This is a preview of "IPC 2221B-2012". Click here to purchase the full version from the ANSI store.

IPC-2221B

Generic Standard on Printed Board Design

Developed by the IPC-2221 Task Group (D-31b) of the Rigid Printed Board Committee (D-30) of IPC

Supersedes:
IPC-2221A - May 2003
IPC-2221 - February 1998

Users of this publication are encouraged to participate in the development of future revisions.

Contact:

IPC
3000 Lakeside Drive, Suite 309S
Bannockburn, Illinois
60015-1249
Tel 847 615.7100
Fax 847 615.7105

This is a preview of "IPC 2221B-2012". Click here to purchase the full version from the ANSI store.

Table of Contents

1 SCOPE 1
1.1 Purpose 1
1.2 Documentation Hierarchy 1
1.3 Presentation 1
1.3.1 Dimensional Units 1
1.4 Interpretation 1
1.5 Definition of Terms 2
1.5.1 Microvia 2
1.6 Classification of Products 2
1.6.1 Printed Board Type 2
1.6.2 Performance Classification 2
1.6.3 Producibility Level 2
1.7 Revision Level Changes 3

2 APPLICABLE DOCUMENTS 3
2.1 IPC 3
2.2 Joint Industry Standards 4
2.3 Society of Automotive Engineers 5
2.4 American Society for Testing and Materials 5
2.5 Underwriters Labs 5
2.6 IEEE 5
2.7 ANSI 5
2.8 ANSI/ESD 5
2.9 PCMCIA 5

3 GENERAL REQUIREMENTS 6
3.1 Information Hierarchy 8
3.1.1 Order of Precedence 8
3.1.2 End-Product Performance Requirements 8
3.2 Design Considerations 8
3.3 Schematic/Logic Diagram 9
3.4 Density Evaluation 9
3.5 Parts List 10
3.6 Test Requirement Considerations 10
3.6.1 Electrical 10
3.6.2 Printed Board Assembly Testability 12
3.6.3 Boundary Scan Testing 13
3.6.4 Functional Test Concern for Printed Board Assemblies 14
3.6.5 In-Circuit Test Concerns for Printed Board Assemblies 15
3.6.6 Mechanical 17
3.7 Layout Evaluation 17
3.7.1 Printed Board Layout Design 17
3.7.2 Feasibility Density Evaluation 18

4 MATERIALS 21
4.1 Material Selection 21
4.1.1 Material Selection for Structural Strength 21
4.1.2 Material Selection for Electrical Properties 21
4.1.3 Material Selection for Environmental Properties 21
4.2 Dielectric Base Materials (Including Prepregs and Adhesives) 21
4.2.1 Preimpregnated Bonding Layer (Prepreg) 22
4.2.2 Adhesives 22
4.2.3 Adhesive Films or Sheets 24
4.2.4 Electrically Conductive Adhesives 24
4.2.5 Thermally Conductive/Electrically Insulating Adhesives 24
4.3 Laminate Materials 25
4.3.1 High T_g Laminates 25
4.3.2 Color Pigmentation 25
4.3.3 Dielectric Thickness/Spacing 25
4.3.4 Thermally Conductive Laminates 25
4.3.5 Minimum Base Material Thickness for PC Card Form Factors 26
4.4 Conductive Materials 26
4.4.1 Electroless Copper Plating 29
4.4.2 Semiconductive Coatings 29
4.4.3 Electrolytic Copper Plating 29
4.4.4 Gold Plating 29
4.4.5 Immersion Silver 31
4.4.6 Immersion Tin 31
4.4.7 Organic Solderability Preservative (OSP) 32
4.4.8 Nickel Plating 32
4.4.9 Tin/Lead Plating 33
4.4.10 Solder Coating 33
4.4.11 Other Metallic Coatings for Edge Printed Board Contacts 34
4.4.12 Metallic Foil/Film 34
4.5 Electronic Component Materials 36
4.5.1 Embedded (Buried) Resistors 36
4.5.2 Embedded (Buried) Capacitors 36
4.5.3 Embedded (Buried Inductors) 36
4.6 Organic Protective Coatings 36
4.6.1 Solder Mask Coatings 36
4.6.2 Conformal Coatings 37
4.6.3 Tarnish Protective Coatings 38
4.7 Marking and Legends 38

This is a preview of "IPC 2221B-2012". Click here to purchase the full version from the ANSI store.

