on the board, which protects the copper against corrosion and leaves all pads ready for soldering after assembly. If the board has a solder resist coating over the tracks, then only the pads will be tinned. The process is best operated on a continuous basis with the solder in the bath allowed to cool and solidify between batches so that oxidation of the molten solder is minimised. By its nature, the process is easily incorporated in an automated, conveyorised line, with a foam or brush fluxing station, and is capable of handling large numbers of boards at minimal labour costs.

Applied properly to clean, solderable copper, roller tinned boards are usually guaranteed for up to six months storage prior to assembly and soldering, but in practice it can be found that good solderability may be retained for periods of four or five years or more under favourable storage conditions. It is not unusual to find electronics laboratories in which pieces of prototype boards of unknown age are in use, but no board manufacturer can be expected to give any guarantees on such long-term retention of good solderability. For bulk production, six months of storage represents a significant amount of money tied up in stored boards, so it is unlikely that long-term solderability should present a serious problem.

Generally, roller tinning leaves a fairly uniform coating of solder, with a slight tendency to leave a thicker deposit near the trailing edges of larger features. Small holes, under 0·8 mm in diameter, may occasionally be partly blocked by solder, but these are easily cleared, and often the component leads will push off any such obstructions as they are inserted.

Failures in roller tinning usually indicate that some fault is present in the manufacture of the board. Warped or bowed boards may not make good contact with the whole of the width of the roller, leaving areas untinned. Any lack of good solderability of the copper surfaces is usually shown as the solder pulls back from the affected areas to leave pinholes with a broad contact angle at their edges, showing that the copper has not been wetted by the solder. Over thickness boards may have the solder squeezed off by the rollers to leave an inadequate layer which forms a grey coating of copper-tin intermetallic instead of the glistening coating of solder normally left.

If the machine is set to leave too thin a coating of solder there may be longer term problems as the tin in the solder diffuses slowly into the base copper to leave a lead rich surface which can oxidise.

The solder in a roller tinning machine can become contaminated if plated boards or other metals are passed over it. Gold is the worst of the likely contaminants, followed by cadmium or aluminium. Copper from the board dissolves in the solder, and accumulations of copper of around 1 part in 250 cause the solder to become gritty and unusable. However, in normal use the rate of dissolution of copper from the boards is more than balanced by the need to replenish the bath with clean solder as the boards are tinned, so the percentage of copper in the bath should remain well within acceptable limits over long periods.

The exposed surface of the solder is often covered with a blanket of molten rosin which minimises oxidation of the hot surface, and keeps down the formation of particles of dross which could be carried round by the roller and pressed into the board surface.

If the boards to be tinned are clean, adequately fluxed and coated with about 0·003 mm of tin-lead, some of the solder will flow over the edges of the pads and tracks. The amount of this edge coverage will vary from plant to plant, depending on the machine settings. The use of a pre-heat stage before the actual roller tinning can help the flow of the solder over the edges of features. Sufficient edge coverage to prevent corrosion in adverse environments is unlikely to be achieved other than on boards made from thinner than the normal 1 oz copper cladding.

A roller tinning machine with a conveyorised pre-heat stage generates a considerable amount of heat and fumes from the flux, so good air extraction over the machine is needed.

10.4 SOLDER LEVELLING[2]

Solder levelling is an alternative to roller tinning on unplated boards, and it can also be used on copper finished through-hole plated boards. The boards are coated with solder by immersion in a solder bath, and then the excess solder is blown off the boards and out of the holes by hot air knives. The air is blown onto the board in a thin, flat jet which leaves a uniform coating of solder over the faces and edges of features and through the holes.

As the name suggests, these machines can leave a more uniform coating of solder than is usually achieved with a roller tinning machine, and edge coverage is generally good.

There can be a tendency for solder levelling machines to spread the solder coating over small areas of copper with reduced solderability, thus masking a defect which could become apparent during soldering after assembly. Like roller tinning, solder levelling can not hide any large defective areas and any bridging of the solder over small local spots should not have serious consequences.

Solder levelling is commonly used on through-hole plated boards made by additive or semi-additive processes which leave holes and pads with a copper finish. The process is sometimes applied to boards which have been pattern plated with tin-lead as the etch mask. If it is considered essential that there be no solder on the tracks beneath a solder resist, the tin-lead is stripped from the board after etching, the solder resist is applied, then the holes and pads are re-tinned by solder levelling. This ensures that all faces and edges are adequately protected without any of the tin-lead being covered by solder resist.

Both roller tinning and solder levelling can suffer from the effects of pumice or other grinding or scrubbing agents which may become embedded in the copper surface during processing. Copper can be burnished over these inclusions to give an apparently clean surface which tins well, but on subsequent wave soldering significant areas of the copper de-wet. This happens because the very thin layer of copper over the inclusions dissolves in the solder so that on re-melting the solder the unsolderable inclusions are exposed and the solder draws back to leave the sharp angled edge indicative of poor wetting. The cure for such problems lies in the elimination of the inclusions, usually by replacing mechanical cleaning processes with chemical processes such as mild etches.

10.5 TIN-LEAD PLATING[2,8]

Tin-lead plating is the most common 'as-made' finish for through-hole plated boards. On pattern plated boards the solder alloy is plated directly onto the clean electrodeposited copper so that with good clean rinsing there is almost no way the copper surface can become unsolderable. The two biggest hazards with tin-lead plating are the risks of burnt plating on isolated features, which should be overcome by good design and the use of suitable robber bars on the edges of panels, and the risk of slivers or whiskers being formed when the etchant undercuts the tin-lead to form an overhang.

Some etchants dull the surface of the tin-lead and a brightener bath may be used to restore the clean surface. Certain manufacturers use an immersion tin dip to leave a clean non-oxidising surface on the tin-lead, but since the earlier 1970s most tin-lead plated boards are reflowed.

Like a roller tinned coating, good quality tin-lead plating should retain guaranteed solderability for at least six months of storage under normal factory conditions. Because the tin-lead is generally used as the etch resist, the edges of tracks and pads are left uncovered, and can corrode during storage or service in adverse environments.

The thickness of plated tin-lead over a board can vary quite widely. Usually about 0·01 mm thickness of tin-lead will be plated on the hole walls in the centre of a panel or in the most densely featured areas, rising somewhat nearer the edges of the panel or in lightly tracked areas. There will be significantly more tin-lead on the surfaces of pads and tracks, depending on the current density during plating, which in turn depends on the size of a feature and the proximity of other features. On a board which is to be left with a tin-lead finish these variations in thickness will have no significant effect, but they can make satisfactory application of a solder resist coating more difficult.

10.6 REFLOW[2,8,47]

Tin-lead plating proved to be an excellent finish for printed circuit boards, but the risk of slivers of overhanging tin-lead breaking away from the edges of tracks and causing short circuits proved excessive. Such short circuits could be intermittent, and they could form to non-adjacent tracks, so their detection and cure could be very difficult and time consuming. The obvious solution was to eliminate the cause of the troublesome short circuits, which could be accomplished by melting the tin-lead after etching, which as well as eliminating

the slivers allows the tin-lead to flow over the freshly etched sides of the tracks and pads.

The first attempts at reflowing the plated tin-lead were made by dipping the boards in a hot oil bath, and these were followed by hydrosqueegees, which worked on a similar principle to the solder levelling machines but used hot oil jets instead of air knives to melt the solder. Both of these produced the desired result, but the oil had to be used near its flash point, and the earlier machines had a tendency to catch fire.

They were superseded by infra-red ovens, mostly now conveyorised. Infra-red reflowing of the solder is not as simple as it might seem. The wavelength used must be correct for the type of work being handled, and simply increasing the power of the heaters can result in less effective heating of the boards. A good reflow machine should heat the board uniformly over its entire area, and the heating should be controlled so as to be adequate to reflow the solder without subjecting the board to a high temperature for any longer than is necessary. The heat or the conveyor speed must be controllable to cater for the quite large differences in thermal masses between a lightly tracked double-sided board and a densely packed multilayer. The boards are coated with flux before reflow, and then after heating the flux residues are cleaned off. If hot boards, especially multilayers, are taken straight from reflow, and immersed in relatively cold cleaning solutions, thermal stresses can be locked into the boards and severe warping or twisting can occur. Thermal shock should be avoided when the boards are above the glass transition temperature.

More recently, increasing interest is being shown in vapour phase reflowing of boards. In this process, the fluxed board is placed in the hot vapour from a boiling, inert fluorocarbon liquid, and the vapour condenses on the board, giving up its latent heat of evaporation to the board and thus melting the solder. The boiling point of the liquid need not be far above the melting point of the solder, so the system minimises any risks of over-heating the boards.

Infra-red systems are mostly conveyorised, so lend themselves to large-scale production, whereas vapour phase systems are at present batch tanks, although there would appear to be no reason why they should not be automated.

Reflowing eliminates all tin-lead slivers and it covers the edges of all features. It also tends to leave only a very thin coating of solder right on the shoulders of through-plated holes with the solder thickening up in the middle of the hole barrels and on the tracks. Rumours have been heard of reflowing causing complete closure of holes in boards or making component insertion impossible. However, a few simple calculations show that for a hole to be closed completely by reflowed solder moving from the ends of the holes to the middle, either the original tin-lead plating must have been grossly over normal thickness or extra solder had been added.

The movement of solder away from the sharp corners at the ends of the holes can result in the solderability of a very small ring round the edge of the hole being impaired because of the formation of copper-tin intermetallics, but any such loss of solderability is confined to a very small area indeed and is of no practical significance. The effect which can have some significance is the slight closing of the holes near the centre which can make it difficult for a goods-in inspector to verify the hole diameter, and it is also no longer possible to measure accurately the thickness of tin-lead which was plated onto the boards. Neither of these difficulties in accurate measuring should pose any problems in practice.

One advantage of reflowing is that it thermally stresses boards, and any manufacturing weaknesses will be revealed, especially in the bonding of multilayer boards and in through-hole plating. Also any defects in the plating processes which might cause subsequent solderability problems will be revealed by the reflowing so the customer can be more certain of the quality he is buying when boards have been reflowed. In the earlier 1970s reflow was often regarded as an optional extra on tin-lead plated boards, but in the 1980s it is regarded as part of the normal process of manufacture for good quality boards.

10.7 GOLD AND PALLADIUM-NICKEL

10.7.1 Gold

In the earlier 1960s gold plating was often used as an etch resist and finish for through-hole plated boards. It made a good etch resist, was not difficult to plate, appeared to solder well, and it gave boards an attractive appearance with some measure of corrosion resistance. The gold used was not thick enough to give complete protection against corrosion, but it did prove thick enough to cause the formation of intermetallics of gold and tin in the soldered joints in the hole barrels—the now well known gold embrittlement of solder. In the earlier years, most of the effective joint area was on the surface of the boards and much of the gold could be washed away with the surplus solder involved in making mass joints. However, as pad sizes decreased and more reliance was placed on joints inside the holes, the gold plated into the holes, and often on the leads of the active devices, was trapped in the holes and unreliable joints became a major problem.

Most companies had abandoned the use of gold as an all over finish by 1970, and during the next decade rises in the price of gold confined its use on boards to edge connector or switch contacts.

Even in these relatively small areas, the cost of gold plated onto boards is far from insignificant, and efforts continue to find a reliable alternative which will offer corrosion resistance, low electrical resistance, good wear properties and ease of application, with the main interest in palladium-nickel alloys.

10.7.2 Palladium-nickel

Palladium-nickel alloys are being investigated as possible substitutes for tin-lead as an etch mask and as an underlay for a thin plating of gold on edge-connector contacts. Pore free palladium-nickel can be plated at less than 0·0002 mm thickness. This thin coating does not have the mechanical strength to form slivers as the etchant dissolves the copper from beneath it, and after etching edge connector fingers require only a thin flash of gold to form a good electrical contact with excellent corrosion resistance.

However, its solderability does not appear to match that of clean copper or of tin-lead finishes, and the application of only thin gold over palladium-nickel which has been used as an etch resist leaves the sides of edge connector fingers with no real protection against corrosion.

Investigations into the use of palladium-nickel are in their early stages. It would appear that its use in high reliability equipments may be limited, but it seems that for commercial equipment the use of palladium-nickel as an etch resist and underlay for thin gold on edge connector fingers may offer significant

savings on board costs. Any such savings in the future must be related to the world metal markets, and any large swing to the use of palladium instead of gold could lead to price changes.

10.8 SOLDER RESIST[2]

Solder resist is a coating added over the tracks on boards, and over all features which are not intended to be soldered or which form contact fingers. Its prime purpose is usually regarded as being to prevent any risk of short circuits or bridges being formed between soldered pads and adjacent tracks. It is sometimes thought of as a coating to prevent corrosion of the tracks, and it has also been described as a means of saving money by preventing valuable solder from being deposited on tracks during wave soldering. Finally, applied to the component side of a board, it can form a better background for a legend than is offered by uncoated closely spaced tin-lead plated tracks.

There is some element of truth in all the above virtues of solder resist, but there is also some element of fiction. Possibly the least controversial aspect is the use of solder resist as a background for a legend, where there can be no doubt that it gives a smoother surface on which to print the legend and its single colour, often a dark green, does allow the legend to be read more easily.

Savings in costs by the elimination of solder pick-up in mass soldering will undoubtedly be realised, but without some other benefit they will not be sufficient to justify the cost of applying the solder resist.

Before the other two reasons for its use can be considered, it is necessary to review the types of solder resist available.

10.8.1 Types of Solder Resist

Solder resists can be divided into three main classes: those which are screen printed onto boards and subsequently dried; dry film resists which are applied by roll laminators and then photographically developed to leave the required mask pattern, and photographically developed resists which are applied as a liquid and exposed when dry.

Screen printable resists are usually epoxy resins, which are oven cured. Registration of the screen print to the copper pattern is not usually very precise by modern board manufacturing standards, and the allowances for this plus those for screen distortion and the flow of the ink mean that it is customary to leave an annular gap of between 0·3 and 0·4 mm between the edges of pads on the board and the nominal edge of the solder resist pattern.

The resist can be allowed to encroach slightly onto the pads without causing any harm, but any such overlap should be kept to a minimum. If any of the solder resist gets into the plated-through holes it can cause problems in soldering, and where the resist overlaps pads there can be a risk of the screen not snapping off cleanly and flooding of the resist occurring. Obtaining a clean print with good coverage of all tracks is not easy on the uneven surface of a typical high density board and the mesh of the screen must be carefully chosen and the printing frame properly set if good results are to be achieved.

Care must be taken in the racking of boards carrying a resist which requires oven drying at temperatures above the glass transition point of the epoxy resin. If the boards are not properly supported, they can warp during stoving.

A gap of 0·3 mm between the edge of a pad and the edge of the solder resist was unlikely to be of any significance when gaps between pads and tracks were comparable, but in the 1980s the use of tracks and gaps of 0·15 mm or less makes the use of a solder resist which can leave a gap of 0·3 mm plus registration tolerances of rather dubious value.

Dry film resists do not require such a large gap to be left to allow for tolerances, and they are often exposed with a mask which has apertures the same size as the pads on the board. Since two different exposures are involved in printing the pads and the resist, there must be some risk of misregistration, but this can be allowed to show as overlap of the resist on the pad, as there can be no risk of any resist being left in the holes or of flooding occurring.

Suitable dry film resists can be exposed, and if the boards are then wrapped the resist can be left over the pads until immediately prior to assembly, when the resist which will have protected the tin-lead on the pads from oxidation is developed and washed off the board.

Dry film resists are in many ways a great improvement over screenable resists, but they can pose problems in their adhesion to the board. For the best results they should be applied to oxidised copper finish tracks, and after the resist has been developed the holes and pads can be cleaned and tinned by solder levelling. Like the cover coats for flexible circuits, it can be difficult to get a dry film resist to conform well over a plated-up pattern of narrow tracks and gaps.

The photo-curable liquid resists combine the best features of the other types in that they can be applied over a dense track pattern with no worries about their not filling the gaps between the tracks properly and they can be exposed with a size for size pattern. Care is needed in their developing and rinsing to ensure that all traces of resist are removed from the holes.

10.8.2 Effect of Board Finish

Screen printed resists gave excellent results on non-plated boards on which the pads were roller tinned after the resist had been cured, and the photo-exposed resists also give good results on bare copper finished boards with holes and pads tinned by solder levelling. However, there can be problems when the resists are applied over tin-lead plated tracks. When the boards are wave or drag soldered, the tin-lead below the resist melts and flows, and the very thin layer of resist breaks down to a pattern of fine cracks. Often an adhesion test with a piece of Sellotape will remove most of the resist from the tracks after mass soldering.

This failure of the resist is worst on broad tracks, so it is common practice to leave all broad tracks on tin-lead plated boards uncovered by the solder resist. This does allow solder to collect on the tracks during wave soldering, but it can be argued that since the presence of this excess solder must lower the electrical resistance of the tracks it is more of an advantage than a disadvantage. It can also be argued that the breakdown of the resist over the tracks is of no consequence since the resist has by then performed its job in preventing the formation of bridges between pads and adjacent tracks. This argument assumes that the tracks are far enough from the pads to have been covered by the resist.

Breakdown of the resist on soldering can be avoided by the use of a solderable metal finish which does not melt at normal soldering temperatures. Tin with a very small percentage of lead can be used on pattern plated circuits, and it is possible that palladium-nickel may be used as an etch resist and track finish.

However, it seems that the most likely process to be used in situations where an undamaged solder resist is considered important is bare copper tracks either with selectively plated pads, or with the pads tinned by solder levelling.

10.8.3 The Need for Solder Resist

Well designed boards with gaps of 0·25 mm can be soldered without resist with no risk of bridging, so it must be regarded as questionable whether solder resists are strictly necessary on many boards. On some boards the design may not be suitable for soldering without resist; companies may prefer to use a resist and to take less care in soldering than would be necessary if no resist were used. There are many arguments both for and against the use of resists, but, as tracks and gaps become smaller, so the need for a good resist increases.

Solder resist offers some degree of protection against contamination. A dry film resist with poor adhesion to the metal of the tracks or which has not conformed right into the corners beside the tracks may not be effective in preventing corrosion of bare copper tracks in an adverse environment. Any resist which has crazed when tin-lead plated tracks are wave soldered will clearly be of little value in preventing corrosion, but the tin-lead offers some protection to the tracks. A screen printed resist may have only a very thin coating over the edges of tracks, and this may not be adequate to prevent corrosion of bare copper tracks. In general, though, it does seem that a dry film resist properly applied will offer the best long-term protection to the tracks it covers, with the photo-exposable liquid resists probably as good provided they are applied in sufficient thickness to give good coverage to the corners of tracks.

10.9 LEGENDS

Most printed circuit boards have printed on the components side a legend which describes the position of the components. To be of any value, the legend must be printed in an ink which will resist attack by commonly used cleaning fluids and solvents, and which is sufficiently durable to withstand a limited amount of mechanical abrasion and handling. The commonly used inks are epoxy resin based, and are similar to solder resists.

Legends are usually screen printed onto the board, and then oven cured. Most legends are printed in white or yellow so they can be easily read over the base laminate or over copper or tin-lead finished tracks. As mentioned in Section 10.8, the presence of a solder resist can help to make a legend more easily readable.

Legends are useful in assembly, when they identify all component positions, and also in field servicing.

However, as boards become more densely packed, there must be doubts about the real value of a legend. Many boards now have their components inserted by automatic machines which 'read' a punched or magnetic tape and not any printed legend. Even when automatic machines are not used, it has been suggested that assembly staff have learned the board pattern after they have assembled the first five or six boards. It is not difficult to find alternative ways of presenting assembly information to staff. A projector can be fitted over the work station to project an image from a legend film onto the board, or the information can be in the form of a drawing based on a print of the component side track pattern film.

On low density boards there is sufficient space beside components to print a legend which a field service engineer can read, but as the packing density increases so it becomes increasingly difficult to find space between components for legend characters, and in many cases the legend has to be printed below the bodies of the components. Such a legend can advise a service engineer which component he has removed only after he has removed it, so its value is dubious.

On low density boards with broad tracks and gaps it is usually possible to arrange a legend so most of the characters are printed either over the tracks or in the spaces between them. As track and gap sizes decrease, so it becomes increasingly difficult to find flat areas on which to print the legend, consequently characters have to be placed over the edges of tracks. Even with a solder resist coating to give the board a more uniform surface and to show off the legend, it can be difficult to achieve clear printing on such non-level surfaces, especially when the characters are small as will usually be the case on high density boards.

