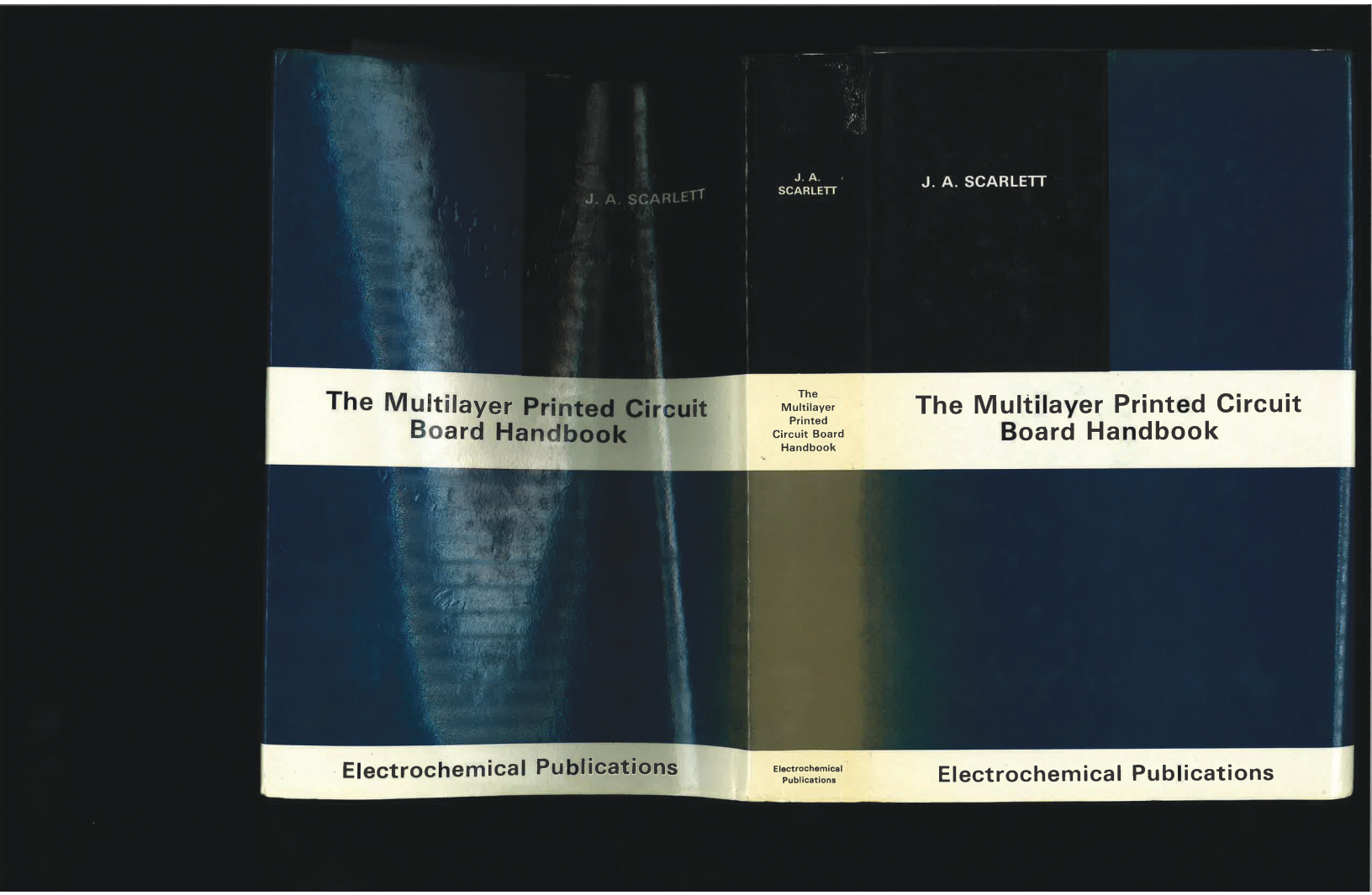

J. A. SCARLETT
The Multilayer Printed Circuit Board Handbook
Electrochemical Publications
J. A. SCARLETT
The Multilayer Printed Circuit Board Handbook
Electrochemical Publications
J. A. SCARLETT
The Multilayer Printed Circuit Board Handbook
Electrochemical Publications

APPLE 1021

THE MULTILAYER PRINTED CIRCUIT BOARD HANDBOOK

Edited by
J. A. SCARLETT

ELECTROCHEMICAL PUBLICATIONS LIMITED
1985

ELECTROCHEMICAL PUBLICATIONS LIMITED
8 Barns Street, Ayr, Scotland

© Electrochemical Publications Ltd. 1985

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

ISBN 0 901150 15 0

Typesetting by Brian Robinson, North Marston, Bucks
Printed and bound in Great Britain by
Anchor Brendon Ltd, Tiptree, Essex

PREFACE

The importance of multilayer printed circuit boards to the electronics industry is at long last becoming more widely recognised, and this internationally multi-authored book is an attempt to offer in one volume authoritative advice on the design, manufacture and use of these vital and often expensive components.

Whilst all chapters of the book are written primarily about multilayer boards, it must be borne in mind that double-sided and non-plated boards are merely multilayer boards reduced to their simplest possible form, and most of the chapters concerned with board manufacture are equally applicable to all simpler types of boards. This book, however, differs from all others so far published on printed circuit boards in general in that, whereas the information offered in other books may sometimes cover multilayer boards, it is usually aimed at the simpler types of boards or at general problems of board assembly, and often does not emphasise the particular requirements of the more sophisticated types. Throughout, this work offers information on the most complex types, and it confines its terms of reference to the design and manufacture of multilayer boards, mentioning only essential differences between multilayer and non-plated boards in subjects such as assembly, which are well covered elsewhere in the literature.

All the individual authors have been selected as being widely recognised as leading authorities in their particular fields, and many are familiar as speakers at international conferences and symposia. Some of the authors express slightly differing views of aspects of multilayer technology, and some chapters may duplicate areas covered in other chapters. In places, there may be what appear to be disagreements or contradictions. These are not accidental, nor do they indicate any errors by any of the authors. They are different views of the same subject, and they serve to highlight the fact that good design and good manufacture may be in direct conflict with requirements for the best electrical, mechanical or thermal performance. Every equipment designer must set his own standards and adopt those compromises which best suit the requirements of the particular equipment being designed.

Editing of the authors' texts has been confined to the minimum necessary to present a reasonably unified overall style. Generally, dimensions have been left in the units chosen by the authors: some metric, some Imperial, and some dual. In certain cases, where dual dimensions are quoted, the metric figures are not exact conversions of the Imperial values, but are those approximations which a designer thinking in one of the systems would naturally use.

In offering this book to the industry, thanks must be expressed to all the authors who have collaborated in its compilation, and who have dealt so patiently with the many queries which have arisen. I hope that the efforts of all concerned will prove to be of value to the electronics industry as a whole and that this book will help to answer at least some of the many queries which arise over multilayer boards.

June, 1984
John A. Scarlett, F.Inst.C.T.
Galashiels

CONTENTS

PREFACE v

CHAPTER ONE
Introduction
J. A. Scarlett

1.1 General 1
1.1.1 Printed Circuit Boards 1
1.1.2 History 2
1.1.3 The Electrical Importance of Multilayer Boards 3
1.1.4 Costs and Applications 3
1.1.4.1 Costs of Boards 3
1.1.4.2 Costs of Alternatives to Boards 5
1.1.4.3 Choice of Board Type 6
1.1.4.4 Thermal Problems 7
1.1.4.5 System Architecture and Board Type 8
1.1.4.6 Probable Need for Multilayers 9

1.2 Types of Multilayer Board 11
1.2.1 Access Hole Boards 11
1.2.1.1 Access Hole Boards with Internal Connections 12
1.2.1.2 Disadvantages of Access Hole Boards 14
1.2.1.3 Solder Buttons 14
1.2.2 Plated Pillar Boards 15
1.2.3 Staked Boards 16
1.2.4 Bonded Through-hole Plated Boards 16
1.2.5 Buried Via-hole Boards 17
1.2.6 Sequential Buried Via-hole Boards 18
1.2.6.1 Two-level Buried Via Holes 18
1.2.6.2 Multiple Level Buried Via Holes 18
1.2.7 Double-double Boards 19
1.2.8 Double-multilayer Boards 19
1.2.9 Flexi-rigid Boards 20
1.2.10 Other Forms of Multilayer Interconnects 20
1.2.10.1 Etched and Welded Interconnects 20
1.2.10.2 Three Dimensional Planar Co-axial Interconnects 21
1.2.10.3 Etched and Filled Boards 22
1.2.10.4 Moulded Boards 22
1.2.10.5 Multiwire 23

1.3 Production of Through-hole Plated Multilayers 23
1.3.1 General 23
1.3.2 Preparation of Inner Layers 24
1.3.2.1 Film Work 24
1.3.2.1.1 Incoming Photo Tools 24
1.3.2.1.2 Film Types and Labels 25
1.3.2.1.3 Step and Repeat 27
1.3.2.1.4 Borders 28
1.3.2.1.5 Environmental Control 29

1.3.2.1.6 Film Punching 30
1.3.2.2 Planning 33
1.3.2.3 Tape Making 33
1.3.2.4 Laminate Preparation 34
1.3.2.5 Inner Layer Printing and Etching 34
1.3.2.6 Preparation for Bonding 34
1.3.3 Bonding 34
1.3.4 Drilling 38
1.3.5 Smear Removal 38
1.3.6 Through-hole Plating and Outer Layers 38
1.3.7 Finishing 41

1.4 Production Variants 41
1.4.1 Pillar Plated Boards 41
1.4.2 Buried Via-hole Boards 42
1.4.2.1 Inner Layer Plating 42
1.4.2.2 Lay Up and Bonding 44
1.4.2.3 Pre-filling of Buried Holes 44
1.4.3 Sequential Buried Via Holes 45
1.4.4 Double Boards 45
1.4.5 Flexi-rigids 45
1.4.6 Mass Lamination 46
1.4.7 Semi-additive Multilayers 46
1.4.8 Bonding of Heat Ladders 46

1.5 Build and Thickness 48
1.5.1 Prepreg Layers 48
1.5.2 Board Thickness 49
1.5.3 Practical Builds 50
1.5.3.1 Four Layer Boards 50
1.5.3.2 Higher Build Boards 50
1.5.3.3 Buried Via-hole and Flexi-rigid Builds 52

1.6 Handling Thin Laminates 52
1.6.1 Storage and Transport 53
1.6.2 Cutting 53
1.6.3 Picking Up Thin Laminates 53
1.6.4 Conveyorised Machines 54
1.6.5 Plating Fixtures 54

1.7 Reliability of Multilayer Boards 54
1.7.1 Delamination 55
1.7.2 Outgassing 55
1.7.3 Short Circuits 56
1.7.3.1 Design and Manufacturing Faults 56
1.7.3.2 Growth of Short Circuits 56
1.7.4 Carbonisation 58
1.7.5 Warping 58
1.7.6 Damage to Through-plated Holes 58
1.7.6.1 Mechanical Damage 58
1.7.6.2 Z-Plane Expansion 59

CHAPTER TWO
Laminates and Prepregs
J. S. Sallo

2.1 Introduction 61
2.1.1 Use of Epoxy Materials in Laminate and Prepreg Manufacture 61

2.2 Epoxy Materials 62

2.2.1 Bisphenol-**A Epi**chlor**ohy**drin 62
2.2.2 Addition of **Epo**xy **Novo**lacs 62
2.2.3 Control of Flow 62
2.2.4 Glass Transition Temperature 63

2.3 Non-epoxy Materials 63
2.3.1 Polyimide 63
2.3.2 B-Triazine 63
2.3.3 **Modi**fied **Epox**ies 64
2.3.4 **Prob**lems **with** Brominated Materials 64

2.4 Glass Cloth 64

2.5 Prepreg Manufacture 64
2.5.1 Treater Types 65
2.5.1.1 Mechanical Techniques 65
2.5.1.2 Thermal Techniques 65
2.5.1.3 Operating Conditions 65
2.5.2 Process Control 66
2.5.3 Prepreg Testing 66
2.5.3.1 Gel Time Test 66
2.5.3.2 Flow Test 67
2.5.3.3 Scaled Flow Testing 67
2.5.3.4 Resin Content Test 68
2.5.3.5 Volatile Content Test 68
2.5.3.6 Dicyaniamid Test 68
2.5.3.7 Test for Ageing 69

2.6 Laminate Manufacture 69
2.6.1 Influential Parameters and Their Measurement 69
2.6.1.1 Dimensional Stability 70
2.6.1.2 Copper Adhesion 71
2.6.1.3 Dielectric Constant 71
2.6.1.4 Thickness 71
2.6.1.5 Solvent Resistance 72
2.6.1.6 Baking 72
2.6.2 Manufacturing Problems 72
2.6.2.1 Smear Removal 72
2.6.2.2 Drilling 73
2.6.2.3 Degree of Cure 73
2.6.3 Solutions to Laminate Problems 73
2.6.3.1 General 74
2.6.3.2 Use of Thin Copper Foil 74
2.6.4 Methods of Promoting Adhesion 74
2.6.4.1 Black Oxide 75
2.6.4.2 Double Treatment of Copper

CHAPTER THREE
The Manufacture of Woven Glass Fabrics
D. J. Vaughan and C. G. Herschdorfer

3.1 Introduction 76

3.2 Glass Fibre Production 76
3.2.1 Glass Composition 79
3.2.2 Yarn 80
3.2.3 Warping 83
3.2.4 Slashing 84
3.2.5 Entering 84
3.2.6 Quilling 84
3.2.7 Weaving 86

3.2.8 Heat Cleaning 86

3.3 Finishing of Glass Fabric 88

3.4 The Coupling Agent-resin Interface 90

3.5 The Application of Finished Glass Fabric to Multilayer Prepreg and Circuit Laminates 91

3.6 Conclusion 93

CHAPTER FOUR
Photoresists and Photomechanical Printing 95
D. Simpkins

4.1 History and Philosophy 96

4.2 The Materials 97
4.2.1 Components 98
4.2.2 Resins 98
4.2.3 Sensitisers 98
4.2.4 Additives 99
4.2.5 Solvents 99

4.3 Substrate Preparation 99
4.3.1 Cleanliness Testing 100

4.4 Application Methods 100
4.4.1 Whirl Coating 100
4.4.2 Dip Coating 101
4.4.3 Spray Coating 101
4.4.4 Roller Coating 101
4.4.5 Curtain Coating 103

4.5 Drying 104
4.5.1 Air Drying 104
4.5.2 Static Oven Drying 104
4.5.3 Conveyorised Infra-red/Air Ovens 104

4.6 Exposure 105
4.6.1 Fixed Source 105
4.6.2 Travelling Source 106
4.6.3 Exposure Time 106
4.6.3.1 Using a Step Tablet 106
4.6.3.2 Precautions 107

4.7 Development 107
4.7.1 Negative Resists 107
4.7.2 Positive Resists 108
4.7.3 Developing Equipment 108
4.7.4 Inspection and Touch-up 109
4.7.5 Resist Removal 109

4.8 Future Developments 110

CHAPTER FIVE
Dry Film Photoresists
L. R. Wallig

5.1 General 111
5.1.1 Introduction 111

5.1.2 Dry Film Resist Structure 111
5.1.3 How the Dry Film System Functions 114
5.1.4 Comparison with Other Image Transfer Processes 114
5.1.5 Dry Film Resist Cost 115

5.2 Photopolymer Make-up 115
5.2.1 Properties Determined by the Photoresist Composition 116
5.2.2 Classification of Dry Film Resists 117
5.2.3 The Dry Processable Photoresists 117
5.2.3.1 Mechanism 117

5.3 Selection of a Dry Film Resist Image Transfer System 120
5.3.1 Solvent Processable Resists 121
5.3.2 Aqueous Processable Resist System 122

5.4 Dry Film Photoresist Processing 122
5.4.1 General 122

5.5 The PCB Fabrication Process 123
5.5.1 The Substrate 123
5.5.2 Substrate Prelamination Cleaning 123
5.5.3 The Copper Surface 124
5.5.4 Resist Lamination 125
5.5.5 Exposure (Photopolymerisation) 128
5.5.5.1 Exposure Control/Measurement 129
5.5.5.2 Effect of Actinic Intensity 131
5.5.5.3 Spectral Sensitivity of the Resist 132
5.5.5.4 Exposure Technology Trends 133
5.5.6 Development of the Resist Image 133
5.5.6.1 Development Mechanism 133
5.5.6.2 Solvent Development 135
5.5.6.3 Aqueous Development 135
5.5.6.4 Future Trends 136
5.5.7 Resist Removal (Stripping) 137

5.6 Other Dry Film Resist Developments 138
5.6.1 Dry Film Solder Mask 138
5.6.2 Phototape 138
5.6.3 Photo Chemical Machining (Chemical Milling) 138

5.7 Dry Film Resist Processes 139
5.7.1 Subtractive Processes 139
5.7.1.1 Print and Etch 139
5.7.1.2 Print, Plate and Etch 139
5.7.1.3 Print and Etch with Copper Plated-through Holes (Tenting) 140
5.7.2 Additive Processes 140
5.7.2.1 Fully Additive Process 140
5.7.2.2 Semi-additive Process 141

5.8 Troubleshooting Guide 142

CHAPTER SIX
Accurate Screen Printing for Multilayer PCBs
C. Walter

6.1 Introduction 154

6.2 Factors Influencing Printing Quality 155
6.2.1 The Original Filmwork 155
6.2.2 The Screen Frame 156
6.2.3 Mesh Tension 156

6.2.4 The Mesh Itself 158
6.2.5 Photomechanical Stencils 159
6.2.5.1 Direct Stencils 159
6.2.5.2 Indirect Stencils 160
6.2.5.3 Direct/Indirect Stencils 160
6.2.5.4 Summary of Stencil Types 160
6.2.6 Exposure 161
6.2.7 Setting Up 161
6.2.8 The Ink 163
6.2.8.1 Summary of Ink Mixing 164

6.3 Conclusion 165

CHAPTER SEVEN
Etching, Etchants and Etching Machines
W. Macleod Ross

7.1 Introduction 166

7.2 Etchants 168
7.2.1 Ferric Chloride 168
7.2.2 Ammonium Persulphate 170
7.2.3 Chromic/Sulphuric Acid Mixtures 170
7.2.4 Cupric Chloride 171
7.2.5 Alkaline Etchants/Ammoniacal Etchants 171
7.2.6 Hydrogen Peroxide Etchants 172

7.3 Etching Machines 174

7.4 Conclusion 179

CHAPTER EIGHT
Lay-up and Bonding
J. A. Scarlett

8.1 Inner Layer Preparation 181
8.1.1 Plain Multilayer Boards 181
8.1.1.1 Tooling Holes for Printing 181
8.1.1.2 Tooling Holes for Bonding 182
8.1.1.3 Milling or Punching Tooling Slots 183
8.1.1.4 Film Tooling Set 183
8.1.1.5 Location of Films 184
8.1.1.6 Copper Around Tooling Holes 185
8.1.2 Buried Via-hole and Sequential Multilayer Boards 185
8.1.2.1 Panel Plating 185
8.1.2.2 Pattern Plating 185
8.1.2.3 Hole Filling 186
8.1.2.4 Surface Dressing 186
8.1.3 Single-sided Layers 187
8.1.4 Oxidation 187
8.1.4.1 Need for Oxidation 187
8.1.4.2 Bond Strength 187
8.1.4.3 Oxidising 188
8.1.4.4 Cleaning and Drying 188
8.1.4.5 Conveyorised Spray Oxidising 189

8.2 Prepreg Preparation 189
8.2.1 Storage 189
8.2.2 Cutting and Testing 190
8.2.3 Punching 190

8.3 Tooling 191
8.3.1 Bonding Pins 191
8.3.1.1 Pin Patterns 191
8.3.1.2 Material Movement 192
8.3.1.3 Pin Sizes 192
8.3.2 Press Plates 193
8.3.3 Liners 194

8.4 Stacking 195
8.4.1 The Lay-up Area 195
8.4.2 Lay-up 195
8.4.2.1 Single Board 195
8.4.2.2 Multiple Boards 196
8.4.3 Handling of Laminates and Prepregs 196

8.5 Packing 196
8.5.1 Kraft Paper 196
8.5.2 Conformal Sheet 198
8.5.2.1 Blotting Paper 198
8.5.2.2 Bonding Under Vacuum 198

8.6 The Action of Prepreg During Bonding 199
8.6.1 The Bonding Operation 199
8.6.2 Inter-relation of Bonding Parameters 199
8.6.3 Board Thickness 200
8.6.3.1 Filling Low Areas 200

8.7 Bonding Defects 201
8.7.1 Types of Defects 201
8.7.1.1 Telegraphing 201
8.7.1.2 Unbonded Voids 201
8.7.1.3 Air Inclusions 202
8.7.1.4 Delamination 202
8.7.1.5 Surface Defects 202
8.7.2 Flatness of Boards 202
8.7.2.1 Thickness Variations 202
8.7.2.2 Bowing, Warping or Twisting 203
8.7.2.3 Measling and Crazing 204
8.7.3 Avoidance of Defects 204

CHAPTER NINE
Presses and Press Cycles
J. A. Scarlett and C. T. Morrisroe

9.1 Presses 205
9.1.1 Introduction 205
9.1.1.1 General Press Features 205
9.1.1.2 Press Platens 206
9.1.1.3 Press Frame 206
9.1.1.4 Installation of a Press 208
9.1.2 Heating 208
9.1.2.1 Electric Heating 208
9.1.2.2 Steam Heating 209
9.1.2.3 Oil Heating 209
9.1.3 Cooling 210
9.1.3.1 Water Cooling 211
9.1.3.2 Air Purging 212
9.1.3.3 Chilled Oil Cooling 213
9.1.3.4 Summary of Heating and Cooling Methods 213
9.1.4 Pressing 213

9.1.4.1	Hydraulic System	213
9.1.4.2	Intermediate Plattens	213
9.1.5	Instrumentation	214
9.1.6	Loading and Unloading	214
9.2	Press Cycles	218
9.2.1	General	218
9.2.1.1	Resin Behaviour	218
9.2.1.2	Kiss Pressure Cycles	220
9.2.1.3	Single Pressure Cycles	221
9.2.2	Temperature Cycles	222
9.2.2.1	Cold Start Cycles	222
9.2.2.2	Hot Start Cycles	222
9.2.2.3	Packing	224
9.2.2.4	Post Cure Baking	225
9.2.3	Automation of Press Cycles	226
9.2.4	Misunderstandings	226
9.2.4.1	Temperature	226
9.2.4.2	Pressure	227
9.3	Transfer Presses	228
9.3.1	General	228
9.3.2	Transfer Press Construction	231
9.4	Press Capacity and Use	232
9.4.1	Durability and Flexibility	232
9.4.2	Capacity	232
9.4.2.1	Board Size	232
9.4.2.2	Number of Blanks Bonded Per Cycle	233
9.4.2.3	Multiple Imaging of Boards	234
9.4.2.4	Overall Cycle	234
9.4.3	Unstacking	234
9.4.3.1	Removing Tooling Pins	234
9.4.3.2	Opening the Press Plates	235

CHAPTER TEN

Machining Multilayer Circuit Boards

J. K. Hagge and J. C. Mather

10.1	Introduction	236
10.2	Fundamental Considerations	236
10.3	Detailed Considerations	238
10.3.1	Tool Geometry	238
10.3.2	Tool Material	239
10.3.3	Design Properties of the Tool	241
10.3.4	Chip Removal	241
10.3.5	Workpiece Materials	241
10.3.6	Equipment	242
10.4	Machining to Establish Geometry	243
10.4.1	Contour Routing	243
10.4.1.1	Acceptability Criteria	243
10.4.1.2	Conclusions from Machining Theory	244
10.4.1.3	Routing Bits	247
10.4.1.3.1	Materials	247
10.4.1.3.2	Geometry	248
10.4.1.3.3	Tool Life	248
10.4.1.4	Contour Routing Equipment	249
10.4.1.5	Workpiece	250

10.4.1.5.1	Material Properties	250
10.4.1.5.2	MLCB Design	252
10.4.1.5.3	Support and Rigidity	252
10.4.1.6	Operating Parameters	252
10.4.1.7	Troubleshooting Contour Routing Problems	254
10.4.2	Chamfering	257
10.5	Machining to Establish Interconnection	258
10.5.1	Drilling	258
10.5.1.1	Acceptability Criteria	258
10.5.1.2	Conclusions from Drilling Theory	261
10.5.1.3	Drill Bits	265
10.5.1.3.1	Drill Bit Materials	265
10.5.1.3.2	Drill Geometry	265
10.5.1.3.3	Tool Life	267
10.5.1.4	Drilling Equipment	268
10.5.1.5	Workpiece	268
10.5.1.5.1	Materials	268
10.5.1.5.2	MLCB Design	269
10.5.1.5.3	Support	271
10.5.1.6	Operating Parameters	271
10.5.1.7	Finishing Operations	273
10.5.1.8	Troubleshooting Drilling Problems	274
10.5.1.8.1	Dielectric Smear	275
10.5.1.8.2	Inner Pad Distortion	275
10.5.1.8.3	Drill Withdrawal Distortion	275
10.5.1.8.4	Entry and Exit Copper Burrs	277
10.5.1.8.5	Rough Hole Walls	278
10.5.1.8.6	Fishbowl Hole Walls	281
10.5.1.8.7	Drilling Delamination	281
10.5.1.8.8	Nailhead	282
10.5.1.8.9	Pad Ripout	285
10.5.1.8.10	Dielectric Tearout	286
10.5.1.9	Troubleshooting Summary	286
10.5.2	Counterboring	287
10.6	Advanced Machining Techniques	288
10.6.1	Mechanical Methods	289
10.6.2	Chemical Methods	289
10.6.3	Laser Methods	289
10.6.4	Adaptive Control	290

CHAPTER ELEVEN

Smear Removal Methods

D. R. Witherell

11.1	Introduction	292
11.1.1	Definition of Drill Smear	292
11.1.2	Causes of Smear Formation	292
11.1.3	Materials Subject to Smear	293
11.1.4	Procedures Leading to a Reduction in Hole Wall Smearing	293
11.2	Criteria for Selection of a Smear Removal Process	293
11.2.1	Laminate Material	293
11.2.2	Process Capabilities	294
11.2.3	Customer Preference	295
11.3	Smear Removal Methods	295
11.3.1	Chemical Methods for Smear Removal	295
11.3.1.1	Sulphuric Acid Systems	295
11.3.1.1.1	Mechanism of Removal	296

11.3.1.1.2	Equipment Required	296
11.3.1.1.3	Typical Process Procedure	297
11.3.1.1.4	Process Parameters	299
11.3.1.1.5	Control Procedures	300
11.3.1.2	Chromic Acid Systems	300
11.3.1.3	Strong Base Systems	301
11.3.2	Mechanical Methods for Smear Removal	301
11.3.3	Gas Plasma	302
11.3.3.1	Principles of Operation	302
11.3.3.2	Materials Subject to Plasma Etch	302
11.3.3.3	Equipment Requirements	302
11.3.3.4	Typical Process Procedures	302
11.4	Inspection Methods	303
11.5	Summary	304

CHAPTER TWELVE
Electroless Plating
D. A. Luke

12.1	Introduction	306
12.2	History of Through-hole Plating of Multilayer Printed Circuits	306
12.3	Special Problems Posed by Multilayer Circuits Compared with Conventional Double-sided Boards	307
12.3.1	Drill Smear on Innerlayer Connections	307
12.3.2	Over Etching of Innerlayers	308
12.3.3	Surface which Rejects Catalyst	308
12.3.4	Bleaching of Black Oxide Layers	309
12.3.5	Thick Boards Requiring High Throwing Power Electroplating Solutions	309
12.3.6	Thermal Shock of Plated Deposits	309
12.4	Alternative to Through-hole Plating for Multilayer Printed Circuits	310
12.4.1	Vacuum Metallising	310
12.4.2	Consecutive Deposition of Metal and Insulant	310
12.5	Special Laminates and Composites	310
12.6	Typical Sequences for Electroless Plating	311
12.6.1	Special Conditioners for Epoxy-glass which has been Back-etched	311
12.6.2	Pre-etch	313
12.6.3	Activation	315
12.6.3.1	Two-stage Activation	315
12.6.3.2	Single-stage Activation	316
12.6.3.3	Catalyst Coverage	318
12.7	Electroless Copper Deposition	321
12.7.1	Basic Constituents	321
12.7.2	Available Processes	322
12.7.3	Analysis	323
12.7.4	Deposit Properties	324
12.7.5	Plant	324

CHAPTER 13
Electroplating
D. A. Luke

13.1	Copper Plating	326

13.1.1	Typical Pattern Plating Sequence	326
13.1.1.1	Soak Cleaner	327
13.1.1.2	Mild Etch	327
13.1.1.3	Acid Dip	327
13.1.1.4	Rinses	327
13.1.2	Copper Plating Solutions Available	328
13.1.2.1	Pyrophosphate Solution	328
13.1.2.2	Acid Sulphate Solution	330
13.1.3	Throwing Power Considerations for Thick Boards	331
13.1.3.1	Throwing Power Measurement	331
13.1.4	Distribution	332
13.1.4.1	Anode Positions	333
13.1.4.2	Anode Masks	333
13.1.4.3	Power Supplies to Anodes	333
13.1.4.4	Agitation	333
13.1.4.5	Solution Constituents	334
13.1.4.6	Robbing Strips	334
13.1.5	Plating of Internal Layers	334
13.1.5.1	Stress Considerations	334
13.1.5.2	Stress Measurements	335
13.1.5.3	Elongation Deposits	336
13.1.5.4	Hardness	338
13.1.5.5	Electrical Conductivity	338
13.1.6	Contamination	339
13.1.6.1	Causes and Effects of Organic Contamination	339
13.1.6.2	Metallic Contamination	339
13.2	Tin-lead Plating	340
13.2.1	Considerations for High Throwing Power Tin-lead Process	340
13.2.1.1	Throwing Power and Plate Distribution	340
13.2.2	Changes of Alloy with Processing Conditions	341
13.2.3	Interference from Organic Matter	343
13.2.3.1	Fusion Time	343
13.2.3.2	Outgassing	343
13.2.3.3	Surface Effects	343
13.2.3.4	Estimation	344
13.2.4	Corrosion Resistance	344
13.3	Nickel and Nickel Alloys	345
13.3.1	Nickel	345
13.3.2	Tin-nickel	347
13.3.3	Palladium-nickel	347
13.4	Gold	347
13.4.1	Substrate Preparation	348
13.4.1.1	Preparation and Methods of Reducing Porosity	348
13.4.1.2	Porosity	348
13.4.1.3	Pretreatment	349
13.4.2	Cathode Efficiency	350
13.4.3	Distribution	351
13.4.4	Internal Stress	351
13.4.5	Elongation	353
13.4.6	Wear	354
13.4.7	Hardness	354
13.4.8	Porosity Measurement	354

CHAPTER FOURTEEN
Cleaning and Preparation of Exposed Copper Surfaces in MLB Manufacture
G. D. M. Menzies

14.1	Introduction	356

14.2 Cleaning Methods 356
14.2.1 The Importance of Rinsing 356
14.2.2 Purposes of Surface Preparation 357
14.2.3 Cleaning Processes Used and Associated Problems 357
14.2.4 Results of Inadequate Cleaning 360
14.2.5 Surface Smoothing 362
14.2.6 Preparation of Boards with Gold-plated Edge Connectors 362
14.2.7 Cleaning Sequence 365

CHAPTER FIFTEEN
Solder Finishes
D. S. Kingsley

15.1 Introduction 367

15.2 Types of Finish 367
15.2.1 Roller Tinning 367
15.2.2 Electrodeposited Tin-lead 368
15.2.3 Flow Melting 368
15.2.4 Hydro-squeegee 369
15.2.5 Hot Oil Fusing 370
15.2.6 Infra-red Fusing 370
15.2.7 Condensation Reflow 371
15.2.8 Hot Air Levelling 371

15.3 Conclusion 372

CHAPTER SIXTEEN
Quality and Inspection
J. A. Scarlett

16.1 Quality 374
16.1.1 Quality 374
16.1.2 Quality Control 374
16.1.3 Quality Assurance 375
16.1.4 Acceptable Quality Level 375

16.2 On-line Quality Control 376
16.2.1 Preliminary Controls 377
16.2.1.1 Filmwork 377
16.2.1.2 Drill and Router Tapes 379
16.2.1.3 Raw Materials 379
16.2.2 Quality Control on Inner Layers 379
16.2.2.1 Printing 379
16.2.2.2 Etching and Oxidising 380
16.2.3 Quality Control of Bonding 380
16.2.4 Quality Control of Drilling 381
16.2.4.1 Smear Removal and Etch Back 381
16.2.5 Quality Control of Plating and Outer Layer Processing 382
16.2.5.1 Electroless Copper 382
16.2.5.2 Electroplating 382
16.2.5.3 Printing and Etching 382

16.3 Final Inspection 382
16.3.1 Test Coupons 383
16.3.1.1 Need for Test Coupons 383
16.3.1.2 Features of Test Coupons 384
16.3.2 Visual Inspection 385
16.3.2.1 General Examination 385
16.3.2.2 Pads and Holes 386

16.3.2.3 Tracks and Clearances 386
16.3.2.4 Solder Resist, Legend and Profile 387
16.3.2.5 Aids to Visual Inspection 387
16.3.3 Destructive Tests 389
16.3.3.1 Track Tests 389
16.3.3.2 Hole Barrel Strength 389
16.3.3.3 Solderability 390
16.3.3.4 Solder Float Test 390
16.3.3.5 Flexing 390
16.3.3.6 Thermal Tests 392

16.4 Microsections 394
16.4.1 Cutting Samples 394
16.4.2 Encapsulation 395
16.4.2.1 Resins 395
16.4.3 Preparation 396
16.4.4 Viewing 396
16.4.4.1 Photographic Records 396
16.4.5 Evaluation of Microsections 396
16.4.5.1 Surface Features 397
16.4.5.2 General Internal Features 398
16.4.5.3 Holes 398
16.4.5.4 Evaluation After Thermal Testing 399
16.4.6 Interpretation of Microsections of Holes 400