4.7.1 ESD Considerations 39

5 MECHANICAL/PHYSICAL PROPERTIES 39

5.1 Fabrication Considerations 39

5.1.1 Bare Printed Board Fabrication 39

5.2 Product/Printed Board Configuration 39

5.2.1 Printed Board Type 40

5.2.2 Printed Board Size 40

5.2.3 Printed Board Geometries (Size and Shape) ... 42

5.2.4 Bow and Twist 42

5.2.5 Structural Strength 42

5.2.6 Composite (Constraining-Core) Printed Boards 42

5.2.7 Vibration Design 43

5.3 Assembly Requirements 44

5.3.1 Mechanical Hardware Attachment 44

5.3.2 Part Support 44

5.3.3 Assembly and Test 45

5.3.4 Tooling Rails for PC Card Form Factor Printed Boards 45

5.4 Dimensioning Systems 45

5.4.1 Dimensions and Tolerances 45

5.4.2 Component and Feature Location 45

5.4.3 Datum Features 46

5.5 Printed Board Thickness Tolerance 49

5.6 Panelization 49

5.7 Palletization 49

6 ELECTRICAL PROPERTIES 53

6.1 Electrical Considerations 53

6.1.1 Electrical Performance 53

6.1.2 Power Distribution Considerations 53

6.1.3 Circuit Type Considerations 53

6.2 Conductive Material Requirements 56

6.3 Electrical Clearance 56

6.3.1 B1-Internal Conductors 57

6.3.2 B2-External Conductors, Uncoated, Sea Level to 3050 m [10,007 feet] 57

6.3.3 B3-External Conductors, Uncoated, Over 3050 m [10,007 feet] 57

6.3.4 B4-External Conductors, with Permanent Polymer Coating (Any Elevation) 58

6.3.5 A5-External Conductors, with Conformal Coating over Assembly (Any Elevation) 58

6.3.6 A6-External Component Lead/Termination, Uncoated, Sea Level to 3050 m [10,007 feet] 58

6.3.7 A7-External Component Lead/Termination, with Conformal Coating (Any Elevation) 58

6.4 Impedance Controls 58

6.4.1 Microstrip 59

6.4.2 Embedded Microstrip 60

6.4.3 Stripline Properties 61

6.4.4 Asymmetric Stripline Properties 61

6.4.5 Capacitance Considerations 62

6.4.6 Inductance Considerations 63

7 THERMAL MANAGEMENT 65

7.1 Cooling Mechanisms 65

7.1.1 Conduction 65

7.1.2 Radiation 65

7.1.3 Convection 66

7.1.4 Altitude Effects 66

7.2 Heat Dissipation Considerations 66

7.2.1 Printed Board Housings 66

7.2.2 Individual Component Heat Dissipation 67

7.2.3 Thermal Management Considerations for Printed Board Heatsinks 67

7.2.4 Assembly of Heatsinks to Printed Boards 68

7.2.5 Special Design Considerations for SMT Printed Board Heatsinks 69

7.3 Heat Transfer Techniques 70

7.3.1 Coefficient of Thermal Expansion (CTE) Characteristics 70

7.3.2 Thermal Transfer 70

7.3.3 Thermal Matching 70

7.4 Thermal Design Reliability 70

8 COMPONENT AND ASSEMBLY ISSUES 72

8.1 General Placement Requirements 73

8.1.1 Automatic Assembly 73

8.1.2 Component Placement 73

8.1.3 Orientation 74

8.1.4 Accessibility 75

8.1.5 Design Envelope 75

8.1.6 Component Body Centering 75

8.1.7 Flush Mounting Over Conductive Areas 75

8.1.8 Clearances 76

8.1.9 Physical Support 76

8.1.10 Heat Dissipation 78

8.1.11 Stress Relief 78

8.2 General Attachment Requirements 79

8.2.1 Through-Hole 79

8.2.2 Surface Mounting 80

8.2.3 Mixed Assemblies 80

8.2.4 Soldering Considerations 80

8.2.5 Connectors and Interconnects 81

This is a preview of "IPC 2221B-2012". Click here to purchase the full version from the ANSI store.