When a legend is being designed, it is most important to avoid placing any characters near any holes in the board. If a character is placed too near a hole so that slight misalignment during printing allows any ink to come through the screen over the edge of the hole, that ink will not be left on the board when the screen snaps up behind the squeegee and after a few boards have been printed a blob of ink will have gathered on the underside of the screen, and sooner or later such a blob will spread over the face of a board, ruining the print. On densely tracked double-sided boards the spaces between components may appear to be sufficient for a good legend, but the presence of via holes can make large areas of these spaces unusable to the point where the design of the legend can become the most difficult part of the whole design work. If the design requirements include an instruction to avoid placing characters over the edges of tracks, it can become impossible to produce a meaningful legend even though the package placement sketch apparently shows adequate space.

One point to be watched when designing or specifying a legend for a board is to avoid the use of black inks. These are liable to contain carbon black, and, if the legend is printed over bare copper or tin-lead finished tracks, the end result can be a collection of unintended resistors connected randomly between odd points on the circuit.

On many digital boards the need for a printed legend in the component area can be avoided by using alpha-numeric designators for the rows and columns of integrated circuit packs. Larger packages, such as microprocessors, may occupy as many as four of the designated spaces, but they will take the designation of the lowest numbered space they occupy. Resistors and capacitors are often used at less than one of each per integrated circuit position, so they can be called up by reference to the nearest integrated circuit position.

When such an alpha-numeric grid is used over a board, it helps assembly and service staff if all decoupling capacitors can be placed in a standard position relative to the integrated circuit packages.

On some board types there may be sufficient space round the edges of the board to print the row and column symbols in copper clear of the track pattern; or in some cases the characters can be printed in negative form within broad earth or power bus tracks up the board edges. More usually a printed ink legend is likely to be necessary to identify the mounting positions for the almost inevitable discrete components which always seem to be necessary somewhere in most large digital circuits.

High densities and automated assembly may remove the need for legends for some types of boards, but over the whole industry it is probable that screened ink legends will be in use for many years to come.

Chapter 11

INSPECTION AND TESTING

11.1 SPECIFICATIONS

Whether printed circuit boards are made in-house by a large systems company or are ordered from an outside supplier, some form of specification is needed so that all concerned can verify that the boards have been made correctly and will be capable of fulfilling their intended function. The specifications used vary widely from company to company. Some firms prefer to use a brief, simple document which refers to national or international standards; others prefer large volumes which detail all possible points.

It may seem that specifying a printed circuit board should be a fairly simple job. The master films define the track patterns, hole sizes and positions; and overall size and thickness are easily defined, as is the laminate to be used. Then a few more questions arise. How closely can the widths of features on the board be toleranced? What are the limits on the strength of the laminate? Or its flammability? What constitutes good solderability? Can pinholes in tracks be accepted? What is the adhesion of the metal to the base? And many, many more. The 'fairly simple job' is very far from simple, and without reference to already existing standard specifications it is almost impossible.

11.1.1 General Standards

Probably the most widely used standard specification is the IPC600, Acceptability of Printed Wiring Boards, published by the Institute for Interconnecting and Packaging Electronic Circuits. This specification, which is currently at issue C, is sometimes referred to directly in company standards used by large multi-nationals, and sometimes information from it is rewritten as a company document.

In the United Kingdom, the normally used standard is BS9760, Printed Wiring of Assessed Quality. Unlike the IPC document, which concentrates on illustrating good, acceptable, and not acceptable practice, the British Standard is directed towards classifying the capability of a manufacturer to produce boards of adequate quality. Board suppliers have to prove their ability to make good boards to defined limits of feature sizes by making special panels which are cut up and tested and submitted for approval to EQD, the Electrical Quality Assurance Division of the Ministry of Defence. The manufacturer must also submit quality and capability manuals, and demonstrate the records of calibration of all equipment involved in testing the samples from the approval panels. When EQD are satisfied that the factory has the capability to produce good boards and to verify the quality, approval will be granted up to the limits of feature and overall sizes claimed. This approval then has to be maintained by regular EQD checks on test records and by periodic submissions of test panels.

The approval limits may not represent the ultimate limits of the manufacturer's capability. For instance, the approval might be for boards with tracks down to 0·2 mm wide. Boards with tracks 0·1 mm wide could be made, but can not be sold as complying with BS9760 unless an extension of the firm's capability is obtained.

The generic specification BS9760 covers all test methods used on boards, but the actual approvals are to sectional specifications which are BS9761 for multilayer boards; BS9762 for through-hole plated double-sided boards; BS9763 for non through-plated boards; BS9764 for non through-plated flexibles, and BS9765 for flexibles with plated-through holes.

In the United Kingdom, some boards for military use are covered by the Ministry of Defence DEF-STAN 5948, and in the USA the MIL series of specifications such as MIL-P-55640 are frequently used.

All these national standards make extensive references to other standards, and a full list of relevant documents covers several pages. Any such list is complicated by the continuing work being done to bring national standards such as BS and the German DIN series into line with European standards and into the IEC (International Electrotechnical Commission) series of documents.

11.1.2 Company Standards

A company which makes all its own boards and does not supply any to other equipment manufacturers can make good use of unique specifications, but when boards are bought or sold on the open market it is clearly advantageous to make the maximum possible use of national or international standards which should be familiar throughout the industry and have been proven as efficient specifications.

There can be pitfalls in the use of a simple drawing or specification with references to other documents. It is not unknown for a board manufacturer, who needs to know exactly what a customer requires, to find the customer's documents state '. . . all other features to BS9760' or similar, and to find on reference to the quoted specification that the detail required is called up 'as specified by the customer' or is referred to other documents which ultimately refer back to the customer's documentation.

Sometimes such references to national standards can lead to specifications which are impossible to meet, such as one current specification which can have the result of boards being designed on both metric and imperial grids simultaneously. It can, and does, take years for some specifications to be updated to match the advances in technology, and this can result in detail requirements being applied to features they were not originally intended to cover. One case of this was on line widths, which were called for in one specification as being 'to within 10 per cent of designed width'. At the time it was first written this was not unreasonable, but when applied to lines 0·05 mm wide it was almost an impossibility, and at the other extreme it was absurd when applied to heavy current bus bars 10 cm or more in width.

Any designer who wishes to use general standards in specifying a product should read the standards carefully and follow through all references to ensure that what he has called for is practicable and contains no 'closed loop' references which do not convey any factual information.

The practicability of using some specifications is becoming increasingly difficult as boards are made larger and with finer features. Reputable board

manufacturers are being faced with the dilemma of knowing that some clauses, usually on mechanical features, are virtually impossible to meet or even to measure to the limits specified. However, if they reject the order on the grounds that the specification makes the job impossible, it is with a virtual certainty that some less scrupulous supplier will take it, and will probably supply boards which will satisfy the customer even though they may fail to meet the literal requirements of the specification.

One golden rule in drafting any company specification is to keep it as brief as possible, which is usually accomplished by making the maximum use possible of references to relevant national standards. Quality staff in any supplier's plant should be fully familiar with these standards, and the less extra information they have to read and understand to be able to appreciate the customer's requirements the more likely it is that those requirements will be met. Two or three bulky folders of extra specifications may look most impressive but they are unlikely to result in efficient production or inspection. In some existing cases, it can take an inspector longer to read the specifications than to check out a full batch of boards.

11.1.3 Concessions

Even in the best run manufacturing plants working to well compiled specifications, boards can be made which do not comply with the specification but which are apparently usable. When this happens, the manufacturer may apply to the customer for a concession which will allow the boards to be used. A customer whose prime concern is to make and sell usable, reliable equipment will often grant a concession rather than see his production delayed while the boards are re-made. Possibly some adjustment in price may be agreed.

Any such concessions granted should be examined closely to see whether the specification for the board should be changed. If it is possible to use a board which does not comply with all the terms of the specification, then there must be a probability that it is the specification which is too stringent rather than the board which is faulty.

There can be cases in which characteristics of the board are interdependent, and if one is well away from its limit another can exceed its limit without any adverse effect. For instance, if a pair of holes are near but within their top limit in diameter, their centre distance could be outwith its tolerance but a rigid lead component will still fit. Such cases suggest that some multi-dimensioned features may be better checked with Go-No Go gauges than by direct measurement of all the dimensions involved.

Sometimes a concession may be given to accept repaired boards. The granting of such a concession raises the question of why the original specification failed to consider the possibility of such repairs.

11.1.4 Fitness for Use

Some specifications or drawings do not specify what the customer requires, but instead concentrate on how the designer thought a desired result should be achieved. There are cases in which such a form of specification is desirable or even essential because the desired end result can not be measured, but in the majority of cases such specifications fail to achieve what should be their main

object of defining a properly usable product in such a way that it can be made at the lowest cost possible. Generally the choice of process to be used is best left to the board manufacturer.

Track widths and metal thicknesses are often afflicted by over-specification. It is not uncommon to find tight tolerances whose sole purpose is 'to ensure that the board manufacturer does a good job'. In one classic instance a manufacturer had to re-make a batch of boards on which a track 0·5 mm wide was locally notched down to 0·38 mm because of a defect on a printing film. It had to be admitted that the defect should not have occurred and the boards did not meet their specification. The customer refused to accept the boards, although the nominal width of the same track was reduced to 0·35 mm within a few cm of the defect, making it obvious to all that the boards were perfectly usable.

Often it turns out that such over-specification is the result of the design authority having little or no idea of the electrical function of the tracks, so the limits are arbitrarily chosen 'to indicate good practice' instead of being fitted to the actual situation. Money can often be saved by the avoidance of unnecessarily tight tolerances.

With the rise in popularity of automatic insertion machines it is not uncommon to find hole centre distances specified at tolerances which approach the natural material movement of the laminate over the centre distance. Since often there is no mention of the temperature at which the distance should be measured, the quoting of such tolerances does little more than raise serious doubts about the competence and relevance of the rest of the specification.

Sometimes if a specification is queried, the answer is in the form of 'because it is our company practice', and further questioning reveals that it has been so for many years. Very often in such cases it turns out that the factor which originally caused the clause to be written has long since been overcome by newer technology, but successive generations of specification writers slavishly copy the old clause without any thought as to whether it has any true relevance.

11.2 INSPECTION

11.2.1 Sampling

To be absolutely certain that every board made meets its full specification, every board must be fully inspected, and for real certainty it is probable that at least double checking would be needed to eliminate risks of errors by the first inspector. Clearly the cost of such absolutely sure inspection would be excessive, so much of the industry relies on sampling inspection to ensure a high probability of adequate quality at a reasonable inspection cost.

Printed circuit boards have to go through a number of different process stages, as shown in Figure 5.1, where any error made on one stage can mean that all subsequent work is wasted, so much of the inspection has to be carried out on-line to minimise the costs of work undertaken on defective boards. Since many of the processes are effectively batch processes where an error on a single board is unlikely much of the on-line inspection can be done on a sampling basis.

Sampling inspection can be performed on a basis of taking boards from a line at regular intervals, such as every tenth board, and when a defect is found, stopping all boards since the last correct check. Checks on processes such as plating may be made on a time interval, or random samples may be taken, with

the number of samples varying from job to job depending on the complexity of the boards. Sampling is usually best carried out to a mathematically based plan which offers well defined risks of defective boards remaining undetected.

The best known of such plans is that defined in the American MIL-STD-105D, Sampling Procedures and Tables for Inspection by Attributes, which was also adopted in BS6001. The use of these tables and the levels of risk inherent in their use is fully described in BS6000, the guide to the use of BS6001.

Very briefly, the tables are used on the basis that absolute perfection can not be guaranteed and would be prohibitively expensive to verify, so a limited, randomly selected sample for a batch is inspected and the entire batch is accepted or rejected depending on the result. The customer and supplier agree on the Acceptable Quality Level or AQL, which for printed circuit boards would usually be 0·65, 1·0 or 1·5 (per cent). The sample size to be inspected varies with batch size and AQL. For a typical multilayer board delivery of 281 to 500 boards, to an AQL of 1·0, the sample size would be 50 boards. For AQLs of up to 0·4, the entire batch must be rejected if a single defective board is found in the sample. To 1·0 AQL the batch is accepted if the sample contains 1 defective board and rejected at 2; whilst for an AQL of 1·5 two defectives are allowed, but 3 is cause for rejection of the whole batch.

Different inspection levels are available so that if a supplier is offering a consistently good quality the level of inspection can be reduced, but when problems arise it can be increased. On small batches or for very low AQLs the size of sample given in the tables may be larger than the batch size, so the inspection will have to be 100 per cent.

The principles of AQLs and of sampling inspection are not difficult to understand when BS6000 is read, but it must be appreciated that sampling inspection to, for instance, an AQL of 1 (per cent) does not mean that there will be only 1 defective board per 100 delivered, but that to a high statistical probability the defectives will average less than 1 per 100 over a period. Clearly, with a batch of 500 boards, there must be some very small but finite possibility that the 450 boards not included in the sample are all defective but the 50 good boards in the sample say the batch must be accepted by the customer.

Defects may be categorised as major or minor. A major defect is one which renders the product unusable, whereas a minor defect is one which indicates some manufacturing fault but which still leaves the product functional. Minor defects may be allowed to a higher AQL than major defects, or they may be added on a basis of perhaps four minor defects being counted as equivalent to one major.

It can be argued that an ability to consider any minor defect as acceptable is an indication of over-specification, or that if any single minor defect still leaves a board usable, or can be said to have zero effect on the usability, then an infinite number of such defects should still leave the board usable, albeit very badly made.

The sampling tables can be used for on-line inspection as well as for finished goods inspection, but, when used on line, a sampling plan or AQL which rejects at 1 defect should be used.

11.2.2 Process Inspection

Inspection of printed circuit boards starts with a thorough check of the master film set for scratches, ragged edges to features, spots of unwanted black or any pinholes in dark areas. Pinholes are most easily found by laying a positive film

over its negative on a lightbox, with the whole of the top of the illuminated face covered and the room lights dimmed. The positive is then moved over the negative by as little as possible, in the direction of the run of the tracks. While the films are aligned, holes in the positive will usually correspond to spots on the negative, but the slight movement destroys the cancellation and allows light to shine up through the middle of a feature. Scratches and kinks are often most noticeable viewed obliquely. Size and squareness of the films should be verified, as described in Chapter 4.

Control tapes for NC drills and routers are verified by working a first off blank and comparing the result against a negative checking film.

Rigid laminates are usually checked for base thickness, copper thickness, copper adhesion and stability as they are delivered, with tests conducted on strips cut from the edges of randomly selected sheets. Prepreg material for multilayer boards should be checked for resin content, flow, and gel time immediately prior to use so that the press parameters can be adjusted to suit the test results.

Each operation on a batch of boards requires checking before the job is allowed through for further work. Much of the necessary checking can be done by the operators, who should have sufficient training to enable them to detect all likely faults, not only at their own process, but also from preceding stages. Such operator inspection should be backed up by a sampling quality audit carried out by QA staff. Whether this is better performed on a regular basis by staff assigned to a particular area or process, or by a roving team who make semi-random spot checks depends on the size of the plant and the grade of operators employed.

Drilling needs careful checking. When NC machines are used, it is advisable to check the bottom board of every stack drilled. The operator can do this by pinning the board against a verified accurate board, holding the pair up to a light, and viewing from both sides. This quick check will find extra holes, or holes missing because a drill broke or slipped back in its collet. Gross positional errors can be found, but marginally out of tolerance hole positions or sizes will usually be found only on a more detailed quality audit. The actual quality of the drilled holes and their suitability for through-hole plating are usually verified by plating and checking periodic test samples.

Track patterns should be printed under such conditions that a periodic sampling check will suffice to confirm that all is well. In addition to checks on the printed panels, the films or screens should also be checked for any scratches or other damage at regular intervals. The operator should look carefully at every screen printed panel as it is removed from the printing frame. Before the ink has dried, it can be wiped off and done again if there is any defect, but once an epoxy ink for a solder resist or legend has cured it is virtually unremovable.

All chemical processes are controlled both by the operator observing the work as it passes through, backed up by a quality audit, and by regular monitoring of solution strengths and temperatures backed by laboratory analysis of all solutions to check for any possible contamination.

Operator checking of plating and etching can be made very much easier if all process panels are provided with special checking features. Size check gauge marks of the type described in Section 4.1.3 can help in quick checking, the operator looking for faults as shown in Figure 4.3.

A more effective and easier check is possible if all panels are printed with process monitor gauges. A process monitor gauge is any pattern which will give a quick, easily determined measure of the effect of any process. The first use of

such gauges on PCBs known to the author was the inclusion within the board area of the full board number, printed on each side in the copper pattern, in characters with line widths and gaps significantly finer than any used in the working area of the board. If the number could be read clearly then there was a high probability of the board being usable.

A far more sophisticated gauge consists of a series of lines on constant pitch measured from one edge, as shown in Figure 11.1. The first line is 0·001 in. wide; the second 0·002 in., and so on to the last in the pattern which is 0·001 in. less than the pitch. The pattern repeats, so that in use one looks for the broadest line, beside which there should be a gap then a line both 0·001 in. wide.

On most practical boards either the gap will have been closed or the line will have vanished. A simple count of the number of gaps or lines missing gives the processing error in units of 0·001 in.

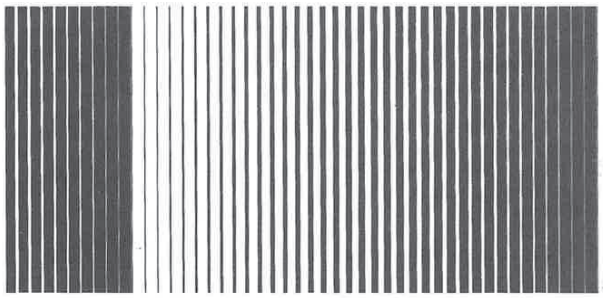


Fig. 11.1 Simple process monitor gauge.

Such a gauge can not detect all possible plating and etching faults, but if applied round the edges of all panels and in the spaces between step and repeat boards it will enable an operator to spot any change in a plating or etching stage. As features on boards are made smaller, the gauge steps can be reduced to 0·0005 in., or whatever smaller dimension is considered suitable. No special space need be left for such gauges, because they can be made any width suitable over about 1 mm, and, if space on a panel is really restricted they can be placed adjacent to the board edge in the path of the routing cutter. The use of such gauges by the operators should not replace in any way the regular quality audit of boards as they are processed.

The QA department can make use of the gauges as an overall monitor of all processes from film production on, and often when such gauges are first introduced the overall result comes as something of a shock, especially to some of the personnel engaged in compiling specifications.

11.2.3 Final Inspection

Most firms give all finished boards a 100 per cent visual inspection. If the boards are to be coated with solder resist or a legend is to be applied, the track

and hole patterns are checked first, and only the resist, legend, and points such as flatness are checked as a last stage.

Final inspection must also cover points such as plating thicknesses, solderability, track adhesion, verification of through-hole plating and the durability of the boards. Most of these involve destructive tests which have to be performed on test coupons or on a sampling basis.

Visual inspection for cut or notched tracks is one of the hardest parts of final inspection. Most inspectors appear to work on the principle of looking for any departure from normal rather than by scanning every track. Work is in progress on automatic scanning systems to relieve inspectors of the burden of checking tracks, but the results so far do not indicate that the detection of hair line breaks in tracks or small, deep notches is likely to be achieved by automatic means on large, densely tracked boards for some years to come. Optical systems can be made to enable the machine to detect such defects, but the problem lies in the enormous amount of information involved and the difficulty in distinguishing true faults from intended patterns, and in not rejecting a board which has not been placed on the bed of the machine in perfect registration. The errors to be found will usually be smaller than the probable tolerance in a normal dowelling system.

11.3 ELECTRICAL TESTING

The difficulties in being certain that a large, densely packed double-sided board is free of short circuits and cut tracks, and the obvious problems of final visual inspection of multilayer boards led to a growth of interest in the electrical testing of bare boards in the mid 1970s.

As bare boards are flat, or are substantially flat, testing on a sprung probe 'bed of nails' is not difficult. When all boards are designed with holes on a regular grid of 2·54 mm pitch a tester can be built with a full matrix of probes, and boards to be tested are provided with a plastic shield drilled with oversize holes from the NC tape for the board. The shield allows only the probes which lie over hole positions to contact the board.

Problems arise when a board carries components such as T077 packages whose leads do not lie on regular grid points. If boards are specifically designed to be tested on a grid pin array, the pads for such off-grid components can be extended to take in the nearest grid intersection, but it is generally better to have a tester which can be used to test any board likely to come along.

One solution is to use a unique head for each type of board, which can be satisfactory when large numbers are made of a limited range of board types. A unique head approach can become quite uneconomical in the average PCB jobbing shop. The probes can be re-usable, but the cost of inserting all the probes into the head and making connections to them makes a unique head approach uneconomic for general use.

