

IPC-7352 **2023 - May** **Generic Guideline for** **Land Pattern Design**

An international standard developed by IPC



BUILD ELECTRONICS BETTER

IPC Mission

IPC is a global trade association dedicated to furthering the competitive excellence and financial success of its members, who are participants in the electronics industry.

In pursuit of these objectives, IPC will devote resources to management improvement and technology enhancement programs, the creation of relevant standards, protection of the environment, and pertinent government relations.

IPC encourages the active participation of all its members in these activities and commits to full cooperation with all related organizations.

About IPC Standards

IPC standards and publications are designed to serve the public interest through eliminating misunderstandings between manufacturers and purchasers, facilitating interchangeability and improvement of products, and assisting the purchaser in selecting and obtaining with minimum delay the proper product for their particular need. Existence of such IPC standards and publications shall not in any respect preclude any entity from manufacturing or selling products not conforming to such IPC standards and publication, nor shall the existence of such IPC standards and publications preclude their voluntary use.

IPC standards and publications are approved by IPC committees without regard to whether the IPC standards or publications may involve patents on articles, materials or processes. By such action, IPC does not assume any liability to any patent owner, nor does it assume any obligation whatsoever to parties adopting an IPC standard or publication. Users are wholly responsible for protecting themselves against all claims of liabilities for patent infringement.

IPC Position Statement on Specification Revision Change

The use and implementation of IPC standards and publications are voluntary and part of a relationship entered into by customer and supplier. When an IPC standard or publication is revised or amended, the use of the latest revision or amendment as part of an existing relationship is not automatic unless required by the contract. IPC recommends the use of the latest revision or amendment.

Standards Improvement Recommendations

IPC welcomes comments for improvements to any standard in its library. All comments will be provided to the appropriate committee.

If a change to technical content is requested, data to support the request is recommended. Technical comments to include new technologies or make changes to published requirements should be accompanied by technical data to support the request. This information will be used by the committee to resolve the comment.

To submit your comments, visit the IPC Status of Standardization page at www.ipc.org/status.



IPC-7352

Generic Guideline for Land Pattern Design

Developed by the 1-13 Land Pattern Subcommittee
of the 1-10 Printed Board Design Committee of IPC

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

Contact:

IPC
3000 Lakeside Drive, Suite 105N
Bannockburn, Illinois
60015-1249
Tel 847 615.7100
Fax 847 615.7105

Currently in preview, click buy full version

This Page Intentionally Left Blank

Acknowledgment

Any document involving a complex technology draws material from a vast number of sources across many continents. While the principal members of the 1-13 Land Pattern Subcommittee of the 1-10 Printed Board Design Committee are shown below, it is not possible to include all of those who assisted in the evolution of this standard. To each of them, the members of the IPC extend their gratitude.

1-10 Printed Board Design Committee

Chair:

Philip Henault, Raytheon

Vice-Chair:

Michelle Gleason,
Plexus Corporation

1-13 DFX Land Pattern Subcommittee

Chair:

Michelle Gleason, Plexus

Technical Liaison of the IPC Board of Directors

Bob Neves

Microtek (Changzou) Laboratories

Land Pattern Subcommittee

Constantino Gonzalez

ACME Training & Consulting

Pietro Vergine

Advanced Rework Technology

Jesus Castane

Arrival Limited

Jeff Schake

ASM Assembly Systems

Timothy Carey

Boeing Research & Development

Miguel Dominguez

Continental Automotive

Russell Steiner

Casco Products

Geok Ang Tan

DSO National Laboratories

Gary Ferrari

EPTAC Corporation

Olga Scheglov

EPTAC Corporation

Francesco di Maio

GESTLABS S.r.l

Paul Bartholomew

HP Inc.

Jennie Hwang

H-Technologic Group

Joe Hughes

Hughes Circuits, Inc.

Perla Wehbe

I.F. Engineering

Thomas Romont

IFTEC

Michael Creeden

Insulectro

Lim L Ming

Jabil Circuit Inc.

Stephen Golemme

JITX Inc.

