

**AN INTRODUCTION TO
PRINTED CIRCUIT BOARD
TECHNOLOGY**

by
J. A. SCARLETT

**ELECTROCHEMICAL PUBLICATIONS LIMITED
1984**

TK 7872.P65
8286
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
McCorquodale (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.

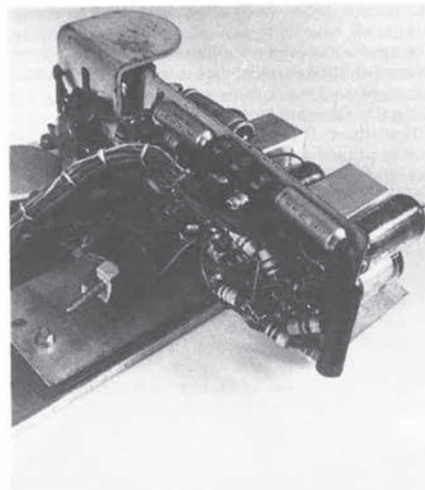


Fig. 1.1 Radio receiver on aluminium chassis with components suspended in the sub-chassis wiring.

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 pre-cut 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

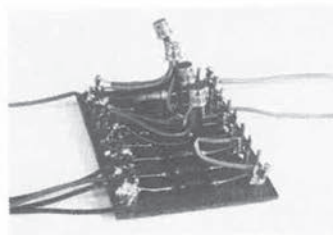


Fig. 1.2 Tagstrip carrying resistors and transistors.

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

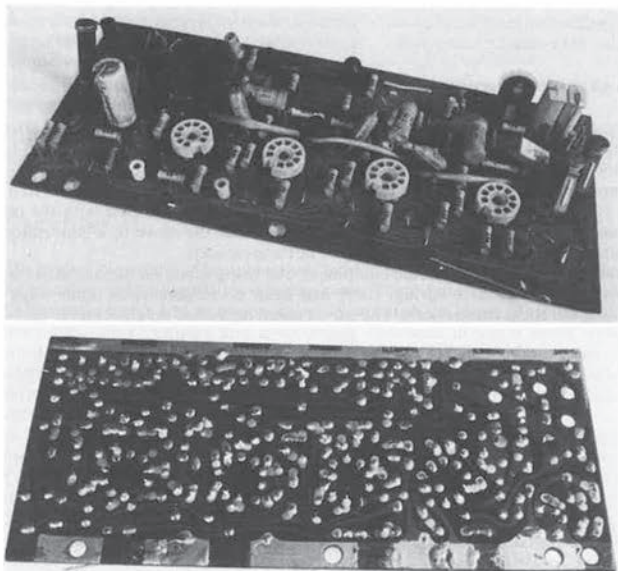


Fig. 1.3 Single-sided board with miniature valve sockets. Note the use of wire links.

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.

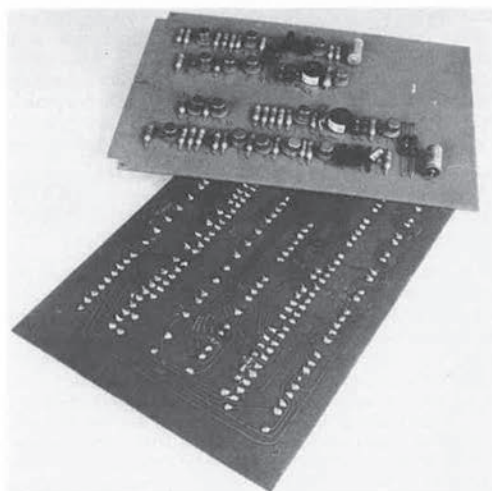


Fig. 1.4 Single-sided board which serves mainly as a component carrier with few actual interconnections.

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 Hazeltine in the USA, of the seeded, plated-

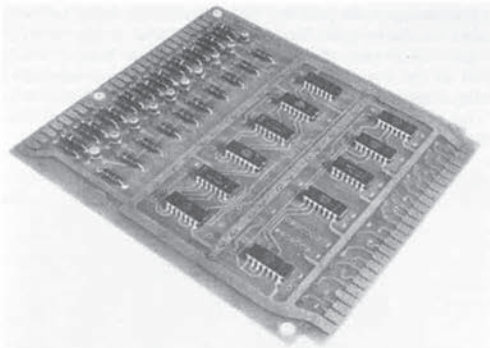


Fig. 1.5 Through-plated hole board of the early 1960s carrying integrated circuits and discrete components.

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

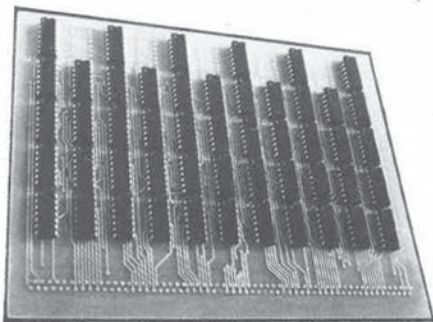


Fig. 1.6 Late 1960s' double-sided board with extensive use of via holes to achieve high packing density.

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.

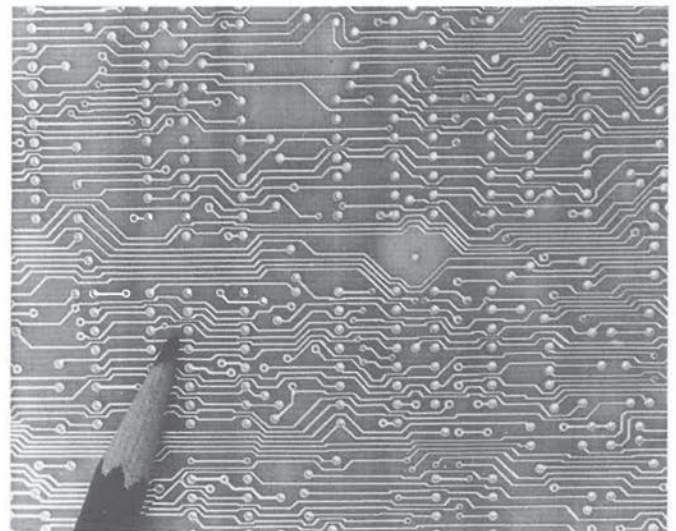


Fig. 1.7 High density tracking on double-sided board.