The alternative is to use a fixed grid head with either a translation board or with flexible probes. A translation board can be a through-hole plated flexible circuit with pads on the grid points on one side and on the other side pads plated up with copper and gold which align with the pads on the board to be tested. The flexible probe approach uses long probes which pass through a plate drilled to the board pattern and which are sufficiently flexible to distort away from the base

grid. If the probes are mounted on 2·54 mm pitch the required distortion can not exceed 1·78 mm.

The use of high speed solid state switches allows a tester to check for continuity from each probe in turn at speeds of up to 2000 points per second, which for most types of boards means that load and unload times will be longer than the actual test times. Most testers work by memorising the interconnection pattern of a known good board, but it is possible to arrange for a tester to be driven by a tape from a CAD system so it tests all boards to the required pattern, thus eliminating any risk of a film error making all boards including the 'known good board' faulty.

High speed testers usually work at around 10 Volts, and test only for short or open circuits. Higher voltage tests to stress insulation, and current measuring tests can be arranged, but only on much slower testers, and the value of such tests is questionable.

11.3.1 Electrical Testing and Visual Inspection

High speed electrical testers are sometimes sold or bought on the basis that they will be able to replace or partially replace visual inspection. They can and do make a fully detailed visual inspection less important, but there are defects which can not be detected by electrical testing. A very high voltage test might find a particle of metal between tracks, but the normal low voltage high speed tester is capable of passing boards with very small gaps indeed. Even if the actual resistance of all tracks is measured, an electrical tester can not distinguish between a track on its upper limit of width which is almost totally cut through at one point and a perfect track near its lower limit of width tolerance. Faults such as non-adherent plating in holes or lifted tracks will pass undetected.

For commercial applications it is probable that the use of an electrical tester may enable visual inspection to be done on a sampling basis, possibly at an AQL of 0·65, but in cases where reliability is important full visual inspection will still be needed. The electrical testing gives the customer an extra assurance that he is buying good boards, but it can not replace in-process and final visual checks.

11.4 TEST COUPONS

Many of the tests which need to be carried out to ensure that boards are of good quality and durability are destructive, so have either to be done on boards made over and above the required number or on special test coupons made on the edges of production panels. For the board manufacturer it is usually better to use external test coupons, as these can then be standard for all types of boards and the records of tests carried out provide a measure of improvements in the processes. Also coupons can be cut from an edge of a panel as it passes through the factory to make spot checks on any given process. The use of special test coupons allows the features to be made in the most suitable forms for specific tests, such as circles of copper of known area for measuring layer to layer insulation or tracks of defined width for checking copper adhesion.

On a few types of boards with complex shapes test coupons can be included in incuts, but generally any test coupon has to be totally external to the working board, so there must always be some element of risk of the features on the coupon not being truly representative of those on the board. In particular, the thickness of plating inside holes may not be representative, therefore it is always advisable to make further tests, on at least a few actual boards. In most printed circuit board factories a reject allowance is added to the quantity of boards needed so there should always be some boards available for testing.

Often customers require test coupons to be delivered with their boards, especially for military work. Whenever this is necessary, it is best to leave the customer's test coupon attached to the board by a narrow stem which can be easily cut through when the tests are to be made. Once the coupons have been separated from their boards, it is not easy to keep track of the specific board to which a given set of test results applies.

Sometimes customers design their own test coupons with specific tests in mind, but generally the patterns described in BS9760 or in similar national standards are used, and the tests are carried out in accordance with the standards.

As well as testing features such as insulation, plating thicknesses and track adhesion, test coupons are also used for reliability tests such as 5 seconds solder float; thermal cycling, and long term heat and humidity tests. Many of these tests involve microsectioning the coupon after the test has been completed to verify the condition of plating inside the holes.

11.5 MICROSECTIONS[8]

Defects in drilling and through-hole plating can render a multilayer board unusable, but often these defects can be found only when the board or a test coupon is cut up and the sides of the holes are examined under a microscope. Similar defects can also affect double-sided boards with through-plated holes, and microsectioning has become the recognised method for proper inspection of all through-plated hole boards.

The preparation of test specimens is not difficult. A piece of board or the test coupon which contains the hole or holes to be examined is cut out, potted in a suitable resin, and then ground and polished until the face of the sample passes through the centre of the hole. Whilst the process is simple enough, misleading results can easily be obtained if some precautions are not observed. The resin used to pot the sample should be a slow setting, low exotherm resin which exhibits the minimum of dimensional change after its cure starts. Epoxies are usually the most suitable. Some firms prefer to use faster setting resins with higher exothermic reactions to obtain quicker results. Sometimes the stresses imposed on the sample by contraction of the resin are not sufficient to damage the sample, but, when fast setting resins are used, care must be taken in interpreting the results. Obviously, if the sample shows no defects the board can be assumed to be satisfactory, but some types of defects, such as lack of contact between the hole barrel and the copper of an inner layer, might have been caused by the contraction of the resin damaging the sample rather than by any fault in the board manufacture.

Grinding of microsections should always be carried out as a wet process, taking care to rinse the sample well between each successive stage. Care must be taken not to overheat the sample or damage the exposed edges of the copper by trying to work too fast. When the sample has been ground down to the centre of the hole to be examined, the surface is polished and then given a very light etch to remove any copper which may have been smeared over an interface or a defect during the grinding.

It is not unknown for a microsection prepared by an inexperienced worker to appear satisfactory, but, after re-polishing and etching properly, to reveal defects such as resin smear, resin recession, or cracks in the hole wall.

A good clear description of the preparation of suitable microsections and of their evaluation can be found in the IPC Microsection Handbook, IPC-M-675.

When microsections are being examined, it has to be remembered that they show only a single section through a hole or possibly through a row of holes. They give a good representation of the quality of drilling and of the plating thickness through the holes, but they may not be so reliable on other aspects such as the presence or absence of resin smear or pinholes in the plating. Many such defects are confined to small portions of the circumference of a hole, and re-grinding and re-polishing a sample may show quite a different picture. Sometimes a section is ground and polished on two faces at right angles to show two views of the one hole. If resin smear is found on one side of a hole but not on the other the section may be ground off parallel to the face of the board to see how far round the hole the defect extends.

Microsections show up a fascinating variety of faults, from rough drilling, through resin smear, resin recession where the copper barrel of the hole is no longer in intimate contact with the base laminate, various stress cracks caused by thermal cycling or shock, inner layer pads which have parted company from the hole barrel, voids in plating which may be the result of excessive etch back and ion depletion, generally poor plating, to misregistration of pads on multilayer boards and poor bonding between cured laminates and prepreg.

Whilst they have their limitations, microsections do give a very good picture of the overall quality of a board, and generally if a defect is found on a first section examined it is highly unlikely that it will prove to have been the only such defect in the board.

Documents such as IPC-M-675 can give a good idea of the types of defects likely to be found in microsections, but proficiency in the examination and interpretation of microsections can be gained only by practice, experience, and by discussing the results found with other proficient workers.

11.6 REPAIRS AND REWORK[8]

Most time and cost conscious equipment manufacturers will allow some level of repair or rework on printed circuit boards. Defects can be repaired in ways which can hardly be seen, and most of which are as reliable as the rest of the board. The repair of printed circuit boards is a very skilled job which can not be hurried, but often the cost of a repair can be far less than the cost of re-making the board, especially if it is a multilayer or flexi-rigid, or if all the components have been soldered to it. The repair techniques can be used to rectify defective boards, to mend boards which have been damaged during assembly or in use, or to modify new boards when post design changes are introduced after the boards have been made.

Mainly repair work uses standard hand tools such as scalpels, scrapers, a small bitted temperature controlled soldering iron and so forth. A hand-held, dentist's air-driven high speed drill with a range of cutters is excellent for removing growths from tracks, and for some of the bigger repair jobs such a drill in a universal mounting on a small milling machine equipped with a precision three axis table can be used. Local electroplating is usually done with a soft swab soaked in electrolyte and wrapped round the lead from a suitable power supply. Ferranti Ltd of Bracknell developed a machine for through-plating single holes, on which the boards are clamped between two O ring seals and the required solutions and rinses are pumped through the hole, with plating currents supplied by anodes lining the tubes either side of the board.

Companies which do allow repairs to boards usually have their own specifications detailing the methods to be used, or referring to suitable national specifications such as BS9760. There are many ways in which satisfactory repairs can be carried out, but space permits mention here of only a few.

11.6.1 Cut Tracks

Cut tracks can be repaired by first cleaning from the area all solder resist and tin-lead plating. If the exposed ends of the bared copper track are very close together, it may be possible to burnish the copper until it makes good electrical contact across the gap, but generally it is more common to use a conductive epoxy resin, often silver loaded, to bridge the gap between the ends. When the epoxy has cured it is cleaned off to the same width as the track, and the top surface of the track in the area is then very carefully milled away down to about half its normal thickness, and tested to ensure that there is still electrical contact across the gap. The local plating equipment is then used to deposit copper over the joint area until the track has been built up to rather more than its original thickness. The repaired section of track is then tinned with the soldering iron, and solder resist is painted back over the area to leave an almost imperceptible repair.

Cuts in tracks can also be repaired by taking a short length of copper track peeled off a scrap board, laying it tinned side downwards over the break, and re-flow soldering it in place. When the solder resist is painted over the repair it is quite hard to see that the track has been patched, but care must be taken not to use this type of repair on the solder side of a board which is to be mass soldered. On boards which must withstand high temperatures a nickel patching wire can be welded over a break on a parallel gap welding machine. Welded repairs are probably the easiest to perform, but they are not generally liked by users of high reliability boards because the copper conducts the heat from the weld rapidly, and the heat can weaken or destroy the bond between the copper track and the laminate.

11.6.2 Replacement Tracks

A length of track which is under width, which contains pinholes, or which requires re-routing, can be replaced by cutting through the track at the ends of the defective section, peeling the track off the board, and glueing down a suitable length of replacement track peeled from a spare board. This leaves two breaks, which are then treated as described in Section 11.6.1.

11.6.3 Inadequate Clearances

Inadequate clearances or short circuits caused by growths on the sides of tracks or by metal particles left between two tracks can be removed with a scalpel or by careful machining with a dental burr. Inverted cone cutters are popular for this work, but such cutters are delicate and need using with great care. After the metal

particles have been removed the board is thoroughly cleaned and the solder resist restored by hand painting.

11.6.4 Defective Edge Connector Plating

Edge connector fingers which show excessive pinholes or which have been slightly damaged in service can have their gold surface restored by local plating. The plating should be continued until both thickness measurements and porosity tests show that the repair is satisfactory. When more than one or two fingers need more plating, or if the plating is badly damaged, it may be better to replace the fingers.

11.6.5 Replacement of Edge Connector Fingers

Badly damaged edge connector fingers can be replaced by milling out the defective finger or fingers, cutting down to about a third of the thickness of the board. Replacement fingers are cut from a spare board, trimming the piece to fit accurately into the milled-out area and taking special care to ensure that the surface of the replacement finger is flush with the remainder of the connector. The replacement piece is bonded in place with epoxy resin, taking care to clean off any surplus resin squeezed out. A simple clamp such as a clothes peg wrapped in polythene or PTFE sheet can be used to hold the replacement finger in place. When the epoxy resin has cured, any small traces left on the surface are cleaned off with the scraper or scalpel, and the joints in the tracks leading to the fingers are made good by loaded epoxy resin and local plating as described in 11.6.1.

11.6.6 Track Repairs on Inner Layers

Inner layers of multilayer boards can be repaired in the same way as outer layers, except that the layers above that to be repaired must first be milled away to expose the defective portion of track. When the repair has been made and covered with epoxy resin, the outer layer tracking may be restored by bonding down a length of track and joining it as described in Section 11.6.2. If more than one outer layer track has to be disturbed for access to the repair it may be better to set in a piece cut from a spare board as for replacing edge connector fingers.

Repairs to tracks on the inner layers of multilayer boards may be effected by drilling into the board to isolate the defective track and replacing it with a new track bonded onto one of the surface layers.

11.6.7 Lifted Pads

On non-plated boards pads which have lifted from the base can be bonded back into place with epoxy resin. Often the pad will be damaged, and the pad and a short length of track to it are then replaced. If the board is not to be mass soldered, the new portion of track can be reflow soldered over the end of the old track, but if the board may be wave or drag soldered then a plated joint is better.

On double-sided through-hole plated boards a lifted pad will often involve re-making the connection to the other side of the board. Component leads may be used to do this, soldering to each side of the board. If the component has an overhanging body the best repair method is to open out the hole slightly and set in a thin walled eyelet. Damage to a pad at a via hole can sometimes be repaired by cutting away the pads both sides, passing a length of track through the hole and making surface joints to the tracks on both sides of the board as described in 11.6.1.

On multilayer boards a lifted pad to which there was no surface connection can be overcome by milling away the board surface to expose the pad on the next layer in. The pad must be carefully cleaned and then hand-tinned.

11.6.8 Missing or Undersize Holes[48]

Missing holes present no problems on unplated boards; they are simply drilled in. On double-sided boards component holes can often be left unplated and the component lead used to make the through-connection. Via holes can be drilled and a wire run through, or an eyelet can be fitted. The real problems arise on multilayer boards, where the only really good cure is a special machine to through-plate individual holes. If no such machine is available, then other ways of making connections to inner layer tracks must be found. A via-hole connection can be made by access hole techniques, drilling the smallest hole possible to expose the lowest pad to which a connection is required, then increasing the hole diameter in steps at each higher level to be connected. A thin strip of copper is then pressed against one side of the stepped hole, soldered in place, and the hole filled with epoxy resin. Sometimes it may be better to avoid the via-hole site altogether and find surface routes for new tracks. On tightly packed boards it may be necessary to re-run one or more connections as discrete insulated wires passed through holes drilled close beside mounting pads and soldered to the component leads.

11.6.9 Warped or Out of Flat Boards

Boards can be warped or twisted from various causes, mostly involving thermal stresses. Badly warped, twisted or bowed boards can be straightened by mechanically over-bending in the opposite direction, after which it is advisable to check for track continuity, or by heating them to above the glass transition temperature, pressing them flat, and allowing them to cool slowly. It has been suggested that two or three slow thermal cycles to above the glass transition point, with the board held flat, produce better results than a single cycle. If the distortion was caused by mishandling, such as stoving a solder resist while the board is not properly supported, or chilling a freshly reflowed board to below the glass transition temperature, such thermal cycling should make a permanent cure, but, if the fault is caused by lack of balance in the board design, then the 'cure' may be only temporary.

Chapter 12

ASSEMBLY AND SOLDERING

12.1 COMPONENT MOUNTING[49]

Most of the components mounted on printed circuit boards are small and light enough to be held to the board by their connecting leads. The most common discrete components are the cylindrical bodied, axial lead resistors, capacitors and diodes which are mounted as shown in Figure 12.1A with their leads bent a short distance from the body and passed through the board. Care is needed when bending the leads to avoid any risk of damage to the body of the component. Many resistors and diodes have small glass tubes as the body, and attempts to bend the lead too close to the end of the body can crack the glass or pull the lead away from its internal contact. The smaller axial lead resistors are available from the manufacturers with their leads ready bent for mounting on 7·62 mm pitch.

Some small capacitors and resistors are made with flat or tubular bodies but with leads emerging from the edge of the body or radially. These can be mounted directly into the board as shown in Figure 12.1B, but such components may be unsuitable for use in some more exacting environments. They are more easily damaged when a board is handled than the axial lead types, and in equipments which are subject to vibration there may be too great a risk of them failing in service. Most firms who manufacture military equipment have calculated tables of sizes and weights of components which can safely be mounted on edge leads.

Transistors in metal cans such as the TO5 or TO18 are ideally suited to mounting on boards, but usually they need to be mounted clear of the face of the board to avoid the formation of a moisture trap beneath their bodies. Some of these transistor cases have a projection in the centre of the base to hold it clear of the board, as shown in 'C' on Figure 12.1. Stand-off mounting for plain based transistors is often arranged by drilling the mounting holes to a larger pitch than the lead spacing on the can so the leads have to be splayed out as in 'A' on Figure 12.2.

This ensures that a moisture trap can not be formed beneath the transistor, but it introduces a risk of the glass seal round the leads being cracked in bending, and for reliable equipment it is better to drill the holes on the correct pitch and to mount the transistors on small plastic stand-off bases. Figure 12.2B shows a transistor being mounted on such a base, but not pushed right down.

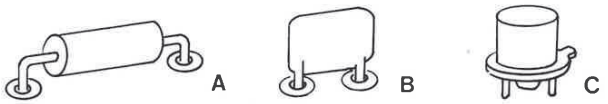


Fig. 12.1 Small component mountings.

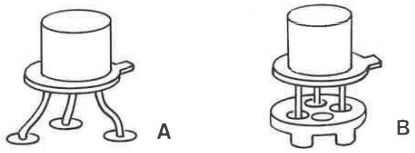


Fig. 12.2 Transistor mountings.

On very tightly packed boards there may be insufficient space to mount all resistors, diodes and capacitors flat as in Figure 12.1, so they may be mounted on one end with the upper lead bent over and brought down beside the body as in 'A' of Figure 12.3. As with the edge mounting capacitors, this is not a vibration resistant mounting, but it may be allowable if groups of four or more components can be taped together or bonded to form what is in effect a multi-lead package as shown in B of Figure 12.3. Similarly, as shown in C, edge leaded capacitors can be safely mounted in pairs.

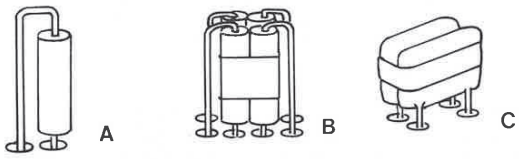


Fig. 12.3 Components mounted on end.

Some large diameter components have leads which are not strong enough to hold them securely. These can be mounted in a metal or plastic P clip screwed to the board, or they can be held in place with two loops of thread tied through holes in the board as shown in Figure 12.4.

If the boards are to be mass soldered it may be better to fit the screws holding P clips after soldering, to avoid any risk of the plating on the screws contaminating the solder bath. If thread is used, it must be of a type which will not melt at normal soldering temperature. When large components are fitted after mass soldering, their mounting holes can be closed by small plastic plugs to keep them open during the mass soldering, or they can be masked over with tape.

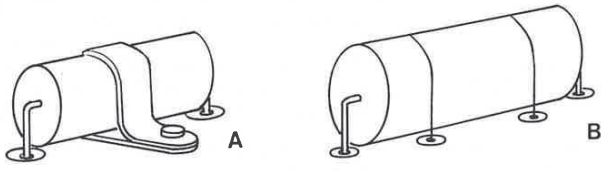


Fig. 12.4 Mechanical securing of large components.

Some medium sized diodes are packaged in awkward shaped cases such as the DO1, 2 or 3. These can be mounted with their leads bent, but generally they are better mounted on stand-off pins or fishbeads as shown in Figure 12.5. Often these slightly larger diodes dissipate considerable heat and therefore should be mounted clear of the face of the board.

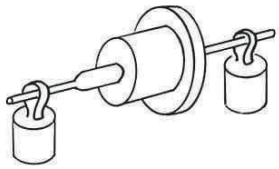


Fig. 12.5 Medium power diode mounted on fishbeads.

Most manufacturers mount small axial lead components with their bodies touching the face of the board and the leads bent with a single bend to pass through the board as shown on the left of Figure 12.6. However, some companies prefer to crimp the component leads to form a self-retaining snap-in mounting which holds the body of the component clear of the board.

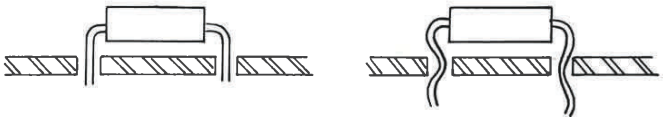


Fig. 12.6 Plain and snap-in component mountings.

The snap-in mounting offers the advantage of being self-retaining until the board is soldered, but forming the leads adds to the cost of assembly and the resulting stand-off mounting can be a vibration hazard. If a component requires a stand-off mounting because of the heat it dissipates, it should be mounted on pillars or on fishbeads rather than by having its leads bent. A component which relies on bent leads to keep it clear of the board can very easily be bent over so its body does contact the board.

Components such as small transformers, coils, variable resistors, delay lines and relays are available in special board mounting forms. Most have leads which emerge from the base for direct insertion into the board, and should also have pins or projections moulded on the base so that they mount with a clear space beneath the body. When any part with a flat base through which the leads emerge is to be mounted on a board, it should always be stood off on some form of spacer both to avoid a moisture trap and to allow vapour from the flux to escape during soldering.

