



ASSOCIATION CONNECTING
ELECTRONICS INDUSTRIES®

IPC-7351

Generic Requirements for Surface Mount Design and Land Pattern Standard

Developed by the Surface Mount Land Patterns Subcommittee (1-13)
of the Printed Board Design Committee (1-10) of IPC

Supersedes:
IPC-SM-782A with
Amendments 1 & 2

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

Contact:

IPC
3000 Lakeside Drive, Suite 309S
Bannockburn, Illinois
60015-1219
Tel 847 615.7100
Fax 847 615.7105

Table of Contents

1 SCOPE	1	3.5.1 Solder Mask Finishes	30
1.1 Purpose	1	3.5.2 Solder Mask Clearances	30
1.2 Documentation Hierarchy	1	3.5.3 Land Pattern Surface Finishes	31
1.2.1 Component and Land Pattern Family Structure	2	4 COMPONENT QUALITY VALIDATION	31
1.3 Performance Classification	2	4.1 Validation Techniques	31
1.3.1 Producibility Levels	2	5 TESTABILITY	32
1.4 Land Pattern Determination	2	5.1 Board and Assembly Test	32
1.5 Terms and Definitions	3	5.1.1 Bare-Board Test	32
2 APPLICABLE DOCUMENTS	5	5.1.2 Assembled Board Test	32
2.1 IPC	5	5.2 Nodal Access	33
2.2 Electronic Industries Association	5	5.2.1 Test Philosophy	33
2.3 Joint Industry Standards (IPC)	5	5.2.2 Test Strategy for Bare Boards	33
2.4 International Electrotechnical Commission	6	5.3 Full Nodal Access for Assembled Board	33
2.5 Joint Electron Device Engineering Council (JEDEC)	6	5.3.1 In-Circuit Test Accommodation	33
3 DESIGN REQUIREMENTS	6	5.3.2 Multi-Probe Testing	34
3.1 Dimensioning Systems	6	5.4 Limited Nodal Access	34
3.1.1 Component Tolerancing	6	5.5 No Nodal Access	34
3.1.2 Land Tolerancing	9	5.6 Clam-Shell Fixtures Impact	34
3.1.3 Fabrication Allowances	9	5.7 Printed Board Test Characteristics	35
3.1.4 Assembly Tolerancing	10	5.7.1 Test Land Pattern Spacing	35
3.1.5 Dimension and Tolerance Analysis	10	5.7.2 Test Land Size and Shape	35
3.2 Design Producibility	18	5.7.3 Design for Test Parameters	35
3.2.1 SMT Land Pattern	18	6 PRINTED BOARD STRUCTURE TYPES	36
3.2.2 Standard Component Selection	18	6.1 General Considerations	38
3.2.3 Circuit Substrate Development	18	6.1.1 Categories	39
3.2.4 Assembly Considerations	18	6.1.2 Thermal Expansion Mismatch	39
3.2.5 Provision for Automated Test	18	6.2 Organic-Base Material	39
3.2.6 Documentation for SMT	18	6.3 Nonorganic Base Materials	39
3.3 Environmental Constraints	18	6.4 Alternative PCB Structures	39
3.3.1 Moisture Sensitive Components	18	6.4.1 Supporting-Plane PCB Structures	39
3.3.2 End-Use Environment Considerations	18	6.4.2 High-Density PCB Technology	39
3.4 Design Rules	20	6.4.3 Constraining Core Structures	39
3.4.1 Component Spacing	20	6.4.4 Porcelainized Metal (Metal Core) Structures ...	39
3.4.2 Single- and Double-Sided Board Assembly	20	7 ASSEMBLY CONSIDERATION FOR SURFACE MOUNT TECHNOLOGY (SMT)	39
3.4.3 Component Stand-off Height for Cleaning	22	7.1 SMT Assembly Process Sequence	39
3.4.4 Fiducial Marks	22	7.2 Substrate Preparation	40
3.4.5 Conductors	24	7.2.1 Adhesive Application	40
3.4.6 Via Guidelines	24	7.2.2 Conductive Adhesive	40
3.4.7 Standard PCB Fabrication Allowances	27	7.2.3 Solder Paste Application	40
3.4.8 Panelization	27	7.2.4 Solder Preforms	41
3.5 Outer Layer Surface Finishes	30	7.3 Component Placement	41