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.

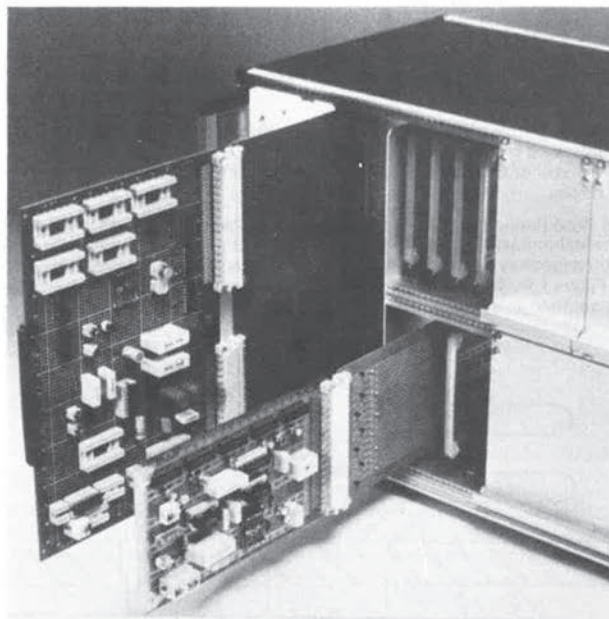


Fig. 1.8 Half and full height boards in a cabinet, plugging into a back wiring frame. Two cards are mounted on extender boards for test access.

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

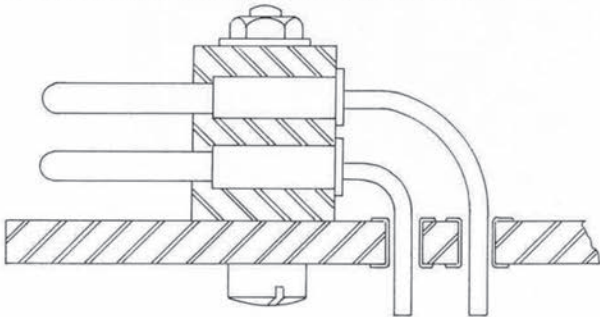


Fig. 1.9 Section through typical PCB connector.

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

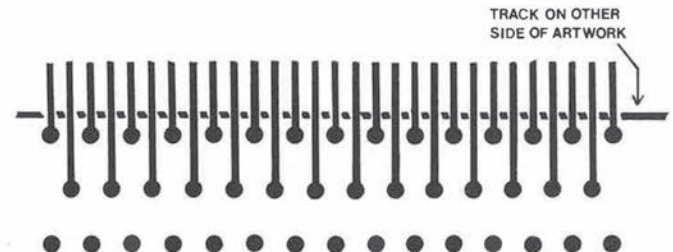


Fig. 1.10 When only one track can pass between pairs of pins on a double-sided board a third row of pins may be unusable.

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.

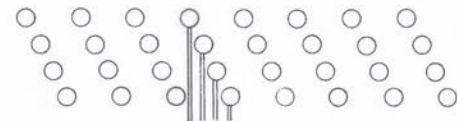


Fig. 1.11b Four row ribbon cable connector pattern.

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

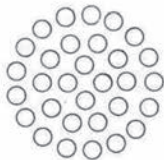


Fig. 1.12 Circular pin pattern of bulkhead connector.

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.

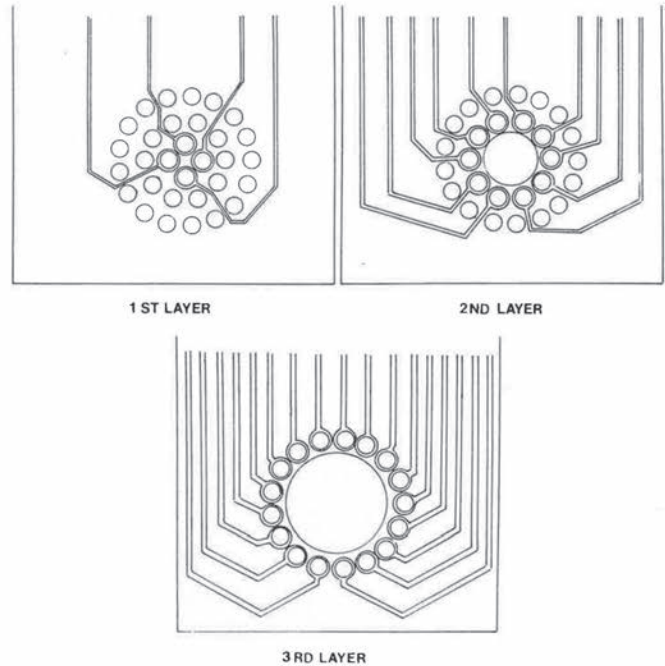


Fig. 1.13 Three layers of flexible circuit for connection to circular bulkhead connector.

1.4.2 Edge Connectors

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 tolerated 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.

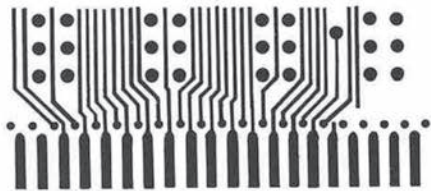


Fig. 1.14 Edge connector finger pitch should be chosen to match the track pitch on the board.

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

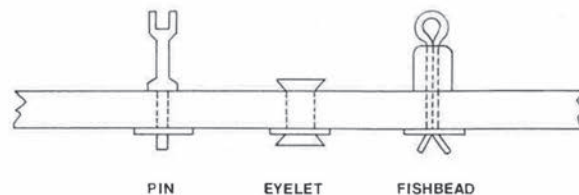


Fig. 1.15 Terminations for discrete wires.

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.