8.2.6 Fastening Hardware 83
8.2.7 Stiffeners 83
8.2.8 Lands for Flattened Round Leads 84
8.2.9 Solder Terminals 84
8.2.10 Eyelets 86
8.2.11 Special Wiring 86
8.2.12 Heat Shrinkable Devices 87
8.2.13 Bus Bar 87
8.2.14 Flexible Cable 87
8.3 Through-Hole Requirements 87
8.3.1 Leads Mounted in Through-Holes 87
8.4 Standard Surface Mount Requirements 91
8.4.1 Surface-Mounted Leaded Components 91
8.4.2 Flat-Pack Components 92
8.4.3 Ribbon Lead Termination 92
8.4.4 Round Lead Termination 92
8.4.5 Component Lead Sockets 92
8.5 Fine Pitch SMT (Peripherals) 93
8.6 Bare Die 93
8.6.1 Wire Bond 93
8.6.2 Flip Chip 93
8.6.3 Chip Scale 93
8.7 Tape Automated Bonding 93
8.8 Grid Array SMT 93
8.9 No-Lead Devices 94
8.9.1 Small Outline and Quad Flat No Lead with Pullback Leads (PQFN, PSON) 94
8.10 Compliant Pin Design Guidelines 95

9 HOLES/INTERCONNECTIONS 95
9.1 General Requirements for Lands with Holes 95
9.1.1 Land Requirements 95
9.1.2 Annular Ring Requirements 96
9.1.3 Thermal Relief in Conductor Planes 97
9.1.4 Lands for Flattened Round Leads 97
9.2 Holes 98
9.2.1 Unsupported Holes 98
9.2.2 Plated Holes 98
9.2.3 Location 100
9.2.4 Hole Pattern Variation 100
9.2.5 Location Tolerances 100
9.2.6 Quantity 101
9.2.7 Spacing of Adjacent Holes 101
9.2.8 Aspect Ratio 101
9.3 Via Protection 101
9.3.1 Via Protection Requirements 101
9.3.2 Via Fill 101

10 GENERAL CIRCUIT FEATURE REQUIREMENTS 102
10.1 Conductor Characteristics 102
10.1.1 Conductor Width and Thickness 102
10.1.2 Electrical Clearance 105
10.1.3 Conductor Routing 105
10.1.4 Conductor Spacing 105
10.1.5 Plating Thieves 106
10.2 Land Characteristics 106
10.2.1 Manufacturing Allowances 106
10.2.2 Lands for Surface Mounting 106
10.2.3 Test Points 106
10.2.4 Orientation Symbols 106
10.3 Large Conductive Areas 106

11 DOCUMENTATION 106
11.1 Special Tooling 108
11.2 Layout 108
11.2.1 Viewing 108
11.2.2 Accuracy and Scale 108
11.2.3 Layout Notes 108
11.2.4 Automated-Layout Techniques 108
11.3 Deviation Requirements 108
11.4 Phototool Considerations 109
11.4.1 Artwork Master Files 109
11.4.2 Film Base Material 109
11.4.3 Solder Mask Coating Phototools 109

12 QUALITY ASSURANCE 109
12.1 Conformance Test Coupons 109
12.2 Material Quality Assurance 110
12.2.1 Laminates 110
12.2.2 Compliant Pin 110
12.3 Conformance Evaluations 110
12.3.1 Coupon Quantity and Location 110
12.3.2 Coupon Identification 114
12.3.3 General Coupon Requirements 114
12.4 Individual Coupon Design 114
12.4.1 Plated Hole Evaluation (Thermal Stress, Rework Simulation, Registration) Coupons 114
12.4.2 Moisture and Insulation Resistance Coupons 115
12.4.3 Hole Solderability Coupons 115
12.4.4 Surface Mount Solderability Coupons 115
12.4.5 Interconnect Resistance and Continuity Coupons 115
12.4.6 Solder Mask Adhesion Coupons 115
12.4.7 Surface Insulation Resistance Coupons 115

This is a preview of "IPC 2221B-2012". Click here to purchase the full version from the ANSI store.