16.5 Electrical Testing 401
16.5.1 Tests on Test Coupons 402
16.5.2 Manual Testing for Continuity 402
16.5.3 Automatic Continuity Testing 403
16.5.3.1 Test Fixtures 403
16.5.3.1.1 Individual Heads 403
16.5.3.1.2 Grid Based Heads 403
16.5.3.1.3 Translation Boards 404
16.5.3.1.4 Flexible Probe Heads 404
16.5.3.1.5 Simulated Component Test Heads 405
16.5.3.1.6 Edge Connectors and Buried Via Holes 406
16.5.3.2 Test Programmes 406
16.5.3.2.1 Comparative Testing 406
16.5.3.2.2 Testing to Circuit Diagram 407
16.5.3.3 Limitations of Automatic Continuity Testing 407

CHAPTER SEVENTEEN
Laminate Stability
N. C. J. Parry

17.1 Introduction 409
17.1.1 X and Y Shift 409
17.1.2 Z-Axis Changes 409

17.2 Causes 410
17.2.1 Raw Materials 410
17.2.1.1 Woven Glass Fabric 410
17.2.1.2 Resin 410
17.2.1.3 Copper Foil 414
17.2.2 Laminate Production Processes 414
17.2.2.1 Impregnation 414
17.2.2.2 Pressing 414

17.3 Quantifying the (In)Stability 415

17.4 Improving Stability 419
17.4.1 The Laminator's Task 419
17.4.2 M/L Designer and Fabricator's Task 420

17.5 The Future 421

17.6 Conclusion 422

CHAPTER EIGHTEEN
The Use of Radiography in the Inspection of Multilayer Boards
D. S. Kingsley

18.1 Introduction 423

18.2 Basic Principles 423

18.3 Film Image 425

18.4 Choice of Equipment 426

18.5 Safety Requirements 428

18.6 Basic Specification 429

CHAPTER NINETEEN
Multilayer Board Repair
D. S. Kingsley

19.1 Introduction 430

19.2 Specifications 431
19.2.1 Rework 431
19.2.2 Repair 431
19.2.3 Modification 431

19.3 Quality Control of Rework 432

19.4 Repair Technician 432

19.5 The Repair Section 433
19.5.1 The Track Cutting Machine 434
19.5.2 Other Equipment 435

19.6 Repair Methods 437
19.6.1 Removal of Inclusions 438
19.6.2 Lifted Pads and/or Conductors 440
19.6.3 Internal Short Circuit 441
19.6.4 Replacement of Missing Surface Pads 441
19.6.5 Repair of Damaged Conductors on Internal Layers Prior to Lamination 441
19.6.5.1 Conductive Compound 441
19.6.5.2 Tinned Copper Foil Bridge 442
19.6.5.3 Parallel Gap Welding Technique 442
19.6.6 Refurbishing Edge Contacts 442
19.6.7 Repair of Damaged Conductors on External Layers 443
19.6.8 Complete Replacement of Edge Contacts 443
19.6.9 Removal of Short Circuits and/or Excess Metal, External Layers 443
19.6.10 Repair of Open Circuits After Lamination (Inner Layer) 443
19.6.11 Re-establishment of Solder Pad 444
19.6.11.1 Method A 444

19.6.11.2 Methods B and C 444
19.6.11.2.1 Method B 444
19.6.11.2.2 Method C 446
19.6.12 Local Through-hole Plating of Holes 447

CHAPTER TWENTY
Flexi-rigid Boards
J. A. Scarlett

20.1 Construction of Flexi-rigid Boards 450

20.2 Types of Flexi-rigid Boards 451

20.3 The Design of Flexi-rigid Boards 454

20.4 Applications for Flexi-rigid Boards in Digital Systems 455
20.4.1 Advantages and Disadvantages of Flexi-rigid Boards 458

20.5 Manufacture of Flexi-rigids 459
20.5.1 Laminate Stability 459
20.5.2 Cover Coat and Plated Holes in Tails 462
20.5.2.1 Cover Coat 462
20.5.2.2 Plated Holes in Tails 462
20.5.3 Final Bonding, Through-plating and Profiling 466
20.5.3.1 Plating and Bonding 466
20.5.3.2 Profiling 470

20.6 Design of Pads in Tails 472
20.6.1 Unplated Tails 472
20.6.2 Plated Tails 474

CHAPTER TWENTY-ONE
Desoldering Multilayer Boards without Damage
J. A. Scarlett

21.1 Introduction 478
21.1.1 Solderability 478
21.1.2 Unplated and Plated Holes 478
21.1.3 Repair Kits 479
21.1.4 Damage to Holes 479

21.2 Types of Joints 480
21.2.1 Non-plated Holes 480
21.2.2 Through-plated Holes 480

21.3 Removal of Components 481
21.3.1 Single Leads 481
21.3.1.1 Removal of Solder 481
21.3.1.2 Softening of Resin 483
21.3.1.3 Damage to Holes 484
21.3.2 Multiple Leads 484

21.4 Correct Techniques for Lead Removal 484
21.4.1 Simultaneous Desoldering 484
21.4.2 Lead Cutting 485

21.5 Crimping 485
21.5.1 Damage Caused by Crimped Leads 485
21.5.2 Assembly Without Crimping 486

21.6 Avoidance of Damage to Holes 486

21.7 Summary 486

CHAPTER TWENTY-TWO
Documentation and Standards
W. Macleod Ross

22.1 Introduction 489

22.2 International Standards and Specifications for Multilayer Materials and Circuits 490
22.2.1 European Standards and Specifications 491
22.2.2 United Kingdom Standards and Specifications 492
22.2.3 American Standards and Specifications 494

22.3 In-house Standards and Specifications 496

CHAPTER TWENTY-THREE
Design
J. A. Scarlett

23.1 When to use Multilayer Boards 499

23.2 Electrical Properties in Design 500
23.2.1 Current Carrying Capacity 500
23.2.2 Voltage Breakdown 501
23.2.3 Characteristic Impedance 501
23.2.3.1 Reflections and Surge Currents 503
23.2.3.2 Propagation Delay 503
23.2.3.3 Effect of Low Characteristic Impedance 503
23.2.4 Capacitance 504
23.2.5 Crosstalk 504

23.3 Mechanical Aspects of Design 507
23.3.1 Balance 507
23.3.2 Build 507
23.3.3 Overall Thickness 509
23.3.4 Laminate Thickness and Stability 509
23.3.5 Feature Sizes 510
23.3.5.1 Track and Gap Widths 510
23.3.5.2 Hole and Pad Sizes 511
23.3.5.3 Pad Size and Reliability 512
23.3.6 Location of Features 513
23.3.7 Planes and Pads 513
23.3.8 Outer Layers and Finishes 520

23.4 Layer Numbering 521

23.5 Solder Resist and Legends 522

23.6 Designing Multilayer Boards 522
23.6.1 The Self-defeating Multilayer Board 522
23.6.2 Placing Via Holes 523

23.7 Buried Via-hole and Double-double Boards 524

23.8 Filmwork for Multilayer Boards 524

CHAPTER TWENTY-FOUR
Multiwire 528
R. P. Burr and G. Messner

24.1 Introduction 530

24.2 Multiwire Product and Process Overview 530
24.2.1 The Multiwire Board 531
24.2.2 The Multiwire Process 532

24.3 Multiwire Manufacturing System 533
24.3.1 Format Preparation 533
24.3.2 Wiring Process 535
24.3.2.1 Wiring Machine Structure 535
24.3.2.2 Wiring Machine Operation 536
24.3.2.3 Wiring Machine Performance 541
24.3.2.3.1 Wiring Speed and Productivity 541
24.3.2.4 Multilevel Wiring 542
24.3.3 Termination Process 543
24.3.3.1 Protection of Board Surface 543
24.3.3.2 Hole Drilling 543
24.3.3.3 Hole Cleaning 544
24.3.3.4 Hole Metallisation 544
24.3.3.5 Alternative Processes 545
24.3.4 Final Fabrication 545
24.3.5 Testing 546

24.4 Multilayer Design (MDT) System 547
24.4.1 Design System Function 547
24.4.2 System Data Interface 548
24.4.3 Input Conversion 548
24.4.4 Routing 550
24.4.5 Hardware and Performance 553

24.5 Multiwire Product Characteristics 554
24.5.1 Reliability 554
24.5.1.1 Electrical Insulation 555
24.5.1.2 Termination (Hole-to-wire) Interface Reliability 557
24.5.1.2.1 Effects of Multiwire Pattern Formation Technology 557
24.5.1.2.2 Effects of Wire-hole Interface Formation Process 558
24.5.1.2.3 Effects of Multiwire Structural Materials 560
24.5.2 High Frequency Characteristics 560
24.5.2.1 Characteristic Impedance 561
24.5.2.2 Effects of Wire Crossovers 563
24.5.2.2.1 Crossover Capacitance 563
24.5.2.2.2 Interwire Capacitance Coupling 564
24.5.2.2.3 Crossover Capacitance Loading Effects 565
24.5.2.3 Crosstalk 566
24.5.3 Special Product Characteristics 569
24.5.3.1 Improved Thermal Conductivity 569
24.5.3.2 Options for Surface Attachment 570

24.6 Multiwire Market and Applications 572
24.6.1 Typical Production Configurations 572
24.6.2 Direction of Product Applications 573
24.6.3 Commercial Status 577
24.6.4 Summary 577

Chapter 1

INTRODUCTION

J. A. SCARLETT
STC Exacta Ltd., Selkirk, Scotland

1.1 GENERAL

1.1.1 Printed Circuit Boards

Printed circuit boards (PCBs) are key components in almost all electronic assemblies. Their basic concept is simple enough. A pattern of metal tracks which serve to interconnect the components is bonded onto an insulated plastic substrate which provides mechanical support for the components. When the density of the interconnect is such that all the tracks required cannot be fitted onto one side of the plastic substrate, the second side can also be used. When the tracking on two sides becomes inadequate, a multilayer structure of alternating layers of plastic and patterned metal tracking is used. Components can be mounted with their leads through holes in the board and their leads soldered to pads around the holes on the interconnect tracking or they can be re-flow soldered to pads on the surface.

The functions of a board can be summarised as:

1 To provide a mechanical support for the components;
2 To provide an electrical interconnect for the components;
3 To provide a safe working environment for the components.

All of these can be provided in other ways, but only a printed circuit board can offer the user the advantages of:

(i) predictability of electrical performance;
(ii) repeatability of manufacture (if all the interconnections are right on the prototype, they will be right on all boards that are properly made);
(iii) reliability;
(iv) low volume;
(v) light weight;
(vi) ease of replacement in field servicing (most boards carry some form of plug which can be inserted into a socket on the frame of the equipment);
(vii) ability to carry a printed legend to aid assembly and field servicing;
(viii) ease with which modifications can be made;

(ix) low unit cost compared with other possibilities;
(x) ease of cooling;
(xi) ability to use mass jointing techniques;
(xii) ability to carry a thermal conducting path.

1.1.2 History

Like so many everyday items, the multilayer board as it is widely used today was not invented all at once. A number of different developments, processes and materials have been involved, and in a brief account such as this only a few of the main developments can be mentioned. Apologies are offered to any readers who feel they have made a major contribution which has been ignored.

The first practical printed circuit boards, which were double-sided, are usually considered to have been those made by Dr Paul Eisler of Pye Ltd in 1942, patented on February 2nd, 1943. The widespread adoption of this new technology was delayed by the lack of really suitable materials, and it was not until the 1950s that boards made by etching copper foil bonded to a glass reinforced epoxide resin substrate were widely used.

The effective use of double-sided boards required an efficient means of making an electrical connection through the board at a point where no component lead is to be mounted (a via hole). Work on this continued through the later 1950s.

Meanwhile, the advent of the transistor as a replacement for the thermionic valve began to tax the capabilities of the PCB as an interconnect. The smaller size and lower dissipation of the transistor made it possible to reduce the overall size of equipments by more than an order of magnitude, and most or all of the components could be mounted on boards instead of, in the case of larger, heavier components and sockets for valves, on a metal chassis. By the late 1950s electronics engineers were finding that the two outer faces of a board were inadequate to carry the necessary track patterns.

The main problem which delayed the universal adoption of multilayer boards was the difficulty of making a reliable interconnection to the buried tracking on the inner layers. Four possibilities were:

(i) access holes
(ii) plated pillars
(iii) metal stakes
(iv) plating through the holes (see Section 1.2).

Multilayer technology as it is known today can really be said to date from the 1961 patent by Hazeltyne in the USA of the seeded plated-through hole in a multilayer structure.

Progress in the development of electroless plating solutions was fairly rapid, and by the mid 1960s the bonded multilayer board with through-plated holes was firmly established as the preferred medium for high density interconnections. By the end of the decade most of the variants in present-day use had been adopted, and the developments throughout the 1970s have been mainly in production techniques and materials. Sizes of boards used have increased steadily and feature sizes have been reduced to enable very high packing densities to be achieved.

1.1.3 The Electrical Importance of Multilayer Boards

When printed circuit boards first came into use the operating speeds of electronic equipment were low, and the performance of any digital system was governed by the switching speed of the valves or transistors used. Delays in a few decimetres of wire or tracking could be ignored. The development of planar silicon transistors and then of integrated circuits altered this situation. With devices capable of changing their output state in less than a nanosecond, the effects of a few centimetres of track can be critical and, instead of being merely an interconnect with no electrical significance other than its current carrying capacity and voltage breakdown capability, the PCB became a component whose electrical properties of inductance, capacitance and line impedance had to be considered along with the properties of the semiconductor devices used.

Kolias of IBM has suggested that, for a rise time of 1 nanosecond, only lines shorter than 0·0033 in. (under 0·1 mm) can be regarded as purely resistive; up to 1·67 in. (42·4 mm), the lumped characteristics may be used, but beyond this length the characteristic impedance of the line must be considered.

With densely packed boards and relatively low line impedances, cross-talk and the effects of ring and line surge currents have to be considered very carefully in the design of any PCB for a digital system. When the importance of the interconnect is considered, it is apparent that the shorter the tracks can be kept for a given type of interconnect, the lower will be the chance of electrical malfunction. Conversely, the lower the packing density, the higher the probability of malfunction (Figure 1.1). It must be noted that this single line graph is valid only for any one specific type of interconnect. Higher packing densities achieved by changing the nature of the interconnect, such as by replacing double-sided boards with multilayers, may introduce discontinuities into the curve as the electrical properties will be different. A gain in packing density achieved by the use of a different style of interconnect may be more than offset by increased signal delays in the denser packing style (see Chapter 23).

1.1.4 Costs and Applications

1.1.4.1 COSTS OF BOARDS

Printed circuit boards, including multilayers and flexi-rigids, usually provide the most cost-effective method for mounting and interconnecting components, but this does not necessarily mean that they are cheap. The simpler single-sided boards in which all holes can be punched and the profile formed with a single press tool can cost only tens of pence each in quantity, but double-sided boards with through-plated holes and gold plated edge connector fingers may cost several pounds, running up to over a hundred pounds each for small quantities of larger, more densely packed

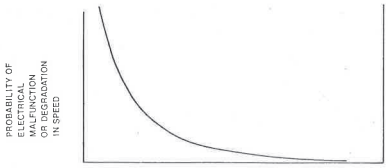


Fig. 1.1 Probability of electrical malfunction against packing density for any one given style of packaging.

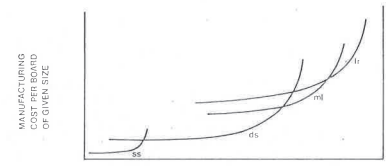


Fig. 1.2 Printed circuit board costs against packing density for single-sided, double-sided, multilayer and flexi-rigid.

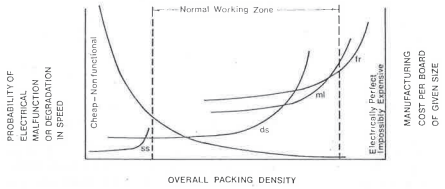


Fig. 1.3 Typical working zone available to a board designer, over which cost and functionality must be balanced.

boards. Even in large quantities a board can cost almost as much as the total cost of the components it carries.

Multilayer boards cost significantly more than double-sided boards of comparable size and complexity. Obviously, costs will vary depending on the size and nature of the boards concerned and the quantities involved, but as a very rough guide it can be taken that a four layer board could cost somewhere between half as much again as a comparable double-sided board to two and a half times as much. The use of six layers raises these figures to two and a half up to three and a half times as much as a double-sided, with comparative costs rising as further layers are added. The use of flexi-rigid or other more specialised types such as those described in Section 1.2.5 onwards raises these cost ratios significantly, some of the more complex types of multilayers costing several hundreds of pounds in production quantities.

It cannot be over-emphasised that these comparative costs are only rough guidelines and that more accurate quotations should be obtained before any detailed design work commences.

Whenever board costs are being considered, the costs of design, the generation of film work, and other tooling costs must not be overlooked. Design and film costs for a complex multilayer or flexi-rigid may amount to thousands of pounds, and when only small quantities will be required the influence of these costs will be considerable. In the Author's experience, a break-even with other possible interconnection methods usually occurs when approximately four boards have been made, but it has been shown that the superior, predictable and controllable electrical properties of multilayer boards can make their use justifiable, even with all the design costs, for a single system.

1.1.4.2 COSTS OF ALTERNATIVES TO BOARDS

With unit costs for large, complex multilayer boards running into hundreds of pounds, plus design and film costs, such boards may seem expensive, but only until the true costs of other possible options are considered. At the smaller end of the size range, epoxy-glass based multilayer PCBs may be in competition with multilayer ceramic substrates for hybrid assemblies, but in general the choice of substrate will depend on the physical properties of the materials rather than on the costs, which tend to favour the epoxy-glass laminates. With larger multilayer boards the usual alternative is some form of discrete wiring or cable harnesses. First, consideration must be given to the provision of some form of mechanical support for the components or sockets likely to be carried by a multilayer board. On back wiring panels for rack mounted systems the cost of providing an accurate, secure mounting for all the connectors is likely to approach that of a simple PCB, whilst for plug-in units the provision of an alternative component mounting in any form other than an untracked board could exceed the costs of a simple board.

All the costs of making and verifying the complex interconnect must then be considered. On a multilayer board there may be two or three thousand points to be interconnected. If a board is not used, each of these points must have a durable, reliable connection made to it, verified, and repaired or

replaced if necessary, at a cost per connection of only two or three pence. Wires must be cut to length, terminated, and dressed into channels or tied into bundles. All this can be automated, but the equipment required is expensive and often its inflexibility imposes limitations on future generations of the basic product.

These obvious direct comparisons consider only a part of the true savings realisable by the use of multilayer boards.

A board contains the interconnect, usually within the volume which would be occupied by any other purely mechanical form of mounting for the components. The use of a board permits mass soldering techniques to be employed, usually with gains in joint reliability of an order of magnitude or more as well as a substantial cost saving. One major advantage of a multilayer or double-sided board is that the electrical properties of the interconnect can be predicted and controlled at the design stage, and will be substantially the same for all units made. The improvements in performance and the reduction in development times made possible by the use of boards may be considered more than adequate justification for their cost, even if an alternative discrete wiring interconnect were available at zero cost.

Flexi-rigid boards offer the benefits of the elimination of connections, either plugs and sockets or wires on pins or in holes, This economises on materials and labour, but of even greater significance it gives appreciable increases in overall system reliability coupled with reductions in size and weight which may be difficult to achieve by any other means.

1.1.4.3 CHOICE OF BOARD TYPE

For any given style of board, such as single-sided, double-sided with through-plated holes, conventional multilayer, or flexi-rigid, there will be a range of packing densities beyond which the cost per unit area of a board will not be significantly affected by the density. Then, as the number of holes increases and track and gap widths decrease, the costs begin to rise more steeply with increased packing density until a cut-off point is reached at the current limit of the technology (see Figure 1.2).

When a new system is being designed, such a graph of costs for the various possible styles of interconnect can be combined with a graph of the probability of electrical malfunction (Figure 1.1) to demonstrate the range of choices open to the designer (Figure 1.3). The curve for electrical malfunction may have to be broken or stepped to allow for the differing electrical characteristics of double-sided and multilayer boards, and it may also have to be sharply biased or cut off to take into account factors like long-term reliability.

The choice of board type may well be dictated by electrical requirements or by limitations on space or weight. Any demand for accurately controlled line impedances or for earth screening will usually necessitate the use of a multilayer board with internal planes, and limits on the available space for the equipment can call for the use of high density multilayers or flexi-rigids. Multilayer boards are occasionally used instead of double-sided because they are a more recent development or sometimes simply because they cost

more and are therefore assumed to be 'better'. The use of a graph such as Figure 1.3 should help to prevent such costly mistakes.

1.1.4.4 THERMAL PROBLEMS

When the use of multilayer boards is being considered to help to achieve higher packing densities, it must be borne in mind that not only may the usually lower line impedances of a multilayer offset or negate any improvements over the performance of lower density double-sided boards, but the higher packing density may create expensive thermal problems.

Overheating is a notorious cause of unreliability in electronic systems, and whilst double-sided boards are usually designed at densities such that air cooling, either natural convection or fan blown, will suffice to keep the commonly used dual in-line packages at a safe temperature, the higher component densities achievable on a multilayer may make some form of conduction cooling essential.

The first stage of any such cooling is normally a copper or aluminium heat ladder etched or punched out of sheet 0·5 to 1 mm thick and bonded to the components side of the board. The heat ladder must have adequate clearances for all component leads to pass through it, and the obvious need to maximise the area on which the heat generating components will be seated, plus the necessity to allow clearance between the metal of the heat ladder and the live pads on the surface of the board, usually means that the tolerancing on the manufacture and bonding in position of the ladder is critical. Such ladders are generally bonded to the face of a finished board with a low flow prepreg (see Section 1.4.8).

Heat generated by the components is carried by the ladder to the board edges, where specially designed metal board guides clamp onto the face of the ladder and conduct the heat away from the board assembly. The design of the clamping mechanism, which must ensure the best possible thermal contact when in use but which must open to allow easy insertion and extraction of boards for servicing, is often a difficult exercise, and manufacture of the necessary metalwork is seldom cheap. The side plates of the unit which carry the guides may have to be deeply finned in order to provide adequate surface area to radiate the heat to the outside air, or on large systems with multiple rows of boards they may have to be provided with channels for pumped liquid cooling.

In most systems, the heat generated by the components will ultimately have to be dissipated to the outside air, which implies that there will have to be a certain area of radiating surface. It can be found that attempts to minimise the overall volume of an equipment by the use of high density conduction-cooled boards may result either in no ultimate saving or even in a unit with greater overall dimensions. Whether space is gained or lost, a conduction cooled unit will be considerably more expensive than a simple air-cooled unit.

Attempts to improve electrical performance on a large system by the use of high density packaging can have the reverse effect, because the need to provide cooling channels within the equipment means that some of the essential inter-board or inter-unit connections must be significantly longer

than they would have been had a more uniform, lower-density packaging been adopted.

1.1.4.5 SYSTEM ARCHITECTURE AND BOARD TYPE

Situations can arise in which some early decisions on system architecture may dictate the use of multilayer boards. Bit or byte split units or boards may have quite different tracking density requirements from function split units, and bussed systems also influence the density of their interconnect. The allocation of board connector pins can have a major impact on the tracking density of both back wiring panels and plug-in boards. A unit should be studied as a three-dimensional entity, and the interconnections can then be organised to provide the best overall tracking density to suit electrical performance, reliability and cost.

Systems in which a limited number of standard plug-in boards can be grouped into different overall configurations will usually require multilayer back wiring panels, as it can be difficult or impossible to organise the connector pinning on the boards to suit all likely applications. However, on single-purpose systems which use dedicated boards it may not be necessary to use a multilayer back wiring. Figure 1.4 shows a typical, small back panel

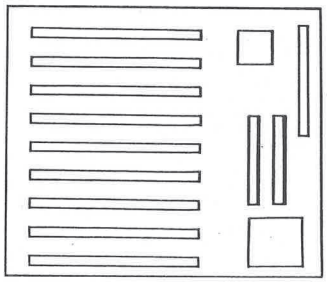


Fig. 1.4 A back-wiring panel layout which will almost certainly have to be multilayer to contain the necessary tracking along and across the board sockets on the left.

layout with sockets for nine boards, two round cables, and three ribbon cables. There can be little doubt that to complete an interconnection on such a layout, with tracks linking the nine board sockets crossed by tracks out to the cable sockets, a six or eight layer board will have to be used. However, if the length of each of the nine board sockets is increased to allow redundant pins over which tracks can be routed and more spaces

between pins, and if the cable philosophy is changed to allow the use of in-line instead of round connectors, then it is possible that the necessary interconnect could be accommodated on a double-sided board (Figure 1.5). The savings in board costs should more than offset the increase in connector costs.

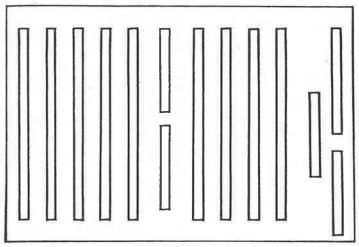


Fig. 1.5 A modification of the back-wiring layout of Fig. 1.4 which might be implemented on a double-sided board.

1.1.4.6 PROBABLE NEED FOR MULTILAYERS

With multilayer boards costing substantially more than double-sided and the possibility of thermal problems, it may be felt that there is only a limited field of application for multilayers. Figure 1.6 illustrates very roughly the probable need to use a multilayer depending on the type of equipment being made and its application. It can be seen that UHF analogue and fast, high power digital logic boards, both of which types require controlled impedance lines, will almost always use multilayers. Military portable and industrial FHP logic boards are shown with their squares divided vertically instead of horizontally because there is only a limited likelihood of such systems being manufactured, but if they were it is almost certain they would use multilayers.

Two columns are shown for back wiring panels, one to cover situations in which a system concept may be amenable to alteration to allow the use of double-sided boards as already described, and the other for those systems in which other requirements preclude any such structuring in order to minimise the cost of the back wiring.

Figure 1.6 reveals a surprisingly high percentage of board types as being likely to require multilayer construction. However, this figure is misleading as it shows equal areas for all types of boards. Figure 1.7 indicates the approximate market split predicted for the various types of through-hole plated boards in the foreseeable future. This graph cannot be regarded as being accurate, as clearly it will be influenced by national and international

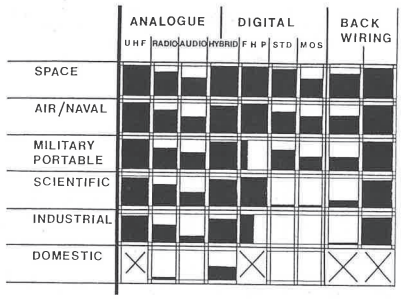


Fig. 1.6 Probability of the need to use a multilayer board for various types of electronic systems.

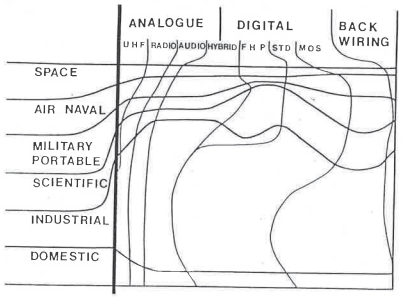


Fig. 1.7 Probable market for electronic systems using through-hole plated boards.

policies on defence spending and in the UK by the future of CB radio. However, the graph does at least give a crude indication of a possible split.

When the two diagrams are combined (Figure 1.8), it can be seen that it is probable that between a quarter and a third of the market for through-hole

plated boards will be for multilayers of various types. As these diagrams represent numbers of boards and not value, and as the multilayers may well have an average price towards three times that of the average double-sided board, it can be appreciated that the multilayer board will become one of the most critical components of the electronics industry.

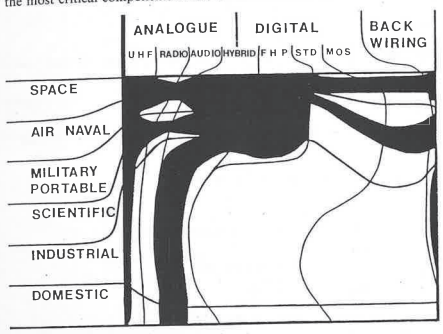


Fig. 1.8 The probable need for multilayer boards.

1.2 TYPES OF MULTILAYER BOARD

The term 'multilayer board' is usually understood to imply a board which has been bonded up from separate thin laminations, with holes plated through the bonded assembly (see Section 1.2.4), other types normally being qualified by a further description. Strictly speaking, however, a multilayer board can be any array of more than two layers of tracking, with or without internal layer-to-layer connections, and such boards can be constructed from a variety of materials by various methods. In this Section some of the major variants are described, but no claims can be made that the list is fully comprehensive.

1.2.1 Access Hole Boards

Access hole boards are probably the oldest type of multilayer board. The general principle is that the necessary interconnect pattern is designed on a number of thin, single-sided laminates with all tracking to any given point completed on one layer. Connections to tracks on all but the outer (solder side) layer are made possible by drilling large clearance holes in all layers between the one requiring the connection and the solder side of the board

(see Figure 1.9). These access holes are usually drilled somewhat larger than the pads used, to allow for misregistration in assembly of the board, and their diameter is also influenced by the need to provide adequate access to the pads for a soldering iron.

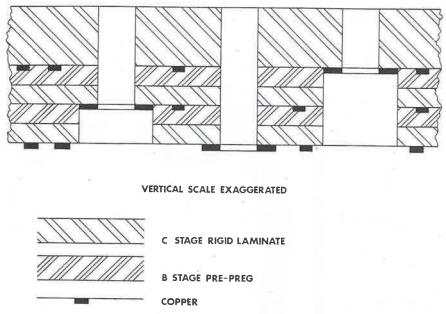


Fig. 1.9 Access hole board.

On a board with three or more layers it may be advisable to use larger holes for access to the layers nearer the components side. In fact, some designers prefer to use progressively larger holes in each lower layer of the board. When all the access holes have been drilled, the board is bonded with a low flow prepreg or adhesive, and then the holes for the component leads are drilled. Control of the flow of the bonding layer during assembly is critical, as it must be sufficient to encapsulate all the tracks on the inner layers, but not so much as to flow out over the etched copper pads to which joints will subsequently be formed.

Boards on which all interconnection nets are completed on one layer do not necessarily have to be bonded, but can be assembled layer by layer onto the component leads, leaving adequate space between the layers to avoid moisture retention. Such an assembly constitutes multilayer interconnect, but it is doubtful whether it should be classed as a multilayer board.

1.2.1.1 ACCESS HOLE BOARDS WITH INTERNAL CONNECTIONS

Situations can arise in which the completion of all tracking to any given point on one layer of a board can be impossible without using an impracticably large number of layers. As a simple example, it may be necessary to interconnect with minimum length tracking four points at the corners of a square. This can be achieved using a simple X pattern. However, if the total interconnection requires a multiplicity of such X

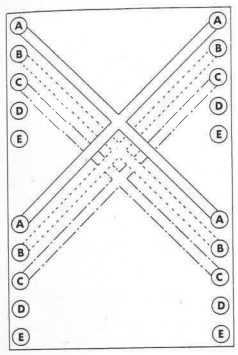


Fig. 1.10 A repetitive X pattern which requires as many layers in the board as there are repeated patterns if via holes are not used.

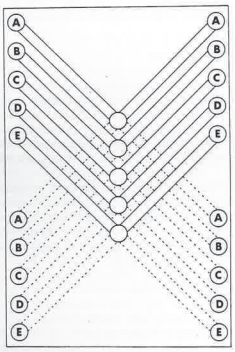


Fig. 1.11 The use of via holes enables the pattern of Fig. 1.10 to be implemented on two layers only.

connections, all superimposed but stepped down the board, then the pattern will require as many layers as there are superimposed Xs (see Figure 1.10). On the other hand, if layer-to-layer connections can be made within the board, this pattern is obtainable on only two layers of nested Vee tracks (see Figure 1.11).

Layer-to-layer interconnections can be achieved on access hole boards by using stepped hole and pad sizes as shown in Figure 1.12. With this method it is not essential to have a small hole right through the board, though some engineers prefer to insert a headed pin into the hole before soldering and to bend the pin so that it contacts the faces of the two pads to be connected. Without such a pin the joint will depend solely on the ability of the solder to bridge the dielectric material.

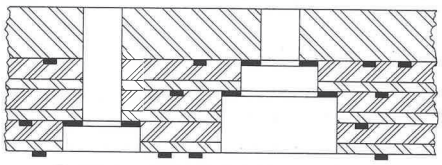


Fig. 1.12 Access hole board with layer-to-layer connection.

1.2.1.2 DISADVANTAGES OF ACCESS HOLE BOARDS

The major disadvantage of an access hole board is the amount of space occupied by the access holes. The land around the component holes must be at least that which would be used on a single-sided board (i.e., a minimum metal condition of more than 0·25 mm), and the access holes must be larger to allow for all manufacturing tolerances and for easy soldering. A double-stepped hole as in Figure 1.12, which is not to scale, increases the space taken up by the hole.

Good bonding which does not flow over the pads is not easy to achieve. The ability to make connections to inner layers by small diameter plated-through holes has rendered the access hole board obsolete for many applications. However, there are instances, especially in development projects, in which an access hole board can provide an elegant, cost effective solution to an interconnect problem.

1.2.1.3 SOLDER BUTTONS

One frequently used variant of the access hole board is found on non through-plated flexible circuits, where the dielectric base can be etched away from behind pads on the component side of the flexible to allow access for soldering either to a component lead or to another flexible layer (see Figure 1.13).

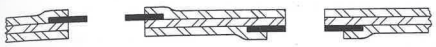


Fig. 1.13 Back-etched flexible circuit.