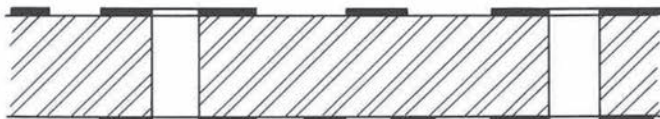


Fig. 1.17 Double-sided, non through-plated board.

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.

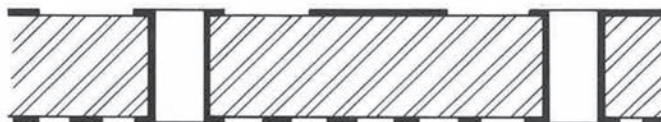


Fig. 1.19 Double-sided PTH board.

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.

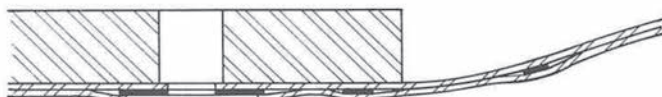


Fig. 1.21 Rigidised single-sided flexible circuit.

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.

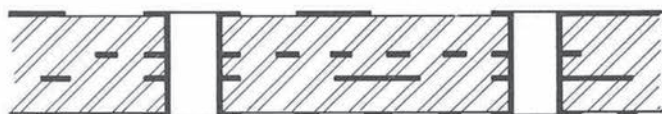


Fig. 1.22 Four layer multilayer board.

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.

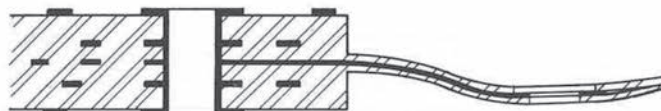


Fig. 1.23 Five layer flexi-rigid with single tail.

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.

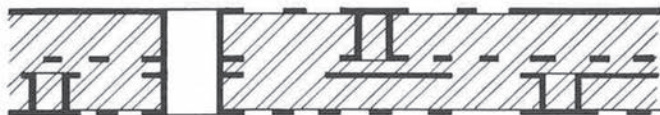


Fig. 1.25 Double-double board with no internal planes.

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.

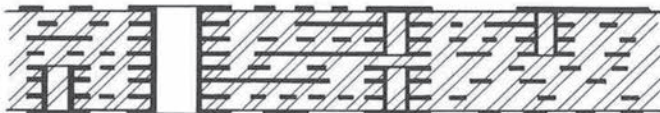


Fig. 1.26 Double multilayer board.

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.

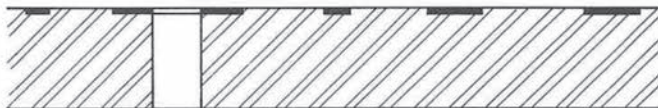


Fig. 1.29 Simple flush bonded board.

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.

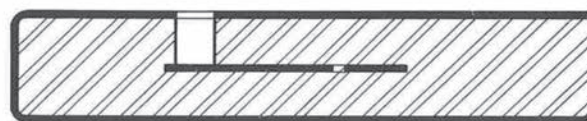


Fig. 1.30 Section through microwave bonded board.

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

$$R = \rho l / wt$$

A commonly used derived term is sheet resistance, which is

$$R_s = \rho / t$$

$$R = R_s l / w$$

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

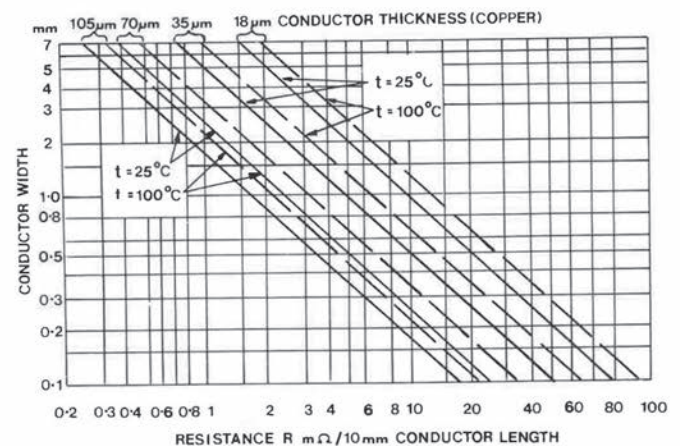


Fig. 2.1 Resistance of copper tracks.

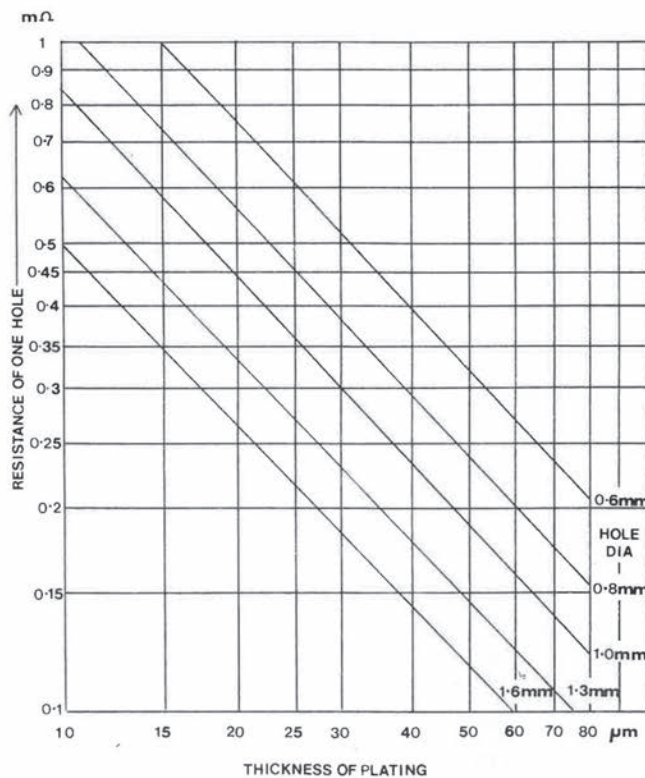


Fig. 2.2 Resistance of through-plated holes in 1.6 mm laminate.

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 mm² 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.

