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

Supersedes:
IPC-7352 -
June 2010

Table of Contents

Figures

Tables

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.