7.3.1	Component Data Transfer	41	8.6.4	Resistance to Soldering	47
7.4	Soldering Processes	42	8.7	SOT89	47
7.4.1	Wave Soldering	42	8.7.1	Basic Construction	47
7.4.2	Vapor Phase (VP) Soldering	42	8.7.2	Marking	48
7.4.3	IR Reflow Soldering	43	8.7.3	Carrier Package Format	48
7.4.4	Hot Air/Gas Convection Soldering	43	8.7.4	Resistance to Soldering	48
7.4.5	Laser Reflow Soldering	43	8.8	SOD123	48
7.4.6	Conduction Reflow Soldering	43	8.8.1	Basic Construction	48
7.5	Cleaning	43	8.8.2	Marking	48
7.6	Repair/Rework	43	8.8.3	Carrier Package Format	48
7.6.1	Heatsink Effects	43	8.8.4	Resistance to Soldering	48
7.6.2	Dependence on Printed Board Material Type	44	8.9	SOT143	48
7.6.3	Dependence on Copper Land and Conductor Layout	44	8.9.1	Basic Construction	48
8	IPC-7352 DISCRETE COMPONENTS	44	8.9.2	Marking	48
8.1	Chip Resistors (RESC)	44	8.9.3	Carrier Package Format	48
8.1.1	Basic Construction	45	8.9.4	Resistance to Soldering	48
8.1.2	Marking	45	8.10	SOT223	48
8.1.3	Carrier Package Format	45	8.10.1	Basic Construction	49
8.1.4	Resistance to Soldering	45	8.10.2	Marking	49
8.2	Chip Capacitors (CAPC)	45	8.10.3	Carrier Package Format	49
8.2.1	Basic Construction	45	8.10.4	Resistance to Soldering	49
8.2.2	Marking	45	8.11	TO252 (DPAK Type)	49
8.2.3	Carrier Package Format	45	8.11.1	Basic Construction	49
8.2.4	Resistance to Soldering	45	8.11.2	Marking	49
8.3	Inductors (INDC, INDM, INDP)	45	8.11.3	Carrier Package Format	49
8.3.1	Basic Construction	45	8.11.4	Resistance to Soldering	49
8.3.2	Marking	45	8.12	Molded Body Diode (DIOSMB)	49
8.3.3	Carrier Package Format	45	9	IPC-7353 GULLWING LEADED COMPONENTS, TWO SIDES	49
8.3.4	Resistance to Soldering	45	9.1	SOIC	50
8.4	Tantalum Capacitors (CAPT)	45	9.1.1	Basic Construction	50
8.4.1	Basic Construction	46	9.1.2	Marking	50
8.4.2	Marking	46	9.1.3	Carrier Package Format	50
8.4.3	Carrier Package Format	46	9.1.4	Resistance to Soldering	50
8.4.4	Resistance to Soldering	46	9.2	SOP8/SOP63 (SSOIC)	50
8.5	Metal Electrode Face Diodes (DIOMELF, RESMELF)	47	9.2.1	Basic Construction	50
8.5.1	Basic Construction	47	9.2.2	Marking	50
8.5.2	Marking	47	9.2.3	Carrier Package Format	50
8.5.3	Carrier Package Format	47	9.2.4	Resistance to Soldering	50
8.5.4	Resistance to Soldering	47	9.3	SOP127 (SOP-IPC-782)	50
8.6	SOT23	47	9.3.1	Marking	51
8.6.1	Basic Construction	47	9.3.2	Carrier Package Format	51
8.6.2	Marking	47	9.3.3	Resistance to Soldering	51
8.6.3	Carrier Package Format	47	9.4	TSSOPS	51
			9.4.1	Marking	51
			9.4.2	Carrier Packages Format	51