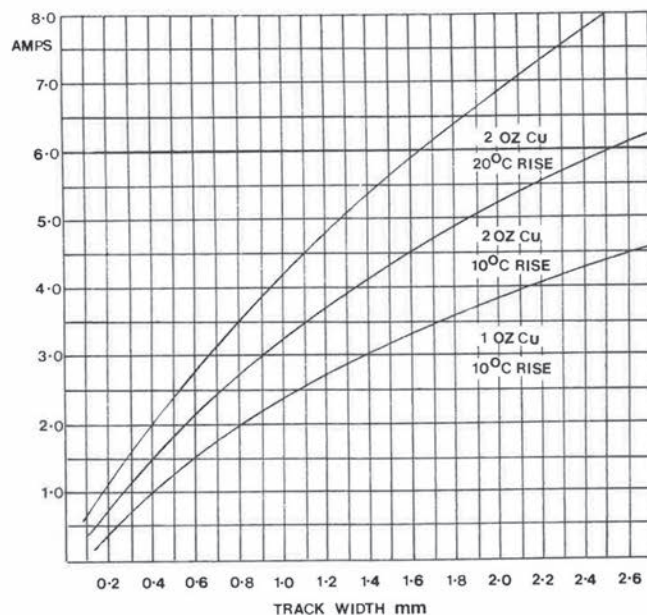


Fig. 2.3 Safe currents in tracks on PCBs.

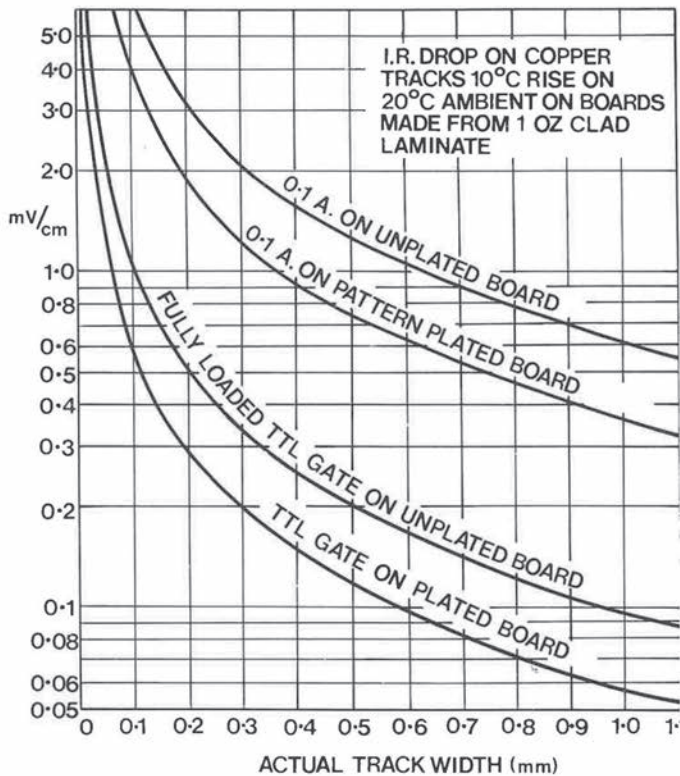


Fig. 2.4 Voltage drop on tracks on PCBs.

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.

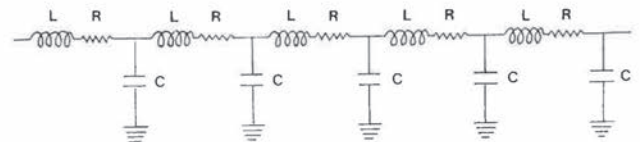


Fig. 2.5 Electrical characteristics of a track on a PCB.

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

laminates, 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.

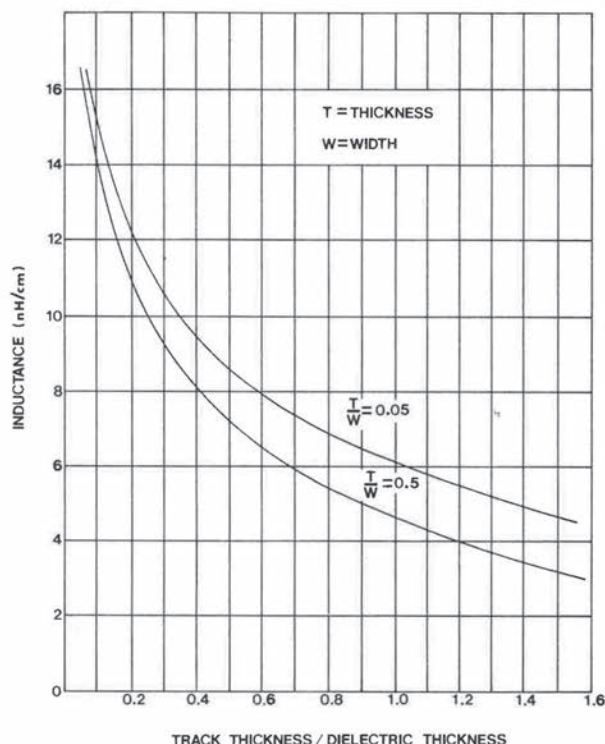


Fig. 2.6 Inductance of tracks on PCBs.