12.4.8 Peel Strength and Plating Adhesion Coupons 116
12.4.9 Controlled Impedance Coupons 116
12.4.10 Optional Legacy Registration Coupons 116
12.4.11 Legacy N Coupon (Peel Strength, Surface Mount Bond Strength - Optional for SMT) .. 116
12.4.12 Coupon X (Bending Flexibility and Endurance, Flexible Printed Board) 116
12.4.13 Process Control Test Coupon 116

APPENDIX A 117

APPENDIX B 142

APPENDIX C 162

Figures

Figure 1-1 Microvia Definition 2
Figure 3-1 Package Size and I/O Count 9
Figure 3-2 Test Land Free Area for Parts and Other Intrusions 16
Figure 3-3 Test Land Free Area for Tall Parts 16
Figure 3-4 Probing Test Lands 16
Figure 3-5 Example of Usable Area Calculation, mm [in] (Usable area determination includes clearance allowance for edge printed board connector area, printed board guides, and printed board extractor.) 18
Figure 3-6 Printed Board Density Evaluation 20
Figure 4-1 HASL Surface Topology Comparison 34
Figure 5-1 Example of Printed Board Size Standardization, mm [in] 41
Figure 5-2 Typical Asymmetrical Constraining-Core Configuration 43
Figure 5-3A Multilayer Metal Core Printed Board with Two Symmetrical Copper-Invar-Copper Constraining Cores (when the Copper-Invar-Copper planes are connected to the plated-through hole, use thermal relief per Figure 9-4) 43
Figure 5-3B Symmetrical Constraining Core Printed Board with a Copper-Invar-Copper Center Core 43
Figure 5-4 Advantages of Positional Tolerance Over Bilateral Tolerance, mm [in] 46
Figure 5-5 Datum Reference Frame 47
Figure 5-6 Example of Location of a Pattern of PTHs, mm [in] 48
Figure 5-7 Example of a Pattern of Tooling/Mounting Holes, mm [in] 48
Figure 5-8 Example of Location of a Conductor Pattern Using Fiducials, mm [in] 49
Figure 5-9 Example of Printed Board Profile Location and Tolerance, mm [in] 50
Figure 5-10 Example of a Printed Board Drawing Utilizing Geometric Dimensioning and Tolerancing, mm [in] 50
Figure 5-11 Fiducial Clearance Requirements 51
Figure 5-12 Printed Board Panelization/ Palletization, mm 51
Figure 5-13 Example of Connector Key Slot Location and Tolerance, mm [in] 52
Figure 6-1 Voltage/Ground Distribution Concepts 54
Figure 6-2 Single Reference Edge Routing 55
Figure 6-3 Circuit Distribution 55
Figure 6-4 Transmission Line Printed Board Construction 59
Figure 6-5 Capacitance vs. Conductor Width and Dielectric Thickness for Microstrip Lines, mm [in] 63
Figure 6-6 Capacitance vs. Conductor Width and Spacing for Striplines, mm [in] 64
Figure 6-7 Single Conductor Crossover 64
Figure 7-1 Component Clearance Requirements for Automatic Component Insertion 68
Figure 7-2 Relative Coefficient of Thermal Expansion (CTE) Comparison 71
Figure 8-1 Component Orientation for Boundaries and/or Wave Solder Applications 75
Figure 8-2 Component Body Centering 75
Figure 8-3 Axial-Leaded Component Mounted Over Conductors 76
Figure 8-4 Uncoated Board Clearance 76
Figure 8-5 Clamp-Mounted Axial-Leaded Component ... 76
Figure 8-6 Adhesive-Bonded Axial-Leaded Component 76
Figure 8-7 Example of Filleting Compared to Bonding .. 77
Figure 8-8 Mounting with Feet or Standoffs 77
Figure 8-9 Heat Dissipation Examples 78
Figure 8-10 Lead Bends 79
Figure 8-11 Typical Lead Configurations 79
Figure 8-12 Typical Keying Arrangement 82
Figure 8-13 Printed Board Edge Tolerancing 82
Figure 8-14 Lead-In Chamfer Configuration 83
Figure 8-15 Two-Part Connector 83
Figure 8-16 Edge-Board Adapter Connector 83
Figure 8-17 Round or Flattened (Coined) Lead Joint Description 85
Figure 8-18 Standoff Terminal Mounting, mm [in] 85
Figure 8-19 Dual Hole Configuration for Interfacial and Interlayer Terminal Mountings 86
Figure 8-20 Partially Clinched Through-Hole Leads 88
Figure 8-21 Dual In-Line Package (DIP) Lead Bends 88
Figure 8-22 Solder in the Lead Bend Radius 88
Figure 8-23 Two-Lead Radial-Leaded Components 89
Figure 8-24 Radial Two-Lead Component Mounting, mm [in] 89
Figure 8-25 Meniscus Clearance, mm [in] 89
Figure 8-26 “TO” Can Radial-Leaded Component, mm [in] 89

This is a preview of "IPC 2221B-2012". Click here to purchase the full version from the ANSI store.