A stand-off mounting also allows for some flexing of the leads to occur when differential thermal expansion causes relative movement between the board and a multi-pin component. Undoubtedly one of the best multi-pin package forms for mounting with its leads through a printed circuit board is the Dual In-line

Package. Its shaped leads ensure that the body sits clear of the board, unless it is mounted in excessively large holes; the shaping of the leads and the shoulder bend allows them to flex in any direction to take up tolerances or thermal movements; the placing of the leads beside the body allows easy inspection of the solder joints on the upper side of the board, and the package has a generous surface area which allows it to dissipate heat up to around half a watt at normal packing densities with air cooling. Also, its internal leads are well spaced and do not offer a high capacitive load on the chip.

Smaller integrated circuit packages such as the flat pack, the leadless inverted device, or the chip carrier usually need to be mounted on heat sinks or ladders, and their leads are surface mounted which can introduce further complications.

12.2 COMPONENT INSERTION[49]

12.2.1 Manual Insertion

The insertion of components into a printed circuit board is a time consuming operation, and efforts are continually being made to ease the work, from the provision of special trays to hold components to the total automation of the job with robots. While there has been considerable progress in automation, some components like transistors have proved difficult to insert by machine, and capital costs for good fully automatic assembly lines are such that they are usable only by larger systems companies.

Manual insertion of two to four lead components is a simple process, if somewhat tedious. It is usually the DIPs and connectors which provide the problems, together with the question of making the whole process more efficient.

Insertion of DIPs can be eased with a very simple tool, a small piece of stiff plastic sheet with one end cut with shallow vee notches on 2·54 mm pitch. The packs should be supplied with their leads splayed out beyond the normal mounting pitch. The guide is held over one row of holes and the legs of the package on one side are pushed against it until they just enter the holes. The guide is then used to push the second row of pins in until they also enter their holes and the pack is pressed down until the upper portions of the leads touch the pads. When DIPs are mounted in rows on a regular pitch, a second guide can be cut which fits against the side of a package in the first row and guides the nearer row of pins of the next pack along.

More elaborate guides can be made which hold the pack and guide the tips of the leads into accurate positions, but some of the more sophisticated guides are more suitable for use on automatic insertion machines than as aids for manual insertion.

Boards on which components are being inserted should be held in a shake-free fixture clear of the surface of the bench. Components to be inserted can be sorted into separate trays for each type, possibly with trays arranged in columns to correspond to rows of components on the board. Board assembly stations can be elaborated up to complex tape controlled affairs which automatically project a beam of light to illuminate the mounting position for the next component and open the front of the tray or box containing the appropriate part ready for the operator to take one. The amount of effort put into such semi-automation of work stations usually depends on the numbers of boards to be assembled, and it is probable that a firm looking at such semi-automated work stations would be inclined to go the whole way and opt for fully automatic component insertion.

12.2.2 Automatic Insertion[2,50]

Automatic insertion machines for DIPs have been available for many years, as have machines which take axial leaded components, usually fed in on a bandolier arranged in the correct sequence to suit successive component sites on the board, bend the leads to the correct pitch, insert the component, then bend and crop the leads below the board. Some machines can sort out component sequences themselves, taking components from large hoppers and arranging the sequences; others have to be fed with boxes, or slides made up in the correct order.

Most automatic insertion machines carry the board on a table which is moved under a fixed position insertion head. Some machines require very tight tolerancing of the position of holes in the boards, or the use of grossly oversize holes to be sure a component lead will enter and not bend over on the board beside the hole. There are machines which use optical sensors to detect when the mounting holes are in the correct positions under the head, and control the movement of the table to adjust for inevitable tolerances on both the board and the machine, allowing the correct hole size for a good soldered joint to be used. Between different firms using automatic insertion for DIPs there is a range of over 0·2 mm in the minimum hole diameter allowable.

Automatic component insertion can be very much faster than manual insertion, and it should eliminate wrongly placed components. With such benefits it may be felt that it should be used universally, but it is not, because there may be some penalties. Most machines can handle only a restricted range of component types which usually leaves one or two components to be manually inserted anyway. It may be necessary to leave fairly generous spaces round the ends of components to allow room for the heads of the machine, and there may be other restrictions such as a need to keep all polarised components orientated the same way. The design restrictions imposed by some automatic insertion machines can add as much as 30 per cent to the size of a board needed to carry a given circuit, which can lead to significantly reduced electrical performance and increased costs in other aspects such as framing. If the limitations include the use of oversize holes which can result in less than optimum quality soldered joints, then it can be that the apparent savings made by automating assembly turn into an overall loss.

Work on robots is proceeding apace and it may not be too long before automated 'hands and memories' are available at prices which will make their use justifiable in board assembly. Such robots should be capable of handling modern high density boards with components placed according to the electrical needs of the circuit instead of to an automation rulebook.

12.2.3 Cutting and Crimping of Leads[2,46]

At some stage during assembly the component leads have to be cut off to length. Some authorities insist that this should be done prior to insertion; some suggest that after insertion but before soldering is the correct time; while others insist that they should be left full length until after soldering. In fact, which is correct depends on the circumstances of each job, whether it is manually soldered or wave or drag soldered, and on the sizes and types of the leads.

Automatic lead cropping machines are available which use high speed rotating cutters to dress the leads to length. Most of these machines give the best results after the board has been soldered. However, mass soldering of components with

long leads is not so successful as soldering after the leads have been cut to length.

Simple semi-automatic guillotine cutters can be made to crop the leads of all components after they have been bent and before they are inserted in the board. Some capacitors have very fine leads which are not at all easy to insert when they have been cut off short. These are usually best cut to length after insertion and prior to soldering.

TO70 or MO002 eight lead integrated circuit packages are most easily inserted if their leads are first cropped to different lengths so that they are inserted one at a time, then after insertion all the leads are cut to length ready for soldering.

When the components have all been inserted, either the board will have to be inverted for manual soldering, or it will have to be taken to a mass soldering machine, probably being shaken or inverted on the way. Something has to be done to prevent the components from falling out between insertion and soldering.

If only a few boards are being made the components can be soldered individually as they are inserted, but this is impracticable on production quantities. The traditional way of holding components in place is by crimping the leads over on the solder side of the board.

On non-plated boards the joint between an unbent lead and its pad must be formed by the mass of solder on the pad and up the lead, as shown on the left of Figure 12.7. A much better joint will result if the lead is bent over to lie close against the face of the board, usually over a broad extension to the pad as shown on the right of Figure 12.7.

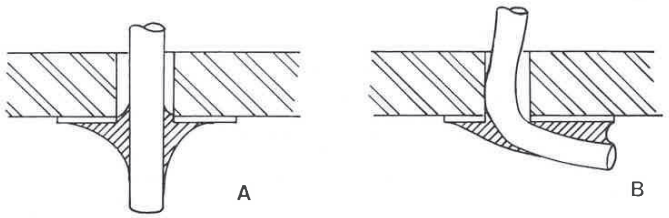


Fig. 12.7 Solder joints on unplated boards.

Most automatic insertion machines have heads below the board which operate as the component is inserted, cutting the leads to length and crimping them over.

On through-hole plated boards the generally higher tracking densities and smaller pads may not allow space for the leads to be bent over fully, so a partial bend to about 40 degrees has to be used. This may be adequate to hold DIPs and most resistors, but depending on the lead and hole sizes it may not hold transistors and some types of capacitors sufficiently securely to prevent their becoming displaced.

When the holes are through-plated the real strength of the joint is in the solder in the barrel of the hole, not on the surface, as shown in Figure 12.8. Bending over the lead will do nothing to add to the quality of the joint, and it may make it harder to make and inspect good joints. However, there is a further point to be considered. On most boards it is possible that a component may have to be replaced at some time. The surface solder on a non-plated board can be removed

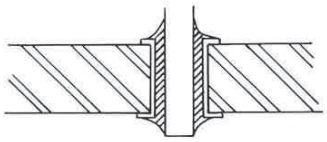


Fig. 12.8 Solder joint in a plated-through hole.

and the lead can be more or less straightened to allow it to be pulled out through the hole.

As anyone who has ever tried to straighten a wire which has been bent will know, it is virtually impossible to restore a bent over lead so that it will come cleanly out of its hole, and a not infrequent result of attempts to remove components, whose leads have been bent, from plated-through holes is that the barrel of the hole is pulled out with the lead. Even if the entire barrel is not pulled out, removal of bent leads from through-plated holes can be expected to cause severe damage to the hole.

This may not matter on double-sided boards where a joint to the component side tracking can be made on the top of the board, but a damaged hole which contacts an inner layer of a multilayer board may mean the whole board has to be scrapped, or on commercial equipment accepted with some connections made by discrete surface wires.

In summary it can be said that on non-plated boards the component leads should be bent over; on double-sided through-hole plated boards bending is undesirable, whilst on multilayer boards the leads should be left straight.

Straight leads leave the problem of how the components can be held in place between assembly and soldering. This is not difficult. Some form of snap-on cover with a compliant blanket to hold the components down can be provided, or automatic insertion machines can be modified to put a small dab of a suitable adhesive on the components as they are inserted. If snap-on covers are used, the provision of a limited number of covers will encourage all staff to ensure that boards are soldered as soon as possible after assembly, as they should be but not always are.

12.3 SOLDERING[46]

12.3.1 Solderability

A soldered joint is a joint between two metals made with a third metal or alloy of lower melting point which alloys with or diffuses into the metals to be joined. In electronic assemblies the use of a 60:40 tin-lead alloy which melts at around 189°C is almost universal.

The solderability of a surface is a measure of the ability of the tin-lead solder to wet and combine with the cleaned, fluxed surface. Flux is used to dissolve minor contamination from the joint surface and to prevent the hot surface from oxidising while the joint is being made. Fluxes may be non-active, often a pure rosin, or they may be active and containing acidic cleaning agents. The residues of a non-active flux can be left on a joint without risk of later corrosion, but all

traces of active fluxes must be cleaned off after soldering. Generally the preference in the electronics industry is for the use of non-active fluxes with good joints ensured by proper pretreatment and protection of the surfaces to be joined[47].

The ability of a surface to be wetted by solder can be assessed by the shape of the meniscus formed when a drop of solder is placed on the hot, fluxed surface. If the meniscus has a noticeably undercut edge, like a drop of water on an oiled surface, then the metal is unsolderable. A sharp edge with no undercut indicates some wetting of the surface, but the solderability is good only when the meniscus draws out to a fine angle as shown in Figure 12.9.

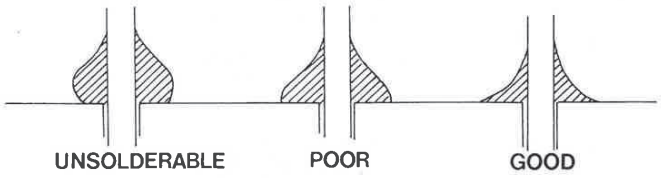


Fig. 12.9 Solderability of board surface.

Copper and its alloys and silver solder well provided they are clean, and tin-lead plated clean copper yields excellent solderability until atmospheric corrosion affects the tin-lead. Gold solders well, but at very low percentages in a solder joint it forms a brittle gold-tin intermetallic compound which can completely surround a component lead to the point at which a straight gold plated lead can be pulled out of the joint like a plug out of its socket. Most firms who have to buy components with gold plated leads dip the leads in solder baths to remove the gold before assembling the components on boards.

Copper also forms intermetallic compounds with tin. These compounds are not solderable, but normally they are well diffused in the lead-rich solder alloy near the interface and cause no problems. If only a very thin layer of solder is applied over a copper surface, the formation of these intermetallic compounds can cause the whole surface to become unsolderable. This should not happen if PCBs are well made and carefully handled.

Over long periods the tin can diffuse out of soldered joints to leave a brittle, lead-rich joint which is easily broken. This diffusion is accelerated at high temperatures, and in cases where solder joints must be kept very thin and diffusion could prove a problem the joint faces may be plated with a flash of nickel under a copper coating to limit the depth to which the tin can diffuse.

Nickel is a difficult metal to solder, and normally good solderability is achieved only with the use of active fluxes. The soldered joints on palladium-nickel seen by the author appeared to exhibit low solderability and were in all probability mechanical encapsulation and not true alloying. Such mechanical encapsulation or dry joints can occur on copper when one surface is inadequately cleaned or has not been heated sufficiently so the solder chills to below its melting point on contact with the surface.

12.3.2 Manual Soldering

The first requirement for manual soldering of PCBs is a skilled trained operator, and the second is a good quality, temperature controlled, low voltage soldering iron with a clean, fine bit which is properly earthed. The bits normally used are copper coated with a layer of iron. These bits must not be filed or cleaned with any abrasive, but are kept clean by touching them with resin cored solder and wiping on a damp natural sponge. The iron should never be put in any form of muffle, but should be kept in its open spring holder on top of the transformer when it is not being used.

A copper bit, or an iron coated bit which has been filed to shape, will dissolve in the solder to leave a ragged, cratered tip with which it is almost impossible to make good joints. Much can be learnt about the quality of an assembly shop by a glance at the soldering irons. The making of good soldered joints is a skilled task which requires proper tools and a high level of cleanliness.

To make a good joint between a component lead and a pad on a board the iron should be coated with sufficient solder to form a film which will ensure good thermal contact between the bit and the component lead and pad. Although the board may feel far more solid than the component lead, the lead has a greater mass of metal than the pad, so the iron should be applied mainly to the lead and should not be pressed hard onto the face of the board. Flux cored solder is then applied to the lead and pad, preferably on the opposite side to the iron. If the surfaces are clean and hot enough the flux coats the joint area, and the solder flows over the pad and up the lead to leave a fine angled meniscus indicative of a good joint. The iron is removed as soon as the joint is formed and the joint is left to cool without movement or vibration which can cause crazing.

In many cases it can be easier to make joints manually if the component leads are left uncut until after soldering, which usually means the operator will make the joints row by row, soldering then cropping the row of leads to gain access to solder the second row. A good operator can make excellent joints at a high rate, but manual soldering of the thousands of joints involved in modern boards is best avoided if there is any reasonable alternative.

12.3.3 Drag Soldering

The earliest form of automated soldering is drag, or dip soldering, in which the assembled and fluxed board is lowered onto a bath of molten solder and drawn along and out of the bath on an inclined conveyor. The boards are mounted on carriers which have a blade at their leading edge to clear dross from the top of the solder bath. Drag soldering can produce high quality joints, usually with a trace of thickening of the solder on the trailing edges of pads, but to solder large modern boards would require a very large bath in which oxidation of the solder would occur quite rapidly so the process is now not commonly used.

12.3.4 Wave Soldering

The preferred method for mass soldering PCBs is by wave soldering, in which the boards are passed over an orifice through which a wave of solder the width of the board is pumped. The solder wave spills over one or both faces of the nozzle and runs back into a narrow bath, the surface of which is usually covered with an oil or rosin, or with small hollow ceramic spheres to reduce oxidation of the solder. The solder may spill over the leading or trailing edge of the nozzle, or over both. The nozzle may have a plain top, possibly adjustable in width, or it may be shaped to force the wave into a required shape.

The conveyor may carry the board directly, or on carriers which clip into the conveyor. The use of carriers enables boards of different widths to be soldered without stopping to alter the conveyor width. On some machines the conveyors are flat, but on others they are sloped so the boards can leave the wave at a less acute angle. On some machines the wave is activated only as a carrier approaches.

Wave soldering machines require careful adjustment of wave height, height of the board above the nozzle, conveyor speed and solder temperature. For the best results on plated-through hole boards, the board should be set such that the natural height of the wave is about one third of the board thickness above the solder side and capillary action causes the solder to rise through the holes to the top of the board. A good joint will be indicated by a bright, low angled meniscus of solder on the component side of the board.

Most wave solder machines are provided with in-line foam fluxers, pre-heat sections, and often post soldering cleaning stages. For operator comfort and health they need a hood with adequate air extraction.

Pre-heat is necessary to dry any solvents from the flux and to minimise the thermal shock to the boards when they contact the solder wave.

12.3.5 Vapour Phase Soldering[51]

In vapour phase soldering the assembled board is lowered into a tank containing the vapour from a fluorocarbon liquid which is boiling at a temperature just above the melting point of the solder. The hot vapour condenses back to liquid on the board, giving up its latent heat of vaporisation and raising the temperature of the assembly rapidly and without risk of overheating. No solder is added in the process, so any necessary solder has to be assembled with the components either in the form of a solder cream, which is a mixture of fine grains of solder and flux which can be screen printed onto a board, or as solder preforms which are usually little eyelets made from sheet solder. The process is more commonly used for soldering surface mounted components than for conventional boards with leads mounted through holes.

Boards can also be soldered by focused infra-red heaters or by jets of hot inert gas, but these methods are not in common use.

12.4 COMMON SOLDERING FAULTS[46]

12.4.1 Dry Joints

Dry joints are usually the result of inadequate cleaning of one surface, the use of an inherently unsolderable material, or insufficient heating. On well made boards assembled with good quality components they are more likely to be found on manually soldered joints than if the boards are mass soldered under properly controlled conditions. Dry joints are the most easily avoided soldering fault.

12.4.2 Bridging

Bridging is the short circuiting of features on a board by solder. It can occur on manually soldered boards if the iron is allowed to touch a track adjacent to a pad while a joint is being made, or if an excess of solder is used. On wave soldered boards it is usually the result of either dirty solder or of bad design. In many cases it can be prevented by the use of a solder resist.

Boards should always be designed so they can be wave soldered along the run of the tracking. Figure 12.10 shows a pair of trial boards made with tin-lead

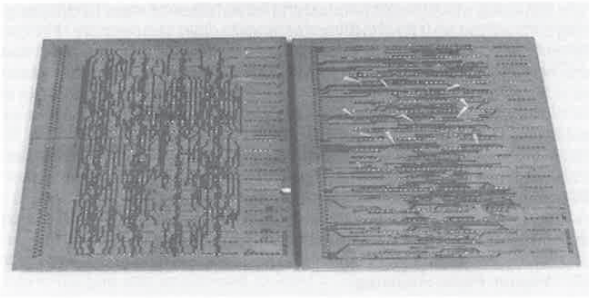


Fig. 12.10 Boards wave soldered along and across the run of tracking.

plated tracks, one of which was soldered along the run of the tracking and one across. All other soldering conditions were identical and the solder in the machine was moderately heavily loaded with impurities and was overdue for renewal. The board soldered with the run of the tracking did not bridge, even though the arrowed track on Figure 12.11 passed within 0·1 mm of a pad on its right, showing that there appears to be no mechanism which will cause sideways bridging.

As shown in Figure 12.12 the board soldered across the run of the tracking bridged badly on most tracks running close past the component pads. It is significant to note that none of the tracks to the unfilled connector pads on the left of the board bridged although the clearances are the same, and neither did the tracks running past via holes.

Subsequent tests confirmed that bridging is a function of the distance between the pad and the track measured along the direction of soldering, the pad diameter, and the amount by which the component lead projects beyond the pad.

12.4.3 Crazing

Crazing of the joint is the breakdown of the solder into large grains which occurs if the lead is allowed to move in the hole while the solder is solidifying. It is a fault which should not be found on mass soldered boards, and is most usually seen on manually soldered joints where the component has to be held while the joint is made.

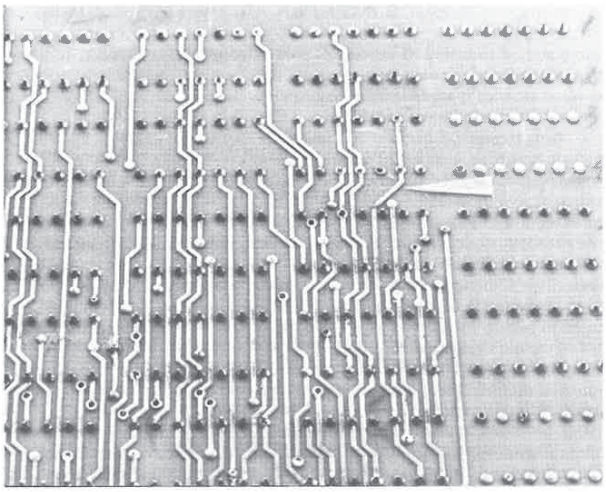


Fig. 12.11 Effect of soldering with the track run.