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

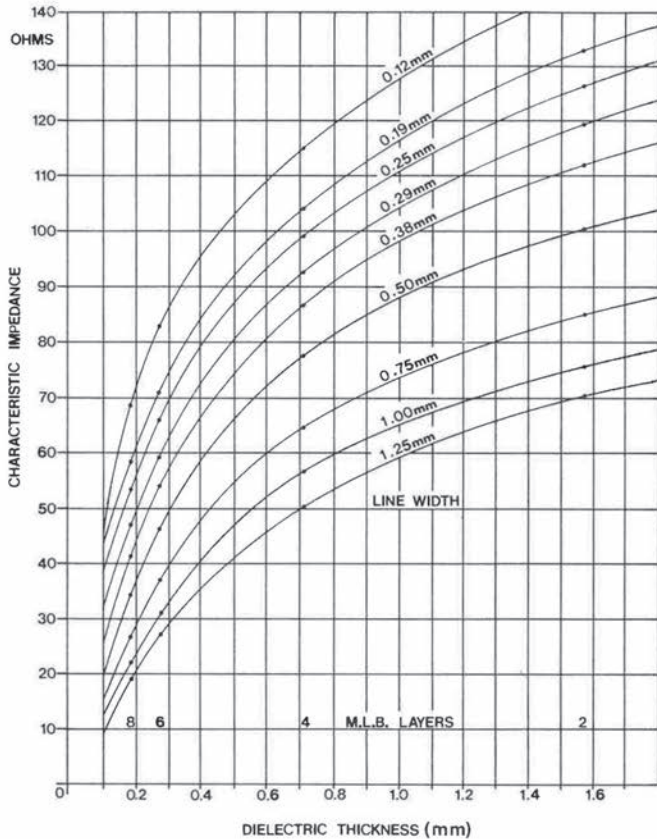


Fig. 2.7 Characteristic impedance of tracks on PCBs.

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

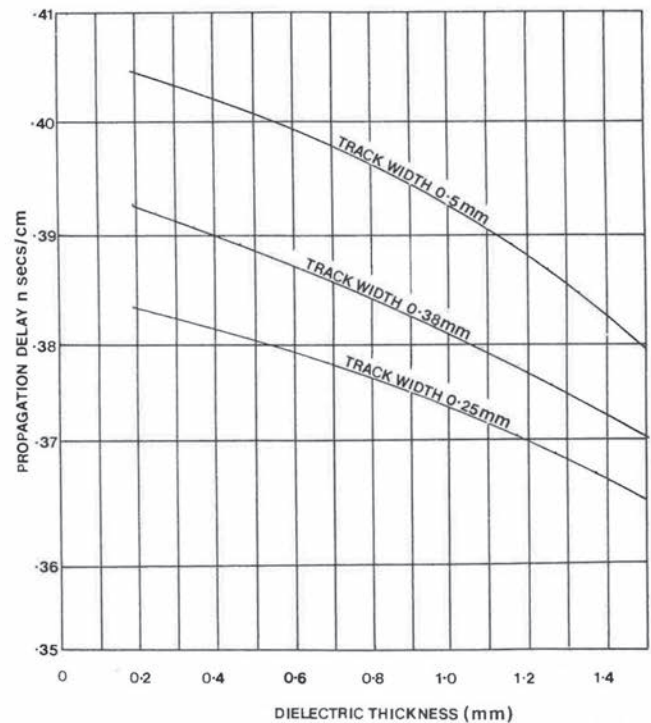


Fig. 2.8 Propagation delay of tracks on PCBs.

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

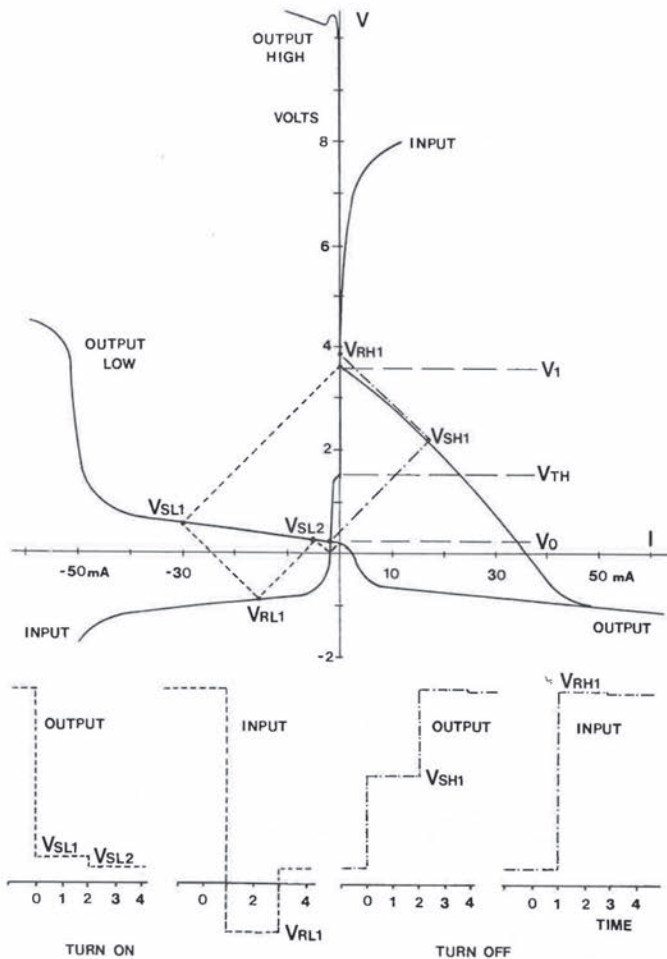


Fig. 2.9 Determination of line reflections on a typical track interconnecting TTL gates.

(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.

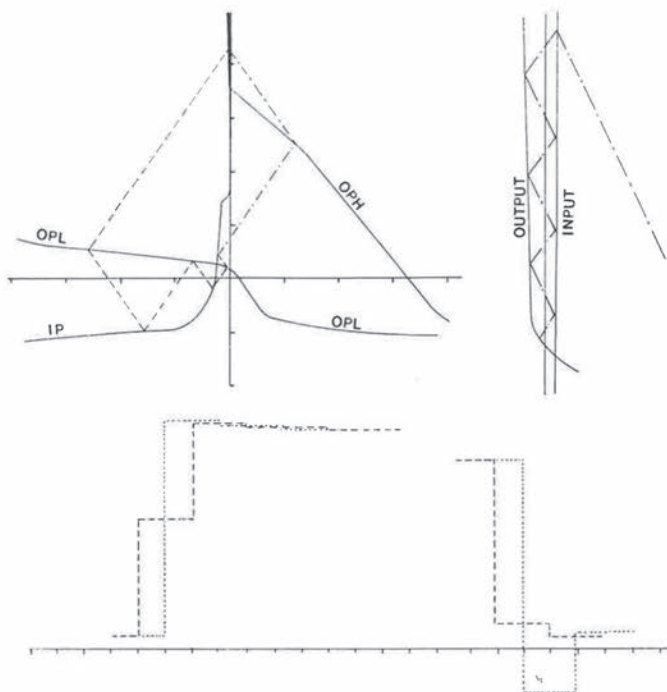


Fig. 2.10 Reflections on a high impedance line.