When employed in flexible circuits such connections are usually called 'solder buttons'. Layer-to-layer connections of this type are generally successful on flexible circuits because the dielectric material which has to be bridged by the solder is normally very much thinner than the rigid laminate and prepreg on an access hole board.

1.2.2 Plated Pillar Boards

Before the plated-through hole became widely accepted as a reliable inter-layer connection, there was considerable interest in plated pillar boards. These boards are similar in many respects to ceramic substrate interconnects, with the board being built up layer by layer by successively plating layers of copper to form the tracking, and depositing layers of dielectric material through which the plated pillars interconnect these layers of tracking (see Figure 1.14).

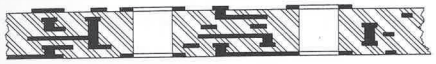


Fig. 1.14 Pillar plated multilayer.

Plated pillar boards have two major advantages over the more conventional through-hole plated boards, the first being that the layer-to-layer pillars can be of any suitable shape and size and need not be any wider than the tracks to which they connect. Their second advantage is that the pillars can be placed at any convenient point without regard to the positions of the pillars between other pairs of layers in the board, and the internal connections are not confined to those between adjacent layers; the pillars can be built up to any required height.

With care during manufacture these advantages combine to offer the capability of forming a very high density interconnect.

Opposed to these advantages are the disadvantages that the boards must be built up layer by layer, making their manufacture slow and expensive (see Section 1.4.1), and that the dielectric is usually relatively thin, which gives the tracks on a plated pillar board a fairly high capacitance and low characteristic impedance.

This electrical disadvantage may be offset by the reduction in track length achievable compared with other forms of interconnects, and the possibility of using a plated pillar board as an alternative to a ceramic substrate should not be overlooked. The fact that the layer-to-layer interconnections can be effected without drilling can make the use of plated pillar boards especially attractive when all the components to be interconnected can be surface mounted.

connections, all superimposed but stepped down the board, then the pattern will require as many layers as there are superimposed Xs (see Figure 1.10). On the other hand, if layer-to-layer connections can be made within the board, this pattern is obtainable on only two layers of nested Vee tracks (see Figure 1.11).

Layer-to-layer interconnections can be achieved on access hole boards by using stepped hole and pad sizes as shown in Figure 1.12. With this method it is not essential to have a small hole right through the board, though some engineers prefer to insert a headed pin into the hole before soldering and to bend the pin so that it contacts the faces of the two pads to be connected. Without such a pin the joint will depend solely on the ability of the solder to bridge the dielectric material.

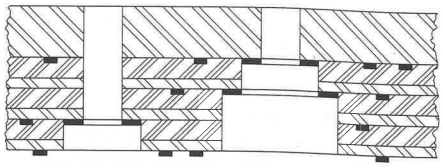


Fig. 1.12 Access hole board with layer-to-layer connection.

1.2.1.2 DISADVANTAGES OF ACCESS HOLE BOARDS

The major disadvantage of an access hole board is the amount of space occupied by the access holes. The land around the component holes must be at least that which would be used on a single-sided board (i.e., a minimum metal condition of more than 0·25 mm), and the access holes must be larger to allow for all manufacturing tolerances and for easy soldering. A double-stepped hole as in Figure 1.12, which is not to scale, increases the space taken up by the hole.

Good bonding which does not flow over the pads is not easy to achieve. The ability to make connections to inner layers by small diameter plated-through holes has rendered the access hole board obsolete for many applications. However, there are instances, especially in development projects, in which an access hole board can provide an elegant, cost effective solution to an interconnect problem.

1.2.1.3 SOLDER BUTTONS

One frequently used variant of the access hole board is found on non through-plated flexible circuits, where the dielectric base can be etched away from behind pads on the component side of the flexible to allow access for soldering either to a component lead or to another flexible layer (see Figure 1.13).

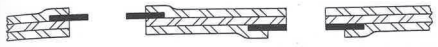


Fig. 1.13 Back-etched flexible circuit.

When employed in flexible circuits such connections are usually called 'solder buttons'. Layer-to-layer connections of this type are generally successful on flexible circuits because the dielectric material which has to be bridged by the solder is normally very much thinner than the rigid laminate and prepreg on an access hole board.

1.2.2 Plated Pillar Boards

Before the plated-through hole became widely accepted as a reliable inter-layer connection, there was considerable interest in plated pillar boards. These boards are similar in many respects to ceramic substrate interconnects, with the board being built up layer by layer by successively plating layers of copper to form the tracking, and depositing layers of dielectric material through which the plated pillars interconnect these layers of tracking (see Figure 1.14).

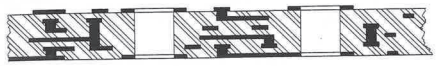


Fig. 1.14 Pillar plated multilayer.

Plated pillar boards have two major advantages over the more conventional through-hole plated boards, the first being that the layer-to-layer pillars can be of any suitable shape and size and need not be any wider than the tracks to which they connect. Their second advantage is that the pillars can be placed at any convenient point without regard to the positions of the pillars between other pairs of layers in the board, and the internal connections are not confined to those between adjacent layers; the pillars can be built up to any required height.

With care during manufacture these advantages combine to offer the capability of forming a very high density interconnect.

Opposed to these advantages are the disadvantages that the boards must be built up layer by layer, making their manufacture slow and expensive (see Section 1.4.1), and that the dielectric is usually relatively thin, which gives the tracks on a plated pillar board a fairly high capacitance and low characteristic impedance.

This electrical disadvantage may be offset by the reduction in track length achievable compared with other forms of interconnects, and the possibility of using a plated pillar board as an alternative to a ceramic substrate should not be overlooked. The fact that the layer-to-layer interconnections can be effected without drilling can make the use of plated pillar boards especially attractive when all the components to be interconnected can be surface mounted.

1.2.3 Staked Boards

The theory of a staked board is that connections can be made to the inner layers of a multilayer stack by metal stakes which are driven down through the board, deforming and making contact with the copper pads on inner layers (see Figure 1.15), thus obviating any need for through-hole plating and for drilling via holes. The stakes can even be driven through an undrilled board.

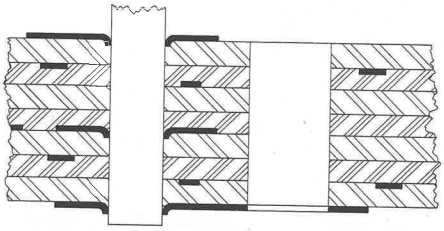


Fig. 1.15 Staked multilayer.

Although many tests have been carried out to demonstrate the reliability of staked connections, field results have not been satisfactory, and in the Author's experience the use of staked connections is best confined to double-sided boards where the joint provided by the deformed metal can be supplemented by soldering. The mass of metal of the stakes can lead to problems with thermal retention during wave or dip soldering of components, which can cause blistering or delamination of multilayer boards. The possibility of using staked connections should not be ignored, but in the Author's opinion they are unlikely to provide a viable alternative to the through-plated hole which has superseded their use on double-sided boards.

1.2.4 Bonded Through-hole Plated Boards

Usually when multilayer boards are mentioned the automatic assumption is that they will be bonded, through-hole plated boards. This widely accepted convention has been followed throughout this book where, unless stated otherwise, the terms 'multilayer', 'multilayer board', or 'multilayer circuit board' do refer to this type.

These boards comprise a number of track-carrying, thin rigid laminates bonded together, with layer-to-layer connections and component mounting holes formed by plated-through holes which are drilled after the boards have been bonded (see Figure 1.16). They may have any number of layers

from 4 to over 20, and can usually be obtained in any size up to about 0·6 metre square.

Boards of this type can offer very powerful interconnects and are relatively simple to manufacture, but for the highest interconnect densities other more sophisticated types of multilayer may prove more cost effective.

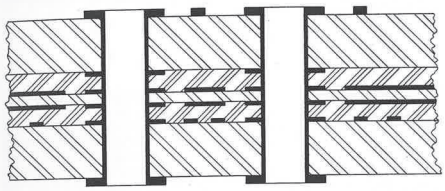


Fig. 1.16 Conventional bonded, through-plated hole multilayer with six layers.

1.2.5 Buried Via-hole Boards

One of the limitations of a bonded PTH board is that a via hole which interconnects tracks on two layers must pass through all the other layers. The minimum size of such a via hole is dictated by the requirement to drill and to plate through the full thickness of the finished board, with the result that a via hole often severely restricts the tracking on all the board layers to which it does not connect.

This disadvantage of the conventional multilayer board can be overcome by using inner rigid laminates which have through-hole plated via holes that interconnect the tracking on their two sides instead of using plain etched laminate. Component mounting holes, and any via holes which must interconnect tracks on non-adjacent layers are drilled and plated after bonding in the usual manner (see Figure 1.17).

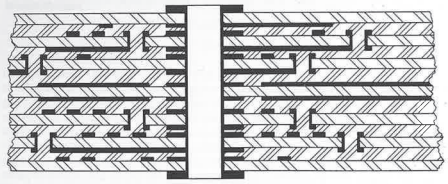


Fig. 1.17 Buried via-hole board.

As on a pillar plated board, via holes between one pair of layers can be superimposed on the same sites as those between other pairs of layers, but the internal connections are restricted to those between pairs of layers which will be formed on the two sides of a rigid inner layer laminate.

The use of buried via holes provides a significant increase in the tracking density compared with that of a conventional multilayer, but the cost involved is also significantly higher.

1.2.6 Sequential Buried Via-hole Boards

1.2.6.1 TWO-LEVEL BURIED VIA HOLES

The buried via-hole concept can be extended by taking pairs of inner layer laminates with first level via holes plated, bonding the pairs together and then drilling and plating further via holes before bonding the whole board together (see Figure 1.18). This allows an interchange of track levels among groups of four inner layers, and can help to solve some tracking problems which may not be easily resolved by a single level of buried via holes. Whilst this technique can simplify the design of the board and permit the implementation of extremely complex interconnects, it is a very expensive way of producing a board and should be used only when all simpler techniques have failed to yield the desired interconnect.

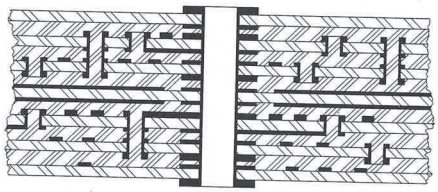


Fig. 1.18 Sequential buried via hole.

1.2.6.2 MULTIPLE LEVEL BURIED VIA HOLES

The sequential technique of plating one level of via holes, bonding, then plating a further level of via holes can be further extended, as in Figure 1.19, to make buried via-hole boards that offer the possibility of linking almost any desired pair of tracks within the board at any point. It must be noted that to avoid an excessive build-up of copper on any layer, successive pairs of layers must be added on alternate sides of the original pair of bonded layers. Because of their sequential manufacture, boards of this type are invariably very expensive, and it is possible that pillar plating techniques may prove more cost effective.

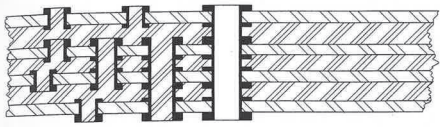


Fig. 1.19 Fully sequential multilayer build.

1.2.7 Double-double Boards

As an alternative to a buried via-hole board, a high density multilayer interconnect can be formed by bonding together a pair of double-sided boards with through-plated via holes. Since power planes and earth planes do not require connections to other layers or to one another, a thin piece of laminate carrying a pair of planes can be sandwiched in the middle of the board (see Figure 1.20). Component mounting holes are drilled and plated after bonding. The essential differences between a double-double and a buried via-hole board are that on the former all holes are exposed on one of the outer faces of the board, and the outer faces carry a substantial percentage of the tracking, whereas on a buried via-hole board the outer layers often carry pads only.

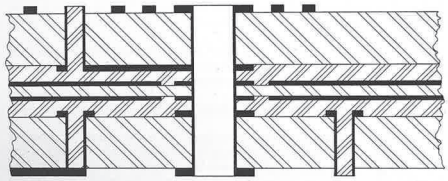


Fig. 1.20 Double-double board.

1.2.8 Double-multilayer Boards

A logical extension of the 'double-double' concept is to use a pair of thin multilayer boards instead of double-sided boards (see Figure 1.21). Because all the via holes in the multilayer portions are filled with resin during the final bonding, pads can be run over the sites of the holes, rendering double-multilayer boards particularly suitable for use with surface-mounted components, which can now be accommodated on both sides of the board. In such cases, it is probably better to include a power and earth plane layer in each of the multilayer halves instead of in the centre of the stack so that

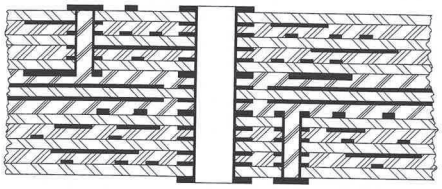


Fig. 1.21 Double multilayer board.

power and earth connections to the surface mounting components can be formed by small, filled via holes directly below the component mounting pads.

1.2.9 Flexi-rigid Boards

So far, all the types of multilayer boards described have been assumed to be constructed of rigid materials. On any type other than pillar plated boards it is possible to replace one or more of the rigid layers with a flexible layer, usually made on a polyimide base. This flexible layer can extend beyond the profile of the rigid area of the board to serve as a connection to a component which is not co-planar with the board face, to act as a hinge between two rigid areas, or to carry electrical signals to a moving part of an equipment.

Of all the possible variants of a 'basic' multilayer board, flexi-rigids have the widest field of application and Chapter 20 will detail the main points to consider in their application, design and manufacture.

1.2.10 Other Forms of Multilayer Interconnects

Sections 1.2.1 to 1.2.9 have described the major range of possible multilayer board forms. There are in existence other minor variants of the types listed, such as fully earth-screened boards which are merely standard bonded PTH boards or buried via-hole types with the earth planes on the surface layers or on the first layer in from the surface. Also, several forms of the basic plated pillar type are available under different names. However, there are also other types of multilayer interconnect which, although not widely used, are worthy of note if only to provide a reasonably complete listing of types.

1.2.10.1 ETCHED AND WELDED INTERCONNECTS

A multilayer back wiring interconnect developed by Elliott Brothers Ltd of London consisted of track layers etched from nickel sheet on an insulating base. Where a connection was required, the base laminate was back-etched away to leave a rectangular aperture, with the end of the nickel

track running over it (Figure 1.22a). The projecting end of the track was then bent over (Figure 1.22b), the layers of the interconnect were stacked on register pins, and the projecting nickel tabs welded together and to the pins of the board sockets.

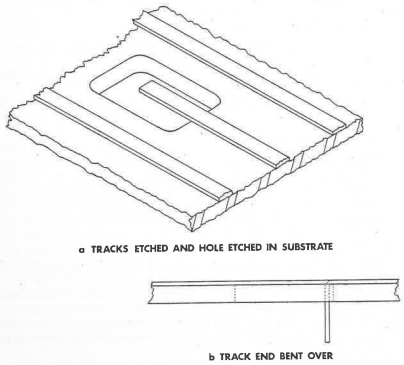


Fig. 1.22 Welded nickel interconnect.

1.2.10.2 THREE DIMENSIONAL PLANAR CO-AXIAL INTERCONNECT

A very high density interconnect with excellent thermal conductivity developed by the Bunker-Ramo Corporation of America consists basically of stacked layers of copper sheet 0·15 mm thick containing small co-axial interconnecting tracks. The tracks are formed by etching into one side of the copper sheet grooves with a step in the centre of their bases (Figure 1.23a). These grooves are filled with plastic, and similar grooves are etched from the other side and filled, thus leaving isolated centre tracks embedded in the plastic (Figure 1.23b). Where a connection is required to another layer, a copper pillar is left, the ends of which are plated with gold to create a small bump (Figure 1.23c, left).

Semiconductors, either as bare chips or in flat packs or chip carriers, are welded into the interconnect in square apertures into which the ends of tracks project as inverse beam leads (Figure 1.23c, right). Some layers of the stack contain the active devices, some are track layers, and alternate layers carry only isolated pillars to provide layer-to-layer connections. The layers of the stack are clamped together, with reliable layer-to-layer contact

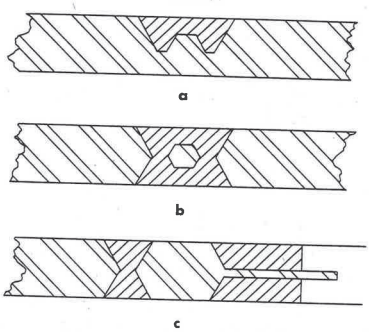


Fig. 1.23 Co-axial interconnect etched and filled in copper sheet.

provided by compression of the gold bumps. Because most of the resulting unit is solid copper, the thermal conductivity from the semiconductor chips is excellent and extremely high packing densities can be realised. The 'pillars only' layers ensure that every track is totally surrounded by copper to give truly co-axial lines.

1.2.10.3 ETCHED AND FILLED BOARDS

Multilayer boards similar to pillar plated types can be formed by etching the conductor pattern from solid copper and filling the spaces left with a dielectric material in a manner similar to the Bunker-Ramo method described in Section 1.2.10.2. Thompson CSF in France patented this type of multilayer in 1975.

1.2.10.4 MOULDED BOARDS

Many of the problems associated with the production of conventional multilayer boards arise in the areas of bonding, drilling and through-hole plating, and some engineers have felt that there ought to be an alternative manufacturing process in which the holes with their metal barrels could be moulded into the boards.

Such boards were developed and manufactured by Mr Pellegrini of Teller Industries Inc., in the USA. The process involves the use of specialised press tooling to form the holes, with track layers electroformed on a plastic matrix. The techniques employed permit the use of holes down to 0·125 mm diameter with 0·05 mm copper walls on boards 2 mm thick, and tracks can

be of any desired cross-section. Some examples had certain tracks 0·25 mm thick. Another advantage of this process was that aluminium heat grids could be moulded into a part of the board area, with the depth of the aluminium being occupied by tracks in other areas. The design of the board could be totally unbalanced, but the final moulding operation resulted in a truly flat end product. Very high interconnection densities are achievable, but the boards are significantly more expensive than those manufactured by the more conventional processes.

1.2.10.5 MULTIWIRE

The multiwire process comprises the laying of insulated wires in a partially cured resin on a base laminate, which may carry etched copper patterns on its surfaces. When all the wires forming the interconnect have been laid and the adhesive has been cured, holes are drilled through the wires and through-plated. The wires are laid by an NC machine and, being insulated, can cross one another on the same layer. Since all wires are laid accurately on pre-determined paths and are rigidly held in position by the cured resin, the performance is predictable and repeatable. However, as the track routing and often the package placement are quite different from that on a conventional multilayer board, multiwire boards cannot be used as performance-proving prototypes for conventional boards. Multiwire is fully described in Chapter 24.

There are other types of 'wired boards' available which are claimed to be usable as prototypes or as substitutes for multilayer boards, but they lack the predictability of ordinary boards or of multiwire and, being in reality little more than formalised 'breadboards' or 'birds-nests', do not receive consideration in this book.

1.3 PRODUCTION OF THROUGH-HOLE PLATED MULTILAYERS

1.3.1 General

The production of bonded, through-hole plated multilayers can be made to sound very simple, but, as the following chapters demonstrate, there are many critical points to be considered and the overall process of producing high quality, cost-effective boards is anything but simple.

As shown in Figure 1.24, the production process can be described in eight or nine basic stages. The purpose of this introductory section is to set the scene so that the subsequent more detailed chapters can be read and considered within their overall context. Sections 1.3.2 to 1.3.7 enlarge on the first simple flow diagram of Figure 1.24, serving as the true introductions to the detail chapters, and Section 1.4 covers possible variants of the basic manufacturing methods.

Some of the processes are used more than once at different stages in manufacture; thus to avoid excessive duplication the detail chapters of the book do not follow precisely the consecutive stages of the operations involved in making a board.

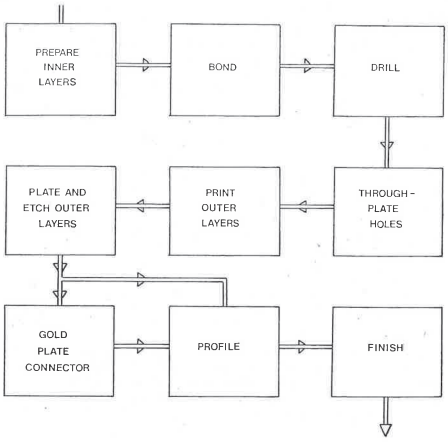


Fig. 1.24 Simplified flow chart of multilayer board manufacture.

1.3.2 Preparation of Inner Layers

1.3.2.1 FILM WORK

1.3.2.1.1 Incoming Photo-tools

Before any work can be started on the actual boards, high quality photo-tools must be available, with all pads on all layers in the most accurate registration possible and with any test coupons required by the customer also in exact registration. Section 23.8 deals briefly with the generation of films and artworks as they will usually be supplied to a board manufacturer. At all times during board manufacture the film set must be regarded as expensive, high quality precision tools. Films must be stored and handled properly, with care being taken not to fold or kink the base material or to scratch the emulsion.

If the design of the board does not include within the board area a panel in which the layer numbers for each layer are visible in sequence, as copper numbers on a clear background, so that the finished board can be inspected

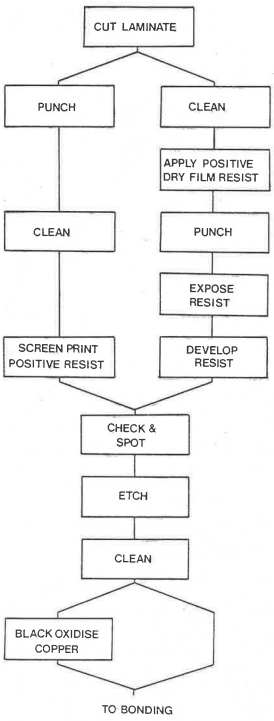


Fig. 1.25 Flow chart of inner layer preparation.

to see that all layers are present and in the correct sequence, then most manufacturers will add such a panel in the border area of the production blank. When a missing-layer numbering sequence, as suggested in Section 23.4, is used it is advisable to add an 'X' before or after the layer number of any films which will be printed on the single-clad laminate. This enables all personnel involved in manufacture to see clearly that the opposite side of the piece of inner layer laminate is intended to be blank and a film has not been mislaid. If the odd layer is missing, the X is put before the even number, but if it is the even layer missing, then the X is put after the odd number.

The photo-tools supplied should be on a stable base material, either glass or 0·007 in. (0·18 mm) film. It can help to prevent costly mistakes occurring if all films in the set carry a clear notation as to their type and orientation.

1.3.2.1.2 Film Types and Labels

It is not unknown for multilayer boards to be made with layers which have copper on areas which should have been clear and *vice versa*, or with layers the wrong way round or on the wrong side of an inner layer laminate. The layer-numbering panel described in Section 1.3.2.1.1 helps to avoid such costly errors, and as a further safeguard the type and orientation of every film should be clearly stated. Films are either positives or negatives, and their orientation is either emulsion side up or emulsion side down. Whilst these statements may appear self-evident, there is a surprising amount of confusion within the industry over these basic definitions.

A film which shows in black the areas which will be metal, solder resist, or legend ink on the finished product is a positive film.

A film which shows as clear the areas which will be metal, solder resist, or legend ink on the finished product is a negative film.

A film which, when laid in register against the product printed or made from it, has its emulsion on the face of the film adjacent to the product is an emulsion side down film.

A film which, when laid in register against the product printed or made from it, has its emulsion on the face of the film away from the product is an emulsion side up film.

The importance of defining whether a film is positive or negative is most easily demonstrated by considering Figure 1.26, in which the colour of the cat can be determined only when one knows the film type. A chess or draughts board is another obvious example of a film which can cause confusion unless the type is known.

Solder resist and earth plane films are sometimes supplied as negatives. In general, these should not cause confusion even if they are not clearly labelled, but some boards contain special screening layers on which there may be no way for a board manufacturer to determine which areas should be copper. Labelling of all films is good practice, and if even the most obvious films have their type clearly defined the risk of errors will be minimised.

THIS IS A POSITIVE OF A BLACK CAT IN A SNOWDRIFT



THIS IS A NEGATIVE OF A WHITE CAT IN A COAL CELLAR

Fig. 1.26 It may not be obvious whether a given film is a positive or a negative.

The orientation of films is a fruitful field for confusion. Sometimes a film set for a multilayer board is generated with films for all layers made as viewed from one side of the board, usually from the components side. This means that films to be printed on the 'upper' side of each piece of laminate will have their emulsion on the reverse side to that of the film for the 'lower' sides. Such a film set is not the best way to provide the essential manufacturing information, as the board manufacturer will have to perform an extra operation on half the film set, either taking a print on reversal film, contact printing through the film, or photographing at one-to-one to bring all the emulsions to the same surface of the films relative to the surfaces on which they are to be printed. Any such operation performed on half the films can introduce small layer-to-layer errors; it is therefore better if a customer supplies a film set with all films of the same type and orientation.

Most quality-conscious film departments will label all films to show the type, orientation, generation, and printing date. Such a label could read:

JOB No. POS. UP 2ND 8/7/84

The generation labelling prevents a second or third generation negative from being used as a master, and the date can provide a warning that a film is old and should be checked for size and overall quality before it is used. The film type, orientation and generation can all be combined into a single letter code, with, for example, A for an off camera emulsion side down negative, B for a plotted emulsion side up positive or a contact print off an A film, and so on, with further letters used for contacts or reversals from the basic masters and for second and third generation copies.

1.3.2.1.3 Step and Repeat

As part of the planning stage, the board manufacturer will decide whether the job will be made as a single board or will be stepped and repeated onto a larger process panel. One factor to be considered in making this decision is the width of the lands available round the holes relative to the panel size. Material movement can render it less economical to make

smallish, high density multilayers as multiples on a large panel than to form them all individually. Once the decision has been taken, the customer's master films will be copied on an accurate step and repeat contact printer to generate master working films.

1.3.2.1.4 Borders

The working films will usually require to have borders added to them. Within the border areas there will need to be markers for punch alignment, any test patterns or coupons required by the manufacturer for process monitoring, and the layer numbers if these are not already included within the finished board area. It is also desirable to include segmented registration and size marks which can be used to verify the size and squareness of any film or piece of etched laminate, and through which microsections can be taken to verify layer-to-layer registration and check for material movement of the bonded panels.

The best type of mark to use for these purposes is either a segmented circular target (Figure 1.27) or a chequered cruciform mark (Figure 1.28). Either of these types can be aligned to a grid or to hair line markers with a very high degree of accuracy, and when one film is reversed over another they show absolute blackness only if the films are exactly the same size, assuming that the markers were properly placed and have not been over-exposed.

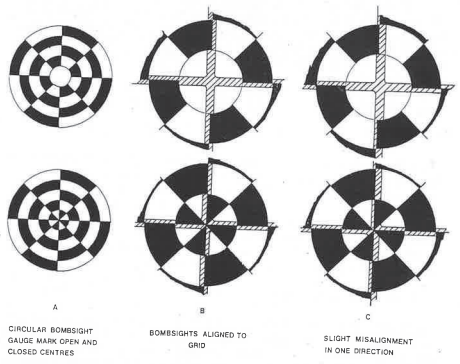


Fig. 1.27 Circular 'bomb-sight' targets for precision films.

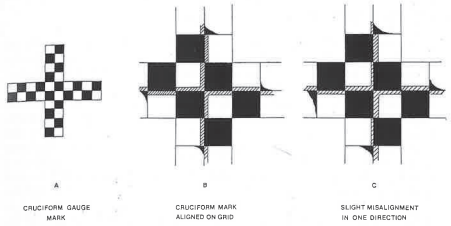


Fig. 1.28 Cruciform targets.

If the films are not exactly to size, one of the pair of targets must be out of alignment, and with the films on a lightbox misalignment of as little as 0·01 mm is immediately evident without the aid of a magnifier. For really fine checks on alignment, an opaque mask with apertures which expose the targets only can be laid over the films on the lightbox. With the room lights dimmed, any misalignment will show very clearly.

If such targets are added, they should be a set distance apart to facilitate measuring against a glass gauge at any stage during manufacture. They should not serve as direct alignment marks for a film punch as this precludes later microsectioning to check for alignment and material movement in the processed panel. They can, however, serve as offset alignment markers.

Added borders for inner layers are usually left as solid copper, broken by gas vent slots, whilst outer layers have a cross-hatched border added to balance plating currents. Board manufacturers will generally use standard borders which fit their process panel sizes and can be added to all working films. The addition will usually be effected by a double photographic exposure, using dowelled negatives to ensure accurate alignment with the board pattern.

1.3.2.1.5 Environmental Control

All work on the master working film set for a multilayer board must be regarded as high precision work, and should be carried out in a strictly controlled environment. Temperature is normally held to within plus or minus one degree of standard, and humidity to 5% or better. The environment in the film area must match that of the photo-mechanical and screen making areas. All incoming films should be allowed time to stabilise to the environmental conditions, usually a minimum of 24 hours. Light boxes used should be indirectly lit so that their working faces remain at the controlled temperature.

1.3.2.1.6 Film Punching

The working film set must be punched with tooling holes to enable all films to be aligned to the laminate blanks for printing, and it will often be necessary to punch films so that they can be aligned for the addition of borders and so forth by double-exposure printing. The punches for this work should be very accurately made and should incorporate facilities for aligning films both to a base grid and to another film. A master alignment punch is required which must match the laminate punch used in the production area, and there will usually also be a darkroom punch which can be used to punch unexposed film so that prints can be made in register on a dowel bar. A guillotine or print trimmer with a marked bed should be used in conjunction with the master alignment punch to guarantee that all films to be punched have sufficient margin to accept the tooling holes, but not so much as to allow the edge of the films to touch the end of the jaws of the punch.

For simplest operation the master alignment punch should be built onto a light box, and should be upwards acting with a stop mechanism so that the punch heads can be left half-retracted. This allows a master film to be retained on the punch pins while leaving space in the jaws of the punch for a second film to be slid in over that retained on the pins. The jaws should be just wide enough to allow two films, and only two, to enter. The second film can be adjusted positionally to align with the master held on the pins; then when it is aligned it can be punched in perfect register (see Figure 1.29).

In addition to the required tooling holes or slots, the master punch should be equipped with an extra pair of holes, preferably arranged such that they can be used either upward or downward acting, with removable punch pins and a pair of well-fitting polished dowel pins having generously tapered ends. To avoid any risk of confusion, these two punches should be of a different diameter from the normal tooling holes. Their position relative to the tooling set is not critical for the normal action of the punch, but to facilitate possible future manufacture of any further tooling it is advisable to locate them at set distances from the tooling holes. These two extra holes are used to punch re-locating holes in a film, so that, with the aid of the dowel pins, the film can later be re-located on the punch and guided back onto the main tooling hole pins without risk of damage.

The glass top of the lightbox may be provided with datum lines or a grid which can be accurately aligned to the tooling punches; alternatively, a separate precision grid film punched with relocation holes and with the holes accurately aligned to the grid may be used.

To punch a film set for a board, the grid film is located on the punch by the relocation pins, and the tooling punch pins are raised, the feel of the punch warning the operator if there is any risk of the punch pins damaging the grid film as they enter their holes. The pins are then drawn back to their half retracted position which leaves them still in the grid sheet, the relocation pins are taken out, and the master film of the set to be punched is aligned accurately over the grid. On a punch fitted with a glass grid pre-aligned to the tooling punch, the master film is simply aligned to the grid. Tooling holes are then punched in the master film and, while the punch is held with its pins still through the master film, relocation holes are punched.

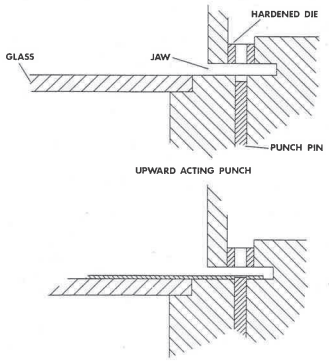


Fig. 1.29 Sections through jaws of a precision film punch.

The grid and the master film are then removed from the punch, the master alone is re-located and the punch pins are entered and held in their partially retracted position.

If the master film is a positive, a negative film for each layer of the board in turn is then aligned over it and punched with its tooling holes. The positive to negative alignment is quite easy, especially if all films carry segmented circle or chequered cruciform marks, and the end result should be a set of punched films all with the holes positioned to a tolerance of less than 0·025 mm. The alignment of the negatives for each layer over the master positive gives an excellent opportunity for a final verification of the layer-to-layer registration of all pads on the board and on any test coupons.

It should be noted that on a punch of the type described all holes are punched with the punch pin working into a hardened steel die. Some cheaper forms of punches use downward working pins, which means that the master film forms the die against which the working films are punched. This is not the best way to punch holes, or to ensure that the master film remains re-usable. Similarly, a punch should never be used to punch more than one piece of film at a time, as when this occurs the first film has to form the punch for the second, with the second film forming the die for the first.

The actual tooling-hole pattern punched in the films may have to be chosen to suit board manufacturing considerations, but whatever pattern is

adopted should always follow kinematic principles. The simplest and a very effective pattern is a central round hole with slots on either side of it (see Figure 1.30). Reversal of a film in use can be prevented by locating the slots to the right of the master hole 2, 4, 6 and 8 inches (50, 100, 150, 200 mm) from the hole and those to the left 3, 5, 7 and 9 inches (75, 125, 175, 225 mm). In use, the centre of the board pattern, or the centre of the most critical area of the board, is aligned with the round hole. Such a single row pattern accommodates material movement along the board, but it cannot cope with movement across the board.

Fig. 1.30 Simple irreversible punching pattern of one hole and multiple slots.

To accommodate fully all linear material movements while using the centre of the board as a datum, an 'all round' system of three slots must be employed (Figure 1.31). This type of system can be very effective when only one or two standard blank sizes are to be handled, but the alignment punch required involves complex precision engineering. Most users of such a three slot system seem to prefer to add a kinematically redundant slot on the fourth side, as this can help to minimise risks of film distortion.

