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
  1.1 Purpose .................................................... 1
  1.2 Documentation Hierarchy ................................. 1
  1.2.1 Component and Land Pattern Family Structure .... 2
  1.3 Performance Classification ............................. 2
  1.3.1 Productivity Levels .................................. 2
  1.4 Land Pattern Determination ............................ 2
  1.5 Terms and Definitions ................................ 3

2 APPLICABLE DOCUMENTS ................................. 5
  2.1 IPC .......................................................... 5
  2.2 Electronic Industries Association ..................... 5
  2.3 Joint Industry Standards (IPC) ......................... 5
  2.4 International Electrotechnical Commission .......... 6
  2.5 Joint Electron Device Engineering Council (JEDEC) .... 6

3 DESIGN REQUIREMENTS ................................. 6
  3.1 Dimensioning Systems .................................. 6
  3.1.1 Component Tolerancing ............................... 6
  3.1.2 Land Tolerancing ....................................... 9
  3.1.3 Fabrication Allowances ............................... 9
  3.1.4 Assembly Tolerancing ................................ 10
  3.1.5 Dimension and Tolerance Analysis .................. 10
  3.2 Design Productability .................................. 18
  3.2.1 SMT Land Pattern ................................. 18
  3.2.2 Standard Component Selection ....................... 18
  3.2.3 Circuit Substrate Development ....................... 18
  3.2.4 Assembly Considerations ............................ 18
  3.2.5 Provision for Automated Test ....................... 18
  3.2.6 Documentation for SMT .............................. 18
  3.3 Environmental Constraints ............................ 18
  3.3.1 Moisture Sensitive Components .................... 18
  3.3.2 End-Use Environment Considerations ............... 18
  3.4 Design Rules ............................................. 20
  3.4.1 Component Spacing .................................. 20
  3.4.2 Single- and Double-Sided Board Assembly ....... 20
  3.4.3 Component Stand-off Height for Cleaning ........ 22
  3.4.4 Fiducial Marks ....................................... 22
  3.4.5 Conductors ............................................ 24
  3.4.6 Via Guidelines ....................................... 24
  3.4.7 Standard PCB Fabrication Allowances .......... 27
  3.4.8 Panelization .......................................... 27
  3.5 Outer Layer Surface Finishes ......................... 30
  3.5.1 Solder Mask Finishes ................................. 30
  3.5.2 Solder Mask Clearances ............................. 30
  3.5.3 Land Pattern Surface Finishes ...................... 31

4 COMPONENT QUALITY VALIDATION .................... 31
  4.1 Validation Techniques ................................ 31

5 TESTABILITY .............................................. 32
  5.1 Board and Assembly Test ............................... 32
  5.1.1 Bare-Board Test ..................................... 32
  5.1.2 Assembled Board Test ............................... 32
  5.2 Nodal Access ............................................ 33
  5.2.1 Test Philosophy ....................................... 33
  5.2.2 Test Strategy for Bare Boards ...................... 33
  5.3 Full Nodal Access for Assembled Board ............... 33
  5.3.1 In-Circuit Test Accommodation ..................... 33
  5.3.2 Multi-Probe Testing .................................. 34
  5.4 Limited Nodal Access ................................... 34
  5.5 No Nodal Access ......................................... 34
  5.6 Clam-Shell Fixtures Impact ............................ 34
  5.7 Printed Board Test Characteristics ................... 35
  5.7.1 Test Land Pattern Spacing ......................... 35
  5.7.2 Test Land Size and Shape ........................... 35
  5.7.3 Design for Test Parameters ......................... 35

6 PRINTED BOARD STRUCTURE TYPES ................. 36
  6.1 General Considerations ................................ 36
  6.1.1 Categories ........................................... 39
  6.1.2 Thermal Expansion Mismatch ........................ 39
  6.2 Organic-Base Material ................................ 39
  6.3 Nonorganic Base Materials ............................ 39
  6.4 Alternative PCB Structures ............................ 39
  6.4.1 Supporting-Plane PCB Structures .................. 39
  6.4.2 High-Density PCB Technology ....................... 39
  6.4.3 Constraining Core Structures ...................... 39
  6.4.4 Porcelainized Metal (Metal Core) Structures .... 39