Figure 8-27 Perpendicular Part Mounting, mm [in] 90

Figure 8-28 Flat-Packs and Quad Flat-Packs 90

Figure 8-29 Examples of Configuration of Ribbon Leads for Through-Hole Mounted Flat-Packs 90

Figure 8-30 Metal Power Packages with Compliant Leads 90

Figure 8-31 Metal Power Package with Resilient Spacers 90

Figure 8-32 Metal Power Package with Noncompliant Leads 90

Figure 8-33 Examples of Flat-Pack Surface Mounting 91

Figure 8-34 Round or Coined Lead 92

Figure 8-35 Configuration of Ribbon Leads for Planar Mounted Flat-Packs 92

Figure 8-36 Heel Mounting Requirements 92

Figure 8-37 TSSOP Package Construction 93

Figure 8-38 SQFP Package Construction 93

Figure 8-39 Examples of Ball Grid Array (BGA) Package Construction 94

Figure 8-40 Ceramic Column Grid Array (CGA) Package Construction 94

Figure 8-41 Land Grid Array (LGA) Package Construction 94

Figure 8-42 Quad Flat No-Lead (QFN) Construction 95

Figure 8-43 Small Outline No-lead (SON) Construction .. 95

Figure 8-44 Pullback Quad Flat No Lead (PQFN) Construction 95

Figure 9-1 Examples of Modified Land Shapes 96

Figure 9-2 External Annular Ring 97

Figure 9-3 Internal Annular Ring 97

Figure 9-4 Typical Thermal Relief in Planes 97

Figure 10-1 Etched Conductor Characteristics 104

Figure 10-2 Example of Conductor Beef-Up or Neck-Down 105

Figure 10-3 Conductor Optimization Between Lands 105

Figure 11-1 Flow Chart of Printed Board Design/ Fabrication Sequence 107

Figure 11-2 Multilayer Printed Board Viewing 108

Figure 11-3 Gang Solder Mask Window 109

Figure 11-4 Pocket Solder Mask Window 109

Figure 12-1 Panel Utilization among IPC-2221B Conformance Coupon Designs 112

Figure 12-2 Panel Utilization among Legacy Conformance Coupon Designs 113

Figure 12-3 Example Stack-up for a Ten Layer Printed Board 113

Figure 12-4 Systematic Path for Implementation of Statistical Process Control (SPC) 116

Figure A.2-1 AB/R Coupon Layout, mm [in] 119

Figure A.2-2 AB/R Coupon Example Layers 120

Figure A.3-1 A/R Coupon Layout, mm [in] 122

Figure A.3-2 A/R Coupon Example Layers 123

Figure A.4-1 B/R Coupon Layout, mm [in] 125

Figure A.5-1 E Coupon Layout, mm [in] 126

Figure A.5-2 E Coupon 127

Figure A.6-1 S Coupon Layout, mm [in] 128

Figure A.6-2 S Coupon Example Layers 129

Figure A.7-1 W Coupon Layout, mm [in] 130

Figure A.7-2 W Coupon Layout 131

Figure A.8-1 D Coupon Layout with A and B Features, mm [in] 132

Figure A.8-2 D Coupon Example Layers with A and B Features 133

Figure A.8-3 D Coupon Layout with Non-through Via B Features, mm [in] 133

Figure A.9-1 G Coupon Layout, mm [in] 135

Figure A.9-2 G Coupon Example Layers 136

Figure A.10-1 H Coupon Layout, mm [in] 137

Figure A.10-2 H Coupon Example Layers 138

Figure A.11-1 P Coupon Layout, mm [in] 139

Figure A.11-2 P Coupon Example Layers 139

Figure A.12-1 Z Coupon Layout (Microstrip and edge-coupled microstrip), mm [in] 140

Figure A.12-2 Z Coupon Example Layers 141

Figure A.12-3 Z Coupon Layout (Microstrip and edge-coupled microstrip using alternative test points), mm [in] 141