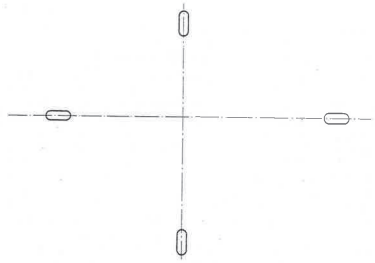


Fig. 1.31 Three or four slot location giving a centre datum.

A three slot system can be made to accommodate films and blanks of any size by angling the slots at 45 degrees and placing them on two adjacent sides of the blank (Figure 1.32). This system must not be employed with more than three pins, and to avoid the risk of an operator trying to use five or more pins on a large blank it is advisable to fit the punch with selectable heads so that only the pattern of three holes which give a working centre near the board centre can be used. Such a system is effective only when all the work handled is more or less square.

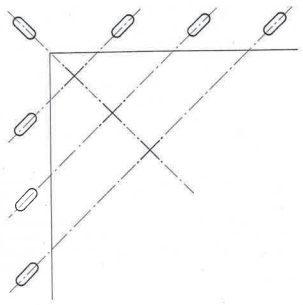


Fig. 1.32 Universal 3-slot system for centre location.

1.3.2.2 PLANNING

Planning of multilayer board manufacture involves all the work common to the planning of any production process, such as the raising of route cards and job tickets, ensuring that plant capacity will be available at the right time, and that all necessary materials are in stock or are ordered for delivery at the appropriate time.

It should also include a capability assessment check to ensure that what the customer has ordered can in fact be made and that the filmwork, documentation supplied and tolerances specified are suitable.

The overall thickness and the build of the board must be correct, and the appropriate grade of prepreg material should be selected to ensure adequate resin flow. The thickness of copper tracks on the inner layers will have to be taken into account in determining the grade, thickness and number of sheets of prepreg to be used.

Stepping and repeating of patterns has already been mentioned. The planning department will also have to ensure that the board is correctly made relative to the warp and weft of the laminate to guarantee maximum stability.

1.3.2.3 TAPE MAKING

Once working films are available, control tapes for NC drills and routers can be produced. Drill tapes may be computer generated from a data base supplied by the customer, or they may be digitised from the master film of the set. In this case, it is easiest if a 'pads only' film can be used. Finished drill tapes are usually verified by drilling a piece of laminate and comparing

the hole pattern with a negative of the master film of the set. Router tapes should be similarly checked by profiling a piece of material and checking the dimensions against the outline drawing.

Although the use of NC or CNC drills and routers is almost universal in multilayer board manufacture, it must not be forgotten that when properly set and operated the older single or multiple head jig drills can yield comparable accuracies, and there could be circumstances within a given facility which would justify the use of jig drilling on small batches.

1.3.2.4 LAMINATE PREPARATION

With all the planning completed, the necessary sheets of laminate and prepreg are cut, usually by guillotine, taking care that the copper thicknesses and orientation of warp and weft are correct. If the job is to have its inner layers screen printed, the sheets of laminate are punched before they are cleaned, but if photo-mechanical printing is used the laminates are cleaned and the resist is applied before the tooling holes are punched. The cleaning of laminates is discussed in detail in later sections, particularly in Chapter 14.

1.3.2.5 INNER LAYER PRINTING AND ETCHING

The cleaned and punched laminate sheets may have the patterns for the inner layers screen printed (see Chapter 6) or they may be photo-mechanically printed (see Chapters 4 and 5). Whichever process is used, the patterns must be accurately aligned to the tooling holes, and the printing must be of high quality to give a usable pattern with the minimum of subsequent re-touching. With improvements in resists, printing techniques, and environmental and dust control the need for inspection and re-work at this stage can be minimised, but it is unlikely that it can ever be totally eliminated. All inner layer patterns are then etched (see Chapter 7), leaving the copper untouched on what will be the outer layers of the finished boards.

1.3.2.6 PREPARATION FOR BONDING

After etching, the resist is stripped and the etched copper patterns are cleaned and oxidised or otherwise treated ready for bonding. The need for oxidation and the preparation necessary are discussed in Chapter 8.

1.3.3 Bonding

The laminates with the etched and prepared inner layer patterns are stacked with the appropriate numbers of prepreg sheets on pins in the press plates, and the stack is bonded under heat and pressure which causes the partially cured resin in the prepreg sheets to flow and then to polymerise to bond the board. Obviously, the preparation for bonding and the actual bonding are key operations in the production of multilayer boards, and these processes are fully described in Chapters 8 and 9.

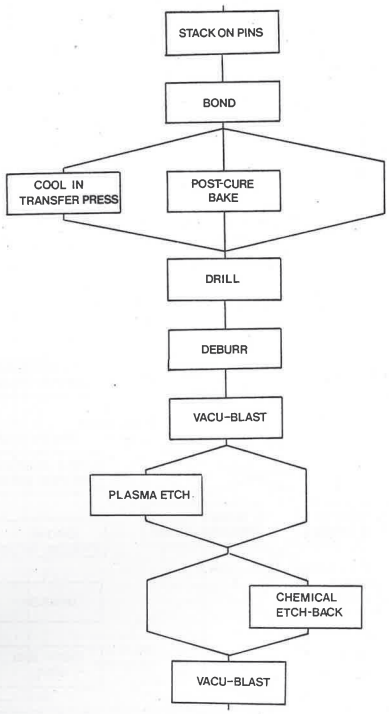
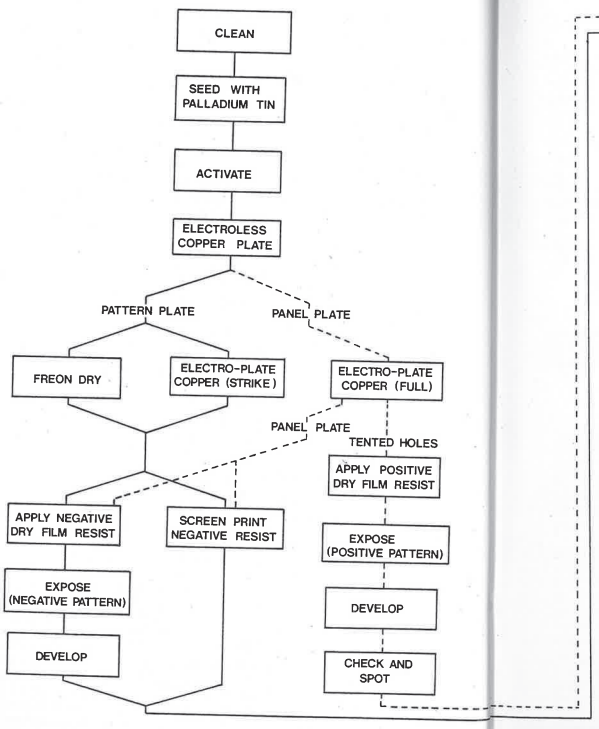


Fig. 1.33 Flow chart of bonding, drilling and pre-plating stages.



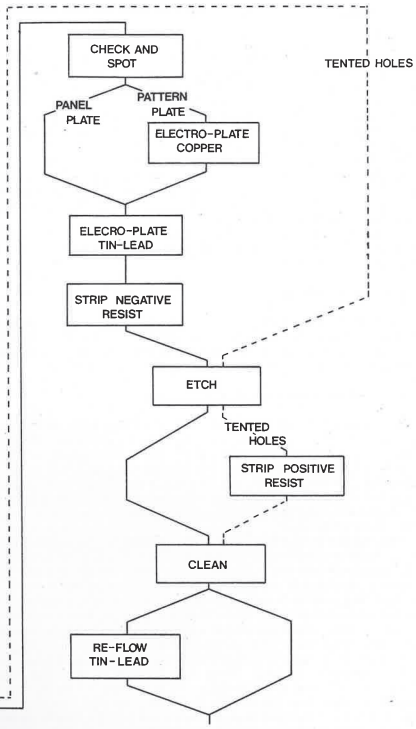


Fig. 1.34 Flow chart of electroless and electro-plating stages.

1.3.4 Drilling

After bonding, the multilayer board follows substantially the same processing as a double-sided board, with drilling as the next major operation (see Chapter 10). If there were no material movement, the bonded board could simply be mounted on the drilling machine on pins through the tooling holes, with the certainty that the holes would pass through the pads in the inner layers. Unfortunately, material movement cannot be ignored (see Chapter 17), and as a rule some adjustments will have to be made. Very often the outer layer copper will be etched off the first blank to be drilled so that the registration of the drilled holes in the inner layer pads can be verified visually. The blank pattern may include special areas in the border in which the outer layer copper can be locally ground off to expose pads, serving as a visual check that material movement has not been excessive and that the drilling is adequately registered.

Most multilayer boards are drilled on multi-spindle NC machines, but jig drilling or even optical drilling on an inverted spindle machine with a film pinned over the board can also be used.

1.3.5 Smear Removal

Drilling and normal de-burring are followed by processes peculiar to multilayer boards for the removal of any resin which may have been smeared over the edges of what should be exposed copper on the inner layers. These processes are described in Chapter 11.

1.3.6 Through-hole Plating and Outer Layers

The cleaned holes are then plated through and the outer layer patterns printed, plated and etched in the same way as for a double-sided board. The boards may be panel or pattern plated, and screen printing or photo-mechanical printing can be used. The flow chart in Figure 1.34 shows the main steps of these processes, and Chapters 12 and 13 deal with the plating processes. Probably the most widely employed process is photo-mechanical printing with tin-lead plating used as the etch resist. Printing and etching are covered by Chapters 4-7.

One problem which can affect the printing and etching of multilayers but is not found on double-sided boards is that, after bonding, the outer layers of a multilayer may not be flat enough to enable the printing of a high precision pattern with good definition. This out-of-flatness is known as 'telegraphing', and is caused by thin outer layers tending to conform to the etched patterns of the inner layers. It is most acute on boards with a large number of layers etched on very thin laminates, and can be aggravated if the designer is not aware of its occurrence and allows some areas of the board to be blank on the majority of layers whilst immediately adjacent areas have copper present on all layers. When there is a risk of such telegraphing occurring, it is sound policy to design the outer layers of the board with pads only. Then, provided all inner layers are made with all pads present round all holes, the bonded blank will be flat and have the same thickness in all areas where the outer layers are to be printed.

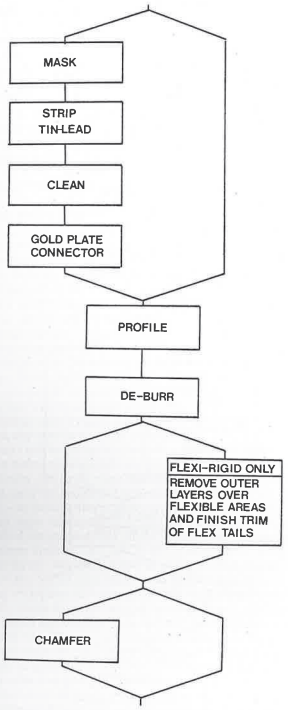


Fig. 1.35 Flow chart of edge connector plating and profiling.

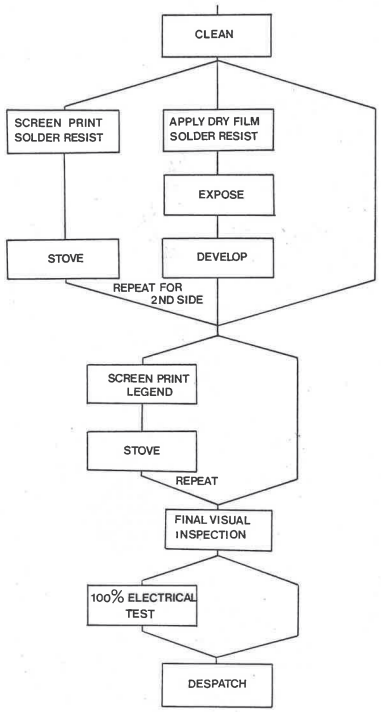


Fig. 1.36 Flow chart of finishing stages.

1.3.7 Finishing

The finishing processes necessary will vary from board to board. Typically, they could include re-flowing the tin-lead plating of the outer layers, masking, stripping the tin-lead from the fingers and gold plating an edge connector (see Figure 1.35), profiling, which is usually performed on an NC router (see Chapter 10) but can also be carried out with a profile template on a freehand router or on a saw, chamfering as necessary, applying solder resist or legend as required (see Figure 1.36 and Chapter 6), a final visual inspection, and possibly a 100% electrical test for short and open circuits (see Chapter 16).

1.4 PRODUCTION VARIANTS

1.4.1 Pillar Plated Boards

The production of pillar plated multilayer boards involves completely different methods from bonded, through-hole plated boards, although the processes of plating, etching and printing are basically similar. As there is no single 'standard process' for producing pillar plated boards, all that will be attempted here is to provide outlines of the various options.

Boards on which all tracks contact one surface can be built up by electroplating, starting with a non-adherent plating on a flat metal base, usually of stainless steel. However, a board may be required to carry surface mounted components on both faces, so that just over half the tracks will contact one face. Such boards are normally built up by high build electroless deposition of copper, and can be built directly on a plastic substrate which will form a part of the finished product. This substrate can be a thin double-sided printed circuit, with through-plated holes, on which pillar plated circuitry is built up on each side.

Track layers can be pattern plated, or a panel plating process can be used in which the entire exposed face is plated with copper to a greater thickness than is actually required, then the surface is mechanically cleaned and dressed flat, and the required track pattern is printed and etched.

Similarly, the layer-to-layer pillars may be formed by differential etching from a thick layer of deposited copper or by pattern plating with either a resist to define the shape of the pillars or with their shapes defined by holes in the dielectric layer.

Dielectric layers may be applied as unreinforced plastic coatings screen printed onto the board, applied around preformed pillars without a screen by squeegee, or by curtain coating. Woven cloth reinforced dielectric layers with partially cured B-stage resin can be bonded over preformed pillars. The use of conformal sheets in the press ensures that the prepreg layer is bonded down around the pillars, and when the resin has cured the resin and reinforcement over the tops of the pillars can be ground off to expose the pillars, ready to plate the next track layer. If the pillars are pattern plated into holes in the dielectric, the dielectric may be in the form of pre-drilled sheets of reinforced laminate bonded with a low flow or 'no-flow' prepreg. The 'pre-drilling' of holes can actually consist of drilling, punching or laser cutting to produce very small or non-circular holes.

Some of the production techniques used to produce pillar plated boards by differential etching from thick layers of deposited copper bear a stronger similarity to the techniques for producing etched and filled interconnects than they do to conventional multilayer manufacture.

When pillar plated boards are being built up, it is usually necessary to dress the surface flat and parallel to the base at each layer, especially when the dielectric material is applied around preformed pillars. The pillars are plated to over their required height, and after the plastic has been applied and cured the whole surface can be ground flat to expose the tops of all the pillars.

When the pillars are formed by plating into holes in the dielectric layer, it is customary to continue the plating of the pillars and of the next layer of copper until the copper layer exceeds the necessary thickness. This can then be dressed off to leave a flat surface on which the etch resist can be printed and to which, after etching and cleaning off the resist, the next dielectric layer can be bonded.

Layer-to-layer separations on pillar plated boards are usually small, between 0·1 and 0·2 mm. If significantly larger separations are required, it may be necessary to duplicate or even triplicate the processes of forming the pillars and adding dielectric material.

1.4.2 Buried Via Hole Boards

The most obvious difference between the manufacture of a buried via hole multilayer and a conventional bonded and through-hole plated board is that the buried via holes must be drilled and through-plated prior to bonding, so the inner layer preparation flow chart of Figure 1.25 will have to be replaced with the significantly more complicated chart of Figure 1.37 or some similar diagram.

1.4.2.1 INNER LAYER PLATING

The processing up to the printing stage is that of a conventional double-sided board except that the laminate is generally much thinner and may require special support, and the holes will usually be considerably smaller. Figure 1.37 shows a commonly used, positive resist, panel plating process in which the resist, which must be a dry film type, tents over the ends of the holes to protect the plated barrels during etching.

It is possible to follow the normal tin-lead plated pattern plating process for double-sided boards on buried hole inner layers, ending with a tin-lead stripping process as the first step in preparing the laminate for bonding. The tin-lead cannot be left on the inner layers as the heat and pressure of bonding would melt it and cause it to be squeezed out from the faces of the tracks, usually to form multiple short circuits between closely spaced tracks.

Possible variants of the pattern plating process are the use of nickel (or another metal which will not melt and flow during bonding) as the overplate and etch resist, or the use of very thin foil cladding on the laminates coupled with generous plating of the tracks so that a brief unmasked etch will remove all unwanted copper without reducing the thickness of tracks and hole walls to an unacceptable level.

INNER LAYER FOR BURIED HOLE BOARD

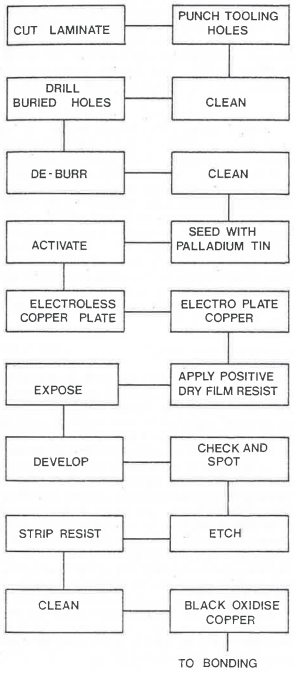


Fig. 1.37 Flow chart of inner layer preparation for buried via-hole boards.

1.4.2.2 LAY UP AND BONDING

Buried via-hole boards may require the use of prepreg with a higher resin content than that used for conventional multilayers to ensure that all air is driven out from the holes and that the holes are totally filled with resin. Whether or not this is necessary will depend on the thickness of the inner layer laminate and on the diameter and number of the buried holes.

As the diameter of the holes will usually be greater than or equal to the laminate thickness, resin flow during bonding should not pose any problems.

In some experiments to establish the nature of the resin flow during bonding, G. C. Wilson, of Ferranti Computer Systems Ltd., used different coloured prepreg sheets on either side of a buried hole laminate.

On microsectioning after bonding it was found that in the centre areas the two resins met in the middle of the holes, but in other areas of the board one or other of the resins had flowed right through to the far side of the inner layer. The distribution of this flow appeared to be random, with no detectable bias towards an 'upwards' or 'downwards' flow. The use of very high resin content prepregs and a malleable glass fabric was found to give good filling of the buried via holes, but tended to produce high side pressures on the inner layer tracks which caused some 'swimming' or sideways movement during bonding. The gel time and bonding cycle should be selected to give a relatively long pre-cure period to allow the resin ample time to fill all holes and expel the air without significant sideways pressure being developed.

1.4.2.3 PRE-FILLING OF BURIED HOLES

When relatively thick inner layers must be used, or there is a large number of buried holes or the holes are fairly large, it may be desirable to fill the holes with resin before bonding the board. A suitable epoxy resin can be squeegeed into the holes, and after the resin has been cured the surfaces of the inner layer are linished, ground or lapped to remove all traces of resin from the copper surfaces. Since the through-plating of the holes will usually have increased the thickness of the copper to the equivalent of 2 oz foil, the inevitable removal of some of the copper can be tolerated. This pre-filling of the holes may also fill the spaces between the tracks on the surfaces of the inner layer, thus eliminating any risk of the inner layer track patterns 'telegraphing' through to the outer layers during final bonding.

On conventional multilayers it is often possible to identify a bonded blank by looking at the inner layer patterns which have telegraphed through to the outer surfaces. Any such patterning of the outer layers can make it difficult to print really fine patterns for surface tracking.

Thus the filling of the holes not only enables a lower flow prepreg to be used, but can also help in allowing the printing of fine patterned outer layers and, in addition, it should eliminate any swimming of the inner layer tracks. Against these advantages must be set the inevitable extra costs of the resin application, curing, and surface cleaning.

1.4.3 Sequential Buried Via Holes

The manufacture of sequential buried via-hole boards differs from that of ordinary buried via-hole boards in that, after through-plating the via holes, only the face of the laminate which will be bonded in the next stage is etched; the face which will be on the outside of the pair or group of laminates after bonding will be masked off and left unetched. If both faces are etched off after through-plating the first level of via holes, there will be no electrical path to conduct the current for the next through-plating stage, and a high build electroless deposition will have to be used.

When electroplating is employed for all levels of via holes, those layers which are on the outside of the stack after each bonding operation will be plated with a further 0·03 to 0·04 mm of copper. On a fully sequential buried via-hole board of six or more inner layers, each individual piece of laminate will have one face etched after a first plating operation; one of the first pair of laminates bonded and one outer layer will have their second faces etched after two plating operations, and all other layers will have been plated three times prior to etching.

This build-up of copper can cause problems in etching unless the board has been carefully designed, with coarser patterns on the layers which must be etched from the thicker copper. If this has not been accomplished, it may be necessary to linish or grind off the excess copper on the outer faces after each bonding operation.

Some cleaning of the outer layers of the bonded stack will be required to remove any resin which has flowed through the via holes.

1.4.4 Doubled Boards

Double-double and double multilayer boards suffer the same build-up of copper on their outer layers as the sequential buried via-hole types. Since doubled boards are likely to be used for surface mounting components with pads over the ends of the via holes, it is important that during the final bonding the resin from the prepreg flows right through the via holes and that the surfaces are thoroughly cleaned off to remove any excess resin.

It has been suggested that doubled boards should be bonded with only a thin release film over a metal plate against the board face, so that the resin will fill the holes but will not spread over the outer faces, but another manufacturer has suggested that the use of several layers of Kraft paper which does allow the displaced resin to spread over the outer surface provides a more reliable hole filling. Satisfactory boards have been produced by both methods, but the difference of opinion between two manufacturers highlights the importance of matching the resin flow to the press parameters for the final bonding.

1.4.5 Flexi-rigids

The basic processes in the manufacture of flexi-rigid boards are the same as those for other types of multilayers, but the necessity of ensuring that

flexible portions are not bonded to rigid outer laminates and of profiling portions prior to bonding introduces special steps unique to the processing of flexi-rigids. These steps are described in Section 20.5. On flexi-rigids with through-plated holes in flexible tails it can be necessary to plate the pads on the tails with a solderable finish prior to bonding, whilst leaving the track patterns with a raw copper finish. This calls for the use of selective plating or etching techniques, and a typical flow chart for the flexible inner layer of a flexi-rigid is shown in Figure 1.38.

1.4.6 Mass Lamination

Mass lamination is basically a service offered to the industry by the laminate supply houses. Instead of their supplying to the board fabricators sheets of copper-clad laminate and prepreg, the laminate suppliers prepare a thin inner layer core for the board, with the etched patterns and tooling holes, and they then laminate this thin core with the appropriate number of prepreg sheets and outer copper foils to supply a large sheet of laminate containing the requisite inner layer patterns ready for the board fabricator to drill and complete in the normal manner. At the time of writing, the process is mainly confined to 4 layer boards, although there are some laminate suppliers offering 6 layer mass lamination, and there would appear to be no technical reasons why the service should not be extended to cover a greater number of layers.

The real difference between a mass laminated board and a conventionally made board lies more in who performs the various stages, and in the point in the cycle at which the B-stage resin is cured, than in any detectable difference in the finished product.

One practical effect of the process is that it enables the smaller board manufacturers whose business would not justify their owning a sophisticated press to enter the multilayer market.

1.4.7 Semi-additive Multilayers

When the capacitance between the outer layers of the board and the next adjacent inner layer can be high, the outer layers can be added by screening or curtain coating a layer of curable dielectric onto an etched inner core (for a 4 layer board), drilling, and then using a high build electroless copper to plate through the holes and to pattern plate the outer layer tracks and pads.

Such additive outer layers can also be applied to bonded stacks of four, six or more layers and, where overall thickness of a board must be kept down, can provide a very useful means of adding pads-only outer layer patterns to avoid any risks of soldering problems. The technique is especially useful on fully screened boards which would normally require their earth planes on the outer faces. Figures 1.39d and 1.39h show semi-additive builds.

1.4.8 Bonding of Heat Ladders

Heat ladders, usually made of copper or aluminium of up to about 1·0 mm thickness, are normally bonded to the components side of an

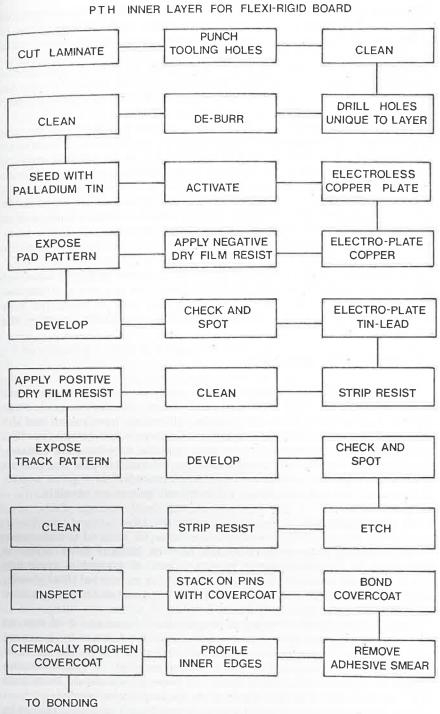


Fig. 1.38 Flow chart of preparation of flexible inner layers with plated-through holes.

otherwise completed board. Since the ladders often overhang the ends of the boards, they cannot be bonded in place before the board is profiled.

The heat ladders may be milled, punched or etched to shape, with clearance holes for all component legs. Copper ladders are usually nickel plated prior to bonding, and aluminium, which must be a soft, ductile aluminium if warping of the finished assembly is to be avoided, should be anodised. Both the board and the heat ladder should be provided with holes for locating pins, preferably in two opposite corners.

The ladder is bonded to the board with two sheets of 'low flow' or 'no flow' prepreg. These sheets are cut to match the profile of the heat ladder, leaving about 0·1 mm plus any tolerances over size all round to ensure that there can be no possibility of the metal ladder touching any live tracks or pads on the board face. The prepreg will usually be shaped with a 'one shot' punch tool, although for small quantity runs it may be hand cut with a knife against a thin steel template. Where the heat ladder consists of long parallel strips to lie beneath the bodies of rows of DIPs, the prepreg sheets may have narrow joining bridges left at intervals between the strips. These bridges are removed with a sharp knife immediately before bonding.

The bonding is carried out at a fairly low pressure, generally at about 150°C, giving a slow melt and cure which minimises any risk of distortion of the finished assembly. The steel tooling pins used to locate the heat ladder and the prepreg to the board are knocked out after the assembly has cooled.

1.5 BUILD AND THICKNESS

The design of a multilayer board may involve strict electrical requirements for careful control over dielectric thicknesses and track widths, in which case the board designer will usually have established the build and method of manufacture at an early stage in the design. Equally, however, the requirement for inter-layer spacing may be quite relaxed, possibly of the form that certain spacings may be minimised (or maximised) to conform with the most economic manufacture within a given overall thickness. In cases like the latter, a number of options are possible.

1.5.1 Prepreg Layers

The minimum separation permitted between bonded inner layers is determined by the need to use at least two sheets of prepreg to avoid any risk of a pinhole or defect in a single sheet causing an internal short circuit, and by the need for prepreg to be of sufficient thickness and resin content to guarantee adequate resin flow during bonding.

The normally used thicknesses of prepreg are 0·1 mm and 0·06 mm, so that with the minimum of two sheets separations of 0·2 mm or 0·12 mm are achievable. The thinner prepreg sheets are often regarded as non-standard and are not so widely used as the 0·1 mm material. A rule of thumb sometimes applied is that the thickness of prepreg used should be at least twice the thickness of copper tracking on the faces to be bonded.

Thus two opposing faces with tracking etched from 1 oz copper should be separated by at least 0·14 mm of prepreg; a single face with tracks plated up

to the equivalent of 2 oz, such as a buried via-hole inner layer, to be bonded to a piece of unclad laminate, also requires 0·14 mm of prepreg, whilst a pair of such plated layers bonded together requires 0·28 mm of prepreg.

Obviously the mixing of 0·06 mm and 0·1 mm prepreg in one bond is undesirable, if only since it complicates the specifying of the board and the stacking, so the simplest practical bonding usable under the rule of thumb between two track layers is two layers of 0·1 mm.

Specification of the bonding layers to be used should also take into account the density of the etched patterns on the faces to be bonded. A track pattern which has only a few narrow tracks will give rise to very high bonding pressures over the tracks; the metal pattern will thus be well embedded into the prepreg sheets and the resin will flow easily to bond the areas between the tracks. On the other hand, an earth or power plane layer will be mainly metal, and the local pressure on the prepreg will consequently be very much lower. It is not uncommon for an extra sheet of prepreg to be called up in bonds between opposing planes over the number specified between track layers.

The presence of pre-drilled holes in inner layers, as in a buried via-hole or flexi-rigid board, will also influence the number of sheets of prepreg to be utilised.

1.5.2 Board Thickness

The overall thickness of a multilayer board is the sum of the thicknesses of the rigid inner layer laminates, plus the prepreg layers, less a small allowance for resin flow which can usually be ignored, plus an allowance for the copper on the inner layer faces. It is this last term of the sum which can cause uncertainties.

If the inner layers carry only isolated tracks and the prepreg layers are fairly thick, then the copper will embed itself into the prepreg and can be ignored, whereas earth and power planes cannot become embedded in the prepreg and their thickness must be included in the overall sum.

Between these two obvious limit cases there is a wide zone of uncertainty in which the thickness and density of the track layers and the nature of the layer which opposes them on the opposite side of the bond must be considered, together with the resin content of the prepreg and the press parameters to be used. Experience and practical testing supply the best, or probably the only, guides in these areas of uncertainty. A good designer or planner should know whether or not to include the copper thickness of a given layer.

In practical manufacture of boards on which the overall thickness needs to be tightly controlled, it may be necessary to do a trial pressing to determine the optimum build, since the sum of the tolerances on the thickness of the individual inner layer laminates can exceed the total tolerance allowable on the final thickness.

Fortunately, the most common need for a tolerance on the overall thickness of a multilayer board lies in the edge connector area, where the inner layers will probably carry earth and power planes to maintain line impedances into the connector areas, but the inner track layers will usually carry very few, if any, tracks.

1.5.3 Practical Builds

1.5.3.1 FOUR LAYER BOARDS

For a four layer board 1·6 mm thick, the classic build comprises two pieces of rigid laminate each about 0·7 mm thick, each with one inner layer pattern etched on one face, bonded together with 0·2 mm of prepreg (Figure 1.39a).

This build offers the advantages of simplicity and the highest practical dimensional stability of the inner layers, which are etched on relatively thick fully cured laminate.

If the separation of 0·2 mm of prepreg between the inner layers is considered excessive, the inner layers can be etched either side of a piece of 0·1 mm clad laminate, which is then included in the centre of the build.

When mass lamination is used, the build will be as shown in Figure 1.39b, with probably seven layers of 0·1 mm prepreg and an outer copper foil each side. Such a build can equally well be used by a board manufacturer, but it is probable that some board manufacturers would prefer to use two layers of prepreg each side of the centre core with rigid outer layers of single-sided laminate 0·5 mm thick (Figure 1.39c).

This type of build involves the stacking of three pieces of rigid laminate and four sheets of prepreg against the two rigid layers and two prepreg sheets of the simplest build and, since the inner layers are etched on thin laminate, problems of dimensional instability may arise. It has the advantage that both sides of the inner laminate are etched, and the single-sided outer laminates will not require any etching. The material for the outer layers should be purchased as 'split laminate' or 'texture finished' material, which needs no special treatment prior to bonding. Ordinary single-sided laminate material has a glazed surface to which it may be difficult to bond satisfactorily.

Figure 1.39d shows the build of a semi-additive four layer board (see Section 1.4.7). It should be noted that in the diagram the layers shown as prepreg are actually an unreinforced dielectric, screen printed or curtain coated onto the blank. The outer track patterns are formed directly by the high build electroless copper.

If the layer numbering used takes into account the side of the piece of laminate on which the layer is printed, the four layer builds shown in b, c and d of Figure 1.39 should be numbered as 6 layers with layers 2 and 5 missing.

The importance of determining the build before the filmwork is completed and of ensuring that the films for the inner layers have their emulsion on the correct side cannot be overstressed. Precision boards cannot be produced if attempts are made to print the track patterns with the film emulsion out of contact with the resist on the laminate face or with the screen resist.

1.5.3.2 HIGHER BUILD BOARDS

Figures 1.39e – h show how a six layer board can be built from three rigid laminates with two bonded zones; from two rigid laminates, one bonded

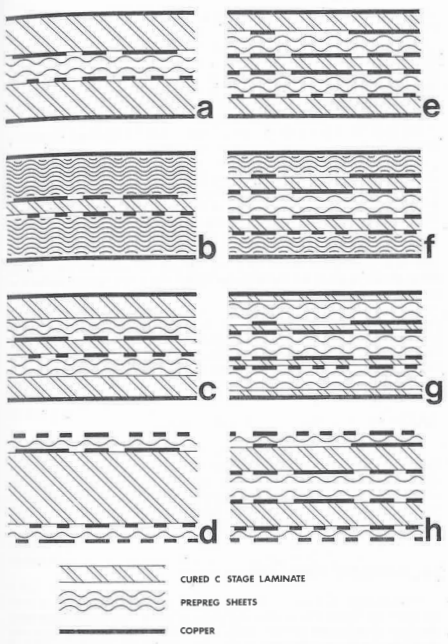


Fig. 1.39 Four and six layer builds (not to scale).

zone and prepreg to outer copper foils; from two double-sided and two single-sided rigid laminates with three bonded zones; or from two rigid laminates, one bonded zone, and additive outer layers. It should be noted that 1.39h with the additive outer layers is a two stage build on which the

outer dielectric layers are applied after the inner laminates have been bonded.

The choice of build will normally be determined by the dielectric thickness required; the copper thicknesses on the bonded faces; the dimensional stability of the inner layer laminates, bearing in mind that laminates over 0·3 mm thick usually yield less instability than thinner laminates; the properties of the prepreg to be used; the finished board thickness required, and the bonding parameters to be used.

The four basic builds of Figures 1.39e–h can be extended by adding extra pairs of layers to cover the required number.