Table 3-17	Conductor Width Tolerances, 0.046 mm [0.00181 in] Copper, mm [in]	28
Table 3-18	Feature Location Accuracy (units: mm [in])	28
Table 6-1	Printed Board Structure Comparison	37
Table 6-2	PCB Structure Selection Considerations	38
Table 6-3	PCB Structure Material Properties	38
Table 14-1	JEDEC Standard JEP95 Allowable Ball Diameter Variations for FBGA (mm)	58
Table 14-2	Ball Diameter Sizes (mm)	61
Table 14-3	Land Approximation (mm)	62
Table 14-4	BGA Variation Attributes (mm)	62
Table 14-5	Land-to-Ball Calculations for Current and Future BGA Packages (mm)	62

Generic Requirements for Surface Mount Design and Land Pattern Standard

1 SCOPE

This document provides information on land pattern geometries used for the surface attachment of electronic components. The intent of the information presented herein is to provide the appropriate size, shape and tolerance of surface mount land patterns to insure sufficient area for the appropriate solder fillet to meet the requirements of IPC/EIA J-STD-001, and also to allow for inspection, testing, and rework of those solder joints.

1.1 Purpose Although, in many instances, the land pattern geometries can be different based on the type of soldering used to attach the electronic part, wherever possible, land patterns are defined with consideration to the attachment process being used. Designers can use the information contained herein to establish standard configurations not only for manual designs but also for computer-aided design systems. Whether parts are mounted on one or both sides of the board, subjected to wave, reflow, or other type of soldering, the land pattern and part dimensions should be optimized to insure proper solder joint and inspection criteria.

Land patterns are dimensionally defined and are a part of the printed board circuitry geometry, as they are subject to the producibility levels and tolerances associated with plating, etching, assembly or other conditions. The producibility aspects also pertain to the use of solder mask and the registration required between the solder mask and the conductor patterns.

Note 1: The dimensions used for component descriptions have been extracted from standards developed by industrial and/or standards bodies. Designers should refer to these standards for additional or specific component package dimensions.

Note 2: For a comprehensive description of the given printed board and for achieving the best possible solder joints to the devices assembled, the whole set of design elements includes, beside the land pattern definition:

- Soldermask.
- Solder paste stencil.
- Clearance between adjacent components.
- Clearance between bottom of component and PCB surface, if relevant.
- Keepout areas, if relevant.
- Suitable rules for adhesive applications.

The whole of design elements is commonly defined as “mounting conditions.” This standard defines land patterns and includes recommendations for clearances between adjacent components and for other design elements.

Note 3: Elements of the mounting conditions, particularly the courtyard, given in this standard are related to the reflow soldering process. Adjustments for wave or other soldering processes, if applicable, have to be carried out by the user. This may also be relevant when solder alloys other than eutectic tin lead solders are used.

Note 4: This standard assumes that the land pattern follows the principle that, even under worst case conditions, the overlap of the component termination and the corresponding soldering land will be complete.

Note 5: Heat dissipation aspects have not been taken into account in this standard. Greater mass may require slower process speed to allow heat transfer.

Note 6: Heavier components (greater weight per land) require larger lands; thus, adding additional land pattern surface will increase surface area of molten solder to enhance capabilities of extra weight. In some cases the lands shown in the standard may not be large enough; in these cases, considering additional measures may be necessary.

Note 7: The land form may be rectangular with straight or rounded corners. In the latter case the area of the smallest circumscribed rectangle shall be equal to that of one with straight corners.

1.2 Documentation Hierarchy This standard identifies the generic physical design principles involved in the creation of land patterns for surface mount components, and is supplemented by a shareware IPC-7351 Land Pattern viewer that provides, through the use of a graphical user interface, the individual component dimensions and corresponding land pattern recommendations based upon families of components. The IPC-7351 Land Pattern Viewer is provided on CD-ROM as part of the IPC-7351. Updates to land pattern dimensions, including patterns for new component families, can be found on the IPC website (www.ipc.org) under “PCB Tools and Calculators.” See Appendix C for more information on the IPC-7351 Land Pattern Viewer.