Figure B.2-1 Test Coupons A and B, mm [in] 143

Figure B.2-2 Test Coupons A and B (Conductor Detail), mm [in] 144

Figure B.2-3 Test Coupon A/B, mm [in] 145

Figure B.2-4 Test Coupon A/B (Conductor Detail), mm [in] 146

Figure B.3-1 Coupon E, mm 147

Figure B.3-2 "Y" Pattern for Chip Component Cleanliness Test Pattern 147

Figure B.4-1 Test Coupon S, mm [in] 148

Figure B.5-1 Test Coupon M, Surface Mounting Solderability Testing, mm [in] 149

Figure B.6-1 Test Coupon D, mm [in] 150

Figure B.6-2 10 Layer Example 151

Figure B.6-3 Example of a 10 Layer Coupon D, Modified to Include Blind and Buried Vias 152

Figure B.6-4 Test Coupon D for Process Control of 4 Layer Printed Boards 153

Figure B.7-1 Test Coupon G, Solder Resist Adhesive, mm [in] 153

Figure B.8-1 Optional Coupon H, mm [in] 154

Figure B.8-2 Comb Pattern Examples 155

Figure B.9-1 Coupon C, External Layers Only, mm [in] .. 155

Figure B.10-1 Test Coupon F, mm [in] 157

Figure B.10-2 Test Coupon R, mm [in] 158

Figure B.10-3 Worst-Case Hole/Land Relationship 158

Figure B.11-1 Test Coupon N, Surface Mounting Bond Strength and Peel Strength, mm [in] 159

This is a preview of "IPC 2221B-2012". Click here to purchase the full version from the ANSI store.

Figure B.12-1 Test Coupon X, mm [in] 161
Figure B.12-2 Bending Test 161

Tables

Table 3-1 PCB Design/Performance Tradeoff Checklist 6
Table 3-2 Component Grid Areas 19
Table 4-1 Typical Properties of Common Dielectric Materials 23
Table 4-2 Final Finish and Coating Requirements 26
Table 4-3 Surface and Hole Copper Plating Minimum Requirements for Buried Vias >2 Layers, Through-Holes, and Blind Vias 27
Table 4-4 Surface and Hole Copper Plating Minimum Requirements for Microvias (Blind and Buried) 27
Table 4-5 Surface and Hole Copper Plating Minimum Requirements for Buried Via Cores (2 Layers) 27
Table 4-6 Surface Finishes 28
Table 4-7 Gold Plating Uses 29
Table 4-8 ENIG Surface Finish Advantages and Disadvantages 30
Table 4-9 ENIG/EG Surface Finish Advantages and Limitations 30
Table 4-10 ENEPIG Surface Finish Advantages and Disadvantages 31
Table 4-11 Immersion Silver Surface Finish Advantages and Disadvantages 31
Table 4-12 Immersion Tin Surface Finish Advantages and Disadvantages 32
Table 4-13 OSP Surface Finish Advantages and Limitations 33
Table 4-14 Copper Foil/Film Requirements 35
Table 4-15 Metal Core Substrates 36
Table 4-16 Typical Minimum Solder Mask Clearances and Dams 37
Table 4-17 Conformal Coating Types and Thickness Range 38
Table 4-18 Conformal Coating Functionality 38
Table 5-1 Fabrication Assumptions and Considerations 40
Table 5-2 PC Card Form Factor Substrate Dimensions 40
Table 5-3 Typical Assembly Equipment Limits 45
Table 6-1 Electrical Conductor Spacing 57
Table 6-2 Typical Relative Bulk Dielectric Constant of Printed Board Material 60
Table 6-3 Example Plane Sequences for a Six Layer Printed Board 62
Table 7-1 Effects of Material Type on Construction 65
Table 7-2 Emissivity Ratings for Certain Materials 66
Table 7-3 Printed Board Heatsink Assembly Preferences 69
Table 7-4 Comparative Reliability Matrix Component Lead/Termination Attachment 70
Table 9-1 Minimum Standard Fabrication Allowance for Interconnection Lands 96
Table 9-2 Annular Rings (Minimum) 97
Table 9-3 Minimum Drilled Hole Size for Buried Vias ... 99
Table 9-4 Minimum Drilled Hole Size for Blind Vias 99
Table 9-5 Minimum Hole Location Tolerance, dtp 101
Table 9-6 Through-Hole Diameters Minimum and Maximum and Aspect Ratio, mm [in] 102
Table 10-1 Internal Layer Foil Thickness After Processing 103
Table 10-2 External Conductor Thickness After Plating 103
Table 12-1 Appendix A Coupon Requirements 111
Table 12-2 Appendix B (Legacy) Coupon Requirements 111
Table A.1-1 IPC Coupons 117
Table A.2-1 AB/R Coupon Parameters, mm [in] 118
Table A.3-1 A/R Coupon Parameters, mm [in] 121
Table A.4-1 B/R Coupon Parameters, mm [in] 124
Table A.5-1 E Coupon Parameters, mm [in] 126
Table A.6-1 S Coupon Parameters, mm [in] 128
Table A.7-1 W Coupon Parameters, mm [in] 130
Table A.8-1 D Coupon Parameters, mm [in] 132
Table A.9-1 G Coupon Parameters, mm [in] 134
Table A.10-1 H Coupon Parameters, mm [in] 137
Table A.11-1 P Coupon Parameters, mm [in] 139
Table A.12-1 Z Coupon Parameters, mm [in] 140
Table B.1-1 IPC-2221 Legacy Coupons 142