Above eight or ten layers, the inner laminates will usually be about the same thickness as the prepreg unless the board is fairly thick. With 0·2 mm of prepreg and 0·2 mm laminates, eight layers are the maximum that can be used on a 1·6 mm thickness, rising to 14 layers if 0·1 mm rigid laminates are used with two sheets of 0·06 mm prepreg per bond, and possibly two sheets of 0·1 mm prepreg between power and earth planes.

Four and six layer boards tend to be fairly well balanced in their build, with clear pairings of layers on either side of the centre of the board, although all too often it is found that a designer has neglected to balance the patterns on the individual layers.

Above six layers the build may not be so well balanced, and it may be desirable to have one or more 'missing layers' in the stack, with some layers printed and etched on single-sided laminate, in order to keep the patterns on the inner layer laminates fairly well balanced. There can be no hard and fast rules to cover such cases: the board designer and manufacturer must collaborate very closely to achieve the best result possible.

1.5.3.3 BURIED VIA-HOLE AND FLEXI-RIGID BUILDS

Buried via-hole boards and doubled board types follow the same basic builds as plain multilayers, with buried via-hole boards usually having single clad or foil outer layers as in Figures 1.39b, 1.39c, 1.39f or 1.39g. Flexi-rigid boards introduce complications in the need to bond some areas and not others, and in the cover coats necessary for flexible portions. These requirements are discussed in Section 20.5.

1.6 HANDLING THIN LAMINATES

The thin inner layer laminates used in the manufacture of boards of over six layers are flimsy and flexible compared with the 1·6 mm laminate used for most double-sided boards, and the flexible layers for flexi-rigids are almost devoid of any inherent stiffness. Thus the inner layers for multilayer boards are prone to damage during manufacture unless special precautions are taken in their handling. The risks of surface contamination from fingering the laminate or from laying a piece of laminate on a dirty surface are common to all types of board manufacture, but multilayer work introduces further hazards of mechanical damage.

1.6.1 Storage and Transport

Laminates must be stored flat and totally supported over their entire area. It is all too easy to balance a parcel of cut pieces of laminate on top of a smaller parcel while it is being opened, so that as the wrapping is undone the lower sheets of the stack can fall away at one outer edge. Rule 1 for the handling of inner layer laminates is that clean, flat surfaces large enough to accept the largest pieces of laminate to be handled, plus space for any wrapping to be undone, must be available at every site at which the laminate will be handled. In most production facilities the need will be for three such areas: one for laminates awaiting work, one for the laminate being handled, and one for finished work. Space can be saved by the use of shelves or trolleys to hold incoming and finished work, but it is essential that all operators involved be trained to ensure that the shelf is clear and clean before any piece of laminate is placed on it. The use of open-fronted wooden or plastic trays of appropriate size for the board being made, on which to stack pieces of laminate and carry them from work station to work station, can assist in avoiding damage.

The copper surfaces of sheets of laminate can be protected from scratches and abrasions by sheets of interleaving Kraft paper or thin foamed plastic. Any such interleaving material must be clean and free from dust, and the re-use of old and often contaminated plastic interleaving should be very firmly discouraged. The material storage area for inner layer laminates will usually have to be quite large with a considerable number of shelves to hold different laminate and cladding thicknesses, and care must be taken never to store full sheets on top of cut pieces. The stored laminate must never be allowed to overhang the edge of its shelf, as corners are very easily bent over or broken off.

1.6.2 Cutting

The first hazard encountered by the laminates when they are drawn from store is the guillotine. This must be properly set to cut the thin material cleanly without any risk of folding it between the blades, and it must also offer adequate support to the piece being cut from the sheet. If the laminate is simply allowed to fall to the floor behind the machine as it is cut, it can be bent or kinked, and the glass fibres may be broken or the surface copper distorted so that it becomes impossible to make a clear print on it.

1.6.3 Picking Up Thin Laminates

Transferring the cut laminate from the guillotine to a tray or trolley is not too difficult as it still has copper on one or both faces, which helps to add stiffness. Thin polyimide based laminates or large pieces of thin epoxy-glass can, however, be kinked if they are picked up carelessly.

Attempts to hold a sheet of thin laminate as one would a piece of paper, with a slight curve forced in by a thumb on one side pressing down between two fingers on the opposite side, will almost invariably result in compound curvature as the far end of the sheet sags down, and often a kink will be

formed in the centre of the sheet. Such a kink may be almost unnoticeable, but it will be permanent and will usually be sufficient to prevent a printing film from making the essential close contact with the resist-coated laminate. The kink may even be deep enough to affect the local adhesion of a dry film resist.

The pieces of laminate should be picked up with two hands, and allowed to sag to a natural catenary curve which will not kink the copper and will minimise the risk of breaking the strands of glass.

1.6.4 Conveyorised Machines

Conveyorised machines designed and built to handle 1·6 mm laminate can cause problems if attempts are made to use them on thin or flexible material. The leading edge of the laminate can fail to ride over a wheel or roller, and flexible inner layers can become completely wrapped round a roller. The thin materials can be fed through such machines by taping them to a normal thickness leader board, but then the laminate can sag down between rollers allowing etchant to collect in puddles on the upper surface.

Machine manufacturers are aware of these problems and more modern equipment is available with rollers or conveyors which cope with thin and flexible laminates without any problems. The smooth operation of even the best conveyorised machine is liable to be disrupted if the laminate has been distorted by careless handling, for instance if one corner is bent down. With cleaning machines, the best practice is usually to feed the blanks into the conveyor at an angle so that the machine has a gradual start and finish.

If boards are fed square on to the rollers there can be a tendency for the back edge to 'flip' down as it passes from under the working heads.

With mechanical cleaning of laminates, excellent results can be obtained by using a rotary brush cleaner with wet pumice, with the laminate resting on a flat, clean, stainless steel bed. If the bed is cleaned and the laminate is laid on the wet bed with a slight rolling down to exclude all air, surface tension effects will hold it in place against the brushing action. Cleaning and rinsing can be followed by drying in a conveyorised infra-red drier.

1.6.5 Plating Fixtures

Through-hole plating in thin laminates must be performed with rigid, all-round framing holding the laminate flat in the bath. Excessive agitation should be avoided as this can overstress the centre of a large laminate and cause stretching and distortion. The material can be clamped into special plating frames, or an all-round pinning system can be used.

1.7 RELIABILITY OF MULTILAYER BOARDS

It may seem that as a multilayer board is a purely passive component there can be little or no possibility of failure. Generally speaking, if the boards are properly designed, well made, and carefully handled during assembly and testing, this is true, but there can be pitfalls for the unwary.

1.7.1 Delamination

Probably the commonest cause of complaints regarding the unreliability of multilayer boards is delamination, especially during wave soldering. There are two possible causes of such delamination: inadequate care during manufacture, and improper handling during assembly and soldering.

Excessive oxidation of inner layer copper, especially on earth or power planes, can cause low bond strengths as the over-thick film of oxide will form a powdery deposit which can be rubbed off the copper prior to bonding. This should not be allowed to occur in a well-run factory. Incorrectly set press parameters or the use of a prepreg with too little resin content or poor flow characteristics for the press cycle used can also cause weak bonds. Again, this should not occur if board manufacture is adequately controlled. The most common cause of delamination during mass soldering is moisture absorption. The thermal shock of the board entering the solder wave vaporises any absorbed moisture, and the resulting steam pressure will blow the board apart at the line of the weakest bond.

This failure mechanism has been recognised and widely reported since the late 1960s, but it is still possible to find assembly shops where it is not appreciated that the boards can and do absorb moisture and must therefore be oven dried, usually at 120°C, prior to soldering, and that the drying must occur virtually immediately before soldering. The best assembly cycle is to mount all components, then dry and solder.

Many multilayer boards find application in military environments where all components must be capable of withstanding temperatures of at least 120°C. In non-military work, where there may be components which cannot withstand baking at 120°C, any such components should be inserted immediately after drying the board prior to wave soldering.

Boards containing polyimide layers, either flexi-rigid or all-polyimide boards for high temperature applications, are particularly prone to delamination caused by absorbed moisture if they are not properly dried before soldering because polyimide can absorb significantly more water than epoxy-glass.

Delamination should not be regarded as an inherent cause of failure, but as an indication that something is wrong in board manufacture or, more likely, in assembly. It should be noted that moisture absorption by polyimide or epoxy-glass is quite normal, and the electrical properties quoted for these materials allow for the moisture content.

1.7.2 Outgassing

Outgassing which causes blowholes in mass soldered joints can be found on all types of through-hole plated boards, its basic cause being the vaporisation of liquid or the release of gas within the solder-filled joint.

Thus, absorbed moisture in a board can cause blowholes in the soldered joints although there may be insufficient moisture to lead to delamination. Other causes of blowholes in joints are processing chemicals adsorbed in the walls of the holes during board manufacture, unevaporated flux solvents trapped in the holes by the solder, hydrogen adsorbed by the metal in the

hole barrels during plating, and partially cured resin in the laminate or in the bonded prepreg layers.

Flux solvents can be eliminated by careful control of fluxing and adequate pre-heat on the wave soldering machine; partially cured resins and absorbed liquids are eliminated or minimised by properly controlled board manufacture. Outgassing during mass soldering has been studied since the early 1960s, but opinions on the subject still vary widely.

Some engineers accept a small amount of outgassing and the formation of minor blowholes as inevitable and acceptable, arguing that the blowholes seldom affect more than 10% of the effective soldered joint area. In a typical joint in a board 1·6 mm thick, there is probably over 2 sq mm of joint area unaffected by a small blowhole, which is usually a greater area than would be designed in as a reflowed joint to hold a comparable surface-mounted component, and is far greater than the joint area would be on a non plated-through hole.

Other engineers argue that the presence of blowholes in the soldered joints indicates that there must be pinholes or cracks in the plating in the hole barrels through which the gases have escaped from the laminate. These arguments ignore flux solvents, and also do not take into account the fact that most board manufacturing specifications do allow the presence of small defects in the hole walls.

Whenever blowholes or outgassing are experienced, it is recommended to take microsections and to investigate all possible causes before deciding whether the situation can be tolerated, whether suspect joints should undergo a manual touch-up process, or whether steps should be taken during board manufacture to eliminate the causes of the outgassing.

If boards are purchased from a reputable manufacturer, are properly dried, coated with the correct amount of a good flux, and are given adequate pre-heat treatment before soldering, outgassing should not prove to be a problem.

1.7.3 Short Circuits

1.7.3.1 DESIGN AND MANUFACTURING FAULTS

If board manufacture is not efficiently controlled, short circuits occur because of inadequate care in printing and etching, overplating, or as a result of metallic inclusions in prepreg layers. Poor design can cause short circuits where tracks on adjacent layers cross with only a single sheet of prepreg called for between the layers, and shorts can also occur where hole barrels are allowed to pass through gaps in internal planes or between tracks without a pad round the hole. Moreover, short circuits can be formed if the designer specifies a black legend and this is applied over the surface tracks or pads with an ink which contains carbon black. None of these short circuits constitutes an inherent reliability hazard; they are, rather, the result of bad design or manufacture, and as such are easily avoided.

1.7.3.2 GROWTH OF SHORT CIRCUITS

There are, however, two mechanisms by which short circuits can form in

a good board under what can be considered as normal usage. The first is the growth of conductive anodic filaments within the board. If the board is used in a humid environment at moderately elevated temperature, hydrolysis will cause a reversible degradation of the glass/epoxy interface. If a strand of glass contacts two metal surfaces, such as the barrels of through-plated holes, which are at different electrical potentials, irreversible corrosion of the copper anode can occur, and migration of the conductive corrosion products along the degraded epoxy-glass interface will occur until a short circuit is completed. It has been calculated that at the voltages typically used on most multilayer boards this failure mechanism could produce faults in 50 to 200 years of use, but at higher voltages (around 200 V) the time to failure could be just a few years.

This mechanism, which would appear to be unavoidable if a conventional epoxy-glass board is used in a moderately humid environment, should actually present a reliability hazard in only a very few cases, and in practical cases where such high voltages are present the risk can be minimised or eliminated for normal life spans by avoiding close pitching of holes which will have high potential differences.

The second mechanism is surface tracking between metal features. This can occur whenever there is ionisable surface contamination between features at different potentials and moisture is present. The basic cause of failure is similar to the conductive anodic filament growth in that it involves migration of metallic ions; in this case, however, filaments can start to grow on the cathode as well as on the anode. The surface contamination may be the result of inadequate cleaning or rinsing during board manufacture, in which case the cure may lie in the substitution of a spray rinse for a dip rinse; it may be the result of inadequate cleaning and defluxing after soldering, for which the cure is to revise the cleaning processes; it may be that the flux used is reacting with a solder resist coating, in which event one or the other will have to be changed; it can be caused by careless handling leaving fingermarks on the board surface, or it may be produced by the deposition on the surface of airborne contaminants during normal use.

If contaminants are being deposited on the board during use, it may be possible to alter air filtration in the equipment, or to arrange for heaters which can prevent condensation of water vapour when the equipment is not in operation. Alternatively, it may be necessary to consider the use of an overall conformal coating.

The occurrence of this type of fault is not so much a reliability hazard caused by the use of multilayer boards as a general environmental hazard. Similar short circuits could affect other portions of the equipment.

When boards are being processed, care should be taken to ensure that if there is any risk of undercutting during etching the subsequent rinse will be adequate to remove any contaminants which may be trapped by the undercut.

The use of solder resist on the board will not prevent the formation of short circuits if there is any surface contamination present on the board, as the polymeric film is not totally impermeable to moisture and the ionisation can occur underneath the solder resist.

With adequate care in design for use in hostile environments and due attention during manufacture, this reliability hazard can be regarded as avoidable.

1.7.4 Carbonisation

Another way in which a short circuit can form in a multilayer board is by carbonisation (burning) of the resin base. This can occur when a component on the board fails and leaves a particular track on the board to carry a very high current. This high current heats up the track, which then burns the resin, leaving a conductive carbon path which can spread to an adjacent track or a track on the next layer up or down. As the carbonisation spreads, more current paths can be connected until the board is effectively destroyed.

Carbonisation can also occur when a conductive anodic filament or surface contamination filament causes a short circuit. If the current carried by the filament is not sufficient to burn out the filament virtually immediately, its heat can char the resin and increase the cross-sectional area of the conductive path.

Carbonisation caused by excess current in a track is a secondary failure, as some other component failed first and caused this excess current. As such, some designers accept that it may happen but adopt the attitude that since the whole board is already scrap the possibility of carbonisation can be ignored. However, equipments do exist on which the boards can be regarded as repairable after possible component failures.

In these cases, the tracks must be designed wide enough to carry the anticipated short circuit current without burning the resin. Figures of up to 10 A per 1 mm of track width are normally used. It should be noted that tracks which connect to integrated circuits do not require such 'fail safe' design, as either the thin aluminium bond wire within the IC package or the metallisation on the chip will burn out if a short circuit occurs.

1.7.5 Warping

A multilayer board can leave the board manufacturer's premises flat, but can warp during wave soldering. This may be remedied by changing the preheat on the soldering machine but, as it is usually a function of the design of the board, the correct precaution is to ensure that the board is designed to be as well balanced as possible.

1.7.6 Damage to Through-plated Holes

1.7.6.1 MECHANICAL DAMAGE

Well-made through-plated holes form one of the most inherently reliable interconnections possible, but they can be damaged by careless handling. Some board users are misled by the external appearance of a board with pads on its outer layers into believing that there is sufficient metal present to withstand any amount of thermal or mechanical abuse (see Chapter 21). They fail to appreciate that, typically, the wall thickness is between one thirtieth and one fortieth of the normal hole diameter.

1.7.6.2 Z-PLANE EXPANSION

Apart from mishandling, the major reliability hazard associated with through-plated holes is Z-plane expansion. The epoxy resin has a much higher coefficient of thermal expansion than the copper of the hole barrels so that, as the board is heated, the barrels of the holes are stretched and the copper has to deform in order to retain continuity.

Often after thermal cycling it can be found that the centres of the pads on the outer layers are noticeably depressed. Provided all holes in the board have pads on all layers, a well-made multilayer board will be able to withstand thermal cycling. However, the omission of pads on inner layers, especially on power and earth planes, can leave resin-rich rings around the holes, where the resin from the prepreg flows into the gap in the copper layer but the glass is unable to flow. Such resin-rich areas have accentuated Z-plane expansion and usually lack some of the micro-rough mechanical key to the plated copper of the glass reinforced areas, so that the stress on the hole barrel will be concentrated at the site of the missing pads. Holes without pads on all layers can be expected to fail after a limited number of thermal cycles through the military temperature range, and failure may occur even after the single thermal shock of wave soldering. Such hole barrel failures at missing pad sites should be regarded as design-induced failures and not as an inherent hazard in the use of multilayer boards.

The problems of Z-plane expansion are accentuated as board thickness increases, and as inner layers are allowed near to the outer surfaces of the board. Clearly there is little problem with the joint between the hole barrels and the inner layers of a four layer board which would usually be closely spaced near the centre of the build. As more pairs of layers are added towards the outer faces of the stack, so the joints will be increasingly stressed when Z-plane expansion occurs. Pads on the outer layers appear to be relatively immune from cracking during thermal cycling because both the resin under the pads and the pads themselves deform under stress. However, the pads on the first layers in from the surfaces are unable to deform in the same way, and the most likely site for failure caused by Z-plane expansion is at the junction between hole barrels and the pads on the internal layer nearest the surface. Because this is the most likely failure site, some specifications (e.g., MIL-STD-275D) call for the use of 2 oz (0·070 mm) copper on layers 2 and n-1. It has been suggested that the use of excessively large diameter pads on inner layers can aggravate such failures caused by Z-plane expansion because the oversize pads are held more firmly in place by the areas of resin in contact with them. Since multilayer boards are normally likely to be used only when there are severe packing density problems, the use of oversize pads should be unlikely to occur.

Boards with pads on all layers for all holes can fail after wave soldering if no pre-heat is used and the holes are abruptly subjected to the temperature of the solder wave. This type of failure is unusual on well-made boards, and is easily avoided by adequate pre-heat which will bring the whole board to above the glass transition temperature of the resin before it contacts the solder wave.

BIBLIOGRAPHY

1 Baillie R. J., 'Mass Lamination as a Laminator's Service to the PCB Industry', *Circuit World*, 7 (3), 71 (1981).
2 Banner H. I., 'Plated Pillar Technology for Multilayer Manufacture with Very High Packing Density', Proc. Inter-Nepcon, 41, Brighton (1974).
3 Foerster R. P., Heath H. E., and Parks H. L., 'High Speed Logic Packaging with Planar Co-axial Interconnections', Proc. Nepcon, 390, New York (1971).
4 Engelmaier W., 'Characteristics of Multilayer PC Boards during Lamination', Proc. Inter-Nepcon, 46, Brighton (1974).
5 Hurd D. E., 'Multilayer Printed Circuits Without Through-Holes', Proc. Nepcon, 208, New York (1964).
6 Hurley H. C., 'Reliability Evaluation of Multilayer Board High Density Packaging Designs', Proc. Nepcon, 15, New York (1971).
7 Jennings C. W., 'Effect of High Moisture Environments on Printed Wiring Board Insulation', Proc. 2nd Printed Circuit World Convention, I, 47, Munich (1981).
8 Kolias J. T., 'Trends in High Speed Computer Packaging Interconnections' Proc. Inter-Nepcon, VII-47, Brighton (1970).
9 Kolias J. T., 'Interboard Interconnections in Digital Computers', Proc. 2nd Printed Circuit World Convention, I, 399, Munich (1981).
10 Luke D. M., 'Factors Affecting Some Properties of Multilayer Printed Circuits', Proc. Inter-Nepcon, 27, Brighton (1974).
11 Mitchell J. P., and Welsher T. L., 'Conductive Anodic Filament Growth in Printed Circuit Materials', Proc. 2nd Printed Circuit World Convention, I, 80, Munich (1981).
12 Nitsch H., 'Very Fine Line Techniques for Making Printed Circuits and Multilayer Boards', Proc. 2nd Printed Circuit World Convention, I, 375, Munich (1981).
13 Scarlett J. A., 'Visits to American Electronics Companies and Nepcon East Conference', Marconi Radar Systems Ltd., EDTM SEC/Y/23/33544/9.
14 Scarlett J. A., 'Printed Circuit Boards for Microelectronics', 2nd Edn, Electrochemical Publications, Ayr, Scotland (1980).
15 Scarlett J. A., 'Transistor-Transistor Logic and its Interconnections', Van Nostrand Reinhold, Wokingham, England (1971).
16 Scarlett J. A., 'Precision in the Printed Circuit Board Industry', *Precision Engineering*, 3 (4), 238.
17 Schuessler P., 'Outgassing Products of Multilayer Board Materials', Proc. Inter-Nepcon 45, Brighton (1980).
18 Schwartz L., 'Stacked Individual Laminates—a Practical Solution to the Mass Interconnection Problem', Proc. Inter-Nepcon 536, Brighton (1968).
19 Smith B. R., 'Double-Multilayer Printed Wiring Boards. A Novel Technique to Overcome a Packing Density Problem', Proc. Inter-Nepcon 41, Brighton (1974).
20 Wild R. N., 'Thermal Characterisation of Multilayer Interconnection Boards', Proc. Inter-Nepcon, 43, Brighton (1975).

Chapter 2

LAMINATES AND PREPREGS

J. S. SALLO

Fortin Laminating Corp, Sylmar, California, USA

2.1 INTRODUCTION

This Chapter will discuss the methods used in manufacturing glass epoxy laminates and prepregs. It will concentrate on the thin laminates and prepregs utilised in the manufacture of multilayer printed circuit boards. The various materials involved in the manufacture of the products as well as manufacturing and testing techniques and some problem and trouble-shooting areas will be dealt with.

The primary product utilised in the manufacture of thin laminates and prepregs is an organic resinous material, and the text will confine itself to the high performance materials employed in this product area, including epoxy and modified epoxy materials, polyimide materials and B-Triazine.

2.1.1 Use of Epoxy Materials in Laminate and Prepreg Manufacture

Epoxy materials have been used in the manufacture of printed circuit boards for a number of years, as well as for a wide range of other applications such as potting, casting and structural materials. In many respects their use in laminate requires different properties from those in other areas. In epoxy laminate, owing to the latency requirement for B-stage partially cured materials to be available, a high temperature cure is mandatory. It is necessary that the material in the partially cured state (B-stage) be very stable, and also that during laminating and prior to cure the resin will become liquid and a reasonable amount of flow take place so that air bubbles and other impurities can flow out of the product during the laminating cycle. It is usually important to obtain good adhesion to copper and other layers of epoxy prepreg during laminating. Equally important is the fact that the cured product should be inert to a wide variety of chemicals including acids, strong bases and solvents such as methylene chloride. The materials must be readily drilled without undue wear to the drills and possess other structural and flexural properties compatible with the needs of the PCB industry. Boards produced from the epoxy laminate must be capable of short time exposure to solder reflow temperatures of approximately 260°C; the epoxy must be able to accept electroless copper with good adhesion, should not smear excessively during drilling, or, if it does, must be susceptible to chemical smear-removal techniques. It must be

possible to flame retard the material so as to meet several very stringent specifications such as UL 94VO. Despite the extreme difficulty of finding materials capable of fulfilling all these requirements, a variety are available that will satisfy the needs of the printed circuit industry.

2.2 EPOXY MATERIALS

2.2.1 Bisphenol-A Epichlorohydrin

The most common epoxy materials in this industry are formed from epichlorohydrin and bisphenol-A. This material is usually supplied in the form of a monomer or slightly polymerised dimer and trimer to the laminator for further reaction. Also a brominated version is used to produce flame retarded material. Some of the effects of the bromine content will be discussed later in this Chapter.

The bisphenol-A epichlorohydrin reactant requires a cross-linking member and some catalysts so that it can be reacted in the laminating process. Normally, the curing material employed by the laminating industry is dicyaniamid (or dicy). Cross-linking can occur so that the resulting epoxy will not be merely a straight chain polymer. This cross-linking is very important, since it produces desirable properties in the final laminate. In addition to the dicyaniamid or other curing material, a catalyst is employed to control the temperature at which cure will take place.

2.2.2 Addition of Epoxy Novolacs

It is not uncommon for epoxy novolacs to be added to the bisphenol-A structure. Addition of these will lead to more cross-linking in the product with a resultant higher solvent resistance, higher moisture resistance and a higher T_g. T_g, the glass transition temperature, is explained in the later text. The major disadvantage of their use is that etchback inside the plated-through hole is extremely difficult, if not impossible, to perform adequately. Many laminate users specify that novolac must not be present in the material supplied to them.

2.2.3 Control of Flow

A simple epoxy formulation containing the brominated resins discussed, and a curing agent will result in a very high flow in the laminating press. Although this type of formulation is often used, it is difficult to control in the manufacture of multilayer materials. For this reason, a variety of agents to control the flow of the prepreg are in use. These materials generally control the viscosity at the melt time so that excessive flow does not occur under pressure while the epoxy is in the molten state. It is important, however, that the materials added to control this flow do not give rise to undesirable additives or unreacted materials in the final product. Most of the more sophisticated prepregs employed in the manufacture of multilayer boards contain materials designed to control the flow, control of viscosity having become a very important factor in the manufacture of multilayer materials.

2.2.4 Glass Transition Temperature

An important parameter in considering the characteristics of epoxy and non-epoxy resin systems is the glass transition temperature, T_g. This is the temperature at which the resinous material appears to change from a solid state to a liquid state, a change accompanied by a rapid increase in the expansion of the material with respect to temperature. Epoxy materials exhibit much greater expansion than do other potential materials. This can lead to hole cracking problems in multilayer boards and to loss of adhesion of copper at the solder temperature. The relative importance of these potential problems depends upon the use to which the eventual circuit board is to be put.

2.3 NON-EPOXY MATERIALS

2.3.1 Polyimide

Despite their widespread use in the electronics industry, the standard epoxy materials have certain limitations, some of the major ones being their relatively low glass transition temperature (T_g) and the fact that they soften at solder pot temperatures and tend to expand greatly at high temperatures. On this account, several non-epoxy materials have been developed in an attempt to overcome these problems. The oldest is the polyimide material which became available for multilayer and other applications a number of years ago. Polyimide materials have a very high glass transition temperature and exhibit excellent adhesion to copper at the solder pot temperature, as well as very low expansion in the Z direction, and therefore have much less tendency to crack the copper plated-through holes in multilayer boards. Polyimides have the disadvantages of high cost, lower copper adhesion at room temperature, some problems with inner laminar bonds probably caused by moisture, and an inability to meet the UL 94VO flammability specification. Polyimide materials have been gaining some acceptance in the laminating industry. They are U.S. MIL spec qualified and, despite their high cost, are finding growing use in high reliability boards requiring thermal stability.

2.3.2 B-Triazine

More recently, the BT resins have been introduced by the Mitsubishi Gas Company. These are a co-reaction of polyimide and triazine. The BT resins have a high glass transition temperature although not so high as that of polyimide. They give good inner laminar bonds, good copper adhesion and can be formulated so as to meet the U.L. 94V0 specification. The cost of the resin is comparable to that of polyimide. However, BT epoxy blends can be used to reduce both the cost and the T_g. BT resins have not as yet received military qualification and so are not used in such programmes. Commercial application of BT resin where Z axis expansion is a problem and high T_g is desired has begun. Additional uses for BT resin appear to relate also to the conductive anodic film problem which has been found in boards exposed to high humidities and voltage during operation. An expansion of the use of BT resins in these areas is to be expected.

2.3.3 Modified Epoxies

A number of other materials have been developed recently in an attempt to improve the performance of epoxies without shifting to completely different types of chemistry such as B-Triazine and polyimide. Among these are the modified epoxies which utilise different curing agents or different epoxy inter-linking materials to produce a higher degree of cross-links. A number of laminators have, in addition, made available proprietary modified epoxies, some of which have excellent high temperature characteristics compared with those of epoxy, although not so good as those of B-Triazine or polyimide. In general, these materials would seem to fill a need in the areas in which they have been introduced, and eventually a range of materials from epoxy to polyimide will emerge for selective use, depending upon the requirement of the printed circuits.

2.3.4 Problems with Brominated Materials

The bromination of epoxy resins has certain attendant problems. The brominated materials are solids rather than liquids, somewhat more difficult to prepreg and rather more difficult to put into solution. These problems have, however, been solved. In order to achieve the specifications required, approximately 20% bromine by weight is used in modern laminates. The brominated materials tend to have poorer bonds to copper than do the non-brominated epoxies and this poorer bond also exists for the copper deposited in the plated-through hole. For this reason, it is necessary that the laminators do not over-use bromine as a flame retardant and its use must be carefully controlled. Furthermore, during thermal cycling it is possible for the carbon-bromine bond to break resulting in brominated by-products.

2.4 GLASS CLOTH

Before dealing with prepreg manufacture, it is necessary to mention briefly the glass-cloth which is used. As Chapter 3 treats the subject more comprehensively, little attention will be devoted to it in this Section. The glass style chosen by the laminator will depend upon the eventual thickness of the laminate desired by the customer along with other requirements such as dimensional stability and cost. In general, a silane finish is used to couple the glass to the resin. The nature of this finish may have very significant effects on the chemical and moisture resistance of the finished product. The quality of the glass in terms of its structural integrity, mechanical integrity, and the finish are very important parameters to the final quality of the laminate produced.

2.5 PREPREG MANUFACTURE

Prepreg is manufactured by pulling a glass-cloth through a solution of the particular resin system chosen. The glass is impregnated with the resin system and then proceeds to a heated tower or oven where the solvents are driven off and the resin is partially cured or B-staged. In circuit board

lamination, the B-stage material is non-tacky and may be supplied to the customer for utilisation in manufacturing multilayers. Prepreg will also be used in laminating for the eventual production of a laminate which may be copper-clad. The impregnation of the glass with the resin is a crucial step and will depend on the glass style, the glass finish, the particular type of treater head which is used, the solvents, and a variety of other factors.

2.5.1 Treater Types

2.5.1.1 MECHANICAL TECHNIQUES

There are a number of types of treaters used, both mechanically and thermally. Mechanically, these may be either horizontal or vertical. With a typical vertical treater 10 metres high, the impregnated glass will go up 10 metres through heating devices, pass across two to four rollers at the top, which may be chilled, and then come down the other side to emerge as finished prepreg. A horizontal treater may be of similar length except that the prepreg is stretched from one end of the oven to the other. Generally speaking, better control of the stress imparted to the glass and resin system can be obtained in a vertical tower, and it is usually easier to run finer glass through a vertical tower than a horizontal one. However, there are so many modifications and drive systems available on both horizontal and vertical treaters that it is almost impossible to generalise about which is better for a particular type of epoxy system and application.

2.5.1.2 THERMAL TECHNIQUES

There also exist a variety of techniques for introducing the required heat to the resin system, the most common being forced air. The air is heated externally to the large oven used in prepregging and then driven through the oven, usually countercurrent to the movement of the prepreg. The main problem associated with this type of heat is the elimination of solvents without blistering and the eventual B-staging which must be uniform both across the web and along its length. Another type of heating which has been used very successfully is infra-red. Although it is extremely difficult to use because various wave lengths can have very dissimilar effects upon the final prepreg, one advantage of infra-red heating is that it has a tendency to heat the glass rather than the resin with the result that the prepreg is heated from the inside out. When prepreg is produced in this way, it is possible for it to have a very low volatile content since volatiles will not be trapped inside the prepreg by the skin curing prior to the internal parts of the prepreg curing. Those groups which have mastered the infra-red techniques attest to their considerable value in making high quality prepreg with very low volatile content and very carefully controlled gel and flow times.

2.5.1.3 OPERATING CONDITIONS

A variety of treater conditions are used in the manufacture of B-staged prepregs. The majority of ovens, either horizontal or vertical, have several zones and it is possible for the first zones to be cooler than subsequent

zones. This may be arranged in order to volatilise solvents more slowly so as to prevent the occurrence of blistering. The speed of the treater is also a very important condition for the prepreg. Although from a production point of view it is highly desirable to run the treaters as rapidly as possible, better results are sometimes achieved with very slow treater speeds. This may be particularly true when very good dimensional stability and flow control are required. Slow treater speeds may also improve impregnation and lead to other desirable characteristics in the finished product.

One common problem is that associated with pollution control. It is vital that the solvents emitted be burned off or trapped for eventual recovery by distillation. The use of these techniques often retards the flow and escape of solvent from the system and causes gross differences in the operating conditions of the treater.

2.5.2 Process Control

Process control in prepreg manufacture is extremely important and it is essential, at intervals during the run, to cut pieces in various places along the width of the moving web and test them for gel, flow, resin content, volatiles and dicy crystals. This must be repeated frequently enough to ensure that changes are not occurring during the length of the prepreg roll which may cause deviations in the type of properties obtained in the product. Any change in glass, resin, speed or temperature of operation should necessitate another check of these parameters.

Recently, the use of scaled flow testing, which will be discussed in a later section, has led to great improvements in the ability of the laminator to control his prepreg product due to the improved nature of the test.

2.5.3 Prepreg Testing

Prepreg testing is a delicate art with many skilled practitioners. Testing for gel, flow, resin content, volatiles and dicy crystals is usually considered sufficient. Sometimes other parameters such as adhesion to copper and exotherms and endotherms during a cure are also evaluated. This Section will review some of the more straightforward test procedures before discussing some of the newer thermal techniques.