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

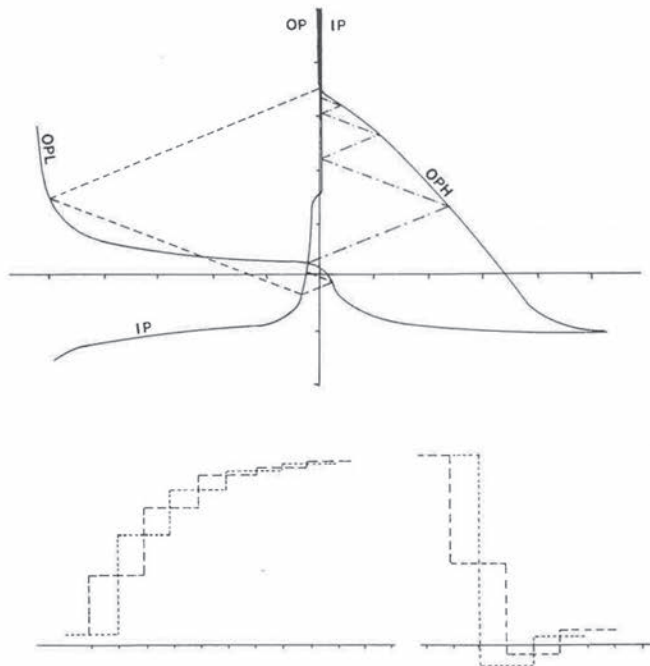


Fig. 2.11 Reflections on a low impedance line.

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).

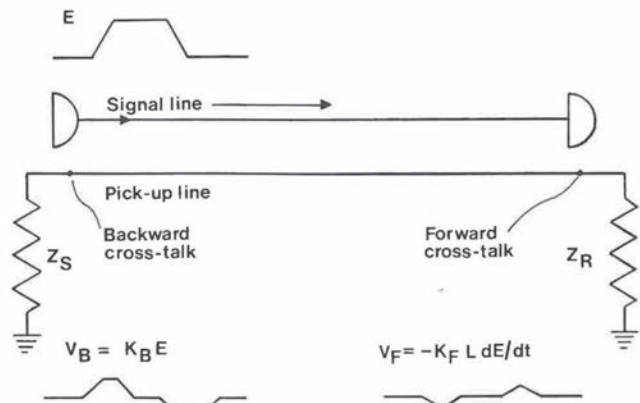


Fig. 2.12 Cross-talk definitions and waveforms.

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.

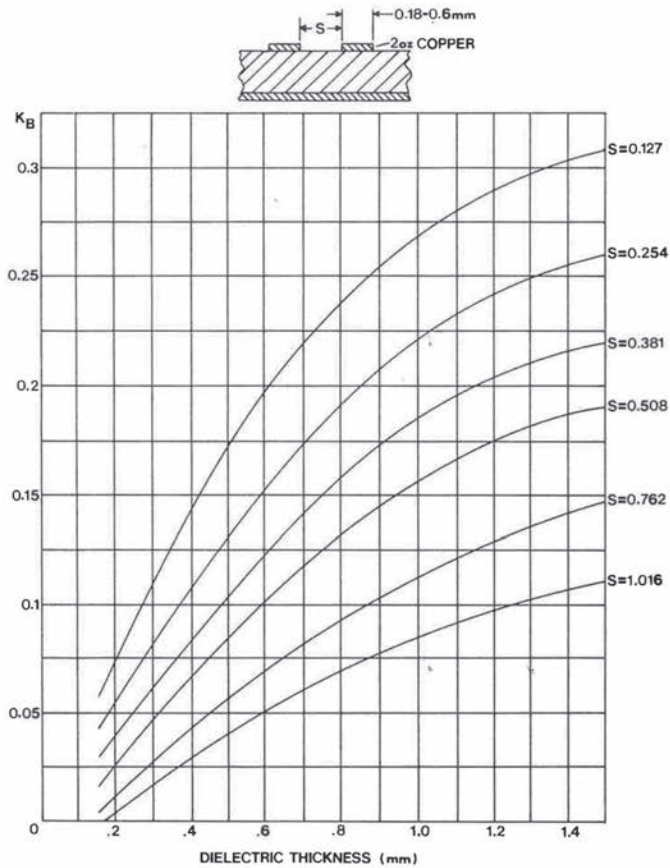


Fig. 2.13 Back cross-talk coefficients.

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

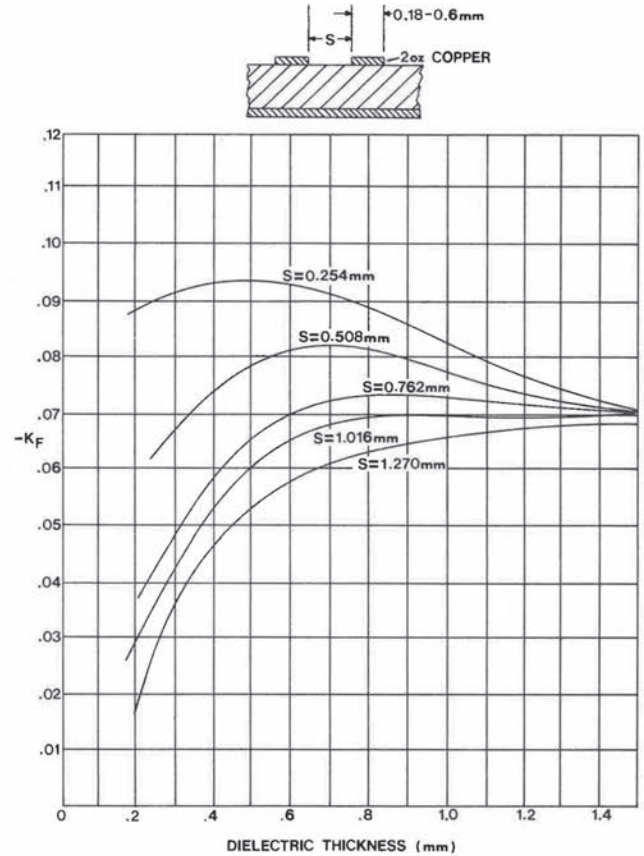


Fig. 2.14 Forward cross-talk coefficients.