Jasbir Bath

Koki Solder America

Keld Maaloe

LEGCO Systems A/S

Scott Rowles

Lockheed Martin Corporation

Kyle Johnson

Lockheed Martin Missiles & Fire

Control

Patti LaRochelle

Miraco, Inc.

Michael Durkan

Mentor Graphics Corporation

Richard Henry

Mentor Graphics Corporation

Gyanesh Mathur

Nanowear Inc.

Karen McConnell

Northrop Grumman Corporation

Michelle Gandhu

Northrop Grumman Space Systems

Karl Sauter

Oracle America, Inc.

Thomas Hausherr

PCB Libraries

Dale Lee

Plexus Corporation

Michelle Gleason

Plexus Corporation

James Daggett

Raytheon Company

Jeff Shubrooks

Raytheon Company

Lance Brack

Raytheon Missile Systems

Rainer Taube

Taube Electronics GmbH

Arnaud Grivon

Thales Global Services

Jose Olivares

Vitesco Technologies

Currently in preview, click buy full version

This Page Intentionally Left Blank

Table of Contents

1	SCOPE	1	4.2	TH Mounting Techniques That Impact Land Pattern	19
1.4	Producibility Levels	2	4.2.1	Axial Land Pattern Design	19
1.5	Density Levels	2	4.2.2	Radial Land Pattern Design	19
1.5	Use of “Lead”	2	4.2.3	Solder Side Land Pattern	19
1.7	Terms and Definitions	2	4.3	Land Pattern Creation	19
2	APPLICABLE DOCUMENTS	3	4.3.1	Courtyard	19
2.2	Joint Industry Guidelines (IPC)	3	4.4	Through-hole Padstacks	19
2.5	Joint Electron Device Engineering Council (JEDEC) ²	3	4.4.1	Nominal Hole Diameter for P vs. b-free Solder Process	19
3	SURFACE-MOUNT TECHNOLOGY (SMT) DESIGN REQUIREMENTS	3	4.4.1.1	Finite Solder Flow	19
3.1.1	Component Tolerancing and Dimensioning	4	4.4.1.2	Infinite Solder Flow	20
3.1.2	Solving for Dimension Z	7	4.4.3	Thermal Relief	21
3.1.3	Land Tolerancing	8	4.4.4	Anti-Pad	21
3.1.4	Dimension and Tolerance Analysis	8	4.5	Press-fit (compliant) Pin Type	21
3.2	Tolerance and Solder Joint Analysis	9	4.6	Through Hole Padstack – Non-Plated	21
3.3	Courtyard Determination	13	4.6.1	Maximum Terminal Dimension	21
3.4	Land Pattern Naming Convention for SMD Packages	13	4.6.2	Nominal Hole Diameter	21
3.4.1	Land Pattern Naming Convention Notes	16	4.6.3	Anti-Pad	21
3.4.2	Naming Convention Special Character Use for Land Patterns	16	4.7	Land Pattern Naming Convention for PTH Packages	25
3.4.3	Suffix Naming Convention for Land Patterns	16	4.7.1	Land Pattern Naming Convention for Unique Packages and Connectors	25
3.4.3.1	SON, QFN, SOP and QFP Components That Have Different Thermal Pad Sizes	16	5	PADSTACK NAMING CONVENTION	25
3.4.3.2	Gull Wing Components That Have Different Lead Tolerances	16	5.1	Basic Land Shape Letters	25
3.4.3.3	Use Of Lead Geometry In Land Pattern Name	17	5.2	Padstack Defaults	25
3.4.3.4	Components With Hidden, Deleted Or Revealed Pins	17	5.2.1	Examples of the Padstack Naming Convention	25
3.4.4	Land Pattern Naming Convention for Unique Packages and Connectors features.	17	5.2.2	Padstack Naming Convention Modifiers	26
	THROUGH-HOLE MOUNTED DEVICES	18	5.2.2.1	Use Of Letter v	27
4.1	Component Type Descriptions	18	5.2.2.2	Use Of Letter w	27
4.1.1	Axial Terminal Components	18	5.2.3	Examples Utilizing Modifiers – The following provide various examples of the padstack naming convention’s usage of modifiers:	27
4.1.2	Radial Terminal Components	18	5.3	Paste Mask for Thermal Tabs	29
4.1.3	Multiple Pin Terminal Components	18	6	LAND PATTERN QUALITY VALIDATION	29
4.1.4	Electrical Connectors	18	7	ZERO COMPONENT ORIENTATIONS	30

