



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

<b>1 SCOPE</b> .....	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	<b>4 COMPONENT QUALITY VALIDATION</b> .....	31
1.3 Performance Classification .....	2	4.1 Validation Techniques .....	31
1.3.1 Producibility Levels .....	2	<b>5 TESTABILITY</b> .....	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
<b>2 APPLICABLE DOCUMENTS</b> .....	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
<b>3 DESIGN REQUIREMENTS</b> .....	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	<b>6 PRINTED BOARD STRUCTURE TYPES</b> .....	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	<b>7 ASSEMBLY CONSIDERATION FOR SURFACE MOUNT TECHNOLOGY (SMT)</b> .....	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
<b>8</b>	<b>IPC-7352 DISCRETE COMPONENTS .....</b>	<b>44</b>	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	<b>9</b>	<b>IPC-7353 GULLWING LEADED COMPONENTS, TWO SIDES .....</b>	<b>49</b>
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](http://www.ipc.org)) under “PCB Tools and Calculators.” See Appendix C for more information on the IPC-7351 Land Pattern Viewer.