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

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.

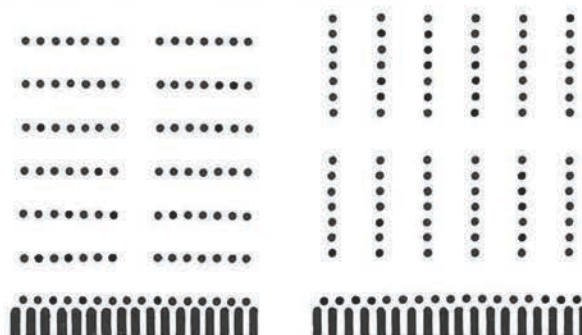


Fig. 3.1 DIPs side on and end on to board connector.

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

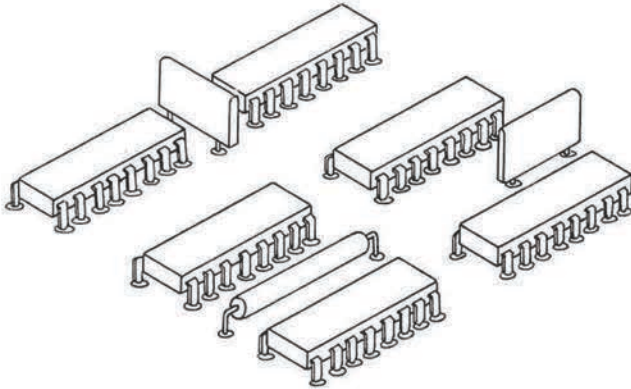


Fig. 3.2 Three ways of mounting decoupling capacitors in an array of DIPs.

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

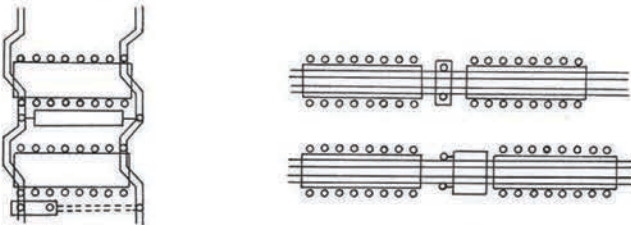


Fig. 3.3 Two ways of routing power tracks to DIPs.

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.

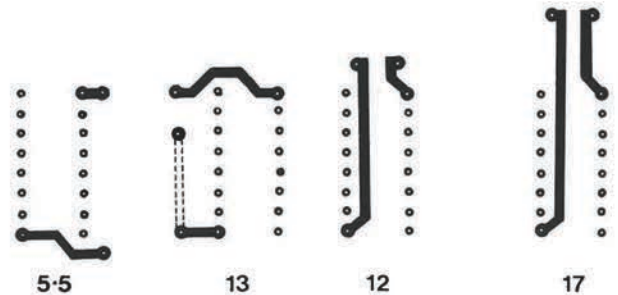


Fig. 3.4 Track length from decoupling capacitors to package pins (in units of 2.54 mm).

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

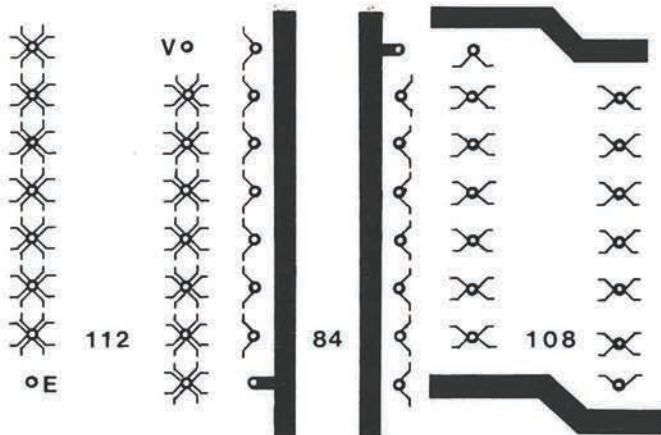


Fig. 3.5 Trackability factors of standard 16 pin DIP.

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.

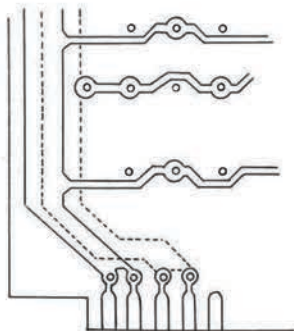


Fig. 3.6 Corner of board viewed from solder side showing power and earth bus bars.

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.

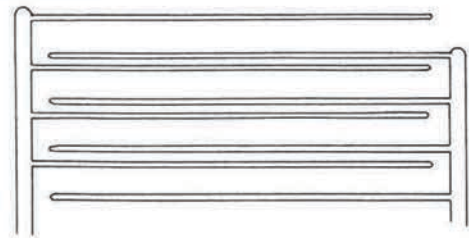


Fig. 3.7 Intermeshed E distribution system.

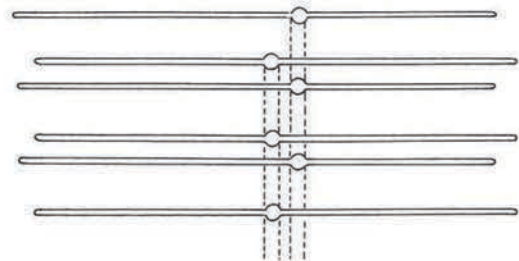


Fig. 3.8 Centre fed distribution system.