Appendix A (Informative) Test Patterns – Process Evaluations	Figures
Evaluations	31
A.1 Test Vehicle.....	31
A.2 Test Patterns -In-Process Validator	31
A.3 Stress Testing	32
Appendix B Polarity Marking Legend	34
B.1 Bottom Only Terminal Packages	34
Appendix C Component Package Naming Reference	35
C.1 Area Array Components (BGA, FBGA, CGA, LGA, Chip Array)	35
C.2 Component Lead Packages.....	35
C.3 Concave Chip Array Packages (RESCAV, CAPCAV, INDCAV, OSCSC, OSCCCC)	35
C.4 Convex Chip Array Packages (RESCAXE, RESCAXS).....	35
C.5 Flat Chip Array Packages (RESCAF, CAPCAF, INDCAF)	36
C.6 IPC-7359 No-Lead Components (QFN, PQFN, SON, PSON, DFN, LCC)	36
C.6.1 Leadless Chip Carrier (LCC).....	36
C.6.2 Quad Flat No-Lead (QFN)	37
C.6.3 Small Outline No-Lead Package (SON).....	37
C.6.4 Small Outline and Quad Flat No-Lead with Pullback Leads (PQFN, PSON).....	38
C.6.5 Dual Flat No-Lead (DFN).....	38
Figure 3-1 Profile Tolerancing Method	4
Figure 3-2 Example of 3216 (1206) Capacitor.....	5
Figure 3-3 Profile Dimensioning of Gull wing Leaded SOIC.....	6
Figure 3-4 Courtyard Boundary Area Illustration.....	13
Figure 4-1 Horizontally Mounted Radial Leaded Component	19
Figure 5-1 Basic Land Shapes	25
Figure A-1 General Description of Process Validation Contact Pattern and Interconnect	31
Figure A-2 Photo image of IPC Test Board for Primary Side.....	32
Figure B-1 Popular Polarity Marking Shapes	33
Figure B-2 Gull Wing Terminal Legend Polarity Marking Location	33
Figure B-3 Sample 0.50 mm Pitch SOP Legend and Polarity Marking Rules.....	34
Figure B-4 Bottom Only Terminal Packages	34
Figure C-1 Polarized Chip Packages	34
Figure C-1 Side Concave Chip Component	35
Figure C-2 Corner Concave Chip Component	35
Figure C-3 Convex Chip Component “E Version”	36
Figure C-4 Convex Chip Component “S Version”	36
Figure C-5 Flat Chip Component.....	36
Figure C-6 LCC Component	36
Figure C-7 Quad Flat No-Lead (QFN) Construction	37
Figure C-8 QFN Devices with Multiple Paste Mask	37
Figure C-9 Small Outline No-Lead Package (SON)	37
Figure C-10 PQFN Device with Pullback Leads	38
Figure C-12 Axial Component Examples	39
Figure C-13 Radial Terminal Components	40
Figure C-15 Pin Grid Array (PGA).....	43
Figure C-16 Connectors.....	43

Tables

Table 3-1	Flat Ribbon L and Gull wing Leads	9	Table 3-12	Flat No-Lead with Pullback or Under Body Leads	12
Table 3-2	J Leads	9	Table 3-13	Corner Concave Lead	12
Table 3-3	Rectangular or Square-End Components with Lead Widths	10	Table 3-14	Aluminum Electrolytic Capacitor and 2-pin Crystal1	12
Table 3-4	Rectangular or Square-End Components with Pin Widths Smaller than 0.5 mm where Leads are 1, 2, 3 or 5 Sided	10	Table 3-15	Small Outline Components, Flat Lead	13
Table 3-5	Cylindrical End Cap Terminations (MELF)	10	Table 3-16	Land Pattern Convention for SMD Packages	14
Table 3-6	Leadless Components with Concave/ Castellated Terminations	10	Table 4-1	Finite Solder Flow (Includes Pin-in-Pack and Captive Solder Charge)	20
Table 3-7	Butt and I Lead	11	Table 4-2	Infinite Solder Flow (Includes Wave, Solder Pot, Selective Solder, Hand Solder)	20
Table 3-8	Inward Flat Ribbon L-Leads	11	Table 4-3	Terminal to Finished Hole Size Adjustments for Board Thickness	21
Table 3-9	Flat Lug Leads ²	11	Table 4-4	Pad Size	21
Table 3-10	Flat No-Lead with Solderable Vertical Surface	11	Table 4-5	Nominal Hole Diameter	22
Table 3-11	Ball Grid Array Components	12	Table 4-6	Land Pattern Convention for TH Packages	22