7 ASSEMBLY CONSIDERATION FOR SURFACE MOUNT TECHNOLOGY (SMT) ............... 39
  7.1 SMT Assembly Process Sequence ....................... 39
  7.2 Substrate Preparation ................................ 40
  7.2.1 Adhesive Application ................................ 40
  7.2.2 Conductive Adhesive ................................. 40
  7.2.3 Solder Paste Application ............................ 40
  7.2.4 Solder Preforms ...................................... 41
  7.3 Component Placement ................................. 41
8 IPC-7352 DISCRETE COMPONENTS

8.1 Chip Resistors (RESC) ........................................ 44
8.1.1 Basic Construction ........................................ 45
8.1.2 Marking ..................................................... 45
8.1.3 Carrier Package Format ................................... 45
8.1.4 Resistance to Soldering ................................... 45
8.2 Chip Capacitors (CAPC) ..................................... 45
8.2.1 Basic Construction ........................................ 45
8.2.2 Marking ..................................................... 45
8.2.3 Carrier Package Format ................................... 45
8.2.4 Resistance to Soldering ................................... 45
8.3 Inductors (INDC, INDM, INDP) ....................... 45
8.3.1 Basic Construction ........................................ 45
8.3.2 Marking ..................................................... 45
8.3.3 Carrier Package Format ................................... 45
8.3.4 Resistance to Soldering ................................... 45
8.4 Tantalum Capacitors (CAPT) ......................... 45
8.4.1 Basic Construction ........................................ 46
8.4.2 Marking ..................................................... 46
8.4.3 Carrier Package Format ................................... 46
8.4.4 Resistance to Soldering ................................... 46
8.5 Metal Electrode Face Diodes (DIOMELF, RESMELF) ........................................ 47
8.5.1 Basic Construction ........................................ 47
8.5.2 Marking ..................................................... 47
8.5.3 Carrier Package Format ................................... 47
8.5.4 Resistance to Soldering ................................... 47
8.6 SOT23 ......................................................... 47
8.6.1 Basic Construction ........................................ 47
8.6.2 Marking ..................................................... 47
8.6.3 Carrier Package Format ................................... 47
8.6.4 Resistance to Soldering ................................... 47
8.7 SOT89 .......................................................... 47
8.7.1 Basic Construction ........................................ 47
8.7.2 Marking ..................................................... 48
8.7.3 Carrier Package Format ................................... 48
8.7.4 Resistance to Soldering ................................... 48
8.8 SOD123 .......................................................... 48
8.8.1 Basic Construction ........................................ 48
8.8.2 Marking ..................................................... 48
8.8.3 Carrier Package Format ................................... 48
8.8.4 Resistance to Soldering ................................... 48
8.9 SOT143 .......................................................... 48
8.9.1 Basic Construction ........................................ 48
8.9.2 Marking ..................................................... 48
8.9.3 Carrier Package Format ................................... 48
8.9.4 Resistance to Soldering ................................... 48
8.10 SOT223 .......................................................... 48
8.10.1 Basic Construction ........................................ 48
8.10.2 Marking ..................................................... 48
8.10.3 Carrier Package Format ................................... 48
8.10.4 Resistance to Soldering ................................... 48
8.11 TO252 (DPAK Type) ....................................... 49
8.11.1 Basic Construction ........................................ 49
8.11.2 Marking ..................................................... 49
8.11.3 Carrier Package Format ................................... 49
8.11.4 Resistance to Soldering ................................... 49
8.12 Molded Body Diode (DIOSMB) ....................... 49

9 IPC-7353 GULLWING LEADED COMPONENTS, TWO SIDES

9.1 SOIC .......................................................... 50
9.1.1 Basic Construction ........................................ 50
9.1.2 Marking ..................................................... 50
9.1.3 Carrier Package Format ................................... 50
9.1.4 Resistance to Soldering ................................... 50
9.2 SOP8/SOP63 (SSOIC) ...................................... 50
9.2.1 Basic Construction ........................................ 50
9.2.2 Marking ..................................................... 50
9.2.3 Carrier Package Format ................................... 50
9.2.4 Resistance to Soldering ................................... 50
9.3 SOP127 (SOP-IPC-782) .................................... 50
9.3.1 Marking ..................................................... 51
9.3.2 Carrier Package Format ................................... 51
9.3.3 Resistance to Soldering ................................... 51
9.4 TSSOP5 .......................................................... 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
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.