2.5.3.1 GEL TIME TEST

The gel time measurement is widely used in determining the potential reactivity of the prepreg material. As a test it is very difficult to repeat from one facility to another, owing to the strong effect of minor differences in technique. However, a skilled operator in his own facility, utilising the same equipment each time, can reproduce gel times within ±5 seconds. Despite its shortcomings, it is still recognised as the best test available for determining prepreg reactivity. Typically, the test is conducted by taking a known amount, preferably weighed, of powdered prepreg removed from the glass. This material is placed on a hot plate at a predetermined temperature, usually 171°C. It is very important that this temperature be maintained accurately, and that the same part of the hot plate, press platen or heating

stage be used each time as temperatures normally vary across the surface. The elimination of stray air currents is also essential to the accuracy of the test. A clock is started as the prepreg material is placed on the hot plate. Subsequent to melting of the powder, a stick, usually wooden, is used to stir or work the melt. The rate of stirring, the size of the stick, and the composition of the stick, in particular whether it is thermally conductive or non-conductive, will have a significant influence on the results obtained. The gel time is arbitrarily defined as that time when the resin is sufficiently sticky or tacky to break as it is pulled in strings from the hot plate. Despite its relative inaccuracy, this test continues to be widely used. Tests have been suggested involving more sophisticated equipment making use of changes in dielectric constants or temperature endotherms and exotherms. Unfortunately, it is almost impossible for the industry to standardise on any of these tests which require expensive or sophisticated equipment and it seems likely that the gel time test will be in vogue for some time.

2.5.3.2 FLOW TEST

Another important parameter in prepreg testing is flow. This test is run by taking a number of plies of prepreg which have been carefully weighed after being cut to a predetermined size (the size chosen having a definite effect upon the results). Typically, these pieces of prepreg are stapled together to prevent shifting and are then placed between two platens and pressed in a known temperature pressure cycle. Following this, the prepreg pieces are again cut to size, discarding that portion which has flowed out beyond the original size of the stack-up. The original size stack-up is reweighed and the flow percent is defined as the second weight divided by the initial weight times 100. This test is subject to many variables and the results are of somewhat questionable value.

2.5.3.3 SCALED FLOW TESTING

Recent work by the Bell Telephone Laboratories and Western Electric has led to the adoption of a scaled flow test method for testing of prepreg. In this method, the final pressed thickness of a test laminate is made to be identical to or related to the final pressed thickness in production. This is done by scaling both the size and the pressures used down to appropriate values for laboratory test. This necessitates the use of very low pressures for incoming inspection; however, the results have shown a spectacular improvement in the ability of both the laminator and the multilayer manufacturer to control their processes. Prepreg utilising scaled flow testing to predict ultimate pressed thickness has recently been introduced by Fortin Laminating under the name CRC® Prepreg. The general test method, based upon the results achieved in production studies, is far superior to the use of gels and flows and it is anticipated that this test will eventually replace gels and flows in prepreg testing for multilayer applications.

®Registered trademark of Fortin Laminating Corporation

2.5.3.4 RESIN CONTENT TEST

The resin content is determined by taking a weighed piece of prepreg, placing it in a very hot muffle furnace with a surplus of air available, and burning off all the material capable of being burned. The remaining glass is then cooled to room temperature and weighed, the resin content being the glass weight divided by the original weight times 100. Resin contents are usually quite repeatable and the measurement is accurate. It is a useful process control in determining the pick-up across a wide web of glass being impregnated with resin system. Resin content testing will also help the multilayer manufacturer to know the eventual thickness of his board. However, it must be combined with the knowledge of scaled flow test results in order to be of real use.

2.5.3.5 VOLATILE CONTENT TEST

Volatile content is determined similarly to resin content, although a lower temperature is used. The measurement is therefore quite arbitrary since the time and temperature chosen will be a major factor in determining the so-called volatile content. Nevertheless, a good indication of volatiles can be obtained from this test and, relatively speaking, the results can be quite meaningful. Normally, prepreg should have less than 0·5% volatiles and it should be borne in mind that water will be considered a volatile by this method, since it has a lower boiling point than some of the solvents used in prepreg manufacture. Consequently, the exposure to moisture of the sample, prior to the test, can be a major factor in determining the volatile content as measured in a multilayer manufacturer's facility.

Volatiles are one factor in determining flow. A prepreg with a relatively high volatile content will have relatively high flow. Unfortunately, if the volatiles are lost with time, a dramatic effect upon flow will result leading to an instability in the prepreg greater than that which might be anticipated by the normal ageing of the product. For this reason, prepreg with a very low volatile content is preferred and it is better to control flow with methods other than the high volatile one.

2.5.3.6 DICYANIAMID TEST

Dicy is normally used as a curing agent in epoxy laminate systems. It is a white crystalline solid which is initially dissolved in the prepreg system. As the solvents are driven off in the treater, unless special precautions are taken dicy crystals will appear in the prepreg. This gives rise to several potential hazards in multilayer laminate production. One of these is that the segregation of the dicy into crystals may leave other areas of the prepreg with a very low dicy concentration, and may lead to sections of the laminate being under-cured after final processing. Dicy itself being hygroscopic will pick up moisture rather rapidly. Consequently, agglomerates of dicy crystals can produce laminates which have moisture sensitivity. Studies have been undertaken showing the deleterious effects of dicy agglomerates on the electrical properties of laminates after humidity exposure. It is therefore important that the free dicy which must be present in the B-staged prepreg

should not be allowed to form crystals but remain evenly dispersed throughout the product. Testing for dicy crystals is readily accomplished by viewing the prepreg under plane polarised light, where they will show up as white particles against a dark background. Whilst a certain level of dicyaniamid crystals is considered acceptable, excessive amounts can cause very serious problems in the laminate.

2.5.3.7 TEST FOR AGEING

Another problem associated with prepreg testing concerns the ageing of the prepreg. Normally assumed to have a six months' life when stored at 5°C, prepreg should be tested by the multilayer manufacturer upon receipt and, assuming some storage time, should be retested prior to use. It is important if the prepreg is maintained in a cold environment that it be given at least a 24 hour period in which to stabilise prior to this testing and prior to use. Prepreg during storage should be protected as much as possible from moisture ingress and other possible atmospheric contaminants. This is best achieved by storing in a sealed bag.

2.6 LAMINATE MANUFACTURE

The manufacture and use of thin copper-clad laminates will now be considered, the laminates being made from prepreg similar to that which has just been described. The laminator may choose to use different resin systems for the manufacture of laminate and prepreg. His prepreg will usually have a controlled rheology and melt point viscosity so as to make the lamination of multilayers easier for the circuit board manufacturer. However, he may not necessarily include these ingredients in laminate for his own production, since he may have other techniques for manufacturing laminate from less expensive and simpler prepreg systems. Provided that the materials behave similarly in etchback, drilling and other required characteristics, this is an acceptable procedure. After the manufacture of the laminate, a number of properties need to be checked to ensure eventual good results in the manufacture of the finished multilayer board. A variety of these tests and properties will be discussed as they may pertain to actual use in PCBs.

2.6.1 Influential Parameters and their Measurement

2.6.1.1 DIMENSIONAL STABILITY

Probably the most important parameter in the manufacture of good quality multilayer boards is dimensional stability. By this is meant the change in the dimensions of the board after etching and baking. It should first be considered why this problem arises. Prepreg is placed in the laminating press with sheets of copper typically on both sides and then heated under pressure until the resin melts and flows. During this time, the copper is allowed to expand along with the prepreg but at a much lesser rate because of the differences in coefficient of thermal expansion. At a certain point, the prepreg cures and bonds itself to the copper, and beyond this the copper and the finished cured epoxy laminate are forced to cool together at

the same rate. As a result, internal stresses are built up: since the epoxy has expanded more than the copper, it also wishes to shrink more and this shrinkage is retarded by the restraining effect of the copper bonded to the epoxy. These resulting strains in the epoxy layers will be relieved to some extent when the copper is etched away to form a circuit pattern and an associated shrinkage in the epoxy material will ensue. Heating of the circuit board will further enhance the strain relieving process and cause even more shrinkage. Therefore considerable shrinkage can occur from the laminating press to the finished multilayer board which has experienced both etching and high temperature during laminating and baking steps.

Measurements of dimensional stability are usually performed in the following way: the laminate is scribed or punched with two marks, approximately ten inches apart, and this spacing is accurately measured and recorded. Small circular pieces of resist, for example tape, are pasted over the scribe marks and the remainder of the copper is etched from the board. The distance between the scribe marks is then measured. Finally the board undergoes a bake cycle which is intended to simulate some of the thermal effects it will experience during processing. The distance between marks is again measured after the board has been allowed to stabilise for some time at room temperature. In a good multilayer material the maximum shrinkage between the initial and final stages of these tests should not be greater than 500 ppm. Laminate with dimensional stability better than this can sometimes be obtained consistently, but this is usually special laminate, specially constructed and moulded and normally available only at a premium price.

Among the factors that determine the final dimensional stability are the resin system and laminating cycle employed, the nature of the copper, and the glass construction used. The last of these may be the most important since very precise dimensionally stable laminate can at times be achieved with the proper glass construction. In general, thinner copper with poorer bond strength will tend to produce better dimensional stability results. The key to the process, however, is in the combination of glass constructions and the specific resin system employed. This is particularly true as the amount of flow in the resin system prior to the gel time is a significant factor in the determination of dimensional stability.

It has recently been shown that scaled flow testing of the prepreg can lead to significant improvements in a number of parameters involved in multilayer laminates. Parameters such as overall thickness and dimensional stability are the most affected.

2.6.1.2 COPPER ADHESION

Copper adhesion is also relatively important in high quality laminate used for multilayer boards. If inadequate adhesion is achieved, it is possible that movement or swimming of circuit lines will result during the laminating cycle and the flow of the prepreg. Usually, however, adhesion above 6 lb per inch will prevent this swimming phenomenon occurring and higher adhesion is probably unnecessary in inner layers of multilayer boards. High adhesion often infers a very heavy tooth on the side of the copper to be laminated to the epoxy prepreg, and this heavy tooth on the copper will

sometimes cause problems due to the extremely rough surface topography left on the laminate after etching of the copper. This is particularly likely to be a problem in very thin laminate where the tooth may actually enter the glass bundles and cause extreme sensitivity to chemical attack, moisture absorption and a variety of electrical problems. For this reason, especially in thin cores, very high adhesion is to be avoided. This clearly applies to the case of copper foil heavier than one ounce, where the tooth is always excessive. There are occasions when special light tooth copper should be employed to minimise the effect.

As contrasted to inner layers, high adhesion is more desirable on cap sheets of multilayers. These circuit lines tend to receive harsh treatment during reflow solder and any re-work which may be required on the board. In some instances, the lines actually provide physical support for components, rendering good adhesion much more desirable on cap sheets and relatively unnecessary for inner layers. Some specific copper problems will be discussed later in the Chapter.

2.6.1.3 DIELECTRIC CONSTANT

A parameter of great interest in certain multilayer boards is the dielectric constant of the laminate. Since glass has a higher dielectric constant than epoxy resin, the dielectric constant is controllable within limits through control of the resin content. Higher glass or lower resin content will always yield a higher dielectric constant. Control of this parameter is, however, only possible within narrow ranges depending upon glass construction and resin content. As a parameter it is usually important only when very carefully matched or controlled impedances are required in the inner layers.

2.6.1.4 THICKNESS

Greater significance is attached to thickness control, and the thickness of thin core laminates is normally a critical parameter since it will determine thicknesses of the ultimate board and will control the spacing between occasional parallel or noise-inducing tracks. Thickness control over a full sheet of laminate can be difficult to achieve and is dependent upon many parameters such as the glass construction used by the laminator, pressure variations within the press, the flow characteristics of the prepreg employed, and the laminating cycle. In some cases, the laminator may be willing to accept very tight thickness tolerances but as a rule only on premium quality and premium priced materials.

2.6.1.5 SOLVENT RESISTANCE

In normal PCB production processes, laminate is exposed to a variety of solvents, to which chemical inertness is very important for the quality of the final product. Solvent resistance may be harmed through the use of partially cured laminates. Excessive exposure to solvents such as multiple passes through methylene chloride stripper will nearly always damage laminate. Also, excessive tooth on the copper as previously described may render the laminate very sensitive to solvent attack. It has recently been observed that

over-baking of laminate in air at too high a temperature for a prolonged period of time may cause degradation of epoxy bonds, a decrease of T_g and excessive sensitivity to solvents. Some resin systems are more inert to solvents than others. Such systems, e.g., those containing epoxy novolacs, are not normally used in multilayer laminates because of the difficulties they present in etchback. Cases do exist, however, where extreme solvent resistance and moisture resistance are required and despite the etchback problem epoxy may be used in these areas.

Recently a new class of resin systems utilising multifunctional epoxies has become available. These materials have higher T_g and are reported to have superior solvent resistance properties. Although not yet widely applied, the use of such materials in very critical multilayer boards will probably increase in the foreseeable future.

2.6.1.6 BAKING

Although the potential deleterious effects have already been described, baking of laminate at various cycles in the process is a necessity. This is normally done to remove either solvents or moisture and is an essential step after exposure to either, and before applying heat. A four hour bake at 135°C will usually remove moisture and solvents without causing damage to the laminate. After exposure to either water or methylene chloride, a bake is essential before heat is applied, either in the laminating press or through reflow or solder pot operations. The bake should always be carried out in a racked rather than a stacked condition so that the moisture or solvent can escape from the board during the temperature cycle.

2.6.2 Manufacturing Problems

2.6.2.1 SMEAR REMOVAL

Etchback has already been mentioned. Normally during the drilling operation there will be some epoxy smear due to the frictional heat developed during drilling. This smear will tend to cover copper which is eventually intended to make contact with the electroless copper employed in the plated-through hole. It is therefore necessary that the smear, if present, be removed prior to electroless copper plating. This is generally accomplished by the use of an etchback solution which will attack the epoxy, warm or hot sulphuric acid often being chosen for the purpose. If it is desired to etch back into the laminate itself, hydrofluoric acid must be added in order to attack the glass. In etchback of epoxy novolac material, a chrome/sulphuric solution tends to be used, although the chrome has recently become highly undesirable owing to pollution problems. It is essential, however, that in one way or another epoxy smear be removed subsequent to drilling and prior to electroless plating.

2.6.2.2 DRILLING

Drilling of the laminate often causes unusual problems. Apart from the smear already discussed, chipping, gouging and other phenomena can occur. The normal cause of these problems lies in the drill bit, drilling temperature or drill speed. Problems will also be experienced if undercured laminates are drilled and this includes the curing of the prepreg layers during the manufacture of the multilayer.

2.6.2.3 DEGREE OF CURE

The use of undercured laminate can be a serious problem in multilayer board manufacture. The best method for determining degree of cure is the glass transition temperature, T_g. Normally the T_g in epoxy materials ranges from 115 to 135°C, depending on modifications to the resin system. Another useful term is the ultimate T_g. This is the T_g which can be achieved with a given laminate upon extreme baking, and is the maximum T_g available with a given resin system and curing agent. A large spread between the T_g of the laminate and the ultimate T_g, typically greater than 4°C, is an indication of an undercured laminate. These have a variety of unpleasant characteristics such as poor drilling qualities and excessive solvent sensitivity; yet, an undercured laminate, providing the catalyst is actually present, is not usually a major problem in multilayer boards because the laminate at some point will undergo a subsequent lamination step which should be sufficient to cause it to cure. Unfortunately, as this step is after exposure to solvents, excessive solvent attack may already have occurred. The checking of T_g by the laminator is an effective way of preventing such materials from being used in multilayer boards.

2.6.3 Solutions to Laminate Problems

2.6.3.1 GENERAL

Having discussed many of the problems that may arise in laminate, it is now appropriate to consider how the circuit facility may act to overcome or minimise the effect of some of the faults which can occur and what the laminate supplier may do to minimise them. The use of heavy tooth thick coppers on thin core materials should be restricted wherever possible. Baking of laminates prior to shipment has a strong tendency to improve dimensional stability and is a practice which the customer should encourage in the laminator, providing the added cost of the baking step is not undesirable. High quality materials will result from this step. The PCB manufacturer should be extremely careful about the nature and quality of solvents which are applied to the laminate. Those materials which are exposed only to the copper do not create such a cause for concern. Baking steps should be introduced after each exposure in a free flowing forced air oven where the laminate is exposed to air on both sides so that the effect of trapped volatiles, including water, can be minimised. If extremely good dimensional stability is required, the practice of compensating for the known laminate shift in the artwork may be employed. This depends on the laminator retaining the shrinkage value constant for the laminate under discussion. In order to improve dimensional stability, the laminator should utilise only high quality glass and resin systems with controlled flow and rheology. The quality and tooth of the copper are also important in

controlling dimensional stability. Cooperation between the supplier and the user is essential in the production of high quality multilayer materials and the foregoing explains some of the interactions between the customer and the vendor which must take place for this product to be optimised.

2.6.3.2 USE OF THIN COPPER FOIL

In multilayer board manufacture the nature of the copper can be of considerable significance. Thin copper foils, typically quarter ounce or 9 micron, have been available to the industry for several years.

This foil is supplied on a sacrificial carrier, usually aluminium. 12 micron foil is now available as a free film and offers substantial advantages over the 9 micron foil, providing that the disadvantage of the slightly greater thickness does not cause problems in line width. These foils are not used in inner layers where simple print and etch steps are normally performed and where the resulting copper tracks would be too thin to pick up the plated-through holes which interconnect most multilayer boards. Thin copper is finding increasing use on the surfaces or cap sheets used in multilayer boards, the reasons for this being two-fold. Thin copper enables very fine line etching without excessive under-cutting. Boards requiring 0·125 mm and 0·25 mm wide conductors on the surface have been readily prepared utilising quarter ounce copper foils. In addition, substantial cost savings have resulted from the use of thin copper foil even though the laminate is slightly more expensive. This is due to various savings in etchants and a variety of labour steps which are minimised or eliminated.

2.6.4 Methods of Promoting Adhesion

2.6.4.1 BLACK OXIDE

The copper surfaces of inner layers are usually modified in some way to promote adhesion to the prepreg layer. There are two ways of achieving this. One method is to use the so-called black oxide technique. A variety of proprietary solutions and a number of in-house solutions are available for this purpose, all of which contain sodium chlorite, are highly alkaline, and produce a black coating of copper oxide on the surface of the copper conductors. This coating then forms a bond between the prepreg and the copper which enhances the adhesion of the copper to the prepreg layer and therefore improves the ability of the board to withstand thermal shock such as reflow soldering. A variety of techniques exist and several opinions are available on what constitutes good oxide coating manufactured in this way. An extremely heavy black coating, although cosmetically pleasing, will often result in black copper oxide particles flowing off into the prepreg layer during the laminating step. This is not a desirable condition. Somewhat lighter black coatings have been employed quite successfully and control of time, temperature and chemical constituents of the bath is required in order to control the thickness and porosity of the coating. Another widely used technique involves a red or brown oxide. In this case, either the black oxide is applied very lightly or after a heavy application some of the black oxiding is removed, typically in a weak sulphuric acid

solution. The proponents of this method claim better adhesion, particularly at high temperature, for this type of oxide coating. Each method has proved successful in the manufacture of boards and has shown greatly improved ability of the board to withstand solder treatment compared with untreated copper.

2.6.4.2 DOUBLE TREATMENT OF COPPER

Another method is to use double treated copper. With this, treatment is applied to both sides of the copper by the copper foil supplier as it is delivered to the laminator. The laminator then puts the primary side treatment down, leaving the smooth copper with its treatment exposed for PCB processing. This eliminates the need for a black oxiding step, and good boards able to withstand solder conditions are produced. Some of the problems with the double treat method include the difficulty of photo exposing this type of surface which may not be uniformly reflective and the difficulty of subsequently removing the polymerised photoresist from the highly porous roughened surface prior to laminating. Cleaning of the double treat is also far from easy because mechanical scrubbing is not possible and the surface is very susceptible to chemical attack. Despite these drawbacks, however, double treated copper has been widely used in the multilayer industry to good effect. The choice of techniques between oxiding after print and etch or the purchase of already treated copper foil depends upon the individual requirements of the customer and the capabilities of the PCB manufacturing facility.

Chapter 3

THE MANUFACTURE OF WOVEN GLASS FABRIC

D. J. VAUGHAN and C. G. HERSCHDORFER
Clark-Schwebel Fiber Glass Corp., White Plains, NY 10604, USA

3.1 INTRODUCTION

Over the centuries, glass has been one of man's most useful materials. According to legend, its discovery is attributed to nomadic tribesmen who, some 10,000 years ago, found molten glass in the bed of a hot campfire built upon sand. The inquisitive tribesmen poked at the molten pool with a stick and drew out coarse glass fibres. These fibres obviously had no immediate utility and, at the time, promised none in the future to the men around the fire.

The first evidence of the production and utilisation of glass fibres can be found in 'L'Arte Vetraria' by Antonio Neri, published in Florence in 1612, and a report to the Paris Academy of Science by René de Reaumer in 1713. However, it was not until the late 1930s that a successful commercial operation for the manufacture of glass fibres became a reality. It was in 1931 that the Owens Illinois Glass Company began its search for an industrial glass fibre process, followed in 1935 by the Corning Glass Works. These two companies formed the Owens-Corning Fiberglass Corporation in 1938 to produce and market glass fibre and glass fibre products. Today, glass fibres are produced in a variety of forms by several major manufacturers.

3.2 GLASS FIBRE PRODUCTION

The commonly accepted definition of glass is: 'An inorganic product of fusion which, when cooled, becomes rigid without crystallising.' The specific chemical and physical properties of glass fibres are controlled by the composition of the glass and the processes by which the fibres are drawn. It is possible to draw many glasses of varying compositions into fibres. These fibres are produced in two general forms—as continuous filaments, or as staple fibres. The largest proportion of continuous filament material is drawn from a stream of molten glass at speeds often exceeding two miles per minute.

In the production of glass fibres, a glass batch is compounded and mixed with great care to ensure consistent glass composition. Production of

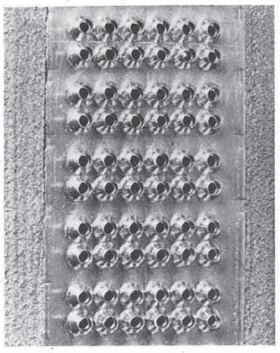


Fig. 3.1 Bottom of typical bushing showing the numerous orifices from which the glass filaments are drawn.

continuous filament fibres commences by feeding the raw glass (in the form of marbles or previously formulated batch) into an electrically heated fibrising element called a 'bushing' (Figure 3.1). The bottom of a typical bushing has 204 or more orifices, each of which is generally in the range of 0·8 mm to 3·1 mm in diameter. Once inside the bushing in a molten state, the glass composition flows through the openings in the form of glass streams. The amount of glass above the orifices, the viscosity of the melt, the number and size of the orifices, and the rate at which fibres are pulled from the bushing influence the diameter of the filaments which are produced (Figure 3.2).

A 'sizing applicator' is located just below the bushing (Figure 3.3). Its function is to apply an organic sizing material in aqueous dispersion to each filament. This sizing performs a threefold function: it acts as a film former to prevent the fibres from abrading each other, as a lubricant so that filaments can move relative to one another and, in some instances, it can provide a compatible interface between the inorganic glass and the organic resin matrices with which it is used.

Below the sizing applicator, the filaments are gathered into a bundle called a 'strand' by means of a 'gathering shoe' (Figure 3.4). The strand passes from the gathering shoe to a winder where it is wound onto a forming tube. This material is often referred to as a 'forming cake' (Figure 3.5). When this cake has been built up to the desired size or weight, it is removed from the winder and dried in an oven. This removes the water associated with the size application.

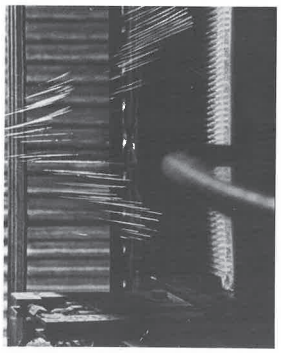


Fig. 3.2 Glass filaments being drawn.

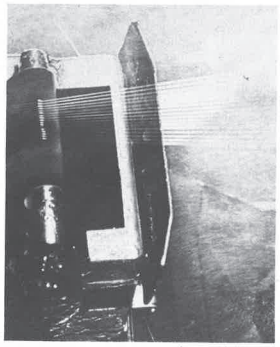


Fig. 3.3 Size applicator.

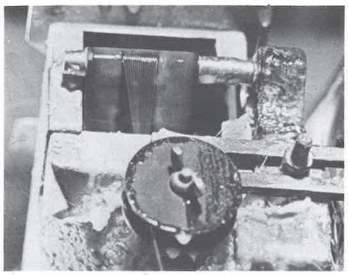


Fig. 3.4 Strands of glass filaments being gathered on the shoe.

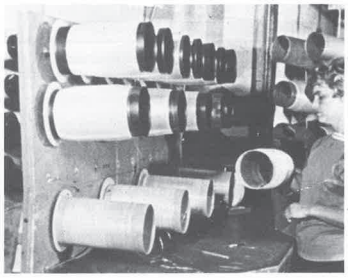


Fig. 3.5 Filaments gathered together to produce a forming cake.

3.2.1 Glass Composition

The major constituent of most commercial glasses is silica. The silicon dioxide molecule has a tetrahedral configuration consisting of a central silicon ion surrounded by four oxygen ions. This three-dimensional network of silicon tetrahedra is the basis for the various properties associated with glass. By the addition of modifying ingredients (such as metallic oxides), the properties of the amorphous glass can be varied and adjusted to give

various levels of performance which may become part of, or disrupt, the silica network.

The first glass developed specifically for production of continuous fibres was a lime-alumina-borosilicate glass that was designed primarily for electrical applications and designated 'E' glass. This glass was found to be adaptable to a great variety of processes, products and applications. 'E' glass does not have a single composition, and can vary within the range shown in Table 3.1.

Table 3.1

Type	*Composition*										
	SiO_2	Al_2O_3	CaO	MgO	B_2O_3	Na_2O	K_2O	ZrO_2	TiO_2	PbO	F_2
Low-alkali, lime-alumina borosilicate	54·5	14·5	18·0–22·0		0–8·5	0–0·5					
Soda-lime borosilicate	65·0	4·0	14·0	3·0	5·5	8·0	0·5				
Soda-lime borosilicate	59·0	4·5	16·0	5·5	3·5	11·0	0·5				
Soda-lime	73·0	2·0	5·5	3·5		16·0					
Lime-free soda borosilicate	59·5	5·0			7·0	14·5		4·0	8·0		2·0
High-lead	34·0	3·0				0·5	3·5			59·0	

Since these variations do not significantly influence its electrical or mechanical properties, each glass producer can tailor the composition to his raw material economics and production processes.

3.2.2 Yarn

Continuous filament glass fibres normally have diameters ranging from 2·5 μm to 19 μm. A yarn is an assembly of such fibres suitable for use in weaving into fabric. The most simple yarn is one consisting of individual strands of glass fibres, usually referred to as 'singles' yarn. Heavier yarns can be obtained by combining single strands in twisting and plying operations (Figure 3.6). This is accomplished by twisting two or more single strands together and subsequently plying. A yarn strand has what is termed an 'S' twist if, when held in a vertical position, the spirals conform in slope to the central portion of the letter 'S' (Figure 3.7). It has a 'Z' twist if the spirals conform in slope to the central portion of the letter 'Z' (Figure 3.8). Strands which are simply twisted will kink, corkscrew and unravel because they are twisted in one direction only. The plying operation normally eliminates this by countering the twist in the singles yarn with an opposite twist in the plied yarn. Twisting and plying of the yarn vary its strength, diameter and flexibility, and are important steps in producing the variety of fabrics necessary to the laminate manufacturer.

A system for identifying glass fibre yarns is required because of the wide variety of the types produced. Yarn nomenclature consists of two basic parts, alphabetical and numerical: (for example, 'ECG 150 2/2'). The first letter identifies the glass composition, in this case electrical glass. The

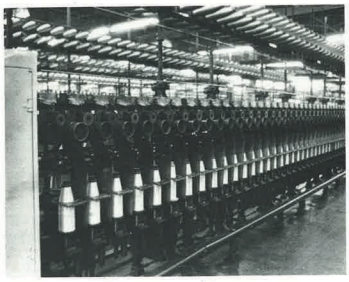


Fig. 3.6 Twisting and plying operation.

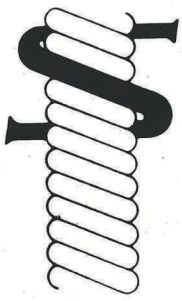


Fig. 3.7 'S' twist yarn.

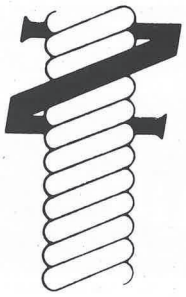


Fig. 3.8 'Z' twist yarn.

second letter 'C' shows that the glass is a continuous fibre. The third letter 'G' indicates the average diameter of the fibre from which the yarn is made. Therefore, the letters 'ECG' identify the yarn as 'E' glass composition, produced as a continuous filament, of average fibre diameter within the range of 8·2 μm to 10 μm. The numbers used in the identification indicate the basic strand weight and yarn construction. The strand weight is shown in the first series of numbers following the letters. It numerically represents approximately 1/100 of the yards per pound of basic glass fibre strand. The second series of numbers (such as 1/2, 2/2, 1/0) designates the yarn construction; this shows how many basic single strands are in the yarn. The first digit indicates the number of basic singles twisted; the second digit separated from the first by a diagonal line designates the number of the twisted strands which are plied together. To establish the total number of strands in the yarn, multiply these two numbers together (0 is multiplied as 1). The approximate yards per pound of fabricated yarn can be computed by dividing the basic strand weight by the number of strands in the yarn.

Table 3.2

Diameter (Inches)	*Diameter (Microns)*	*Filament Designation*	*Filament Strand Count*	*Number of Filaments/ Strand*	*Decitex*
0·00021	5	D	1800	51	28
0·00021	5	D	900	102	55
0·00021	5	D	450	204	110
0·00029	7	E	225	204	220
0·00025	6	DE	150	408	330
0·00036	9	G	150	204	330
0·00051	13	K	75	204	660
0·00051	13	K	37	408	1340
0·00036	9	G	75	408	660
0·00036	9	G	37	816	1340

An example of how the glass yarn nomenclature can be interpreted is 'ECG 150 1/2':

E: Electrical glass
C: Continuous filament
G: G Filament diameter range 8·2 μm to 10 μm
150: 15,000 yards per pound of basic strand
1/2: Single strands were twisted and two of these twisted strands were plied together

$$\frac{15{,}000}{2} = 7{,}500 \text{ yards per pound for fabricated yarn.}$$

3.2.3 Warping

The first step in the weaving process is from the package of yarn to the

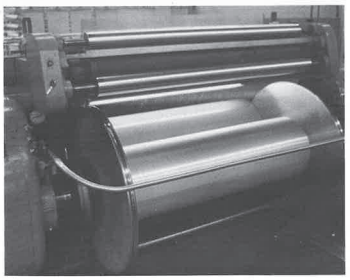


Fig. 3.9 Warp yarn gathered together into a beam.

warp beam. Here, literally millions of yards of yarn are drawn into parallel formation onto large, polished metal cylinders known as 'beams' (Figure 3.9). Each beam contains approximately 2,200 pounds of yarn.

3.2.4 Slashing

The continuous glass filaments on the beam are sensitive to abrasion and are difficult to process without causing severe damage to their surface. One method of reducing this and future damage is to treat the yarn chemically in a process known as 'slashing'. In this operation, the singles yarns pass through a formulated size bath where they are coated with the size material; the coated yarns then pass through a drying unit that removes excess moisture and sets the size on the yarns' surface (Figure 3.10).

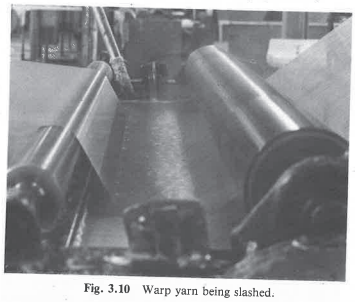


Fig. 3.10 Warp yarn being slashed.

3.2.5 Entering

After slashing, the loom beam is ready for the process known as 'entering'. In this step, individual yarn ends from the loom beam are drawn through an intricate series of harness dropwires and heddles which produce the desired pattern during the weaving operation (Figure 3.11).

3.2.6 Quilling

Woven fabrics (and glass fabric is no exception) are traditionally composed of two sets of perpendicularly interlaced yarns. The preparation of the loom beam produces the warp threads and, in order to obtain the filling or weft yarns, the glass filaments are wound onto a small spool or quill in an operation known as 'quilling' (Figure 3.12). The quill is inserted into the shuttle which, during the weaving operation, deposits a strand of filling yarn perpendicular to the warp yarn as it travels back and forth on the loom.

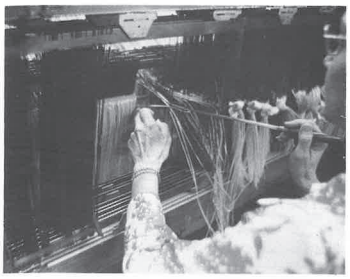


Fig. 3.11 Yarn from the loom beam in the process of 'entering'.

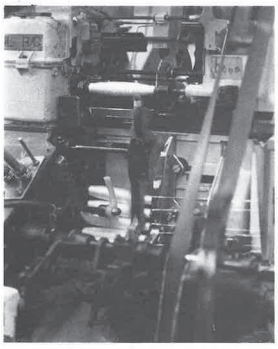


Fig. 3.12 Fill yarns wound onto bobbins in a process known as 'quilling'.

3.2.7 Weaving

Fabric is usually described as 'a material constructed of interlaced yarns, fibres or filaments'. Typical glass fibre fabrics are manufactured by the interlacing of warp and fill yarns on a conventional weaving loom. Such fabrics can be woven in a variety of styles which allow for good control over thickness, weight and strength. The principal factors that designate a given fabric style are count, warp yarn, fill yarn and weave.

The yarn count refers to the number of warp yarns or ends per inch and the number of filling yarns or picks per inch. For example, a fabric count of 44×32 means that there are 44 ends per inch in the warp or lengthwise direction and 32 picks per inch in the fill or crosswise direction. This fabric count and the properties of the warp and fill yarns used in the weaving of the fabric are the main factors which determine the fabric strength.