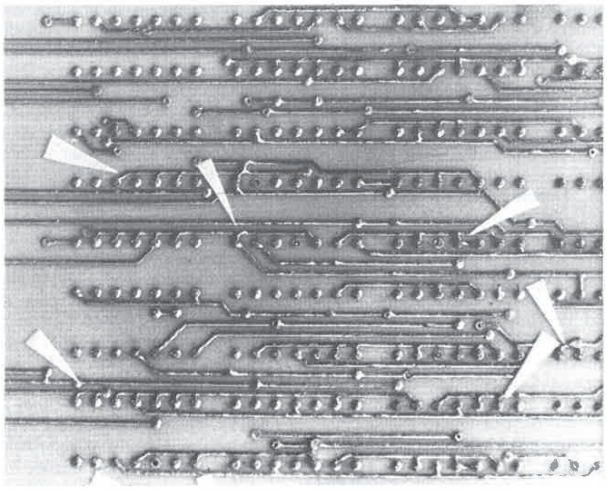


Fig. 12.12 Effect of soldering across the track run.

12.4.4 Icicling

Icicling is the formation of strands of solder from joints or tracks. It is a mass soldering fault, and is usually the result of attempts to solder at too low a temperature or with inadequate pre-heat.

12.4.5 Outgassing

Outgassing, also called blowholing or cratering, occurs when moisture trapped in the boards is vaporised by the hot solder and blows out through the joint as the solder is cooling. The moisture may be from water vapour absorbed by the boards during storage, or it may be processing fluids absorbed during manufacture, or it can be volatile parts of improperly cured resins in the laminate. It is sometimes blamed on flux solvents trapped in the joints, but these are likely only if the boards are given inadequate pre-heat or if components are mounted in such a way as to block the upper ends of the holes leaving no escape route for vapours from the flux.

If the boards are well made, with tin-lead plating which has been reflowed as part of their manufacture, any outgassing is most likely to be caused by moisture absorbed during storage, and indicates a failure to dry the boards properly before soldering. It should be noted that all components for military use have to be capable of working at 125°C, so boards can be baked after assembly and immediately prior to fluxing and soldering. Boards for commercial use may carry components which can not stand a high temperature bake. These should be dried before assembly and soldered as soon as possible, with adequate pre-heat.

Some companies allow a moderate amount of outgassing on the grounds that it seldom reduces the amount of solder in a joint to an unacceptable level. Others reject boards which show outgassing on the grounds that it indicates that the plating in the holes must be porous. This is a weak argument since most board specifications allow the presence of some pinholes in the hole walls. Outgassing can be seen in some of the holes shown in Figures 12.11 and 12.12. Some of the pins remained unsoldered because inadequate pre-heat had not removed all the flux solvent and bubbles masked the pins.

12.4.6 Delamination or Blistering

Delamination, especially of multilayer boards, during soldering is usually a much more violent form of outgassing and the cure is the same: eliminate the moisture before soldering. Flexi-rigids or rigid boards made from polyimide are particularly susceptible to moisture absorption and problems on soldering, and should be thoroughly dried. The absorbed moisture is allowed for in the published electrical performance specifications for polyimide materials, and its only adverse effect is on the bonds if boards are mass soldered without adequate drying.

Blistering and other laminate defects may be caused by moisture, and they can also be caused if the board is overheated in any way. The presence of any unduly heavy mass of metal to be soldered can cause blistering. Closely spaced rows of turret head pins in one area of a board can absorb sufficient heat during wave soldering to damage the laminate and to raise blisters.

12.5 DE-SOLDERING THROUGH-PLATED HOLES

Component leads on non-plated boards can be de-soldered by removing the solder either with a vacuum sucker or by the capillary action of a length of fluxed copper braid. These techniques are not so successful on plated-through holes. The component lead almost always lies at an angle through the hole, leaving two sharp corners where a thin film of solder will be trapped between the lead and the wall of the hole. Any vacuum sucker will pull cold air through the rest of the hole, chilling the small residual joint at the top of the hole. A vacuum sucker or braid also removes all the free solder from the bit of the iron, destroying the thermal conductive path. This usually persuades the operator to press the iron hard against the surface of the pad. The epoxy resin in the board round the hole is by then above its glass transition temperature, so the result is almost invariably a cracked or badly distorted hole barrel. When the lead is eventually pulled out after attempts to remove all solder by sucking or with braid, the cracked barrel of the hole often comes out of the hot resin with the lead.

The correct method for removing components from through-plated holes is to melt all the joints on a solder bath or with a specially designed multi-head iron, and when the solder in all the holes is molten to pull the component out with the liquid solder acting as a lubricant. If only a single-head iron is available, the leads should be cut and removed individually. The solder can be removed from the holes with a hot air jet or fluxed braid after the leads are out. Whatever methods are used, it must be borne in mind that the bond strength of the epoxy resin is very much reduced at soldering temperatures, and the walls of the holes are only some 0·02 mm thick, which means they are not very strong once they lose the support of the epoxy glass backing.

There can be problems in removing packages from multilayer boards if the power and earth planes have not been properly designed. The planes should have normal sized pads around all holes, separated from the body of the plane by annular clearances, with bridging tracks about 0·5 mm wide where a connection to the plane is required. Some multilayer boards are wrongly designed with the through-plated holes allowed to contact the main body of the plane all round where a connection is needed. When attempts are made to remove component leads from such holes the internal plane acts as a heat sink and makes it very difficult to melt the solder in the hole above the level of the plane.

When such holes are wave soldered, the heat sink effect of the planes usually prevents the solder from rising right to the upper side of the board, but sufficient solder will usually rise above the level of the plane to pose a serious hazard on desoldering.

12.6 SURFACE MOUNTING

Leads soldered into holes with pads round the holes take up a lot of space on printed circuit boards compared with that needed for tracks and, since the earliest days of integrated circuits, parts of the electronics industry have mounted devices on the surface of boards, soldering or welding the leads to rectangular pads which need not be associated with holes through the board, although in practice they are often directly associated with via holes or with plated pillars.

By the late 1970s the growth in the numbers of leads associated with

microprocessor chips and with uncommitted logic arrays led to a revival of interest in the surface mounting of small bodied integrated circuit packages. Leaded and leadless chip carriers were hailed by the technical press as new package forms to revolutionise electronics packaging, with a fine disregard for the fact that the surface mounting flat pack is in fact an older package form than the DIP, and that the Leadless Inverted Device, which is merely a two row chip carrier, dates from the later 1960s. Surface mounting resistors and capacitors had been familiar to the makers of ceramic substrate hybrids for some years, and had been mounted in limited numbers on printed circuit boards.

Leaded surface mounting devices can be attached to boards by welding or by reflow soldering; leadless devices which have their connections formed as metallised areas of the base and often up over the side of the package can be attached by reflow soldering or, much less commonly, by ultrasonic bonding.

12.6.1 Welding

Devices such as flat packs or leaded chip carriers can be welded to boards on machines which pick up the body of the device, usually by a vacuum head, place it on the board, whose position is controlled by a pantograph or numerically controlled table, then make the welds with parallel gap heads as shown in Figure 12.13.

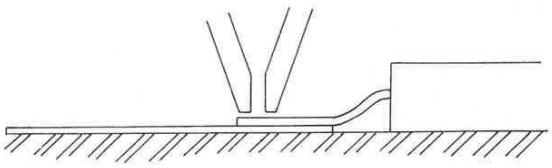


Fig. 12.13 Parallel gap welding.

The parallel gap heads are pairs of heavy electrodes separated by a small gap. When the electrodes are pressed firmly on the leads a high current is passed which melts the metal of the lead and its mounting pad between the heads. The welds are usually made far enough out from the body of the device to enable the leads of a faulty device to be cut and a replacement welded in with the welds nearer to the body. Welding achieved moderate success, but the need to leave space for a second weld meant that high packing densities could not be achieved, and the heat involved in making the welds impaired the adhesion of the copper to the base laminate. Because of its high thermal conductivity copper is a difficult metal to weld satisfactorily, and reflow soldering became the preferred means of attaching surface mounted devices.

12.6.2 Reflow Soldering

Leaded devices can be soldered to boards with machines similar to parallel gap welders which press a solid heated head onto the tin-lead coated ribbon leads of the device, pressing them down onto the fluxed, tin-lead plated pads on the board until the solder on both surfaces melts and forms the joints. The head is then cooled and pressure is held on until the solder has solidified. Leaded devices can also be soldered manually with a fine iron and a great deal of care. One obvious advantage of reflow soldered leaded devices is that they are fairly easy to replace if field servicing is required.

Leadless devices and chip resistors and capacitors are not so suitable for manual soldering as the real strength of the joint lies under the body of the package. Generally they are soldered on wave machines with the body of the device passing through the wave, or by vapour phase soldering. Either the board or the component pads are usually screen printed with a solder paste or cream, and the components are then placed on the board by vacuum head 'pick and place' machines similar to the pick-up head of a parallel gap welder. Small components can be held by the tackiness of the solder paste, and on reflowing the surface tension of the solder aligns the pads on the component with those on the board. Larger components may have to be held with an adhesive on their base while they are soldered, especially if a wave machine is used.

12.6.3 Thermal Expansion

When components are assembled on boards with their leads through holes the flexibility in the leads, aided in most cases by the leads being bent, accommodates any relative movements caused by differential thermal expansion. Surface mounted leaded components usually have a double bend in the leads between the body and the board but leadless components offer no mechanism to accommodate any relative movement. Either the substrate must be made of a material with a coefficient of thermal expansion matching that of the bodies of the devices, the mounting pads on the board must be given a sufficient degree of flexibility, or the risk of fractured joints must be accepted.

Flexibility in the mounting pads can be achieved by using an inherently flexible dielectric, like unreinforced polyimide, or by making the outer dielectric layers of a multilayer board of an elastomeric material and shaping the surface tracking from the plated pillars or via holes to the mounting pads in such a way that the pad can have a limited amount of elastic movement about its normal position.

Chapter 13

ACHIEVABLE DENSITIES AND PERFORMANCE

13.1 PACKING DENSITIES[1,8,52]

Packing densities are measurable in two ways. For analogue or discrete component boards it is customary to measure the area actually covered by components, or rather that area which would be covered if all the components were on their maximum size limit, and to express it as a percentage or as a decimal value of the total board area. For digital boards which usually have similar packages arranged in rows and columns the packing density is usually expressed either as the area occupied per 16 pin package or as the number of packs per unit area. In both cases areas occupied by board guides, handles, and any other banned areas are generally excluded.

Discrete component boards do not usually achieve high packing densities because of the variations in size and shape of the components and the fact that components often have to be placed to suit the electrical requirements of the circuit and not to achieve high density packing. Generally a packaging factor of 0·5 can be regarded as good, although factors of 0·85 or 0·9 are sometimes realised. The limit on packing density is normally set by the bodies of the various components rather than by the tracking, which on well designed boards is often mainly fairly short links between adjacent components.

Dual in-line packages can be packaged on double-sided boards with tracks and gaps of around 0·3 mm at up to about 4 sq cm per package. Large boards which carry well over 100 packs may need a slightly lower density to accommodate all the tracks, and any requirements for large numbers of pull-up resistors or capacitors can also lower the density. If thinner tracks are used, possibly with pairs of tracks between pads on 2·54 mm centres, the area per DIP can be reduced to 3 sq cm, but this value is commonly achieved on multilayer boards rather than on double-sided. On multilayer boards, the practical limits of packing for DIPs of around 2 sq cm per pack can be approached, but usually the presence of discrete components limits the density achievable.

Memory chips in 24 pin DIPs can be packed at up to 6·5 sq cm per pack, and 40 pin DIPs require 9 sq cm or more each. Since memory circuits are largely pin for pin matrices, high densities are possible on double-sided boards, which yield the best possible electrical performance to suit the MOS devices. The multilayer boards which are usually needed to achieve the highest packing densities on general logic circuits tend to slow the working of the devices because of the high capacitive loading on the tracks.

The use of flat packs on double-sided boards normally gives a lower packing density than DIPs, but on multilayer boards flat packs or leaded chip carriers can yield about 50 per cent better packing densities than are possible with dual in-line packages.

Leadless chip carriers on multilayer boards without heat sink pillars beneath

the package bodies can give twice the density achievable with DIPs, and mounting the chip carriers on both sides of the board can double the density again, with a limit of 0·8 sq cm of board required per chip carried.

Only a very small percentage of boards in use approach limit packing densities, mainly in the aerospace industry, in large computers, and in telecommunications. Even in these areas few of the boards carry packages in every possible position. Actual packing densities achieved in terms of total usable board area divided by the actual number of chips mounted are usually well down on the 'matrix' values quoted above.

13.1.1 Track Densities and Board Size

As can be seen from Figures 1.5 to 1.7, track densities have increased over the years. Over the whole of the electronics industry there is a wide spread in the smallest allowable tracks and gaps on a board. Some firms design on what they feel are the best figures possible for production, possibly accepting price premiums or higher than normal reject rates as the cost of using advanced technology. Others prefer to hold back until they are certain that all possible snags have been ironed out. Attempts to assess figures for 'mean current use' are often hampered by further orders for old designs which may be in production in large quantities.

Generally, taking what appear to be reasonable average values for better class boards in bulk production and not including development work or older repeat orders, track and gap widths have decreased by a fairly constant 30 per cent per decade since 1950, with a 1980 value of 0·25 mm. At the time of writing, in 1983, it seems that the printed circuit board industry as a whole regards 0·1 mm lines and gaps as a reasonably practical limit for economic bulk manufacture in the future. The 'limit' takes into account the problems met in film generation, printing, plating, and etching.

It appears that to go beyond the 0·1 mm 'limit' the industry may have to adopt new techniques, possibly the use of ion or electron beams in place of photographic resists, vacuum metallisation, and extended use of lasers and plasma etching. The printed circuit board industry has a long way to go before reaching the 0·001 mm 'limit' of the silicon chip processors.

Board sizes suffer the same vagueness as track widths. The sizes of boards for airborne military use have remained constant over the years, to fit the standard ATR cases, and boards for products such as pocket pagers or calculators have diminished in size. Taking the portion of the market where the use of the equipment places no limits on board size, it appears that up to the mid 1970s the average size of boards increased by about 30 per cent per decade, but this growth slowed and may have halted at just over 0·5 m square.

Through the 1970s this growth in size coupled with the use of finer tracks and gaps resulted in a very significant increase in the overall interconnect capability. Often this increase enabled double-sided boards to be used where in earlier models multilayer boards had been considered the only practical method.

There can be little doubt that the financial climate of the later 1970s and the early 80s has encouraged firms to review design practices to ensure that they are receiving value for money, and the benefits of high density packing on double-sided boards, both in financial and electrical terms, are much better understood than they were in the mid 1970s.

13.2 PERFORMANCE

13.2.1 Electrical Performance

Often the electrical performance of printed circuit boards is so good and the boards are so reliable that engineers bother to look at the performance of the boards only when an assembled unit fails to function as was hoped. Many boards are used in situations where their effects on the devices they carry need to be quite large before any malfunction could become apparent. Digital logic devices are designed to offer the best possible noise rejection! On seeing some of the designs produced in the early 1980s, one may sometimes wonder how much safety margin they leave on cross talk, and in the case of multilayer boards by how much they are slowing the working of a system. Since the boards are being ordered in large production batches it has to be assumed that their performance is adequate.

All the data on resistance, line impedances, line delays, surge currents and cross-talk have been available since the mid 1960s and have been verified thoroughly both by controlled experiments and by sampling of practical circuits so there should be no reasons why electrical performances should not be satisfactory. In most design work graphs are used to gain an overall impression of the likely magnitude of any effects, and detailed calculations are not often carried out.

In the later 1970s a massive swing to the use of multilayer boards was predicted, mainly by commercial authorities who did not realise that there is a significant difference in performance between a multilayer board and an apparently similar double-sided board. After some five years, the predicted swing has not occurred. Many firms are now making better use of the full potentials of fine line double-sided boards, and some are converting multilayer designs to double-sided. Better CAD programs and more use of interactive systems are helping to keep large boards for digital systems on two layers. Where space is restricted or where controlled line impedances or electrical screening are necessary multilayer boards still offer the most cost-effective solution, and their use is increasing, but not at the predicted rate.

Surprisingly, there are still some engineers about who automatically associate large digital logic circuits with multilayer boards, and one can still see memory boards for ground based commercial systems where size is virtually unlimited being designed as multilayers. The two most obvious effects of this are that costs will be increased and the working speed of the system will be decreased.

Reports of boards which do not function properly are sometimes made. High frequency analogue boards may fail to perform as expected and require some of the components to be moved or extra earth screens to be added. Mixed digital and analogue circuits sometimes suffer from noise carried by the power plane or from common earth current effects. Digital boards usually give 'right first time' results, although isolated instances of cross-talk, surge current problems, or earth current effects on E type power distribution systems on which end links have been omitted have occurred. All these faults are basically design failings and can, one hopes, be eliminated from similar designs in the future.

13.2.2 Mechanical Performance and Reliability[35,37,53]

Reliability figures on printed circuit boards are very hard to find, and it

appears that, once installed, most boards cause no reliability problems. Connector fingers sometimes fail; some boards appear to warp or twist over long periods; corrosion of tracks and pads can cause problems in hot humid environments if the board finish was not well applied, and the biggest hazard of all appears to be Z axis movement of multilayer boards under thermal cycling or shock which causes hole barrel failures.

Lifted pads on non-plated boards almost certainly occur while the boards are in use but are found only when a repair is necessary because a component has failed. Other reported instances of failed hole barrels, delaminated multilayer boards, and similar faults are usually traceable to faulty manufacture or to mishandling during assembly.

Boards may be charred or burnt when a component failure results in short circuit currents flowing in thin tracks or through other components. Such mishaps are not caused by any unreliability of the board but are secondary results of some other fault in a component.

Manually soldered joints can be a cause of reliability problems, but this is not a failing of the board and will also affect other soldered joints in the equipment. Dry joints may not reveal themselves immediately; the author's rented television set worked well for some five years then failed several times in fairly quick succession as badly made soldered joints finally went open circuit.

Mass soldered joints which have been properly made appear to have excellent reliability over long periods. An American survey read in the early 1970s (but not noted as a reference) listed all failures over the life of a large equipment and gave calculated mean times between failures. Soldered joints did not appear to be very good, matching the better passive components, but a footnote explained that the only joints considered were manually soldered joints on discrete wires—the wave soldered joints on printed circuit boards showed no failures.

An interesting thought on reliability is that if the current attempts to find an alternative to tin-lead as an etch resist and solderable coating for pads do succeed, the reliability of boards could decrease. The present widely used reflowing of the tin-lead plating serves as a very good screening to detect a variety of faults such as poor solderability, faulty bonding of multilayers, porous plating or non-adherent plating, improperly cured base laminate or prepreg, and other potential reliability hazards which are accelerated by the heat of reflowing.

13.3 SIZE, PERFORMANCE AND COOLING[8]

The reduction in size of equipments has made possible high speed electrical performances which would not have been possible with active devices further apart. Systems can be designed with most of the key interconnections well under their critical lengths. Sometimes this reduction in size brings acute thermal problems, and the overall system designer must decide whether it is better to relax the packing density at the risk of losing some of the high speed of the system or to keep up the speed but jeopardise long term reliability because active devices overheat.

These problems are most likely to be found on large high speed digital systems. Most designers of airborne equipment have had to work with the conflicting demands of thermal reliability against space and weight, and the requirements of high speed devices do not really add to their problems. On lower speed ground

based equipments satisfactory electrical results are usually achievable at densities which do not introduce acute thermal problems.

Boards at densities down to 4 sq cm per DIP can mostly be air cooled, either relying on natural convection and the 'chimney' effect of adjacent boards, or fan blown. At higher packing densities some form of conduction cooling usually becomes necessary. Heat ladders directly beneath package bodies are usually the most effective method of conducting the heat to the side of a case or to a 'cold wall'. Metal cored boards can be used, but as the thermal conducting layer is normally isolated from the packages by an insulant they tend to be less efficient than surface ladders. Also the metal core makes soldering the board more difficult.

Ladders conduct the heat directly from the base of the package, and the chip is usually mounted on the base. On metal cored boards the main heat path from the chip to the core is usually through the package leads and the soldered joints. For the best results packages may need to be bonded to a heat ladder. Other methods of cooling such as immersion in inert liquids are being considered, and it is probable that new generations of high speed equipments built with chip carriers or with directly mounted bare chips will use newly evolved cooling techniques.

Some of the high speed semiconductor devices now being developed require cooling to sub-zero temperatures to work, so their use will bring major changes in packaging.

When overall systems are being considered, it must be borne in mind that for the great majority of systems the cooling is complete only when the heat generated by the active devices has been dissipated to the atmosphere. The tighter the various units of a system are packed, the harder it becomes to conduct the heat to suitable radiating areas and the overall equipment may be larger, not smaller than it would be if the active devices were directly air cooled. The electrical interconnections on large systems may have to run from one cooled block to the next, passing the radiating surfaces. This can mean that, whilst the connections within each unit are reduced in length by the high density packing, the overall average interconnection length may be greater than if the components were more evenly spread in the total volume and were directly air cooled. Whether this point need be considered depends on the working of the system. Some systems lend themselves naturally to partitioning into a number of tight, high speed blocks, with only lower speed connections between blocks.