This is a preview of "IPC 2221B-2012". Click here to purchase the full version from the ANSI store.

Generic Standard on Printed Board Design

1 SCOPE

This standard establishes the generic requirements for the design of organic printed boards and other forms of component mounting or interconnecting structures, including PC card form factors. The organic materials may be homogeneous, reinforced, or used in combination with inorganic materials; the interconnections may be single, double, or multilayered.

1.1 Purpose The requirements contained herein are intended to establish design principles and recommendations that **shall** be used in conjunction with the detailed requirements of a specific interconnecting structure sectional standard (see 1.2) to produce detailed designs intended to mount and connect components. This standard is not intended for use as a performance specification for finished printed boards nor as an acceptance document for electronic assemblies.

1.2 Documentation Hierarchy This standard identifies generic physical design principles, and is supplemented by various sectional standards that provide sharper focus on specific aspects of printed board technology. These include:

IPC-2222 Rigid organic printed board design
IPC-2223 Flexible printed board design
IPC-2225 Organic, MCM-L, printed board design
IPC-2226 High Density Interconnect (HDI) printed board design

The documents are a part of the Family of Design Documents which is identified as IPC-2220. The number IPC-2220 is for ordering purposes only and includes this standard and the four listed above.

Note: IPC-2224, a sectional design standard for PC card form factors, was cancelled by the IPC. Relevant PC form factor design information has been transferred to this revision of IPC-2221 and to IPC-2222.

1.3 Presentation All dimensions and tolerances in this standard are expressed in hard SI (metric) units and parenthetical soft imperial (inch) units. Users of this standard are expected to use metric dimensions. All dimensions greater than or equal to 0.1 mm [0.0039 in] will be expressed in millimeters and inches. All dimensions less than 0.1 mm [0.0039 in] will be expressed in micrometers and microinches.

1.3.1 Dimensional Units The following is taken from National Institute of Standards and Technology - Metric Information and Conversions: ''Beginning January 1, 2010, the European Union Council Directive 80/181/EEC (Metric Directive) allowed the use of only metric units, and prohibited the use of any other measurements for most products sold in the European Union (EU). The Metric Directive made the sole use of metric units obligatory in all aspects of life in the European Union, extending to areas such as product literature and advertising.''

Most component datasheets are provided in metric units. Many printed board designers spend a lot of time converting between imperial (inch) and SI (metric). Round-off errors, when converting units, can result in inaccuracies that result in marginal or failed designs. However, the printed board fabrication vendors often default to imperial units. Electronic Computer Aided Design (ECAD) tools accommodate both metric and imperial library components being placed on the same printed board because dimensional precision is large enough to describe most standard components accurately.

Problems arise when importing information from third party software or trying to mix units during printed board layout. For example, if a portion of the printed board design is an imported Drawing Exchange Format (.DXF) file with metric units that needs to interface with a digital portion done in imperial units, a problem can occur where the data from the two grids are mixed. Unlike importing from libraries, a conversion to printed board units is not always done when importing DXF.

While a user can convert printed board units from metric to imperial in modern day tools without problems, this should not be done too often during the design phase as repeated conversions can introduce unexpected errors. A single set of units should be used in the layout of the printed board. If imported data is in metric units, the layout portion of the process should use metric units. Once the layout is complete and verified, the designer can convert the printed board to imperial units for documentation, if necessary.

1.4 Interpretation ''**Shall**,'' the imperative form of the verb, is used throughout this standard whenever a requirement is intended to express a provision that is mandatory. Deviation from a ''**shall**'' requirement may be considered if sufficient data is supplied to justify the exception.