The weave of a fabric refers to the manner in which warp yarns and fill yarns are interlaced. The weave also determines the appearance and some of the handling and functional characteristics of a fabric. Among the most popular weave patterns are the following:

1 Plain Weave: This is the oldest and most common of the textile weaves. In this weave, one warp end passes over and then under one filling pick.
2 Twill Weave: This weave has one or more warp ends passing over or under two or three or more fill picks in a regular pattern.
3 Crowfoot Satin: The weave of this fabric is one warp over three and under one fill pick.
4 Longshaft Satin: This weave is constructed with one warp end weaving over four or more and under one filling pick.
5 Leno: This weave has two or more parallel warp ends interlocked.
6 Unidirectional: A weave involving a great number of larger yarns in one direction with fewer and generally smaller yarns in the other direction.

Owing to its dimensional stability, the plain weave style fabric is the one most commonly used in the production of printed circuit board materials.

3.2.8 Heat Cleaning

The sizing material previously described in this Chapter serves as a continuous filament binder which affords a measure of protection to the filament during the weaving operation. This size is usually of an organic nature, consisting of starch or dextrinated starch gelatin, a complex amine and, in the majority of cases, a lubricant (such as mineral, animal or vegetable oil). The size (or 'binder', as it is sometimes called) is present on the glass fabric to the extent of 1% to 4% (Figure 3.13), and must be more or less completely removed from the fabric prior to the application of any form of coupling agent. It is important that the fabric be de-sized in a

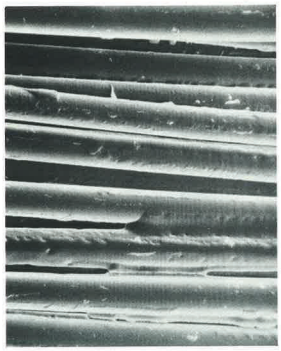


Fig. 3.13 SEM showing the organic size present on the glass fabric.

manner that will produce the whitest fabric possible, while maintaining a good proportion of the initial strength of the material. This is normally achieved by reducing the organic content of the fabric to 0·8%.

The methods currently used in commercial production to achieve this de-sizing are predominantly thermal, and can be classified as follows:

1 Low temperature or batch oven cleaning: In this operation, the glass fabric is rolled onto a perforated metal mandrel; the rolls are then placed in an oven maintained at a temperature range of 260°C to 371°C for between 55 to 80 hours. The fabric thus treated has an organic content of approximately 0·1%.
2 Continuous high temperature cleaning: This de-sizing operation allows the glass fabric to pass in one continuous operation through a vertical or horizontal furnace that subjects it to temperatures of 593°C to 816°C, along with atmospheric conditions which cause rapid burning of the size in a band across the width of the fabric.
3 Continuous low temperature cleaning or carmellising: In the case of continuous low temperature heat cleaning (or 'carmellising', as it is sometimes called), the furnace temperature is kept sufficiently low to prevent actual burning of the size at the fabric's surface. The large amount of carmellised organic residue left at the fabric's surface imparts a distinct brown colour to the material, and generally renders it unsuitable for use in electronic applications.

3.3 Finishing of Glass Fabric

The earliest attempts to apply organic sizes to glass fibres were carried out in the mid-1930s by Slater and Thomas[1], and were directed toward maintaining the physical integrity of the glass to which they were applied immediately following the drawing operation. Continuation of this approach led to the development of the starch oil size by Steiman *et al*[2], which not only gave protection to the fibre after drawing, but also afforded sufficient protection for the weaving operation.

During this period, Hyde[3] carried out work covering the use of silicon based materials as sizing components. These materials were capable of reacting with the glass surface, but not with the resin matrix. Further evaluations were performed abandoning all forms of treatment and using heat cleaned glass fabric. The composites produced gave adequate properties in the dry state but, upon exposure to high humidities, the glass-resin bond failed in less than two hours (Figure 3.14). The development of

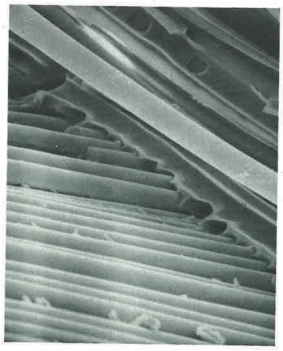


Fig. 3.14 SEM showing the failure of adhesion at the glass filament-resin interface.

coupling agents as they are known today was undertaken in the mid-1940s with the work of Steinman[4] on methallyl silicate, and Goebel and Iler[5] on methacrylate chromic chlorides. Although this work was intended to improve the adhesion of the glass fibre and resin matrix, both the vinyl silanes and chrome complex interfaces are susceptible to attack under aqueous conditions, thereby weakening the glass-resin interface and, ultimately, the properties of the composite (Figure 3.15).

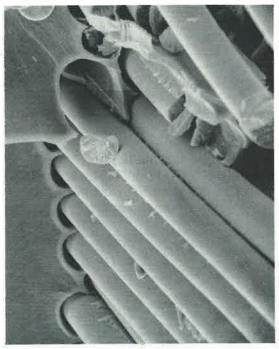


Fig. 3.15 SEM showing the effects of moisture on the glass resin interface where the glass fabric had been previously finished with a chrome complex.

A variety of materials (predominantly organo-functional silanes) have since been developed in an attempt to provide a stable interface under a varying number of adverse environments. These coupling agents can generally be described as molecules which possess two different kinds of reactivity. The silicon portion of these molecules has an affinity for the glass fibre, while the organic portion of the molecule is a stable graft onto the silicone atom that is tailored to combine with many of the organic thermosetting resins used in composite manufacture. The main function of this coupling agent is to provide a stable bond between two dissimilar surfaces. Although the standard properties of composites manufactured from fabric finished with these coupling agents showed improvements in the dry state, the most dramatic differences could be observed after exposure to water, atmospheric moisture, chemical solutions and high temperatures.

The majority of coupling agents have the general formula: $RCH_2CH_2CH_2Si(OCH_3)_3$, where R is a reactive organic group tailored to match the reactivity of the resin system with which it will be used. The silane coupling agents will perform with a wide variety of inorganic surfaces, the degree of performance depending on the number of reactive sites at the surface of the substrate which are capable of chemically reacting with the silanols (glass, for example). This is dependent on the density of SiOH groups present on the surface. In general, performance increases with silanol density:

$$\text{Glass - SiOH} + \text{RSi(OH}_3\text{)} \rightarrow \text{Glass-SiOSiOH} + \text{H}_2\text{O}$$

OH

R

It is usually thought that the silicon portion of the coupling agent has orientated towards the surface, and that the organo-functional group orientates away from the surface to react with the resin system being used. Coupling agents are usually applied to the woven glass fabric from a water solution, although non-aqueous solvent forms have also been used. The equipment used to apply the coupling agents is, in many ways, comparable to the impregnating machines used by the prepregger (Figure 3.16), consisting of a treatment bath, metering rolls and drying chambers.

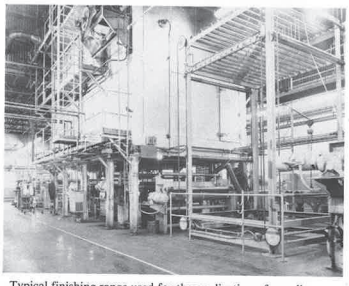


Fig. 3.16 Typical finishing range used for the application of coupling agents to the glass fabric.

3.4 THE COUPLING AGENT-RESIN INTERFACE

In surveying the technical literature, it appears that there are many theories of adhesion systems[6-10]. However, most of the current data indicate that the formation of covalent bonds between the resin and the coupling agent provides one of the strongest and most enduring interfaces. Other factors (such as wettability, compatibility and surface energies) certainly influence the quality of the reinforced composite, but the degree of chemical reaction at the interface usually determines the resistance of the composite to the everyday industrial environmental conditions to which it is exposed. It has been demonstrated that both resin composition and processing temperatures can affect the performance of a given coupling agent. This is to be expected since there are several competing reactions that

can take place at the coupling agent-resin interface: homopolymerisation of the resin or the coupling agent, and copolymerisation of the two. Matching these rates of reactivity as closely as possible should result in the formation of a maximum number of bonds across the interface.

In the development of reinforced composites, the fabricator is usually concerned only with the coupling agent-resin interface because the bonding between the siliceous substrate and the coupling agent is generally taken for granted. Therefore, greater effort goes into grafting the best organo-functional groups onto the silicon than into modifying the trimethoxysilyl portion of the molecule.

3.5 THE APPLICATION OF FINISHED GLASS FABRIC TO MULTILAYER PREPREG AND CIRCUIT LAMINATES

The production of a multilayer printed circuit board is the result of combining three or more vastly different raw materials in different physical states into one compact product. The multilayer printed circuit laminate consists of two or more copper-clad epoxy glass laminates with specific circuit patterns. These laminates are bonded together with prepreg which has, in turn, been manufactured from woven glass fabric and epoxy resin. This prepreg, upon application of heat and pressure, flows to fill the voids in the circuit pattern and forms the bond with the adjoining laminates. During the manufacturing process, the laminates are subjected to prolonged heat and pressure. Also, these substrates are much thinner than those encountered in conventional printed circuit laminates and, as such, approach the limits of the material. These combined factors make the selection of the base materials even more critical than for the standard printed circuit laminate.

The majority of fabrics presently being used in multilayer applications are plain weaves manufactured from plied or singles yarns. These plain weave fabrics produce the most consistent and uniform thickness, and offer a smooth, even surface for resin impregnation. The fabric's construction must also be taken into consideration as this can affect the dimensional stability of the laminate. Although there is no conclusive evidence at present that any one parameter contributes more to the dimensional stability of thin laminates than another, it is known that certain fabric styles perform better than others in this respect. For example, a fabric with a balanced construction (that is, one with the same number of yarns in the warp and fill directions, also the number of filaments per yarn and the filament diameter being similar) will present a material that imparts good dimensional stability to the final laminate.

Glass fabric styles 104,108,112 and 116 as specified in military specification MIL-G-55636A are commonly employed in multilayer applications. Of course, this list is not complete and, in practice, a large number of plied and singles yarn constructions are used, the choice being based on the thickness of the fabric.

At a given resin content, the prepreg thickness bears a distinct relation to the choice of the original fabric used, allowing the engineer to design the lay-up of the multilayer laminate to be used.

Table 3.3

Multilayer Fabric Styles

Plied Yarn	Singles Yarn	Thickness (mm)	Thickness (in.)
104	–	0·025	0·0010
325	–	0·036	0·0014
105	–	0·038	0·0015
106	–	0·038	0·0015
107	1070	0·043	0·0017
108	1080	0·050	0·0020
10860	–	0·056	0·0022
112	2125	0·076	0·0030
113	2113	0·076	0·0030
116	216	0·122	0·0048

Because the fibres used in multilayer fabrics are smaller, finer and more fragile, great care is taken during preparation of these yarns to ensure that they are adequately protected by the size that is applied. The accumulation of fuzz, often small enough to be insignificant on other fabric styles, creates magnified problems in the finer multilayer styles (Figure 3.17). However, most of the fuzz accumulation can be attributed to broken filaments, a problem created in the yarn manufacturing operation (Figure 3.18). In view of the very limited number of yarn suppliers world wide and the variation in quality of these fibres, it is not always possible for the glass fabric weaver to supply the high quality sought by the multilayer laminator.



Fig. 3.17 The accumulation of fuzz on the surface of the woven fabric.

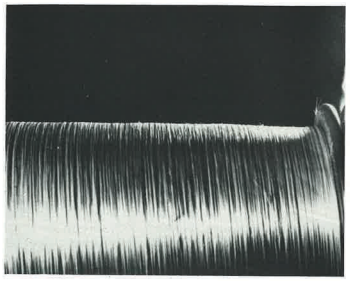


Fig. 3.18 Broken filaments present on the surface of incoming yarn.

3.6 CONCLUSION

In order to improve the performance of glass fabrics as a reinforcing medium in the electronics industry, work has been carried out on many aspects of production, with continuing progress being made in manufacture, construction, and in the coupling agents employed. To continue to improve the laminate's performance, however, it is imperative that yarn producers, glass fabric weavers, resin chemists, copper foil producers, and laminators work in close liaison.

Knowledgeable industrial sources suggest that glass reinforced electrical grade laminates will continue their growth pattern during the next few years. A high percentage of this growth will be in the ultra-thin, multilayer market. These high strength, lightweight constructions of resin and glass fabrics are continually proving their value in the electronics field, often replacing more conventional materials and adding their improved performance and flexibility of design to the final product.

REFERENCES

1 Slayter, J. and Thomas, J.H., US Patent 2, 234, 986, 'Mechanically Drawing Fibres', March 18 (1941).
2 Steinman, R., *et al*, US Patent 2,688,006, 'Composition and Process for Improving the Adhesion of Resins to Glass Fibres Utilising Hydrolised Vinyl Alkoxy Silane', August 31 (1954), and US Patent 2,688,007, 'Composition for Coating Glass Fibres', August 31 (1954).
3 Hyde, J.F., US Patent 2, 315, 259, 'Treating Glass Fibres', March 30 (1943), and US Patent 2,439,689, 'Method of Rendering Glass Water Repellent', April 13 (1948).
4 Steinman, US Patent 2, 688, 006.

5 Goebel, M.T. and Iler, R.K., US Patent 2,544,667, 'Werner-Type Chromium Compounds as Laminating and Casting Composition', March 13 (1951) and US Patent 2,544,688, 'Werner-Type Chromium Compounds', March 13 (1951).
6 Adler, O.E., *Adhesives Age*, **12** (1), 30 (1969).
7 Ashbee, K.H., *et al, Proc Roy. Soc.* A300 415.
8 Barrie, J.A. and Machin, D., *A.C.S., Div. Polymer Chemistry Preprints*, 9 (2) 1489.
9 Bascom, W.D. and Ramans, J.B., *Ind. Eng. Chem. Res. Development*, 7, 172 (1968).
10 Erickson, P.W., *Proc. SPI, Reinforced Plastics/Composites Div.*, 24th Sec. 19-B (1969).

PHOTOMECHANICAL PRINTING

INTRODUCTION

Photomechanical printing comprises the transferring of the required track and pad pattern to the board laminate by the use of a light sensitive layer applied to the face of the laminate.

The two forms of such photoresists in common use in the PCB industry are Liquid Resists and Dry Film Resists. The most widely used are the dry films, which have many advantages over the older liquid resists, chief of which are ease of application and their thickness which allows them to serve as near vertical walls to limit any spread of plating on pattern plated boards.

However, when very fine lines or gaps (less than 0·05 mm or 0·002 in.) are required, the thickness of dry film resists becomes a major disadvantage. Effects such as refraction in the dry film limit the resolution achievable and recourse has to be made to the older and thinner wet resists.

After bonding, the outer faces of a multilayer board can be out-of-flat where the outer thin laminates have conformed to the track patterns on inner layers. This out-of-flatness can be great enough to affect the satisfactory lamination of a dry film resist, making the use of a wet resist desirable.

On buried via-hole and flexi-rigid boards it may be necessary to apply a resist over an etched and possibly plated track pattern to plate selectively pads or through holes. Here again the conformation of a dry film may prove inadequate and a wet resist must be used.

Whenever either type of photoresist is used on an out-of-flat surface, care must be taken in the design to ensure that the actual edges of the patterns required are all on portions of the board which are flat and at the same height so that good contact can be made with the printing film.

J. A. Scarlett

Author Index

(see also separate Subject Index)

Abolafia, O. R. 305
Adler, O. E. 94
Armstrong, E. 180
Armstrong, R. 422
Arrowsmith, D. F. 355
Ashbee, K. H. 94
Ayerst, G. G. 180

Baillie, R. J. 60
Baker, R. G. 290
Banner, H. I. 60
Barrie, J. A. 94
Bartlett, C. J. 291
Bascomb, W. D. 94
Bauer, J. 153
Bellows, G. 290
Benton, R. R. C. 180
Bhattacharyya, A. 290
Blum, W. 355
Borth, P. 180(2)
Bowden, F. P. 291
Brenner, A. 355
Burr, R. P. 530–77

Calder, A. S. 487, 527
Cannon, R. 290
Carboni, R. A. 153
Caropreso, F. E. 180
Carter, R. C. 291
Castonguay, R. 305
Cheale, L. F. 487
Cook, N. H. 291

DeForest, W. S. 110, 152
Devries, M. F. 291
Diggory, B. A. 180
Dorfman, H. 305
Doughty, H. W. 180
Duckett, R. 488
Duffek, E. 180
Dunn, B. D. 372
Dyke, D. E. 291

Edwards, J. 355
Eisler, P. 110, 179
Elmore, G. V. 305
Engelmaier, W. 60
Erickson, P. W. 94
Ernst, H. 290

Foerster, R. P. 60
Forest, P. 180(2)
Form, G. 291
Franklin, B. 180
Fullwood, L. 153

Galloway, D. 290(2)
Gergenda, C. G. 422
Gillespie, L. K. 291
Goebel, M. T. 94
Gray, F. 422
Grondahl, V. J. 291

Hacke, H. V. 527
Hackett, E. 153
Hagge, J. K. 236–90, 291
Haring, H. 355
Harris, D. L. 152
Harris, I. R. 291
Hartmann, H. 422
Heath, H. E. 60
Hedgecock, R. 527
Held, J. R. 527
Henningsen, C. G. 291
Herschdorfer, C. G. 76–93
Hickman, J. J. 153
Hoar, T. P. 355
Howe, L. 527
Hurd, D. E. 60
Hurley, H. C. 60
Hutt, M. J. 179
Hyde, J. F. 93

Iler, R. K. 94

Jacques, M. E. 291
Jennings, C. W. 60
Johnson, D. D. 152

Kaupp, H. R. 527(2)
Kelemann, D. G. 153
Kingsley, D. S. 367–73, 423–9, 430–49
Klein Wassink, R. J. 373, 488, 527
Kobayashi, A. 290(2)
Kocsis, S. 290
Kolias, J. T. 60(2)
Kolkman, J. 422
Kosar, J. 110

Kosmowski, W. B. 291(2), 305

Landall, A. 291
Leach, F. A. 527(2)
Leonida, G. 373
Lowenheim, F. 355
Luke, D. A. 306–25, 326–55, 355
Luke, D. M. 60, 527
Lukes, R. M. 325
Luttmer, W. 527
Lyde, D. M. 355

McGowan, D. R. 422
Machin, D. 94
McKeone, J. 180(2)
McNeal, T. I. G. 422
Manko, H. H. 488, 527
Mather, J. C. 236–90
Mayberry, D. W. 291, 422
Mayhew, A. J. 487(2), 527
Menzies, G. D. M. 153, 356–66
Messner, G. 530–77
Missel, L. 180
Mitchell, J. P. 60
Morris, R. E. 180
Morrisroe, C. T. 205–35
Morton, A. 291
Muckett, S. 488
Murphy, F. D. 180

Nitsch, H. 60
Noller, C. R. 305

Ogden, C. A. 305
Oxford, C. 290

Parks, H. L. 60
Parry, N. J. C. 409–22, 422(2)
Paul, G. R. 305
Powell, H. 153
Price, J. W. 373

Radimer, K. J. 180
Ramans, J. B. 94
Rider, D. K. 110
Roller, M. B. 422
Ross, W. M. 166–80, 179, 489–97
Rust, R. D. 291

Sallo, J. S. 61–75
Sard, R. 290
Scarlett, J. A. 1–59, 60(4), 95, 181–204, 205–35, 374–408, 450–77, 478–88, 499–526, 527(5), 528–9

Schlabach, T. D. 110, 180
Schuessler, P. 60
Schwartz, L. 60
Scrupski, S. E. 152
Senderoff, S. 355
Severin, G. 152
Shaw, M. 290
Shaw, M. C. 290
Shoho, R. 290
Simpkins, D. 96–110
Simpkins, D. C. 180
Slayter, J. 93
Slominski, L. J. 180
Smith, B. R. 60, 527
Smith, G. A. 291
Smith, J. 153
Springfield, W. K. 527
Steinman, R. 93
Stone, K. R. 488
Strong, G. G. 355
Suh, N. P. 290

Tabor, D. 291
Tench, D. M. 305
Thiveyrat, M. 291
Thomas, J. H. 93
Tsukada, T. 290
Tucker, W. B. 152
Tull, L. E. 291

Van Krevelen, D. W. 422
Vaughan, D. J. 76–93

Wallace, T. J. 152
Wallig, L. R. 111–53, 152(2), 153
Warwick, M. 488
Watter, C. 154–65
Weber, W. E. 291
Weinhold, M. 153
Weintz, W. W. 305
Weiss, R. E. 291(3)
Wells, F. H. 355
Welsher, T. L. 60
Weng, G. 290
Weyde, B. 290
Whitlaw, K. J. 355
Wild, R. N. 60, 355, 527
Williams, R. A. 290
Wilson, G. C. 372, 497
Wise, R. 305
Witherell, D. R. 292–305, 305
Wopschall, R. H. 153(2)
Wrenner, W. R. 291

Subject Index

(see also separate Author Index)

abrasive brushing 124–5, 359–60
abrasive jet machining 289
acceptable quality level 375–6
access holes 11–14
acid copper sulphate solutions 330–31
acid dip
  electroplating 327
actinic intensity 131–2
activation
  electroless plating 315–21
adaptive control
  machining 290
additive processes
  dry film resists 140–42
additives
  liquid photoresists 99
adhesion
  copper
    cleaning problems 358
    electroless plating 310–11, 324
    flexi-rigid boards 451
    laminate manufacture 70–71, 74–5, 187–9
    testing 389
  metal-to-metal
    troubleshooting 148, 152
  photoresists
    laminate manufacture 126
    troubleshooting 147–8, 150
  see also bonding
ageing
  prepreg testing 69
  thermal tests 392–4
agitation
  electroplating 333
air drying
  liquid photoresists 104
air inclusions
  bonding defects 201–2
air purging
  water-cooled presses 212
alignment
  filmwork 28–9, 30–33
alkaline etchants 172
ammoniacal etchants 172
ammonium persulphate
  etchants 170
analysis
  electroless copper solutions 323–4
  pre-etchants 314–15
anodes
  affecting distribution
    copper plating 333, 334
aqueous processable resists 118–19, 120–21, 122, 135–6
  troubleshooting 151–2
architecture
  design considerations 8–9
artwork *see* filmwork
asymptotic heating 370
automation
  electrical continuity testing 403–8
  press cycles 226

B-triazine 63
baking
  laminate manufacture 72, 73, 225–6, 417, 421
beams
  glass yarn 83
bevelling 257–8
binder
  glass cloth manufacture 86–7
  photopolymerisation 115
bisphenol-A epichlorohydrin 62
bits
  drilling 265–8
  routing 247–9
black oxide *see* oxidation
blanks
  contour routing 244
'bleed and feed'
  dry film resists 136
blotting paper
  packing 198
blow holes
  soldering 55–6, 318–19
'bomb-sight' targets
  filmwork 28, 461
bonding 34
  action of prepreg 199–200
  and oxidation 187–9
  buried via-hole boards 44
  defects 201–4
  delamination 55
  doubled boards 45
  flexi-rigid boards 460, 466–70
  heat ladders 46–8

  no. of blanks per cycle 233–4
  pins 191–2
  quality control 380
  tooling holes 182–3
  see also adhesion
bonding presses *see* presses
borders
  design considerations 513
  filmwork 28–9
bowing
  board defects 202–3
breakout
  drilling 258–61
British Standards
  4584 493, 495
  4597 493
  5370 431
  6221 494
  9760 431–2
  9761 493
bromine
  and epoxy resins 64
brushing
  substrate preparation 99–100, 124–5, 359–60
bubble etching 174
bulk polymerisable resists 132
Burghard Report 492
buried via holes 17–19, 42–5
  build and thickness of board 52
  design considerations 524
  electrical continuity testing 406
  inner layer preparation 185–6
burr cutters 248–9
burrs
  drilling 277–8
    removal 273–4
bushing
  glass fibre production 77

capacitance
  design considerations 504
  Multiwire boards 563–6
carbide
  tool material 239–41, 265
carbonisation
  board faults 58
carmellising 87
catalysts
  electroless plating 308–9, 315–21
cathodes
  gold plating 350–51, 352
CECC 492
CENELEC 491–2
chamfering 257–8
characteristic impedance
  design considerations 501–4
  Multiwire boards 561–3
chemical cleaning
  smear removal 294–301
  substrates 124
Chemical Discharge Test 546
chemical milling 138–9, 289
chip carriers
  and laminate stability 421–2
chips
  machining 238, 244–5, 253–4, 262, 272–3
    drilling problems 280
    removal 241, 256, 257
Chipstrate 422
chromic acid
  etchants 170–71
  smear removal 300–301
circuit diagrams
  electrical continuity testing 407
cleaning
  copper 356–66
    and oxidation 188
  glass cloth 86–7
  holes
    Multiwire boards 544
  pretreatment
    electroplating 326, 327
  substrates, 99–100, 123–5, 134–5
clearances, thermal
  design of planes 513–20
coating
  liquid photoresists 100–104
co-axial interconnects 21–2
cold start press cycles 222
colourants *see* dyes
component simulating heads
  electrical continuity testing 405–6
computer-aided design
  Multiwire boards 547–54
condensation reflow
  solder finishes 371
conditioners
  electroless plating 311–13
conductive anodic filaments
  short circuits 57
conductive compound
  repair methods 441
conductivity, electrical
  plated copper 338
conductivity, thermal
  Multiwire boards 569–70
conductors *see* tracks
conformal sheet 197–8
  flexi-rigid boards 459
contamination
  and short circuits 57
  copper plating 339–40
  prepreg 66, 197
  see also cleaning
contour routing 243–57
conveyorised machines
  bonding press loading 214–18
  laminate handling 54, 461
  oxidation 189
  photoresist development 109
  photoresist drying 104–5

cooling
  bonding presses 210–13, 229–30
  thermal problems 7–8
copper
  adhesion *see* adhesion: copper
  and laminate stability 414
  around tooling holes 185
  burrs after drilling 277–8
  cleaning 356–66
    effect on dry film resists 124–5
  electroless plating 306–25
  electroplating 326–40
  PCB design 269, 271, 420
  properties 251
  use of thin copper foil 74
corrosion
  gold plating 349
  nickel plating 346–7
  tin-lead plating 344
costs
  boards 3–6
  dry film photoresists 115
  Multiwire boards 573–5
counterboring 287–8
coupling agents
  glass cloth finishing 88–91
coupons
  testing 383–5, 400–401, 402
    Multiwire boards 547
cover-coats
  flexi-rigid boards 451, 462, 463
crazing
  board defects 203–4
CRC prepreg 67
crimping
  leads 485–6
crosstalk
  design considerations 504–7
  Multiwire boards 566–9
crowfoot satin
  glass cloth 86
cupric chloride
  etchants 171
curing
  problems of 73
curing inks 164
current carrying capacity 500–501
current density
  copper plating 331
curtain coating
  liquid photoresists 103–4
cutting
  machining theory 244–7
  microsectioning 394–5
  prepreg 190
  repair work 434–5, 437
  thin laminates 53
  see also routing

deburring
  after drilling 273–4
defects
  acceptable quality level 375–6
  bonding 201–4
  see also quality control
delamination 55
  bonding defects 202
  drilling defects 281–2, 283–4
design 499–527
  affecting board flatness 202–3
  affecting laminate stability 420–21
  affecting tooling 252, 269, 271
  flexi-rigid boards 454–5, 462, 472–7
  Multiwire boards 547–54
desoldering 439, 478–88, 514
development
  dry film photoresists 118–19, 133–6
    troubleshooting 142–6
  liquid photoresists 107–10
  quality control of filmwork 378
'dewing'
  prepreg 190
dichromate
  emulsions
    screen printing 159
dicyaniamid 62, 68–9
dielectric constant
  laminate manufacture 71
dielectric smear *see* smears
dielectric tearout 286
Differential Scanning Calorimetry 413
dimensional stability
  laminate manufacture 69–70, 73, 91, 409–22
    flexi-rigids 459–61
dip coating
  liquid photoresists 101
distortion
  caused by drilling 275, 277
distribution
  copper plating 332–4
  gold plating 351
  tin-lead plating 340–41
double-double boards 19, 45, 524
double-multilayer boards 19–20, 45
double-sided PCBs
  costs 3
  history 2
  versus multilayer 499–500
double treated copper 75
drill smear *see* smears
drilling 38, 258–87
  control tapes 33–4
    quality control 379
  Multiwire boards 543–4
  problems caused by 72–3
  quality control 381
  repair work 433–4
dry film photoresists 95, 111–53
dry processable resists 117, 119, 120
drying
  copper

    after oxidation 188–9
  filmwork
    screen printing 156
  liquid photoresists 104–5
Ductilomat 337
dyes
  dry film photoresists 128–9

E glass 80
earth-screened boards 20
ECQAC 492
edge connectors
  electrical continuity testing 406
  gold plating 348, 349–50
    preparation 362–5
  plating and profiling 39
  repair methods 442–3
edge fusion 126
edge gapping 134
edge lifting
  dry film resists
    troubleshooting 150
Eisler, P. 2, 96
electric heating
  bonding presses 208–9
electrical parameters
  importance of 3, 4, 6
electrical properties
  design considerations 500–507
  Multiwire boards 528, 555–7, 560–69
electrical testing 401–8
  Multiwire boards 546
electroless plating 306–25
  affecting substrate cleaning 124
  dry film resist processes 140–141
  Multiwire boards 544–5
  quality control 382
electrolytic etching 166
electroplating *see* plating
electropolishing
  cleaning of copper 362
Elliott Bros. 20
elongation
  copper plating 328, 336–8
  gold plating 353–4
emulsions
  screen printing 159–61
encapsulation
  microsectioning 395
entering
  glass yarn 84, 85
environmental control
  etching 168
  filmwork 29
epoxy materials 61–3
  properties 251
epoxy smears *see* smears
etchants 168–74
etchback
  and solvent resistance 72
  plating problems 308
  smear removal 72, 294–301
    quality control 381
etched and filled boards 22
etched and welded interconnects 20–21
etching 166–80
  flexi-rigid boards 459
  pretreatment 358–9
    electroless plating 313–15, 324
    electroplating 327
  quality control 380, 382
evaluation
  microsectioning 396–401
exfoliation
  troubleshooting 148, 152
exposure
  dry film photoresists 116, 128–33
    troubleshooting 143, 144, 145, 146
  liquid photoresists 105–7
  screen printing 161
eyelets
  repair of pads 445–6

fabrication processes *see* manufacturing processes
faults *see* defects
features
  design considerations 510–13
ferric chloride
  etchants 168–9
fibres
  glass
    production 76–87
    removal after etchback 296
Field formula 332
filmwork 24–33
  design considerations 524–6
  flexi-rigid boards 461
  influencing screen printing 155–6
  location 184–5
  quality control 377–8
  tooling 183–4
fingerprints
  on copper 361–2
finishing processes 41
  drilling 273–4
  glass cloth 88–90
  microsectioning 396
fishbowl holes 281, 282
flaking
  dry film resists 151
flatness
  lack of in PCBs 202–4, 386
flexible probe heads
  electrical continuity testing 404–5
flexing
  testing 390–92
flexi-rigid boards 20, 450–77
  build and thickness 52
  costs 5
  production variants 45–6, 47
flow

control
epoxy materials 62, 68
testing
prepreg 67
flow charts
board manufacture 24
bonding, drilling, pre-plating 35
edge connector plating 39
finishing stages 40
flexi-rigid preparation 47
inner layer preparation 25
buried via-holes 43
plating 36–7
flow melting
solder finishes 368–9
fluted cutters 248–9
flutes
drills 263
'flying tails'
flexi-rigid boards 452, 470, 472–7
folding flexi-rigid boards 453, 455
formats
Multiwire boards 532–5, 548, 549
forming cake
glass fibre production 77, 79
Fortin Laminating 67
four-layer boards 50, 51
frames
bonding presses 206–8
screen printing 156, 157
friction
during machining 237, 264
fusing
tin-lead plating 370–71
effect of organic matter 343–4
fuzz
glass cloth 92

gas plasma
smear removal 302–3
gas vents
flexi-rigid boards 459
gathering shoe
glass fibre production 77, 79
gel time
prepreg testing 66–7
glass cloth 64, 76–94
laminate stability 410, 411, 417–19
properties 251
glass fibres
removal after etchback 296
glass transition temperature 63, 73, 292–3, 411–13
gold plating 347–55
edge connectors
preparation 362–5
flexi-rigid boards 464
grinding
microsectioning 396

halo bleaching 317
hardness
copper plating 338
gold plating 354
Haring Cell 331–2
heat
affecting board flatness 203
affecting machining 237, 265, 269
affecting performance 7–8
electroless plating 309
etchants 168–9, 173
exposure of photoresists 107
filmwork 155, 156
press cycles 219–20, 222–6, 226–7, 414–15
affecting viscosity
dry film resists 126–7
and elongation of deposit
copper plating 336–8
causing smear 275, 276, 292–3
dissipation
Multiwire boards 569–70
thermal testing 392–4
evaluation 399
heat cleaning
glass cloth 86–7
heat ladders 7
bonding 46–8
heating
bonding presses 208–10
electroless plating 325
prepreg manufacture 65
vapour phase heating 371
high frequency characteristics
Multiwire boards 560–69
hinged boards
testing 390–92
holes *see* access holes; buried via holes; **plated-through** holes; tooling holes
hot air **levelling**
solder finishes 371–2
hot oil fusing 370
hot start press cycles 222–4
hydraulic systems
bonding presses 213
hydrochloric acid
etching additive 169
hydrogen peroxide
etchants 171, 173–4
hydro-squeegee process
solder finishes 369–70

impedance
design considerations 501–4
Multiwire boards 561–3
inclusions
repair methods 438
infra-red fusing
solder finishes 370–71
infra-red heating
photoresist drying 104–5
prepreg manufacture 65