13.4 ALTERNATIVE INTERCONNECTION TECHNIQUES[2]

13.4.1 Wires

Printed circuit boards are not the only possible mounting base and interconnect for components. Simple prototype circuits can be built up on plain drilled panels with wires soldered to the component leads. Logic circuits can be made up on boards which carry DIP sockets, with wires clipped, welded or soldered to the legs of the sockets. Several such systems are available. All suffer from the disadvantage that once the logical or electrical working of the circuit has been proven a proper interconnect, usually a printed circuit board, must be designed and its function verified.

A more durable use of wire is made in the 'Multiwire' system, in which thin insulated wires are laid into a tacky adhesive on the face of a board which can

carry etched track patterns for power and earth distribution. When all the wire has been laid the adhesive is cured, the holes are drilled cutting through the wires, and then through-plated. The wire laying machines are expensive, and as each individual board has to have its wires laid separately the system is not cost effective for large production runs. However, it eliminates the need for a film set so for companies which have a need for small numbers off a variety of board types the system can be cost effective. The densities achievable are roughly equivalent to those of a well designed 6 layer multilayer board, and the embedding of the wires on pre-determined routes allows the electrical properties to be predicted in the same way as those of a conventional board.

In the larger interconnection field wires are steadily being replaced by flexible circuits or by optical fibres.

13.4.2 Thin Film Circuits

The traditional thin film circuit made on a glass substrate with a vacuum deposited metal film is used mainly for specialised applications for which printed circuit boards would be unsuitable, and similarly the thin film circuit would not form a viable substitute for a printed circuit board in most applications. The metallisation layer on integrated circuit chips is a thin film circuit, and these metallisation layers have replaced large numbers of PCBs in the past 20 years. However, for every board which has been replaced by an integrated circuit a larger number of more complex boards has been made to carry the integrated circuit packages and there is no reason not to believe that, as the complexity of integrated circuits increases still further, systems designers will continue to design ever larger systems which will require printed circuit boards in growing numbers.

One advantage of thin film circuits is that it is often possible to deposit high quality resistors and capacitors concomitant with the wiring pattern.

13.4.3 Thick Film Circuits

Most of the advantages which have led to the printed circuit board's virtual dominance in interconnect technology also apply to the ceramic substrate thick film hybrid. It offers reliability, repeatability, predictability, light weight, low volume, and it is capable of forming highly complex inter-connect patterns. It also offers its own advantages of good thermal conductivity, a match for the thermal expansion of semiconductor devices, and the ability to form reliable resistors which will age and track thermally as a matched set within the interconnect package. Against this impressive set of advantages, it suffers from the disadvantages of being limited in size, having a somewhat fragile base, and requiring totally serial manufacture when large numbers of layers are involved. The line and gap widths achievable are similar to those of a good quality PCB, but the dielectric constant is higher so that size for size the ceramic substrate thick film circuit does not offer as good an electrical performance as a printed circuit board. Most ceramic based circuits are fairly small, and the lines are seldom long enough for the inferior electrical properties to have any significance.

The ability to form matched sets of resistors in a thick film hybrid circuit makes it ideal for digital to analogue and A to D converters and other similar circuits where reliance is placed on ratios between resistors. The ceramic base can

be fitted with an hermetically sealed lid so that full protection can be afforded to silicon chips mounted directly onto the substrate.

The techniques used for making printed circuit boards and ceramic substrate hybrids are becoming increasingly interchangeable with the wider use of surface mounting, pillar plating and polymer substrate hybrids.

For making small hybrid circuits the PCB competes very poorly with ceramic thick film circuits, but for large, tough, durable applications the ceramic based circuit is unable to compete. Thus it is quite wrong to view the two technologies as being in any real way competitive; they are complementary and where one is unsuitable the other is likely to be successful. Many equipments use large numbers of ceramic substrate hybrid circuits mounted on PCBs to achieve high density packing of circuits which must contain significant numbers of resistors as well as semiconductors.

13.4.4 Porcelain on Steel

Glazed steel, or porcelain fired onto steel, is available as a thick film substrate which overcomes the two biggest snags of ceramic—size limitation and fragility.

Unfortunately it still has the high capacitance of the ceramic based circuits, and with the longer tracks which become possible on large substrates this could prove to be a major disadvantage. There have been reports of the porcelain crazing with minute cracks, and there must be doubts about the effect of vibration on large panels.

At the time of writing it appears that the technique has some interesting possible applications where circuits can be formed directly onto the steel case of a small unit, but it does not appear to offer any significant advantages over a conventional printed circuit board. The ability to fire on resistors could possibly be an advantage, but for most circuits which require such resistors the existing smaller ceramic substrates can be used. It is probable that like Multiwire and various other alternatives to normal PCB manufacturing processes porcelain on steel will find a place in interconnection technology. It does not seem likely that it will rival conventional boards for the bulk of electronic equipments.

13.5 THE FUTURE FOR PRINTED CIRCUIT BOARDS

There can be little doubt that the not too far distant future will see changes in the processing of PCBs as finer lines are demanded by users at ever lower prices in real terms. The accent will be more on processes which have the flexibility to accommodate the widest possible variety of types of work. At the time of writing two large plants built to mass produce boards have been closed and the equipment sold, and at least two other board producers who aimed to cut costs by large scale production are struggling to keep going.

There can also be little doubt that there will be some changes in the markets for boards. Small electronic units for the domestic and leisure markets can be expected to move increasingly to single chip integrated circuits, but there can also be little doubt that as fast as one market area is lost another will open.Many of the newer markets are likely to be for smaller runs of boards for specialised systems for which existing VLSI circuits are unsuitable and for which quantities do not justify making special semiconductor devices.

GLOSSARY OF PCB TERMS

Access Hole: Hole drilled prior to bonding in layer(s) of a multilayer board to allow access for soldering to a pad on an inner layer.

Additive Circuits: Boards on which the metal of the tracks is formed mainly or entirely of metal plated onto the board.

Artwork: The master of the pattern of the board. Usually larger than the final size; may be drawn or produced by sticking black tapes on a suitable film base.

Back Etching: Chemical removal of plastic substrate beneath a pad (usually on a flexible circuit).

Backing Board: Extra sheet of laminate placed on the outside of a stack of blanks to be drilled. The use of backing boards minimises the formation of entry and exit burrs around the holes.

Base Laminate: The insulating substrate on which the board is made.

Beta Ray Backscatter: A method of measuring the thickness of a plated layer non-destructively.

Blank: See Process Blank.

Blue Grid: Film material on which is printed a grid pattern (usually 0·100 in. squares) in a blue colour to which the film used in reduction will not respond.

Bomb Sights: Special marks used to define datum positions, or to ensure layer to layer registration of films. May be simple, such as a cross or a circle, or may be more elaborate.

Bonding: The pressing together of the separate layers of a multilayer board.

Bowing: The departure from flat of a finished board. (cf. warp; twist).

Bridging: Short-circuiting of a pad to an adjacent track or pad on a board during soldering.

B-Stage: Partially cured epoxide resin which will soften under heat, and which can be used to bond together cured laminate sheets to form a multilayer board when reinforced with glass cloth.

Burnt Plating: Electro-plating which has been carried out at too high a current density. Can occur on isolated features near the edges of a pattern plated board if no precautions are taken.

BVH (Buried Via Hole) Board: A multilayer board in which pairs of layers are interconnected by via holes, some or all of which do not contact either of the outer faces of the board.

Cast Adhesive: Special sheet adhesive material used for bonding polyimide multilayer boards and flexi-rigids. (Similar to prepreg.)

Catalysed Material (Laminate, Substrate): Unclad laminate which has a surface specially treated to make it responsive to electroless plating.

Clad Laminate: Laminate which has an outer metal layer on one or both sides.

Clearance: Metal to metal gaps on a board.

Component Side: The side of the board on which components will be mounted.

Conformal Sheet: Softenable, non-adherent plastic sheeting which may be used during bonding of a multilayer board to equalise the bonding pressure over the whole area of the board.

Contact Print: A film print which has its emulsion on the opposite side to the original from which it was made, and is of reverse polarity, e.g., a contact print taken from an emulsion up negative will be emulsion down positive.

Co-ordinatograph: A table equipped with calibrated cross-slides, by means of which a knife blade can be moved over cut-and-strip film (q.v.), or on which films can be dimensionally inspected through a magnifier on the carriage.

Corner Marks: Marks, usually two intersecting short lines, on artworks and films, which define the corners of the finished board. Should be placed immediately outside the finished board area.

Coupon: Small board usually made in the scrap border of a blank and used for destructive testing.

Cover Coat: Polyimide film with holes to expose pads for soldering stuck down over tracks on flexible circuits.

Cross-talk: Electrical interaction between tracks which can cause faulty working of the board.

Cut and Strip: A method of generating artworks from a special material which has an opaque and a clear layer laminated together. The opaque layer is cut through and pieces are lifted out to leave the final pattern.

Datum Hole: A master hole in a board from which all points on the board are dimensioned.

Datum Line (Mark, Point): A line, mark or point on the board pattern from which all features of the board are dimensioned.

DDB (Double-double Board): A multilayer board which comprises two THP boards bonded together such that all via holes joining the pairs of layers access to an outer face of the board (cf. BVHB).

Dip Soldering: The making of soldered joints on a board by lowering the solder side of a board into a bath of molten solder.

DMLB (Double-multilayer Board): As DDB, but with two multilayer boards.

Drag Soldering: See Dip Soldering.

Drill Chart: Film with pads only used to make drill tape.

Drill Tape: The control tape for a numerically controlled drill, which gives the co-ordinates of the holes to be drilled in the board.

Drop-out Blue: Features in blue on an artwork to which the film used in reduction will not respond.

Dry Film (Resist): A monomer film, polymerisable by exposure to light (usually U-V), which is applied to the surface of a blank to form an etchant or plating resist after exposure through an appropriate film pattern and appropriate development.

Dry Film Solder Resist: Film similar to dry film resist but which is resistant to high temperatures and is applied to a finished board as solder resist. Clearances to pads, etc., can be closer than for screened-on resists.

Dry Film (Transfer Pads): Pads printed in special ink or film which can be transferred from a carrier strip onto an artwork by pressure (similar to 'Letraset').

Earth Plane: More or less continuous sheet of metal forming one layer of a board, and used to connect together the earth or ground pins of all components. May also be used to provide electrical screening, and to control the characteristic impedance of signal tracks. Generally used in multilayer boards, but partial earth planes may be found on some double-sided boards. See also Plane.

Edge Connector: Connector formed on one edge of a board which plugs into a socket mounted on the frame of an equipment. See also Finger.

Electroless Copper (or other metal): Metal deposited on the board by chemical reaction, which does not require any plating current.

Emulsion (side): The 'active' side of a film, on which the pattern is formed.

Etch-back: A process in which the epoxy glass substrate is cut back in the barrels of the holes to leave the copper layers more exposed where they will join the barrels of the through-plated holes.

Etch Factor: See Undercut.

Etch-trap: A feature in a board pattern in which etchant might be retained. Usually applied to any angle significantly sharper than about 80 degrees between metal edges.

Etching: The process of removing unwanted metal from the surface of a board by chemical action.

Eyeballing: Setting up (usually a printing film over a drilled blank) by eye, relying on the skill of the operator to achieve good alignment. With a skilled worker, eyeballed films can be set more accurately than is possible with any tooling system.

Film Stretch (Movement): Changes in size of the pattern on a film, usually with environmental changes. Applied to glass as well as film.

Finger: A strip of metal running towards the edge of a board which makes contact with a spring in the socket into which the board is plugged.

Flexible (Circuit): A single or double-sided circuit printed on a thin, flexible base plastic.

Flexi-rigid: A board, usually multilayer, which has bonded rigid areas from which flexible layers extend to form hinges or tails. Connection to tracks on all layers is by plated-through holes.

Flush-bonded Board: A board on which the metal track pattern is sunk into the base laminate to give a flat surface.

Flying Tail: Projecting flexible portion of a flexi-rigid board.

Glass Transition Temperature: Temperature at which resin ceases to act as a solid.

Grid: A regular orthogonal pattern on which features of a board may be located.

Grid Round-off: The movement of features which are near a grid intersection onto the intersection. Often applied to drill tapes.

Heat Sink—Heat Ladder: A layer of thick metal, usually copper, and usually etched to a pattern, which is bonded to the component side of a board to conduct heat from the components to the edge of the board.

Jig: Frame used to hold blanks during plating. See also Template.

Kapton: Du Pont de Nemours trade name for Polyimide film.

Land: The metal between the barrel of a hole and the edge of the pad around the hole on the surface of the board.

Landless Hole (Board): A hole (or boards with such holes) which has no pads around the hole, such that any track to the hole joins directly into the through-plated barrel of the hole.

Layout: See Red-Green.

Lay-up: The definition of the composition by layers of a multilayer board. Also applied to the process of generating artwork.
Legend: Pattern printed onto the surface of a board which defines the positions of the components to be mounted on the board.
Light Box: A large box with illuminated front or top on which artworks can be made, and on which film can be checked.
Locating Slot: A slot in the edge of the board which fits over a key in a connector, etc., and locates the board in the connector.
Mass Soldering: The making of all soldered joints on a board by machine. (See Wave Soldering or Dip Soldering).
Master Film: The accurate set of 1 to 1 films from which all phototools required during production of the board will be made.
Microsection: A small piece cut from a board to enable microscopic examination to be made.
MLB (Multilayer Board): A board with three or more layers of conductive tracks. Electrical contact to the inner layers is usually made by through-plated holes.
Moisture Trap: A small gap on an assembled board in which moisture can be retained, and from which corrosion of the board may start.
Negative (Film, Artwork, etc.) A pattern on which the features on the finished board which will appear in copper, solder resist ink, or legend ink show as clear areas.
Off-camera Negative: See Reduction Negative.
Orientation Marker: A mark on a film or artwork which indicates the correct orientation of the film.
Pad: The metal annulus on the surface of a board round a hole. Also applied to the circle of material stuck onto an artwork to define the pad on the board. N.B. Usually circular, but pads may be square, octagonal, or any other shape.
Panel Plating: A process in which the whole of the board surface is plated with copper.
Pattern Plating: A process in which only the required track pattern on the board surface is plated with copper.
Peel Strength: A measure of the adhesion of the copper to the board substrate.
Photo-resist: A light-sensitive material applied to the surface of a blank to define the track pattern. (See Dry Film Resist).
Pillar—Plated Pillar: Solid inter-layer connection sometimes used in multilayers as an alternative to buried via holes.
Plane (Usually Earth or Power): A layer of metal which has clearance holes to avoid pads or other features which must not be connected to the plane.
Plasma Etch: Removal of material by ionised gas plasma. Normally used in PCB factories to remove resin smear.
Polarising Slot: A slot in the edge of a board which fits loosely over a key or pin in an edge connector, etc., to ensure that only the correct board can enter the connector.
Polyester: Low melting point plastic film used for cheap flexible circuits. (Polyester plastic has many other applications.)
Polyimide: Higher melting point plastic film used as base for flexible portions of flexi-rigid boards and for many flexible circuits.
Positive: A film or artwork on which the areas which will appear in copper, solder resist, or legend ink on the finished board show as dark areas. (See also negative.)

Power Plane: Similar to earth plane (q.v.) but used to connect a power supply to the appropriate pins of all components. (See also Plane.)
Pre-heat: The warming of an assembled board prior to mass soldering.
Prepreg: Sheets of glass cloth impregnated with B-Stage epoxy resin, used to bond multilayer boards.
Print and Etch (Boards): Boards or portions of boards, produced by etching with resist coating over the metal pattern, on which no plating is carried out.
Process Blank: The panel in which boards are manufactured up to the trim stage. May comprise only one board or may be a number of boards.
Prototype: Board which the customer would like to have delivered before the artwork is finished.
PTH (B): See TPH.
Punch and Crunch (Board): Term usually applied to paper-based print and etch boards which are mass produced by punching all the holes.
Rail (usually Earth and Power): Broad track used especially on double-sided boards to link earth or power supply pins on all components. Also a metal bar which holds board guides or sockets in an equipment frame.
Red-Green: A coloured drawing of the board pattern in which the colour denotes on which side of the board a feature is to be placed. May also be Red-Blue, or other combinations of colours.
Reduction Gauges (Marks): Accurately placed marks on an artwork which must appear at a specified distance apart after photo-reduction. See also Bomb Sights.
Reduction Negative: The master negative produced when an artwork is photo-reduced.
Reflow: Heating of a board to melt the tin/lead coating over the tracks.
Registration: The layer to layer alignment of features on the board—more commonly met as misregistration of varying extent.
Release Sheet (or Film): Non-adherent plastic sheet, usually PTFE based, used to prevent a multilayer board sticking to the press plattens or packing sheets during bonding.
Resin Recession: The pulling away of resin from the hole barrel after thermal shock.
Resin Smear: Resin, smeared over the edge of an internal copper layer of a multilayer board during drilling, which prevents the layer from making electrical contact with the barrel of the hole.
Reversal Print: A film which is of the same polarity as the original from which it is made, but has its emulsion on the opposite side.
Rigidised Flexible: A flexible circuit on which component mounting areas are glued to a rigid plate, which can be plastic or metal.
Robber Bars: Areas of metal outside the finished board area which are included on the master films to minimise the risks of burnt plating within the board area.
Roller Tinning: The coating of a finished board (copper tracks) with hot tin/lead by rollers.
Screened MLB: A multilayer board which has earth planes (or earth and power) on its outer layers to minimise the possibilities of signal radiation from the board.
Scribed Film: A film with a special surface into which pads, tracks, etc., can be scribed, usually on a co-ordinatograph.

Secondary Master (Negative, Positive, Film): A copy of a master negative, positive or film which has been made under controlled conditions and which is used for subsequent copying to avoid risk of damaging the masters.
Slivering: The breaking away of the edges of tin/lead plating when a track has been undercut on etching.
Smear Removal: The chemical removal of resin smear, usually by concentrated sulphuric acid.
Solder Resist: An epoxide resin applied to the board surface which leaves any pads exposed but covers all tracks to eliminate any risk of bridging on soldering. May also be applied for environmental protection or appearance.
Spotting (in): Re-touching local defects on a film or printed pattern on a blank.
Stability (films, materials, etc.): Resistance of a material to movement during changes in environmental conditions, usually lacking in materials used for PCB production. Even glass is not absolutely stable.
Step and Repeat: Process of copying films, etc., to lay down identical images at regular intervals over a large film.
Subtractive Circuits (Process): Boards on which the track pattern is formed mainly or entirely by etching away the unwanted areas from a full sheet of metal.
Swimming: Sideways movement of a track on a multilayer board during bonding.
Tape (See Drill Tape): Also applied to the material used to define tracks on an artwork.
Teardrop: Pad which runs to a point to blend into the track running from it.
Template: A master pattern (usually metal) which guides a routing cutter during trimming. Also a master pattern which guides the drill position on non numerically controlled machines—often called a drill jig.
Test Coupon: Small pieces of board carrying a special pattern, made alongside a required board, which can be used for destructive testing such as micro-sectioning.
Tg: See glass transition temperature.
Tolerance: The amount of variation allowed on any design parameter. Often inadequate for economic manufacture.
Tooling Hole: A hole (usually not through-plated) in a board or blank which is used to locate a drilling template, routing template, etc.
TPH (B): Through-plated hole. Board with through-plated holes.
Track: The electrical connection between two pads.
Triazine: Dielectric material with higher glass transition temperature and better thermal stability than epoxy resin. More expensive than epoxy and not in common use.
Twist: Similar to Bow (q.v.).
Unclad (Laminate): Laminate which does not have any copper sheet bonded to its surface during manufacturing of the laminate.
Undercut: The amount by which an etchant cuts beneath the resist, or tin/lead or gold plating, into the side of a track or other feature.
Via Hole: A hole whose only purpose is to connect a track on one side of the board through to a track on another layer (or side).
Warp: Similar to Bow (q.v.)
Wash-off Film: Film on which the emulsion can be removed with a wet eraser. Used for copies of artworks for long-term storage, whenever there is any possibility that a future change may be needed.

Wave Soldering: The making of soldered joints on a board by passing the board over a wave of molten solder.
Wet Resist: A photographically polymerisable etch or plating resist which is applied to the board in liquid form.