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

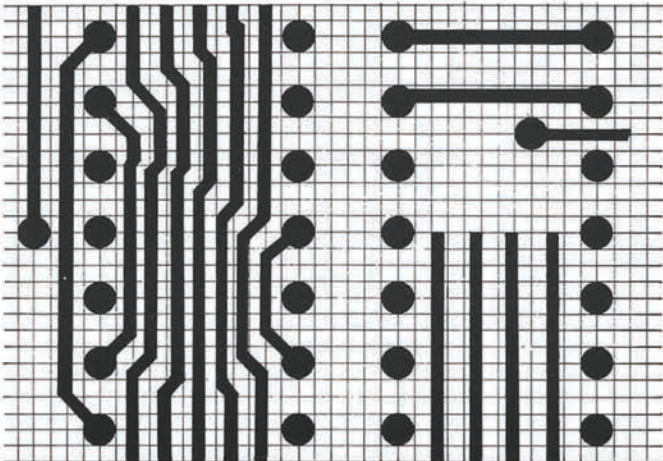


Fig. 3.9 Advantages of the use of off-grid tracks.

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.

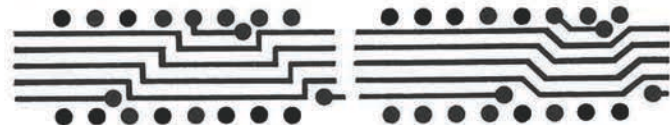


Fig. 3.10 Use of angled bends to improve track density.

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.

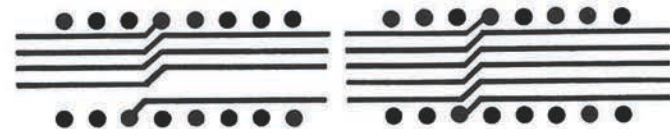


Fig. 3.11 Radial and tangential tracks to pads.

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).

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

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 platens 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 Microelectronics*, 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).

194

References

195

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 catalyst 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 Welshe, 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 Laminates)
- 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
- Hydrosquegee, 144
- Ice, 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
- Retractable, 129
- Ribbon cable, 8, 11, 129
- Rigidflex 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

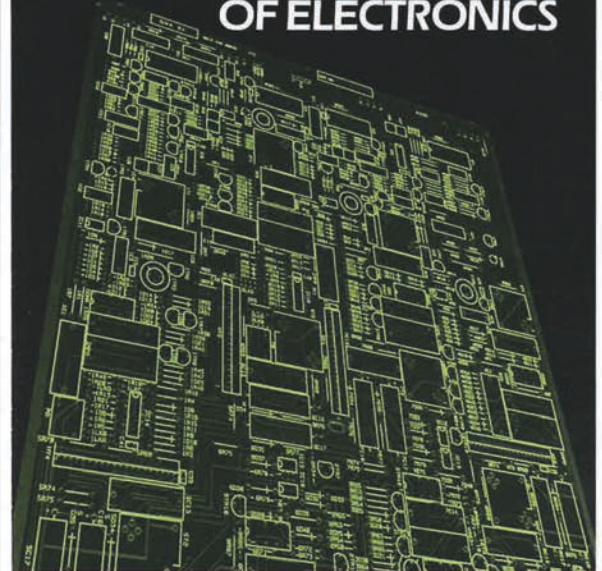
 Undercutting, 86, 100, 119
 Ultra-violet light, 83, 84, 86, 120, 124, 127

 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

COATES IN THE WORLD OF ELECTRONICS



A major supplier of Thermal and U.V. drying products to the Electronics Industry:

- * Etch Resists * Reducers and Cleaners
- * Solder Resists * Plating Resists
- * Preservative Lacquers * 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.

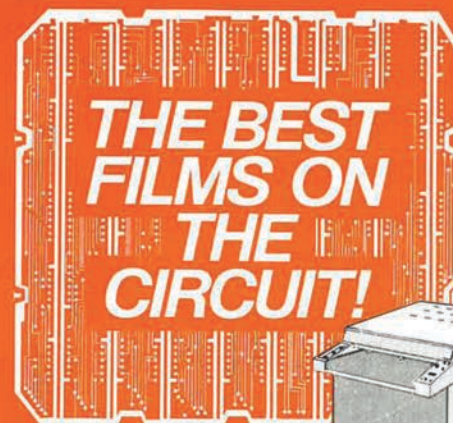
If you require a machine that is
not listed for a conveyorised
bare board process, then we
can almost certainly offer a
suitable machine for your exact
requirements.



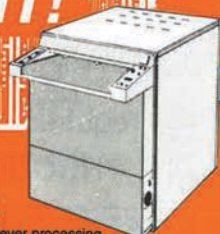
NUBAL

Tribune Drive,
Trinity Trading Estate,
Sittingbourne, Kent.

Tel: (0795) 77478
Telex: 965286



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 ELECTROPLATING 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



THE WAY TO BETTER CIRCUITS

Norplex[®] laminates by UOP

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)



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

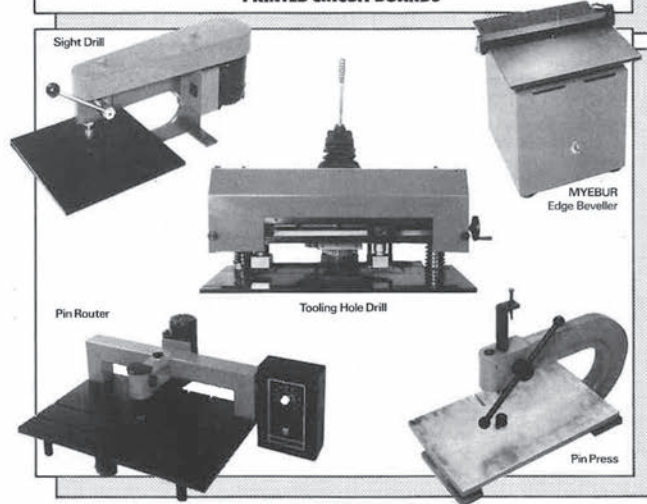
Multiwire is a Registered Trade Mark of Kollmorgen Corporation
Solder-Wrap is a Registered Trade Mark of BEPI Circuits

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 attempts 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