This Page Intentionally Left Blank

Currently in preview, click buy full version

IPC-7352

Generic Guideline for Land Pattern Design

1 SCOPE

This document provides generic guidelines on land pattern geometries used for the attachment of electronic components to a printed board, as well as design recommendations for achieving the best possible solder joints to the devices assembled. Adjustments to the information in this guideline may be required to meet company and/or board technology requirements. It is recommended that a company should document the modifications to the IPC-7352 content in corporate command media documentation.

A land pattern is the representation of the area and features on a printed board needed for a component to be placed and attached to the printed board during an assembly process. The land pattern is usually built using ECAD Library tools.

1.1 Purpose The intent of the information presented herein is to provide the appropriate size, shape and tolerance of through-hole and surface mount land patterns to ensure sufficient area for the appropriate solder fillet to meet the requirements of IPC J-STD-001, and to allow for inspection, testing and rework of those solder joints. Designers can use the information contained herein to establish guideline land pattern geometries not only for manual designs but also for computer-aided design systems. Whether parts are mounted on one or both sides of the printed board and are subjected to wave, reflow, or other type of soldering, the land pattern and part dimensions should be optimized to ensure proper solder joint and inspection criteria.

Land patterns become a part of the printed board circuitry and they are subject to the producibility levels and tolerances associated with fabrication and assembly processes. The producibility aspects also pertain to the use of solder mask and the registration required between the solder mask and the conductor patterns.

In addition to the land pattern geometries required for proper solder joint formation, other mounting conditions should be considered, such as solder mask clearance, solder paste stencil aperture sizes, clearance between adjacent components, clearance between the bottom of the component and the printed board surface (if relevant), keep-out areas (if relevant) and adhesive applications. These additional features become part of the overall land pattern guidelines for each component type.

Note 1: The dimensions used for component descriptions have been extracted from the documents listed in 2 Applicable Documents. Designers should refer to the manufacturer's datasheet for specific component package dimensions.

Caution: Users should be aware that individual component datasheets may not meet standardized component outlines (e.g., JEDEC standard component outlines).

Note 2: Elements of the mounting conditions, particularly the courtyard, given in this guideline are related to the reflow soldering process. Adjustments for wave or other soldering processes, if applicable, should be carried out by the user. This may also be relevant when solder alloys other than eutectic SnPb or SnAgCu solders are used.

Note 3: Heat dissipation aspects have not been considered in this guideline.

Note 4: In some cases, the lands shown in this guideline may not apply for a particular application and may need to be altered based on the end-item environmental requirements. For surface mount components, the solder joints provide not only the electrical connection, but the mechanical support as well.

Note 5: Shock and vibration effects are not considered in this guideline.

1.2 Classification This guideline identifies the generic physical design principles involved in the creation of land patterns for surface mount and through-hole components.

1.3 Performance Classification IPC-J-STD-001 recognizes that electrical and electronic products are subject to classifications by intended end-item use. Three general end-product classes have been established to reflect differences in producibility, complexity, functional performance requirements and verification (i.e., inspection or test) frequency:

CLASS 1 General Electronic Products

Includes products suitable for applications where the major requirement is function of the completed assembly.

CLASS 2 Dedicated Service Electronic Products

Includes products where continued performance and extended life is required and for which uninterrupted service is desired but not critical. Typically, the end-use environment would not cause failures.