REFERENCES

1. Scarlett, J. A., *Printed Circuit Boards for Microelectonics*, 2nd Ed., Electrochemical Publications, Ayr (1980).
2. Herrmann, G., *Handbuch der Leiterplattentechnik*, Eng. Trans., Electrochemical Publications, Ayr (1984).
3. Kendall, J. P., 'The PWB as a mounting for semiconductors', Proc. Internepcon, p. 40 (1981).
4. Lane, W. V., 'Two part versus one part printed circuit connectors', Proc. Nepcon, p. 238 (1971).
5. Menzies, G. M., Scarlett, J. A. and Taylor, M., 'Making printed circuit boards fit their edge connectors', Proc. Connector Symposium, Internepcon, p. 60 (1973).
6. Dyke, D. E. and Fedor, J. R., 'Electrolytic copper foil for new PC applications', Proc. 2nd PC World Convention, **2**, p. 37 (1981).
7. Holt, J. S., 'The production of glass fibre fabrics for use in printed circuit boards', Proc. Internepcon, p. 17 (1976).
8. Scarlett, J. A., *The Multilayer Printed Circuit Board Handbook*, Electrochemical Publications, Ayr (1984).
9. Kolias, J. T., 'Interboard interconnections in digital computers', Proc. 2nd PC World Convention, **1**, p. 399 (1981).
10. Leach, F. A. and Scarlett, J. A., 'Suppressing spurious signals in saturated switching systems', Proc. Internepcon, p. 1 (1972).
11. Scarlett, J. A., *Transistor-Transistor Logic and its Interconnections*, Van Nostrand Reinhold, London (1972).
12. Scarlett, J. A., 'Printed circuit board size for digital systems', Proc. Internepcon, p. 1–41 (1970).
13. Scarlett, J. A., 'The distribution of power and earth on boards for high speed digital logic', Proc. 2nd. PC World Convention, **1**, p. 58 (1981).
14. Howell, W. C., 'Artwork and the precision photographer', Proc. Internepcon, p. x–28 (1970)
15. Leach, F. A. and Scarlett, J. A., 'Practical problems in precision artwork', Proc. Internepcon, p. 183 (1971).
16. Menzies, G. D., 'Treatment of copper surfaces in printed circuit manufacture', *Circuit World*, **7**, No. 2, p. 39 (1981).
17. Fullwood, L and Kurisu, V., 'Resist adhesion—It's the surface that counts', Proc. Internepcon, p. 19 (1980).
18. Dolby, R. F., 'Photo-resists for etching and electroplating', Proc. Internepcon, p. 246 (1968).
19. Haywood, S. J., 'Dry film resist in the printed circuit industry', Proc. Internepcon, p. 1–53 (1970).
20. Lucas, J. F., 'Predicting performance of dry film photo-resist by analysis of the resist sidewall', Proc. 2nd PC World Convention, **1**, p. 111 (1981).

21. Walter, C. A., 'Screen printing for printed circuit boards', *Circuit World*, **7**, No. 2, p. 35 (1981).
22. Swartzell, J. C., 'Etching parameters: Their individual and collective impact on high density circuit production', Proc. 2nd PC World Convention, **2**, p. 16 (1981).
23. Hillis, M. R., 'Electrolytic regeneration of etching solutions', Proc. Internepcon, p. 9 (1980).
24. Mullaney, P., 'Recent developments in hydrogen peroxide sulphuric acid etching', Proc. Internepcon, p. 6 (1980).
25. Hill, P. W., 'In search of the better hole', Proc. Internepcon, p. 251 (1972).
26. Weiss, R. E., 'Evaluation of drilled hole quality as a function of speed and feed in multilayer boards', *Circuit World*, **7**, No. 1, p. 16 (1980).
27. Harris, I. R., 'Smear free holes for printed wiring boards', Proc. Internepcon, p. 9 (1973).
28. Luke, D. A., 'New non-precious metal calalyst for through-hole plating', *Circuit World*, **7**, No. 4, p. 36 (1981).
29. Courduvelis, C. I. and Sutcliffe, G. R., 'Getting the best from electroless copper solutions', Proc. Internepcon, p. 14 (1980).
30. Heywood, D. M., 'The multilayer lamination process', Proc. Internepcon, p. 27 (1974).
31. Engelmaier, W., 'Characteristics of multilayer PC boards during lamination', Proc. Internepcon, p. 46 (1974).
32. Bramel, J. R. and Craven, P. M., 'Optimization of prepreg for high temperature performance', Proc. 2nd PC World Convention, **2**, p. 27 (1981).
33. Luke, D. M., 'Factors affecting some properties of multilayer printed circuits', Proc. Internepcon, p. 27 (1974).
34. Perrone, L. S., 'A review of gas plasma desmearing and etch-back techniques', Proc. 2nd PC World Convention, **1**, p. 127 (1981).
35. Dane, A. J., 'Multilayer board problems in equipment production', Proc. Internepcon, p. 25 (1981).
36. Mitchell, J. P. and Welsher, T. L., 'Conductive anodic filament growth in printed circuit materials', Proc. 2nd World Convention, **1**, p. 80 (1981).
37. Wild R. N., 'Thermal characterization of multilayer interconnection boards', Proc. Internepcon, p. 43 (1975).
38. Messner, G. and Powers, D. W., 'Additive processes for large volume circuit manufacture', Proc. Nepcon, p. 45 (1971).
39. Nitsch, H., 'Very fine line techniques for making printed circuits and multilayer boards', Proc. 2nd PC World Convention, **1**, p. 375 (1981).
40. Smith, B. R., 'Double multilayer printed wiring boards. A novel technique to overcome a packing density problem', Proc. Internepcon, p. 41 (1974).
41. Hurd, D. E., 'Multilayer printed circuits without through holes', Proc. Nepcon, p. 208 (1964).
42. Engelmaier, W., 'Improving the design for flexibility of flex circuits', Proc. Internepcon, p. 12 (1974).
43. Hogan, T. P., 'Some practical aspects of flexible circuit design', *Circuit World*, **7**, No. 1, p. 63 (1980).
44. Cannizzaro, J. R., 'The development of polyimide multilayer boards containing integral flexible circuitry', Proc. Internepcon, p. 1 (1970).

45. Scarlett, J. A., 'Flexi-rigid boards—A total interconnect system', *Circuit World*, **5**, No. 2, p. 19 (1979).
46. Manko, H. H., *Solders and Soldering,* McGraw Hill, NY.
47. Rubin, W., 'The relative merits of water soluble and non corrosive resin based fluxes for mass soldering', Proc. Internepcon, p. 116 (1981).
48. Kingsley, D. S. and Worroll, A., 'Rework of boards by through-hole plating of individual holes', Proc. 2nd PC World Convention, **1**, p. 346 (1981).
49. Leonida, G., *Handbook of Printed Circuit Design, Manufacture, Components and Assembly,* Electrochemical Publications, Ayr (1981).
50. Smith, R. B., 'The implications of automated assembly', *Circuit World,* **4**, No. 1, p. 46 (1977).
51. Rahn, A., 'In-line vapour phase soldering', Proc. Internepcon, p. 112 (1981).
52. Scarlett, J. A., 'Packing Densities', *Electronic Data Library*, Morgan Grampian, London, Vol. 5, p. 54 (1969).
53. Wild, R. N., 'Some fatigue properties of solders and solder alloys', Proc. Internepcon, p. 194 (1975).

INDEX

Acceptability of boards, 150
Acceptable Quality Level, 154, 158
Access holes, 110, 163
Acrylic (resin), 89, 115, 133, 135
Acrylonitrile butadiene, 121
Additive process, 97, **119**, 142
Adhesion, 55, 107, 120, 130, 147, 150, 158
Adhesive, 133, 170
Agitation, 87, 97, 120, 132
Air cooling, 57, 121, 167, 184
Aluminium, 109, 119, 121, 127, 142
Ammoniacal etchants, 87
Amplifiers, 17, 56
Analogue circuits, 15, 17, 29, 33, 49, 56, 59, 182
Annuli, 74
Approval, 78, 150
Artwork, 72, 75
Assembly, 18, 48, 50, 164
Automation of
  inspection, 157
  insertion, 26, 54, 56, 148, 153, 168
  printing, 86
  production, 132, 141
  plating, 97
  testing, 53, 157

B-stage resin, 78, 112
Back etching, **131**, 133
Backing sheet, 92, 115
Back wiring, 8, **16**, 29, 66, 137
Base material (see Laminate)
Bends in
  package leads, 27, 164, 166, 169
  tracks, 65, 74
Blistering, 176
Blotting paper, 109, 114, 131
Blue grid, 72, 73
Bond strength, 107, 114, 138, 161, 178
Bonding, 24, 27, 78, 104, **111**, 129, 131, 136, 160, 183
Bowing, 52, 114, 141, 163
Bridging, 57, 146, 148, 174
Brightener, 97, 102, 143
Build, 110, 114
Buried via hole, **21**, 66, **120**, 135
Burnt plating, 101, 102, 119, 143
Bus bars, 61, 102, 149

Cableforms, 8, 133, 137
Camera, 75
Capability, 150
Capacitance, 17, **33**, 110, 127, 133
Capacitor, 8, 56, 59, 149, 164, 180
Case (equipment), 16, 48, 51, 133, 137, 186
Ceramic, 4, 5, 178, 185
Chamfering, **93**
Characteristic impedance, 33, **35**, 55, 59, 127, 182
Chassis, 2, 3, 4, 49
Checking gauge, 152
Chip carrier, 9, 56, 121, 133, 167, 178, 180
Chromic acid, 87, 115
Cleaning, 81, 95
Clearances, 32, 54, 55, 117, 146, 161, 181
Colour of boards, 24, 25
Coloured artworks, 73
Computer Aided Design, 41, 47, 58, 64, 90, 182
Concessions, **152**
Conduction cooling (see Heat ladders, sinks)
Configuration constant, 46
Conformal sheet, 109, 114, 126, 131
Connectors, 10, 26, 47, 49, 52, 56, 61, 94, 129, 130, 137
Contact prints, 73, 74, 76, 123
Contamination of solder, 142, 165
Convergence, 77
Cooling, 17, 50, 57, 121, 127, 134, 167, 183
Co-ordinatograph, 72
Copper, 86, 94, 121, 127, 129, 140, 171
  acid, 97
  electroless, 97
  foil, 24, 110, 115, 119
  pyro-phosphate, 97
Cordwood module, 8
Corrosion, 10, 13, 94, 129, 140, 145, 146, 148, 183
Costs, 13, 47, 48, **51**, 53, 55, 78, 104, 138, 140, 145, 153, 168, 182
Coupon (see Test pattern)
Cover coat, **129**, 135, 136, 138
Cracking of hole barrel, 28, 118, 160, 177, 183
Cratering (see Outgassing)
Crazing, **174**
Critical length, 42, 43, 44
Cropping of leads, 168
Cross talk, 2, 16, 17, **43**, 55, 65, 182
Current carrying capacity, 31, 60
Cut and strip, 72
Cut tracks, 130, 157, 161

Decoupling, 42, **59**, 149
Defects, 154, 157, 159

Definition of print, 84, 116, 123
Delamination, 114, 176, 183
Delay, 17, 35, 182
Delivery times, 48
Design, 7, 17, **47**, 54, 182
Diameter of holes and pads, 54, 95, 100, 117
Diazo prints, 76
Differential expansion, 27, 121, 123, 167, 179
Diffusion, 142, 171
Digital circuits, 16, 29, 55, 56, 59, 149, 182
Digitising, 71, 90
Diodes, 8, 164, 166
Dissipation (see Cooling)
Doubled boards, 21, 22, 50, 66, 120
Double-sided boards, 18, 49, 50
Dowelling, 74, 104, 157
Drag soldering, 147, 162, **172**
Drilling, 26, 88, 115, 127, 155
  jig, 89
  machine, 88, 90, 160
  tape, 90
Dry film
  resist, 83, 99, 100, 107, 123
  solder resist, 146
  transfers, 72, 75
Dry joint, 171, **173**, 183
Dual In-line Package, 6, 54, 56, 121, 166, 178, 180

Earth plane, 33, 42, 59, 74, 110, 127, 177
Edge connector, 10, 13, 94
  finger, 13, 94, 162
Eisler, Dr Paul, 4
Electrical testing, 157
Electroless plating, 5, **95**, 116, 119
Electroplating, 5, 94, 119, 121, 155, 160
Emulsion, 67, 68, 84, 111
Epoxide resin, 28, 78, 107, 115, 127, 129, 146, 148, 159, 179
Epoxy-glass, 24, 78, 80, 88, 135
Etchants, 84, **86**, 107
Etch back, 115, 160
Etching, 81, **86**, 94, 99, 104, 107, 116, 119, 131, 155
Etching machine, **87**, 107
Etch resist, 81, 94, 99, 131, 145
Etch trap, 65
Exposure, 69, 76, 84, 107
Eyelets, 5, 15, 163

Feed back, 17
Ferric chloride, 86
Film, 2, **67**, 76, 106, 150, 154
  stretch, 68, 76
Finishes, **140**
Fishbead, 15, 166
Flammability, 25, 150
Flat pack, 167, 178
Flexible p.c., 8, **19**, 25, 49, 80, **129**, 151
Flexi-rigid, **20**, **50**, **134**, 176
Flow rate, 110, 112
Flush bonding, 23
Flux, 140, 144, 166, 170, 173
Flying tail, 134
Frame, 16, 48
Frame wiring, 8
Freon, 100

Gap (see Clearance)
Gas vent slot, 107
Gauge marks, 68, 73, 75, 107, 155
Gel time, 113
Glass
  masters, 76
  reinforcement, 25, 27, 78, 117, 118
Glass transition temperature, 113, 144, 146, 163
Gold, 47, 49, 80, 94, 123, **145**, 171
  embrittlement of solder, 145, 171
Grid-layout, 64, 70, 90, 151, 157
Growth of feature size, 77
Guides
  board, 10, 26, 51, 56, 123, 180
  insertion, 167
Guillotine, 80, 92, 169

Hand soldering (see Manual soldering)
Handle, 26, 51, 56, 180
Heat, 31, 58, 133, 167, 183
  ladders, 110, **121**, 127, 133, 167, 183
  sinks, 123, 167, 177
Hinged board, 134
Holes, 1, 25, 54, 88, 95, 129, 149, 155, 163
Hot gas soldering, 173
Hybrid artwork, 72, **73**
Hydrochloric acid, 97
Hydrofluoric acid, 116
Hydrogen peroxide, 87
Hydrosqueegee, 144

Icicle, **176**
Inclusions, 117, 143
Inductance, 17, 33, **35**, 59, 129
Infra-red
  reflow, 144
  soldering, 173
Ink, 84, 86, 148, 155
Insertion
  force, 11
  of components, 26, 52, 54, 148, 152, **167**
Inspection, 144, **153**
Intermetallic compounds, 141, 144, 145, 171
Inverse beam lead, 133
Ion depletion, 116, 126, 160
Iron (soldering), 114, 160, 172, 177

Jig (see Drilling)
Jones diagram, 37

Key (locating), 14
Kiss pressure, 113

Label, 67, 111
Lacquer, **140**
Laminate, 24, 78
Land, (see also Pad) 55
Landless hole, 28, 117, 118
Laser, 77, 126, 131, 132, 181
Layer numbering, **111**
Lay up, **109**
Leadless Inverted Device, 167, 178
Legend, 146, **148**, 157
Length of track, 33, 42, 50, 58, 184
Life of boards, 28, 130, 182
Lifted pad, 15, 28, 130, **162**, 182
Light box, 68, 155
Line reflections, 2, **35**, 46, 59
Linishing, 95, 120, 126
Malfunction of circuit, 2, 42, 58
Manual artwork, **72**
Manual soldering, 15, 132, 168, **172**, 173, 174, 179, 183
Master films, 76
Master pad sheets, 73
Mean Time Between Failures, 183
Melamine, 25
Metal cored board, **127**, 184
Methylene chloride, 84
Microsection, **159**
Microwave circuit, 1, 23, 34
Misalignment of connector, 12
Misregistration (see Registration)
Moisture absorption, 24, 176
Moisture trap, 127, 164, 166
Monitoring of signal, 46
Mounting of components, **164**
Movement of material, 26, 55, 68, 104, 114, 137
Moving of tracks, 64
Multilayer board, 6, 11, **20**, 42, 47, **49**, 65, 104, 135, 162, 181, 182
Multiwire, 184

Naked chips, 8, 133
Necking of tracks, 124
Negative, 67, 70, 73, 101, 155
Negative artwork, 74
NEMA specifications, 25
Nickel, 24, 145, 161, 171
Noise rejection, 182
Numbers—board, 67, 156
  —layer, 67, **111**

Optical fibres, 8, 185
Orientation, 56, 63, 168
Outgassing, **176**
Oxidation, 86, 104, **107**, 114, 141, 147

Package placement, 41, 47, **56**
Packing density, 49, 54, 59, 63, 121, **180**
Packing of press, 109
Pads, 1, 28, 54, 72, 94, 116, 131, 162
Paint, 81
Palladium, 97, 145, 171
Panel plating, 95, **97**, 119
Paper base, 24, 49, 78
Parallax error, 70
Pattern plating, 95, **100**, 119
Performance, 18, 50, 55, 63, 127, 176, **182**, 183
Phenolic resin, 24, 49, 50
Photography, 71
Photomechanical printing, **83**
Photoplotting, 71, 74
Photoreduction, 71, 73, **75**
Photoresist, 83, 88
Pillar plating, 1, 5, 22
Pin, 15, 133, 138, 166, 176
Pinhole, 86, 101, 140, 150, 154, 160, 161, 176
Planning, 78, 100
Plasma etching, 115, 181
Plug and socket, **10**
Polyethylene, 25
Polyester, 25, 85, 129
Polyimide, 25, 115, 121, 129, 176, 179
Polysulphone, 25
Porcelain on steel, **186**
Positive, 67, 68, 154
Power
  distribution, **59**, 64
  plane, 42, 50, 59, 74, 110, 114, 177
Pre-heat, 128, 142, 173
Prepreg, 78, 104, 110, 112, 117, 120, 122, 126, 155, 183
Press, 104, **111**
  cycle, **112**
  plates, 109
Printed back wiring, 16, 29
Printing, **83**, 146
Process monitor gauge, 107, 155
Profiling, 14, 25, **92**, 131, 136
Prototypes, 81, 140
PTFE, 25, 109, 136
Punches and punching, 24, 81, **88**, 92, 104, 132

Reduction gauge, 68, 75
Reflections (see Line)
Reflow
  of tin-lead, 30, **143**, 183
  soldering, 133, 161, 178
Regenerative etchants, 87, 120
Registration, **70**, 104, 124, 130, 146, 160
Release sheet, 109
Reliability, 8, 50, 118, 182
Removal of leads, 170, 177
Repairs, 53, 152, **160**
Replacement of tracks, 161
Resin recession, 160
Resin smear, 115, 160
Resistance, 17, **29**
Resistors, 1, 8, 15, 56, 149, 164, 180, 185
Resonance, 26
Retractile, 129
Ribbon cable, 8, 11, 129
Rigidised flexible, **20**, 49, **133**

Rinsing, 88, 102, 115, 132, 147
Rivets, 5
Robber bar, 101, 107, 143
Robot, 167
Roller tinning, **141**
Rosin, 142, 170, 173
Routing of profile, 92, 131
Routing of tracks, 47, 60, **64**
Rules of design, 17

Sampling, **153**
Sawing, 80, 92
Screening (electrical), 34, 49
Screen printing, 83, **84**, 107, 146, 155
Seeding, 5, 97, 120
Selective plating, 83, **123**, 132, 139, 148
Sensitised laminate, 77, **120**
Sequential boards, 22, 49, 120, 126, 135
Servicing, 48, 52, 149
Short circuit, 111, 117, 143, 146
Short track, 33, 42
Sight drilling, **88**
Silicone resin, 32
Single-sided board, **18**, 49, 131
Size of
  boards, 47, 51, 181
  laminate sheet, 51, 78
Slivers, 143
Slots, 14, 93
Solder, 147, 169
Solderability, 140, 143, **170**, 183
Solder buttons, **132**
Solder cream, 173
Soldering, 15, 54, 128, **170**
Solder levelling, **142**, 147
Solder resist, 74, 121, 130, 141, **146**, 156, 161, 174
Solvents, 78, 84, 86
Spares holding, 52
Specifications, **150**
Spider bond, 8, 133
Split laminate, 110
Spotting, 84, 101
Squareness, 71
Stability, **26**, 68, 76, 84, 137
Stakes, 5
Standards, **47**, 150
Stencils, 85
Step and repeat, 76, 156
Stiffeners, 26, 56
Strength, 25
Strike plating, 101
Strip line circuit, 23
Subtractive process, **81**, 120
Sulphuric acid, 87, 115
Surface mounting, 1, **50**, 121, 126, 167, **177**
Surge current, 2, 35, **42**, 59, 182
Swimming, 114
System design, 52