ink
screen printing 158–9, 163–5
inner layers
electroplating 334–8
evaluation of microsectioning 398
preparation 181–9
buried via-holes 42–4
general introduction 24–34
quality control 379–80
inspection
etchback 303–4
final inspection 382–94
liquid photoresists 109
quality control 374–82
use of radiography 423–9
see also testing
instrumentation
bonding presses 214
insulation, electrical
Multiwire boards 555–7
intensity
light
photopolymerisation 131–2
intermetallic compounds
solder finishes 367–8
International Electrotechnical Commission 490–91, 493
Invar 422
IPC standards 495–6

jet machining 289
joints
desoldering 478–88
Multiwire boards 557

Kevlar 421
kiss pressure cycles 220–21, 225
Kraft paper
packing 197, 221, 225

laminates 61–4, 69–75
design considerations 507–10
general 34
handling problems 52–4, 196–7
quality control 379
stability *see* stability: laminates
smear removal 293–4
lamination
dry film resists 125–7
mass lamination 46
Multiwire boards 543
solder masks 138
laser machining 289
layer numbering 521–2
laying-up 195–7
buried via-hole boards 44
lead
tin-lead plating 340–45
leads
desoldering 481–6
legends
design considerations 522
final inspection 387
leno
glass cloth 86
'lift-off'
screen printing 162
light
exposure
liquid photoresists 105–7
screen printing 161
light boxes
repair methods 440
liner plates 194
liquid honing
smear removal 301–2
liquid photoresists 96–110
compared with other processes 114
loading
bonding presses 214–18
longshaft satin
glass cloth 86

machining 191–4, 236–90
magnifiers
quality control 378, 387
manual testing
electrical continuity 402
manufacturing processes
dry film photoresists 123–37
flexi-rigid boards 459–72
general 23–59
glass cloth 76–93
laminates 69–75
causing instability 414–15
Multiwire boards 533–46
prepreg 64–6
marketing
analysis of needs 9–11
Multiwire boards 572–7
masking
of solder 138
mass lamination 46
measling
board defects 203–4
measurement *see* testing
mercury vapour lamps 105
mesh
screen printing 156–9
metal core Multiwire boards 569–70, 571
microetching
defined 358
microscopes
microsectioning 396
repair work 439
microsectioning 394–401
Microwire boards 529, 572
MIL specifications 91, 495
milling
tooling slots 183
milliohmmeters 438
modification

repair methods 431–2
modified epoxy materials 64
moulded boards 22–3
multiple imaging
pressing 234
Multiwire boards 23, 528–77
Multiwire Design Technology 547–54, 577

nail heading
drilling 259–61, 282, 285
negative films 26–7
negative resists
dry film 117
liquid
development 107–8
NEMA 494–5
nickel plating 345–7
flexi-rigid boards 464
non-epoxy materials 63–4
novolac 62

oil cooling
bonding presses 212–13
oil heating
bonding presses 209–10
on-line quality control 376–82
open circuits
repair methods 443–4, 448–9
organic matter
contamination
electroplating 339, 343–4
outer layers
design considerations 520–21
outgassing 55–6, 318–19, 343
oven drying
liquid photoresists 104–5
oxidation
copper adhesion 74–5, 187–9
plating problems 309
quality control 380
oxygen
affecting photopolymerisation 128

packing
pressing 197–8, 224–5
packing density
and costs 4, 6
and electrical parameters 3, 4
Multiwire boards 529, 572–5
thermal problems 7–8
paddle etching 174
pads
desoldering 481
distortion from drilling 275, 277
final inspection 386
flexi-rigid boards 472–7
Multiwire boards 545
PCB design 271, 513–20, 524–5
repair methods 440–41, 444–6
sizes 511–13
Z-plane expansion 59
palladium
electroless plating 311, 315–17
palladium-nickel plating 347
panel plating
buried via-hole boards 185
parallel gap welding 442
pattern plating
and dry film photoresists 111, 112
buried via-hole boards 185–6
peel strength
testing 389
pegboard drilling 289
photo chemical machining 138–9
photographs
microsectioning 396
photometers
exposure control 130
photopolymerisation 115–20, 127, 128–33
photoresists
dry film 95, 111–53
flexi-rigid boards 465
inadequately cleaned 360–61
liquid 96–110
compared with other processes 114
phototape 138
pigtails
copper burrs 278
pillar frames
bonding presses 206–8
pillar plating 15, 41–2
pinholes
filmwork 377
pins
bonding 191–2, 234–5
planes
design considerations 513–20, 525–6
Multiwire boards 534
planning
production processes 33
plasma etching
smear removal 302–3
plate frames
bonding presses 206–8
plated pillars 15, 41–2
plated-through holes 16–17, 38
damage 58–9
design of planes 513–20
desoldering 478–9, 479, 480–81, 484, 486–7
destructive testing 389–90
drilling problems 278–81
electroless plating 306–25
evaluation of microsectioning 398–9, 400
final inspection 386
flexi-rigid boards 453–4, 462–6, 466–70, 474–7
Multiwire boards 544–5, 559–60
repair methods 447
repair of pads 444–6
size 511
solder finishes 368–9

see also drilling
plates
presses 193–4, 217, 221, 225, 232–3, 235
plating 36–7, 38, 326–55
buried via-holes 42, 185–6
quality control 382
solder finishes 368–70
thin laminates 54
troubleshooting
dry film resists 147–8, 148–9
see also electroless plating; pattern plating
plattens
bonding presses 206, 213–14, 232
plying
glass yarn 80–83
point angle
drilling 263
polishing
pretreatment
copper 362
gold plating 349–50
pollution *see* contamination
polyimides 63
and electroless plating 310
flexi-rigid boards 450–51
smear removal 301
polymerisation
dry film photoresists 115–20, 127, 128–33
polystyrene
dry film resist structure 112, 128
polytetrafluoroethylene *see* PTFE
porosity
gold plating 348–9, 354–5
positive films 26–7
positive resists
dry film 117
liquid
development 108
pre-etching
electroless plating 313–15, 324
pre-filling
buried via-holes 44
prepreg 48–9
during bonding 199–200, 218–20
handling problems 197
manufacture 64–9
Multiwire boards 543
preparation 189–90
quality control 379
use of epoxy materials 61–2
press plates *see* plates; presses
presses and pressing 205–35
and laminate stability 414–15, 421
pressure
press cycles 218–22, 227–8
pretreatment
copper 356–66
gold plating 349–50
print and etch process
dry film resists 139–40
printing 34
photomechanical 95–153
quality control 378, 379–80, 382
tooling holes 181–2
see also screen printing
probes
electrical continuity testing 403–6
process control
prepreg manufacture 66
production processes *see* manufacturing processes
profiling
flexi-rigid boards 470–72
projection imaging 133
propagation delay
design considerations 503, 504
proximity imaging 133
PTFE
electroless plating 310
use in flexi-rigid boards, 462, 463, 467, 469, 470
PTHs *see* plated-through holes
punching
filmwork 30–33, 183–4
laminates 181, 183
prepreg 190
purging
of water from press 212
pyrophosphate solutions
copper plating 328–30

quality control 374–82
rework 432
see also inspection
quilling
glass yarn 84–5

radiography
use in inspection 423–9
rake angle
tool geometry 238–9, 245–6, 263
rams
bonding presses 213
reflections
and characteristic impedance 503
refrigeration
prepreg 189–90
release sheets 195, 196
reliability 54–9
Multiwire boards 554–60
see also quality control
relief angle
tool geometry 238–9, 245–6
repair kits
desoldering 479
repair methods 430–49
resin content
and laminate stability 417–19
prepreg testing 68
resins
action in desoldering 483
affecting machining 241–2

encapsulation of microsections 395
laminate stability 410–14
liquid photoresists 98
prepreg action during bonding, 199–200 218–20
see also epoxy materials
resistance
electrical testing 407–8
resists *see* photoresists; solder resist
resolution gauges 376–7
rework
defined 431
quality control 432
rigidised flexibles 450
rinsing
cleaning of copper 356–7
development
dry film resists 134
electroplating 327–8
etchback equipment 297, 298–9
ripout
drilling 259–61, 285–6
'robbing strips'
electroplating 334
roller coating
liquid photoresists 101–3
roller tinning
solder finishes 367–8
rotation
cutting tools 246–7
routing (cutting) 243–57
control tapes 33–4
quality control 379
routing (planning)
Multiwire design 550–53

safety
radiography 428–9
sampling
quality control 375–6
scaled flow testing 67
scanners
visual inspection 388
scratches
substrates
effect on dry film resists 124
screen frames 156, 157
screen printing 154–65
compared with other processes 114
quality control 380
scrubbing
substrate preparation 99–100
scum
dry film resists 134, 135
troubleshooting 144–5, 150–51
semi-additive multilayers 46
semi-additive processes
dry film resists 141–2
sensitisers
liquid photoresists 98–9
sequential buried via-holes 18, 45, 185–6
short circuits 56–8, 345, 516
repair methods 441, 443
shrinkage
laminate manufacture 70
silica
glass composition 79–80
single-sided PCBs
costs 3
inner layer preparation 186–7
six-layer boards 50–52
sizes
of features
design considerations 510–13
of PCBs
Multiwire 572
press capacity 232–3
sizing
glass fibre production 77, 78, 86–7
glass yarn 84
skip plating
troubleshooting 148–9, 151
slashing
glass yarn 84
smears
drilling 261, 275, 276, 307–8
removal 38, 72, 292–305
quality control 381
Multiwire boards 558–9
smoothing
cleaning of copper 362
'snap-off'
screen printing 161–2
soak cleaning
electroplating 327
solder buttons 14–15
solder finishes 367–73
solder float test 390
solder mask 138, 369
solder resist
design considerations 522
final inspection 387
solder rise 338
solderability 478
gold 348
testing 390
soldering
outgassing 55–6, 318–19, 343
plating problems 309
repairs 444–6
see also desoldering
solutions
electroless plating 321–3
electroplating 328–31
gold 349
nickel 345–6, 347
tin-lead 340, 341
solvents
effect on laminates 71–2
dry film photoresists 118, 120–22, 135, 136
troubleshooting 150–51

liquid photoresists 99
specifications *see* standards
spectral sensitivity
dry film resists 132
spraying
coatings
liquid photoresists 101
development
dry film resists 133–4, 135
etching machines 175–7
oxidation 189
squeegees
screen printing 162–3
stability
laminates 69–70, 73, 91, 409–22
design considerations 509–10
flexi-rigids 459–61
stacking and unstacking 195–7, 234–5
staked boards 16
standards 489–98
drill bits 265
see also British Standards
steam heating
bonding presses 209
stencils
screen printing 159–61
step and repeat
filmwork 27–8, 526
step plating 328–30
troubleshooting 148–9
step tablet
exposure time 106–7, 129–30
storage
prepreg 189–90
thin laminates 53
stoving *see* baking
stress
internal layers
electroplating 334–6, 351–2
stripping
photoresists 109–10, 118–19, 137
troubleshooting 149–50
substrates
preparation
dry film resists 123–4
gold plating 348–50
liquid photoresists 99–100
subtractive processes
dry film resists 139–40
sulphuric acid
etchants 170–71
smear removal 295–300
surface dressing
sequential multilayer boards 186
surface polymerisable resists 132
surface tracking
short circuits 57
surge currents
design considerations 503

Talbot, W.H.F. 96
tank etching 174
tapes
control of drills, etc. 33–4
quality control 379
targets
filmwork 28–9
'tear drops' *see* step plating
tearout
drilling problems 286
'telegraphing' 38
bonding defects 201
temperature *see* heat
tension
mesh
screen printing 156–8
tenting 140
test coupons 383–5, 400–401, 402
Multiwire boards 547
test heads
electrical continuity testing 403–6
testing
cleanliness
substrates 100, 125
destructive 389–401
electrical 401–8
laminate stability 415–17
Multiwire boards 546–7, 556, 560–61
porosity
gold plating 354–5
prepreg manufacture 66–9, 190
see also inspection
thermal aspects *see* heat; heating
thermal clearances
design of planes 513–20
Thermal Mechanical Analysis 411–13
thickness
laminates 71
design considerations 509–10
PCBs 49
affected by prepreg 199–200
affecting characteristic impedance 501–2
design considerations 509
variations 202
prepreg 48–9
thiourea compounds
etching additives 169
through-hole plated boards *see* plated-through holes
throwing power
copper plating 331–2
electroless plating 309
nickel plating 346
tin-lead plating 340–41
tin-lead
plating 340–45
solder finishes 367–72
tin-nickel plating 347
tin-palladium process
electroless plating 311, 315–17
tin plating

flexi-rigid boards 464
tinning
solder finishes 367–8
tool design 241
tool geometry 238–9, 248, 265–7
tool life 248–9, 267–8
tool material 239–41, 247, 265
tooling 191–4, 236–90
flexi-rigid boards 459
tooling holes 181–5
tracks
capacitance 504
characteristic impedance 501–2
cutting 434–5, 437
design considerations 510–11, 513
final inspection 386–7
flexi-rigid boards 451
repair methods 440–41, 441–2, 443
troubleshooting
dry film resists 143–4
transfer presses 228–31
translation boards
electrical continuity testing 404
treaters
prepreg manufacture 65–6
troubleshooting
contour routing 254–7
drilling 274–87
dry film photoresists 142–52
tungsten carbide
tool material 239–41, 265
twill weave
glass cloth 86
twisting
board defects 202–3
glass yarn 80–83

ultrasonic drilling 289
ultrasonic wiring
Multiwire boards 539–40
ultra-violet light
curing inks 164
undercutting
etching 169, 172, 173–4
underplating
troubleshooting
dry film resists 147–8
United States
standards 494–6

vacuum bonding 198, 460
vacuum cleaning
chip removal 241
vacuum metallising 310
vapour blasting
smear removal 301–2, 308
vapour phase heating
solder finishes 371
via holes
design considerations 522–4
via holes, buried *see* buried via holes
viewers
visual inspection 387–8
viscosity
dry film resists 126–7
epoxy materials 62, 219–20
inks 164
visual inspection 385–8
voids
bonding 200, 201
electroless plating 312
volatile content
prepreg testing 68
voltage breakdown 501

warping 58
board defects 202–3
glass yarn 83–4
water
jet machining 289
'water break' test 125
water cooling
bonding presses 211
wear
drills 269, 270, 279
gold plating 354
weaving
glass cloth 86
wedge angle
tool geometry 238–9
welded nickel interconnects 20–21
welding
repair methods 442
whirl coating
liquid photoresists 100–101
Wilson, G. C. 44
window dye concept 128–9
wiring
Multiwire boards
insulation 555–7
process 535–43
woven glass fabric *see* glass cloth
woven quartz fibre 421

X-rays
use in inspection 423–9

yarn
glass fibre 80–83
yarn count 86

Z-plane expansion 59, 409, 560

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

Chemistry for MLB Production

De-Smear
Copper Cleaners
Microetches
Black Oxide
Bronze Oxide
Screen Inks
Conformal Coatings
Resist Strippers
De-foamers
Developers
Passivates
P.T.H. Systems
Etchants
Solder Masks & Idents

Equipment for MLB Production

Vertical Spray Modules for develop etc.
Automatic Handling Equipment
Effluent Treatment & Metal Reclamation
In-Line De-smear Plant
Multi-layer Presses
Automatic Solution Dosing Units
Microprocessor Process Controls
P.T.H. Plugs
Diazo Films

CHEMLEC

Chemlec Solutions Limited
Gladstone Rd., Farnworth, Bolton BL4 7EH. Tel: 0204 700472 Telex 63411

THE WAY TO BETTER CIRCUITS

Norplex® laminates by uop

G10 FR
NP424
G30
G10 M/L

COPPER-CLAD LAMINATES

We offer a wide selection of epoxy-fiberglass laminates, the most widely used substrates for printed circuitry.

THIN CONDUCTOR FOILS

Base laminates are now available with copper foil in thicknesses of 5 microns and 9 microns. Thin clad laminates offer many of the advantages of additive systems, while using conventional plating and processing procedures.

HIGH-PERFORMANCE LAMINATES

Printed circuitry for high-temperature applications has been made possible by the development of a number of unique systems. Polyimide resin contributed a major advancement to printed circuits by its temperature resistance, superior processability and improved reliability.

MULTILAYER BASE MATERIALS

Norplex has long recognized the need for custom-designed core laminates and bonding sheets in multilayer applications. We have the manufacturing versatility to build core laminates with those properties most important to the application, such as dimensional stability, dielectric constant, and coefficient of thermal expansion.

For more information contact

Norplex UK Division
uop Limited

Johnson House · Browells Lane · Feltham, Middlesex, TW13 7EQ,
Telephone 01-890 1269 · Telex 935479 (JSELON G)

AT LAST...

FROM THE COMPANY WHERE SERVICE ALWAYS COMES FIRST



A High Speed, BRITISH BUILT, Dedicated Router at a Price You Can Afford . . .

- ★ Fully interpolating microprocessor CNC
- ★ Two high productivity belt-driven spindles
- ★ Closed loop servo controlled X and Y axes
- ★ 24,000/48,000 rpm operator selected spindle speeds
- ★ One full 24" × 24" capability
- ★ Two 18" × 24" capability
- ★ System resolution to 0·0001"
- ★ Will cut FR4, four high under both heads using 0·125" cutters
- ★ Factory training in programming and operation
- ★ **Reliable, quick, guaranteed after-sales service**
- ★ **Full one-year warranty**

Dedicated printed circuit board routing has long been a consideration only for the larger production company.

NOW THAT IS IN THE PAST!

Systems Efficiency Limited have been selling dedicated routers with a great deal of success over the years.

With the experience gained during that time, and with an intimate knowledge of the problems involved, we have produced a fully interpolating CNC router for the small to medium sized user . . . **and at a decidedly small price!**

You can now start high quality, dedicated routing, **WITH BELT–DRIVEN PURPOSE BUILT ROUTING SPINDLES**, for a fraction of the cost previously quoted.

Let the R100 take the strain of producing your High Quality finished panels (four high if you like) at a price and speed which make it the most cost effective answer to your profiling bottleneck.

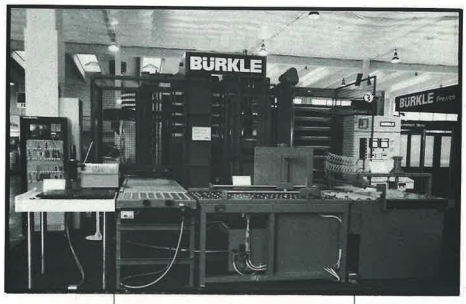
NOW IF YOU WANT TO ROUT PROPERLY . . . YOU CAN AFFORD THE RIGHT EQUIPMENT!



SYSTEMS EFFICIENCY LIMITED
Foundry Close, Foundry Lane
Horsham, West Sussex RH13 5TJ
Tel. (0403) 69149. 62604 Telex 877710



Higher Performance . . .



Automatically Controlled Multilayer Hot-Transfer Press Plant – from the MHK range

The integrated handling system incorporates tool-plate control and multilayer lay-up stations

10 years of experience in the manufacture of multi-layer presses and a close co-operation with material and multilayer manufacturers are a warrant for you that we are the right partners.

Profit from our "know-how". Let us advise on your application.

Contact us direct or our UK agents:
Brian Davis & Associates Ltd.
125 Wells Road
Malvern Wells, Worcs
WR 144PD
Tel.: 06846-65522
Telex: 335284

BÜRKLE
Press & Surface-Treatment Technology

Robert Bürkle GmbH & Co. • Maschinenfabrik • Stuttgarter Straße 123 • D-7290 Freudenstadt • Telefon (07441) 58 1 • Telex 0764227

THREE OF THE BEST FROM I.V.P.

PLASMA

Our existing PC 1100 established for desmearing of MLBs offers a high capacity chamber up to 75 ft² (7 m²) of PCB area. Our smaller chamber machine, the PC 760, is now introduced essentially for the specialised manufacture of MLBs, particularly for polyimide rigid and flex-rigid circuits and many types of flex-circuits.
In addition plasma desmearing of PTFE dielectric is now a possibility with our PC 1100 and PC 760 systems.
Our PC 760 offers a more versatile system for other processes including dry etching or cleaning, surface activation of plastics and solder flux residue removal.
The PC 240 offers the ideal solution for specialised R & D facilities and hybrid microcircuit manufacturers for a range of dry chemical processes described above, and for surface conditioning of conductor metallisations.



• PC 760

• PC 240

PLASMA PROCESSING SERVICES

Extending our already established plasma desmearing services, both PC 1100 and PC 760 systems are now available for a comprehensive range of processes including solder flux residue removal, surface etching, surface activation of plastics and general conditioning of inorganic and organic materials, backed by comprehensive analytical/testing facilities including metallurgical microsectioning with plating capability, optical and SEM equipment.

VAPOUR PHASE SYSTEMS

NEW

The VPR 4000 series systems, proven and tested in both prototype and production environments in the PCB manufacturing and hybrid microcircuit assembly industries, offers its proven features to a new range.
Introducing the VPR 5000 with full microprocessor electronics control. Our VPR 5000 systems offer a user an even more flexible system with different levels of software packages.
Programme facilities for all essential processing parameters are available. In addition the VPR 5000 series is offered with full safety interlock on heater power control.
I.V.P. offer the best all-round package for a vapour phase heating system backed by the expertise established in semi-conductor processing industries with field support unequalled in Europe.



I.V. Products Limited,
Prince of Wales Industrial Estate,
Abercarn, Newport, Gwent, NP1 5AR.
Tel: Newbridge (0495) 246454 – 247070
Telex: 497758

I.V. Products Limited

LeaRonal, first name in electroplating, introduces the last word in screen ink technology.



LeaRonal inks give you ultra-sharp resolution, dense images, swift low-energy curing, prolonged shelf life. And they're formulated to speed production, give you consistent screening board after board, for manual and semi-automatic operations.

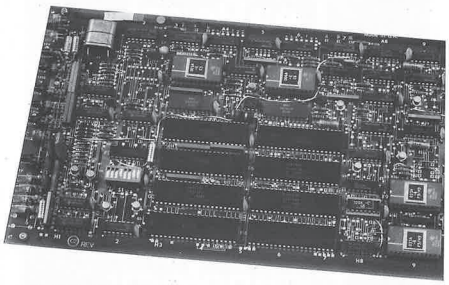
Ask your LeaRonal Service Engineer for technical details and free test samples. Our new inks have passed our screen test—they'll pass yours, too.

For years we've wanted to offer a line of screen printing inks for the printed circuit industry that's as good as our line of LeaRonal plating processes. After extensive testing in production, we've finally developed products that are good enough for us—and for you.

We now offer a full range of plating and etch resists, solder masks, lettering inks, all in your choice of thermal or ultraviolet curables.

LeaRonal (UK) plc

High Peak Laboratories, Ashbourne Road, Buxton, Derbyshire SK17 9SS, England. Tel. Buxton 5391 (4 Lines) Telex: 669768 RONAL BUXTON



CIBA-GEIGY has the material to satisfy the demands of modern technology and reliability.

Araldite Laminating Resins
A range of standard and upgraded electronics laminating resins to suit your specific requirement either for Multilayer or rigid board applications. Materials satisfy required international standards, eg NEMA, UL, MIL, BS etc.

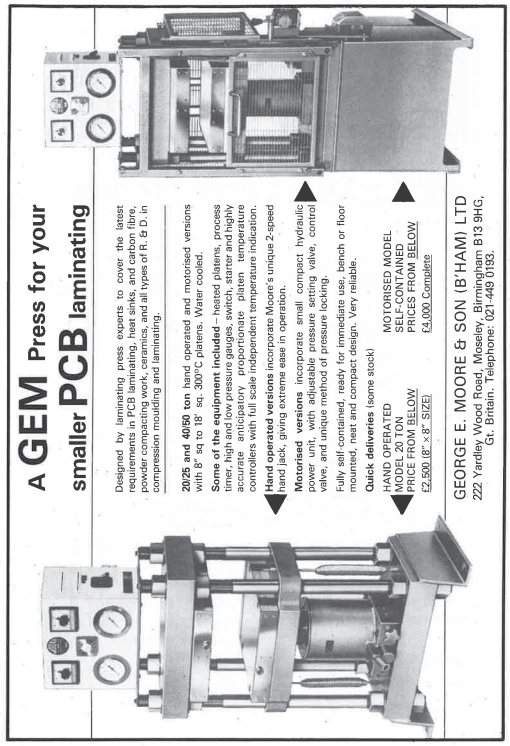
Probimer Photopolymers
PROBIMER 48 — for superior fine line inner layers with enhanced performance.
PROBIMER 52 — epoxy performance, photographic accuracy, highly automated production. Elegant solder masking now and for the future.

Electronics and Microelectronics Products
A comprehensive international range of epoxy resin based products is available for the protection of electronics and microelectronics devices. For further information on the above materials please contact our Technical Sales Office on Cambridge 832121

CIBA–GEIGY
Araldite Electronics Systems
for positive protection

Ciba-Geigy pioneers 50 YEARS

—CIBA-GEIGY PLASTICS, DUXFORD, CAMBRIDGE CB2 4QA. TEL: CAMBRIDGE (0223) 832121—



A GEM Press for your smaller PCB laminating

Designed by laminating press experts to cover the latest requirements in PCB laminating, heat sinks, and carbon fibre, powder compacting work, ceramics, and all types of R. & D. in compression moulding and laminating.

20/25 and 40/50 ton hand operated and motorised versions with 8" sq to 18" sq. 300°C platens. Water cooled.

Some of the equipment included – heated platens, process timer, high and low pressure gauges, switch, starter and highly accurate anticipatory proportionate platen temperature controllers with full scale independent temperature indication.

Hand operated versions incorporate Moore's unique 2-speed hand jack, giving extreme ease in operation.

Motorised versions incorporate small compact hydraulic power unit, with adjustable pressure setting valve, control valve, and unique method of pressure locking.

Fully self-contained, ready for immediate use, bench or floor mounted, neat and compact design. Very reliable.

Quick deliveries (some stock)

HAND OPERATED	MOTORISED MODEL
MODEL 20 TON	SELF-CONTAINED
PRICE FROM BELOW	PRICES FROM BELOW
£2,500 (8" × 8" SIZE)	£4,000 Complete

GEORGE E. MOORE & SON (B'HAM) LTD
222 Yardley Wood Road, Moseley, Birmingham B13 9HG,
Gt. Britain. Telephone: 021-449 0193.

RESCO

Specialists in fine, dependable equipment for Multilayers

- Etch Back
- Blackening
- Cleaning of flexible inner layers
- Fine Line etching (Bi-Etching System)
- Fine Line Developing (Omnidirectional System)

Any other production machine from the cutting (Resco Roller Cutter) to the final cleaning (Resco Flux Remover)

RESCO, Italy,
Via Massena 2/A, Milano

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

TOMORROW'S PCBS REQUIRE RELIABLE BASE MATERIALS

PERSTORP
— Your Technology Partner —
Offers:
GI 250 Polyimide Laminates
GE 313 Epoxy Laminates
Perform Multilayer System
UTC Thin Foil

CALL OR WRITE TO LEARN MORE ABOUT OUR WIDE PRODUCT RANGE.

Perstorp

PERSTORP FERGUSON LTD.
Aycliffe Ind. Estate
Newton Aycliffe
Co. Durham DL5 6EF
Phone (0325) 315141
Telex 587214

X-Y TABLES

FOR MULTI-LAYER PRODUCTION

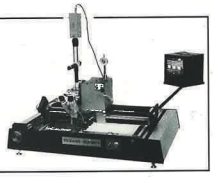


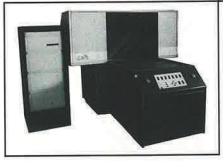
INSPECTION AND MEASURING TABLE

For filmwork control. Also suitable for printed circuit boards and various 2 axis measuring applications.

MODIFICATION AND REWORK TABLE

For multi-layer and high density boards. Precise optics and controls for track cutting etc on inner layers.



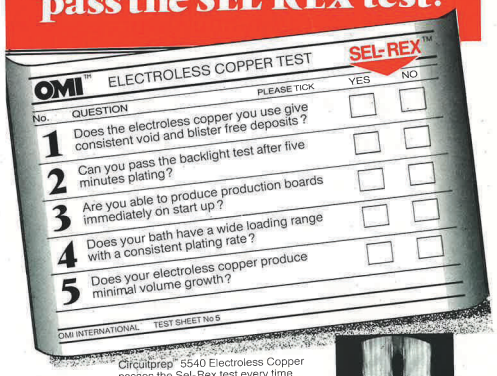


AUTOMATIC OPTICAL INSPECTION

X-Y table designed for CRS system of inspecting layers before lamination and other auto-inspection work.

PACKMAN RESEARCH LIMITED
Willow Drive, Twyford, Berkshire RG10 9BB, England
Telephone: (0734) 341167, Telex: 847839 Pacres

Does your Electroless Copper pass the SEL-REX test?



Circuitprep® 5540 Electroless Copper passes the Sel-Rex test every time and has many additional benefits for you the user.



OMI
OMI INTERNATIONAL (G.B.) LIMITED
FORSYTH ROAD, SHEERWATER, WOKING, SURREY GU21 5RS

Apply for the test today

I WANT TO SEE CIRCUITPREP 5540 RESULTS FOR MYSELF

NAME _____

COMPANY _____

ADDRESS _____ TEL _____

OMI OMI INTERNATIONAL (GB) LTD. FORSYTH ROAD, SHEERWATER, WOKING, SURREY GU21 5RZ TEL: WOKING (04863) 5971 TELEX: 859121

FRY
A Cookson Group Company

The World's leader in Research, manufacture and supply of

SOLDERS, FLUXES, SOLDER CREAMS
for the Electronics Industry

TECHNICAL SERVICE

A complete technical advice service is available to users of soft solders. With their background of research and development in the Industry, Fry's Metals continue to be recognized as the authority for all matters relating to the soldering processes for Electronics and Micro Electronics production.

FRY'S METALS LTD
Tandem Works, Merton Abbey, London SW19 2PD.
Telephone 01-648 7020. Telex 265732.

Branches at GLASGOW, KIDDERMINSTER, LEEDS, ROCHDALE, DUBLIN, ALTOONA (U.S.A.)

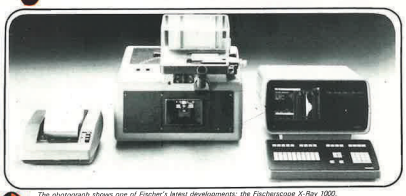
S 15

COATING THICKNESS ON ANY SHAPE

Over two decades of continuous development and production for worldwide markets back the FISCHER range of thickness, porosity, material testing and process control equipment.

FISCHER'S instrumentation utilises the latest technical innovations plus full measurement data processing to meet modern industrial requirements. From the simplest of gauges to the most sophisticated control instrumentation FISCHER uses the latest internationally specified methods of:

X-RAY FLUORESCENCE	ISO 3497
BETA BACKSCATTER	ISO 3543
EDDY CURRENT	ISO 2360
ELECTROMAGNETIC INDUCTION	ISO 2178
COULOMETRIC	ISO 2177



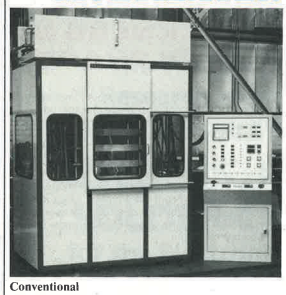
The photograph shows one of Fischer's latest developments: the Fischerscope X-Ray 1000.

Write, telephone or telex for details of the FISCHERSCOPE range of instruments.

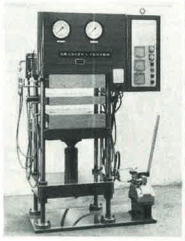
FISCHER INSTRUMENTATION (GB) LTD
Arnhem Road, Bone Lane Industrial Estate Newbury/Berks, England.
Telephone Newbury (0635) 45191 Telex: 849111 FIBRIT G

Fischer®

MULTILAYER PRESSES



Conventional



Laboratory



Hot/Cold Transfer Press

Presses to suit your needs

Hot/Cold Transfer
'Conventional' Multi-Layer PCB Laminating
Heat Sinks
Flexible Circuits
Flush Bonding
Laboratory

YOU deal DIRECT with US. We deal DIRECT with YOU.

We also manufacture MICROPROCESSOR controlled machines and complete EXTRUSION lines for all types of WIRE AND CABLE including FIBRE OPTICS.

FRANCIS SHAW

Bradley & Turton Division,
Francis Shaw & Co. (Manchester) Ltd.,

P.O. Box 12, Corbett Street,
Manchester M11 4BB, England.
Tel: 061-223 1313. Telex: 667057.

A Member of the EIS Group

Can you name a commercially available Printed Circuit Board Innerlayer Imaging System that is;

- Fully automated, (75 panels per man hour)
- Produces fine line circuits (100 microns) with high yields
- Uses a resist that is less than half the cost of dry film?

We can . . . it's called Accutrace and complete technical data is available from

GRACE

Electronic Chemicals

W.R. Grace Ltd., Northdale House, North Circular Road, London NW10 7UH. Telephone: 01-965 0611. Telex: 25139.

QUALITY MACHINES MATERIALS AND TOOLS FOR MULTILAYER PRODUCTION

*Howe Industries Inc. - High temperature laminates.
Polyimide glass laminate and prepreg
Polyimide Kevlar* laminate and prepreg
Polyimide Quartz laminate
Polyimide Invar** laminate
Modified Epoxy laminate

*Advanced Controls - Drilling Machines, designed for and with special software for multilayer drilling. Automatic measuring machines and programming units.

*L.C.O.A. - 3003 Foilboard. Drill entry and backup systems, a must for multilayer drilling.

*Kemmer - Solid Tungsten Carbide Drills & Routers. Special multilayer drills.

*Keene, Chase Foster Div. - Press pad silicone glass materials.

*Services - Subcontract programming and drilling, panel cutting and drill regrinding.

* TM Dupont Co.
**TM Texas Instruments Co.

Holders Technology
A HOLDERS GROUP COMPANY
HOLDERS TECHNOLOGY LIMITED
108 RIPON WAY, BOREHAMWOOD,
HERTFORDSHIRE WD6 2HZ.
TELEPHONE: 01-953-4313 TELEX 892996 HOLMAR