TN0018
Technical note

Surface mounting guidelines for MEMS sensors in an LGA package

Introduction

This technical note provides general guidelines for soldering MEMS sensor products housed in an LGA surface-mount package.

Note: Information provided in this document is to be intended for use as reference material concerning PCB design and soldering processes. For device specifications, refer to the corresponding datasheet.

Contents

1 General guidelines for soldering surface-mount MEMS sensors

The following three elements must be considered in order to adhere to common PCB design and good industrial practices when soldering MEMS sensors:

- PCB design should be as symmetrical as possible
  - large traces on Vdd / Gnd lines are not required (very low power consumption)
  - no vias or traces below the sensor footprint
- Solder paste must be as thick as possible (after soldering) in order to:
  - reduce the decoupling stress from the PCB to the sensor
  - avoid that the PCB solder mask touches the device package
- Solder paste thickness must be as uniform as possible (after soldering) to avoid uneven stress:
  - Final volume of soldering paste within 20% among lands is possible using the SPI (Solder Paste Inspection) control technique

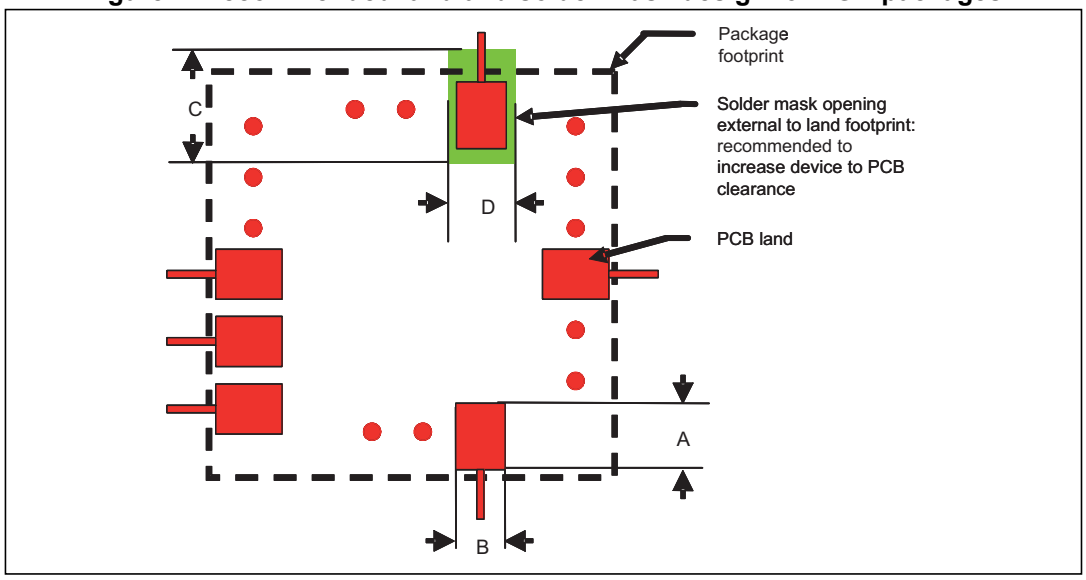
2 PCB design guidelines

PCB land and solder mask general recommendations are shown in *Figure 1*. Refer to the device datasheet for pad count, size and pitch.

- It is recommended to open the solder mask external to the PCB land;
- It is strongly recommended not to place any structure on the top metal layer underneath the sensor (on the same side of the board). This must be defined as a keepout area.
- Traces connected to pads should be as much symmetric as possible. Symmetry and balance for pad connection will help component self-alignment and will lead to better control of solder paste reduction after reflow;
- For optimal performance of the device, it is strongly recommended to place screw mounting holes at a distance greater than 2 mm from the sensor.
- If present, the pin #1 indicator must be left unconnected to ensure proper device functionality.
- In order to prevent noise coupling and thermo-mechanical stress, following standard industry design practices for component placement is advised.

2.1 PCB design rules

Figure 1. Recommended land and solder mask design for LGA packages



PCB land design and connecting traces should be designed symmetrically.

For LGA pin spacing greater than 200 µm:

A = PCB land length = LGA solder pin length + 0.1 mm

B = PCB land width = LGA solder pin width + 0.1 mm

For LGA pin spacing equal to or less than 200 µm:

A = PCB land length = LGA solder pin length

B = PCB land width = LGA solder pin width

C = Solder mask opening length (when applicable) = PCB land length + 0.1 mm

D = Solder mask opening width = PCB land width + 0.1 mm

3 Stencil design and solder paste application

The thickness and the pattern of the soldering paste are important for the proper MEMS sensor mounting process.

- Stainless steel stencils are recommended for solder paste application;
- A stencil thickness of 90 - 150 µm (3.5 - 6 mils) is recommended for screen printing;
- The openings of the stencil for the signal pads should be between 70% and 90% of the PCB pad area;
- Optionally, for better solder paste release, the aperture walls should be trapezoidal and the corners rounded;
- The fine pitch of the IC leads requires accurate alignment of the stencil to the printed circuit board. The stencil and printed circuit assembly should be aligned to within 25 µm (1 mil) prior to application of the solder paste.

4 Process considerations

- The soldering profile depends on the number, size and placement of components in the application board. For this reason it is not possible to define a unique soldering profile for the sensor only. The customer should use a time and temperature reflow profile based on PCB design and manufacturing expertise.
- In order to reduce residual stress on the components, the recommended ramp-down temperature slope should not exceed -3 °C/s.
- No solder material reflow on the side of the package is allowed since LGA packages show metal traces on the side of the package.
- If “self-cleaning” solder paste is not used, the board must be properly cleaned after soldering to eliminate any possible source of leakage between adjacent pads due to flux residues.
- The final volume of soldering paste applied to each PCB land is recommended to be within 20% among (all) the PCB land pads.
- Based on the Jedec 9702 standard, a component shows negligible output variation up to stress intensity of 500 me (microstrain).

5 Solder heat resistance and environmental specifications

In order to meet environmental requirements, ST offers these devices in ECOPACK® packages. These packages have a lead-free second level interconnect. The category of second level interconnect is marked on the inner box label, in compliance with JEDEC Standard JESD97. The maximum ratings related to soldering conditions are also marked on the inner box label.

LGA packages for MEMS sensors are qualified for soldering heat resistance according to JEDEC J-STD-020, in MSL3 condition.

6 Revision history

Table 1. Document revision history

Date	Revision	Changes
12-Oct-2006	1	Initial release
30-Apr-2008	2	Added appendix with mechanical information
30-Jul-2013	3	Updated *Section 2: PCB design guidelines* Updated *Section 4: Process considerations* Removed Appendix A with LGA package drawings and dimensions Minor textual updates throughout technical note
31-Oct-2013	4	Textual update in *Note on page 5*
24-Mar-2014	5	Updated *Section 2: PCB design guidelines*; *Section 3: Stencil design and solder paste application*; and *Section 4: Process considerations*