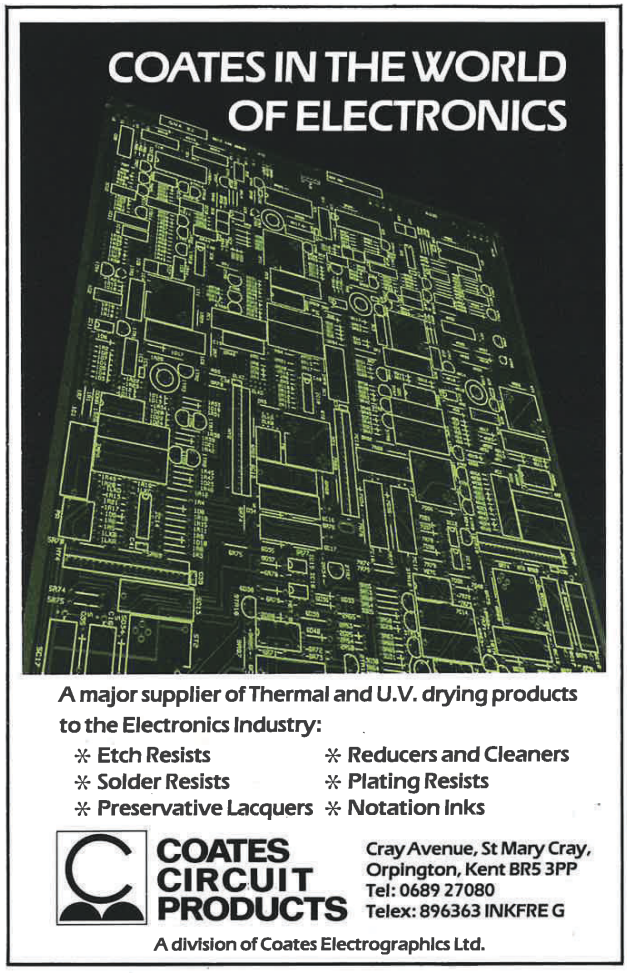
Tagstrip, 2, 5
Tape, 72, 74
Tape Automated Bond, 8, 133
Tear-drop pad, 124
Tenting, 99, 102, 120, 124
Testing, 18, 47, **53**, **157**
Test pattern, 107, 158
Thermal
  expansion (see Differential expansion)
  cycling, 28, 183
  shock and stress, 27, 118, 144, 173, 183
Thick film circuit, 4, 6, 178, **185**
Thickness of
  boards, 111, 150
  copper foil, 24, 111, 118, 119, 139
  plating, 94, 99, 101, 143, 145, 153, 157, 159
Thin film circuit, 3, **185**
Through-plated hole, 30, 49, 55, 88, **95**, 120, 132, 151, 157, 159, 163, 169
Tin-lead plating, 30, 100, 102, 124, **143**, 178, 183
Tolerances, 15, 26, 55, 76, 92, 93, 117, 127, 138, 146, 150, 157
Tooling holes, 26, 74, 81, **104**, 127, 137
Track, 1, 5, 29, 40, 57, 72, 94, 114, 117, 133, 138, 139, 148, 150, 157, 161, 162, 180
Trackability factor, 61, 64
Transfer press, 113
Transistor, 2, 5, 8, 164
Translation board, 157
Triazine, 25
Twist, 52, 114, 163, 183
Type of
  board, **18**, 48, 134
  film, 67

Undercutting, 86, 100, 119
Ultra-violet light, 83, 84, 86, 120, 124, 127

Vapour phase
  reflow, 144
  soldering, **173**, 179
Via hole, 6, 40, 49, 55, 62, 64, 118, 133, 149, 163
Vibration, 26, 52, 60, 164, 165, 186
Visual inspection, 156
Voids, 114, 117, 139, 160
Voltage clearance, 32
Voltage drop, 31, 60

Warping, 113, 114, 141, 146, 163, 183
Wave soldering, 57, 63, 118, 132, 147, 172, 174, 179
Welding, 161, **178**
Wetting, 84, 86, 140, 143, 171
Whiskers, 33, 143
Width of track, 7, 42, 47, 49, 55, 119, 150, 151, 153, 180
Wire, 1, 3, 14, 137, 163, 183, **184**
Wiring harness, (see also Cableform) 129

Z axis expansion, 37, 118, 183



A major supplier of Thermal and U.V. drying products to the Electronics Industry:

- ✲ Etch Resists
- ✲ Solder Resists
- ✲ Preservative Lacquers
- ✲ Reducers and Cleaners
- ✲ Plating Resists
- ✲ Notation Inks

COATES CIRCUIT PRODUCTS

Cray Avenue, St Mary Cray, Orpington, Kent BR5 3PP
Tel: 0689 27080
Telex: 896363 INKFRE G

A division of Coates Electrographics Ltd.

NUBAL MANUFACTURE EQUIPMENT FOR:

Scrubbing. De-burring. Drying – hi-speed and super hi-speed. Rinse drying and super rinse drying. Solder levelling. Etching. Developing. Dry film and ink stripping. Re-circulating and filter systems. Resist/film separators. Feeders. Stackers. Fluxing. Roller tinning. Chemical spray processors for rigid, inner layers and flex. Hi-pressure rinses. Film area measurement. Flux removal after reflow. Vacuum cleaner prior to screening.

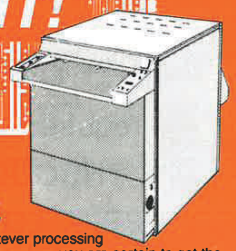
And our own range of circuit board preparation brushes.

If you require a machine that is not listed for a conveyorised bare board process, then we can almost certainly offer a suitable machine for your exact requirements.

NUBAL

Tribune Drive,
Trinity Trading Estate,
Sittingbourne, Kent.
Tel: (0795) 77478
Telex: 965286

THE BEST FILMS ON THE CIRCUIT!



AGFA PHOTOGRAPHIC FILMS FOR THE ELECTRONICS INDUSTRY.

Quite simply Agfa-Gevaert supply the highest quality reprographic films available. Especially for printed circuits. Where specific photographic and physical requirements are extremely demanding. Like:

- High contrast
- Optimum line sharpness
- Wide exposure and development latitude
- Extreme dimensional stability.

Agfa have perfected films for making masters and working copies to meet these exacting demands.

- **HDA3p** for lith or line processing
- **HDU3p** for line processing
- **HDS3p** for lith processing
- **HDP3p** for plotters with low light intensity.

Whatever processing method you use, you are certain to get the very best results, time after time, with Agfa photographic films.

And if you're interested in the very latest automatic processors, we can supply them too. Economical, trouble-free processing with a dry-to-dry access time of only 80 seconds. The dot quality and line sharpness is unequalled by any other system.

Ask for Industrial Products, Graphic Systems Division and we'll gladly send you all the information you'll need.

Agfa-Gevaert Ltd,
Graphic Systems Division,
27 Great West Road, Brentford,
Middlesex TW8 9AX Tel: 01-560 2131

GRAPHIC
SYSTEMS DIVISION
AGFA-GEVAERT

Imasa manufacture and distribute the complete range of ELECTROLESS AND ELECTROLYTIC PROCESSES, together with related STRIPPERS, ETCHANTS, FLUXES and AUTOMATED ELECTOPLATING MACHINERY required by manufacturers of THROUGH HOLE PLATED PRINTED CIRCUIT BOARDS

IMASA

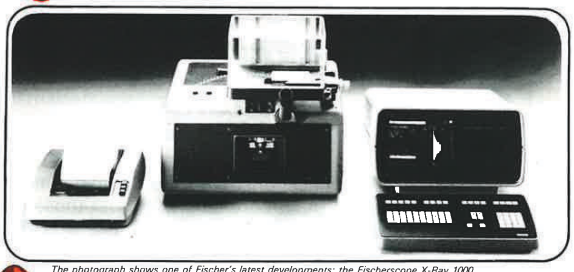
A complete service for
Surface Treatments from:—
IMASA Limited,
188 Bath Rd., Slough. SL1 4DU. England.
Tel: 0753 28282. Telex: 847220

COATING THICKNESS ON ANY SHAPE

Over two decades of continuous development and production for worldwide markets back the FISCHER range of thickness, porosity, material testing and process control equipment.

FISCHER'S instrumentation utilises the latest technical innovations plus full measurement data processing to meet modern industrial requirements. From the simplest of gauges to the most sophisticated control instrumentation FISCHER uses the latest internationally specified methods of:

X-RAY FLUORESCENCE	ISO 3497
BETA BACKSCATTER	ISO 3543
EDDY CURRENT	ISO 2360
ELECTROMAGNETIC INDUCTION	ISO 2178
COULOMETRIC	ISO 2177



The photograph shows one of Fischer's latest developments: the Fischerscope X-Ray 1000.

Write, telephone or telex for details of the FISCHERSCOPE range of instruments.

FISCHER INSTRUMENTATION (GB) LTD
Arnhem Road, Bone Lane Industrial Estate Newbury/Berks. England.
Telephone Newbury (0635) 45191 Telex: 849111 FIBRIT G

Fischer®

THE WAY TO BETTER CIRCUITS

Norplex® laminates by UOP

G10 FR

NP424

G30

G10 M/L

COPPER-CLAD LAMINATES

We offer a wide selection of epoxy-fiberglass laminates, the most widely used substrates for printed circuitry.

THIN CONDUCTOR FOILS

Base laminates are now available with copper foil in thicknesses of 5 microns and 9 microns. Thin clad laminates offer many of the advantages of additive systems, while using conventional plating and processing procedures.

HIGH-PERFORMANCE LAMINATES

Printed circuitry for high-temperature applications has been made possible by the development of a number of unique systems. Polyimide resin contributed a major advancement to printed circuits by its temperature resistance, superior processability and improved reliability.

MULTILAYER BASE MATERIALS

Norplex has long recognized the need for custom-designed core laminates and bonding sheets in multilayer applications. We have the manufacturing versatility to build core laminates with those properties most important to the application, such as dimensional stability, dielectric constant, and coefficient of thermal expansion.

For more information contact

Norplex UK Division
UOP Limited
Johnson House · Browells Lane · Feltham, Middlesex, TW13 7EQ,
Telephone 01-890 1269 · Telex 935479 (JSELON G)

FRY
A Cookson Group Company

The World's leader in Research, manufacture and supply of

SOLDERS, FLUXES, SOLDER CREAMS
for the Electronics Industry

TECHNICAL SERVICE

A complete technical advice service is available to users of soft solders. With their background of research and development in the Industry, Fry's Metals continue to be recognized as the authority for all matters relating to the soldering processes for Electronics and Micro Electronics production.

FRY'S METALS LTD
Tandem Works, Merton Abbey, London SW19 2PD.
Telephone 01-648 7020. Telex 265732.

Branches at GLASGOW, KIDDERMINSTER, LEEDS, ROCHDALE, DUBLIN, ALTOONA (U.S.A.)

S 15

BEPI MATRIX SYSTEMS

Dedicated to reducing your time to market.

BEPI Matrix Systems offers two automatic discrete wiring technologies for circuit boards – SOLDER-WRAP for prototypes, and MULTIWIRE for production applications and high speed switching.

SOLDER-WRAP is unique and intended specifically for circuit-proving prototypes. It cuts the time from schematic to finished board to only seven days, eliminates CAD costs and is easily modified.

MULTIWIRE is for production applications and high speed switching. It cuts the time from circuit diagram to dedicated circuit to as little as four weeks from documentation package and provides manufacturing tools, thus reducing expensive CAD charges.

For more information about these discrete wiring technologies, contact:

BEPI MATRIX SYSTEMS

Galashiels, Scotland TD1 1PR. Telephone: Galashiels (0896) 4321. Telex: 727857 MATRIX G.

A Division of BEPI Circuits *A member of the Cambridge Electronic Industries Group*

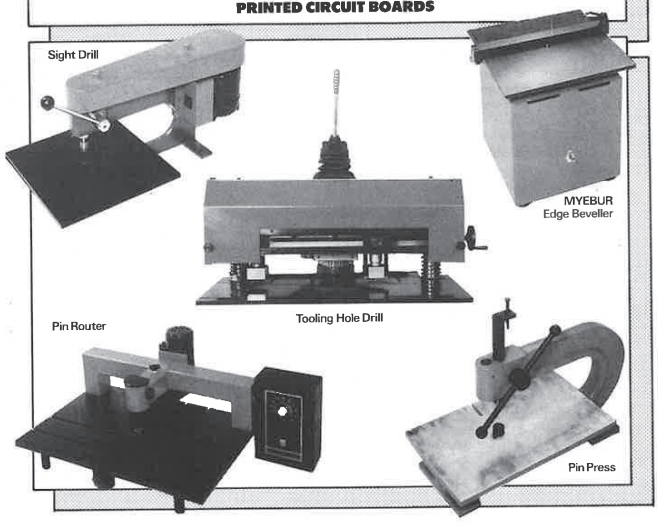
Multiwire is a Registered Trade Mark of Kollmorgan Corporation
Solder-Wrap is a Registered Trade Mark of BEPI Circuits

MICROMAT

Market leader for prototype and small batch production.



MICROPROCESSOR CONTROLLED DRILLING AND ROUTING SYSTEMS FOR PRINTED CIRCUIT BOARDS



DORNIVER LIMITED

Sanders Lodge Industrial Estate, Wellingborough Road, Rushden, Northants NN10 9BQ, England.
Telephone: Rushden (0933) 313093 Telex: 317180 MICMAT G

Gold Plating Technology

By F. H. Reid and W. Goldie

ISBN 901150 02 9

SOME WORLDWIDE OPINIONS

"There does not appear to be any aspect of cleaning techniques, plating solutions, methods of deposition, physical properties, deposit evaluation, and decorative and industrial applications of gold deposits that has been overlooked in this extremely worthwhile addition to the technical literature on gold plating. To paraphrase the title of a recent best seller, it seems that an appropriate alternative title might well be 'Everything You Wanted to Know About Gold Plating but Were Afraid to Ask'. The Editors as well as the 24 individual contributors are, without exception, to be commended for their comprehensive treatment of the multitude of subjects contained within the covers of this book.

It is to the credit of all those individuals involved that there is now available a book on gold which is so complete in its coverage of the subject matter that it should now be possible to find a needed answer concerning gold in a single volume, thereby saving all of us countless hours, sometimes fruitless ones, of searching through seemingly endless journals and periodicals for answers to our questions ... There is little doubt that this book will find wide acceptance and be referred to frequently by anyone who is interested or involved in the deposition, use, or evaluation of gold deposits".

Plating and Surface Finishing (USA)

"The contributors' credits for this, the first comprehensive volume on gold plating, reads like a 'Who's Who' of precious metal deposition ... Anyone who has had to search through the literature to find data on solderability, for example, a very important aspect connected with gold plating, will appreciate the convenience of having it available in one volume where, if what is included does not meet his total need for information, at least directs him to the sources of more. This alone makes the book a very worthwhile investment. And, the ideas which present themselves in the chapters on applications, could suggest further uses offering the possibility of significant economies. When dealing with a metal as costly as gold these days, it may take only a little pointer to open the way to save thousands of dollars".

Metal Finishing (USA)

"This book contains a wealth of data and guidelines not available elsewhere in a single source. It can be highly recommended for our reading audience".

IEEE (USA)

"... embraces the work of 24 world authorities in the field of Gold Plating, and must rank as a unique presentation. Unlike many major technical works, the subject is treated in a manner which is both interesting and informative ..."

Electronics Weekly (UK)

"Twenty-four of the World's leading experts have been persuaded to join together to produce what must surely be the definitive work on this subject".

Design Engineering (UK)

"This comprehensive work—greater in scope and depth than any attemps so far made to survey the electroplating of any other metal—will assuredly be of outstanding value for a long time to come."

Gold Bulletin (South Africa)

"This is a superb book in which a wealth of information written by leading specialists has been gathered into a very lucid form. It is a book which encompasses all aspects of gold plating technology from the ordinary to the exotic ... It is extremely difficult to fault this publication in any manner."

Metals (Australia)

"The volume, which has been contributed by leading specialists in the various branches of gold plating technology in the United States, Europe and the United Kingdom, is a block-buster in the field.

The book is an invaluable reference guide to all those even remotely concerned with this important subject. Undoubtedly, this book will be a standard reference work for many years to come."

Science Reporter (India)

Electrochemical Publications Ltd.
8 Barns Street, Ayr KA7 1XA, Scotland

Printed Circuit Troubleshooting

by H. R. Shemilt

ISBN 901150 03 7

SOME WORLDWIDE OPINIONS:

"Throughout, the author is concerned with practical solutions to common problems. Subject treatment is up-to-date and accurate. The layout of the book, with large pages, many drawings and photographs, and bold headings, is excellent" ... "a work on troubleshooting is unique in the field, and this volume goes far to fill a void. It is recommended for production technologists in the printed circuit field".

Plating and Surface Finishing (USA)

"It is a very convenient book, a ready troubleshooting reference and an indispensable tool for anyone involved in the fabrication of PC boards and in the research laboratories".

Electrochemical Progress (USA)

"This book is an excellent guidebook and will be indispensable to technical and research workers in this field".

Journal of the Metal Finishing Society (Japan)

"Since it covers all aspects of PC manufacture from artwork to testing, the book will be of use to designers, production engineers, chemists, inspectors, process workers and students, all of whom are likely to regard it as an investment".

Pulse (South Africa)

"Printed circuit technicians regard this work as a 'must', as it is a particularly useful book which is written in a crisp and condensed style. It gets to the root of the matter and distinguishes between the important and the trivial."

Product Finishing (UK)

"Half of each chapter describes the how and why and the other half gives a problem-cause-remedy table. By such listing of all the things which can go wrong, the book gives a better picture of the whole technology than one would gain from a conventional, straight-forward account. For this reason it could be invaluable training and reference material for apprentices, process workers, inspectors, design staff, chemists and metallurgists".

The Engineer (UK)

"Each chapter of this unique volume details with a specific stage of the printed-circuit process, presenting a comprehensive review of techniques and concluding with a tabular fault-finding guide which lists faults, causes and remedies. Although the scope ranges from artwork to testing specifications, particularly valuable sections cover plating and multilayer boards. Mr Shemilt's compendium will prove an excellent reference for technologists in industry involved in setting-up and running a printed circuit facility".

ASLIB Book List (UK)

"... a first class manual for the practical engineer".

Design Engineering (UK)

Electrochemical Publications Ltd.
8 Barns Street, Ayr KA7 1XA, Scotland

HANDBOOK OF PRINTED CIRCUIT DESIGN, MANUFACTURE, COMPONENTS & ASSEMBLY

by G. Leonida

ISBN 0 901150 09 6

Pages—569+xvii; Tables—40; Figures—394; Size—23× 15 cm.

Over the past decade, electronic packaging has become an accepted discipline in most electronic companies. Since the large majority of modern electronic assemblies are supported by either a rigid or flexible circuit, the correct assembling and soldering techniques are of paramount importance if low costs and reliability are to be achieved.

Despite the importance of PCB assembly, it is somewhat surprising that there is a noticeable dearth of published literature on the subject and although a number of text books are now available on PCB manufacture, little, if any, space has been devoted in these publications to assembly. Consequently, the Author's primary aim was to rectify this situation with the result that this is the first reference book to treat the subject of assembly in depth. Furthermore, as all other aspects of printed circuits such as Design, Components, Quality Control and in particular, Manufacture—the latter being a book in itself—are covered in considerable detail, this Handbook must rank as one of the most comprehensive and important reference texts ever to be published in this field.

Some Worldwide Opinions:

'For all those engaged in the PCB industry, whether as a manufacturer or user, this book is highly recommended and will undoubtedly become a standard reference work on this subject.' *Electronic Production (UK)*

'Unlike many books on printed circuits, the book is full of facts, all of which are based on obvious practical experience' . . . 'This is, undoubtedly, the best book in the reviewer's opinion yet published. It has all the necessary practical information in one book and is therefore money well spent.' *Microelectronics and Reliability (UK)*

'The overall impression is that the technological chapters of this book are excellent' . . . 'the author has achieved a self-contained and "interconnected" account of printed circuit board design, manufacture and component assembly which will surely make it a standard text for designers, production engineers, chemists and metallurgists who are engaged in this fast-growing industry.' *The Metallurgist and Materials Technologist (UK)*

May be ordered direct from

Electrochemical Publications Ltd.
8 Barns Street, Ayr KA7 1XA, Scotland