Surface mounting guidelines for MEMS sensors in a QFPN package

Introduction

This document provides general guidelines for soldering MEMS products housed in a **Q**uad **F**lat **P**ackage **N**o lead (QFPN) 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.

1 General guidelines for soldering surface-mount MEMS sensors

The following 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
- PCB placement should avoid locations close to hot spots (microprocessors, graphic controllers, batteries, …), close to pushbuttons, screws and/or PCB anchor points since these locations can produce mechanical stress affecting sensor precision.
- High-amplitude resonances (vibrations) of the PCB should be avoided or sensors should be placed in positions in which these resonances are minimized.

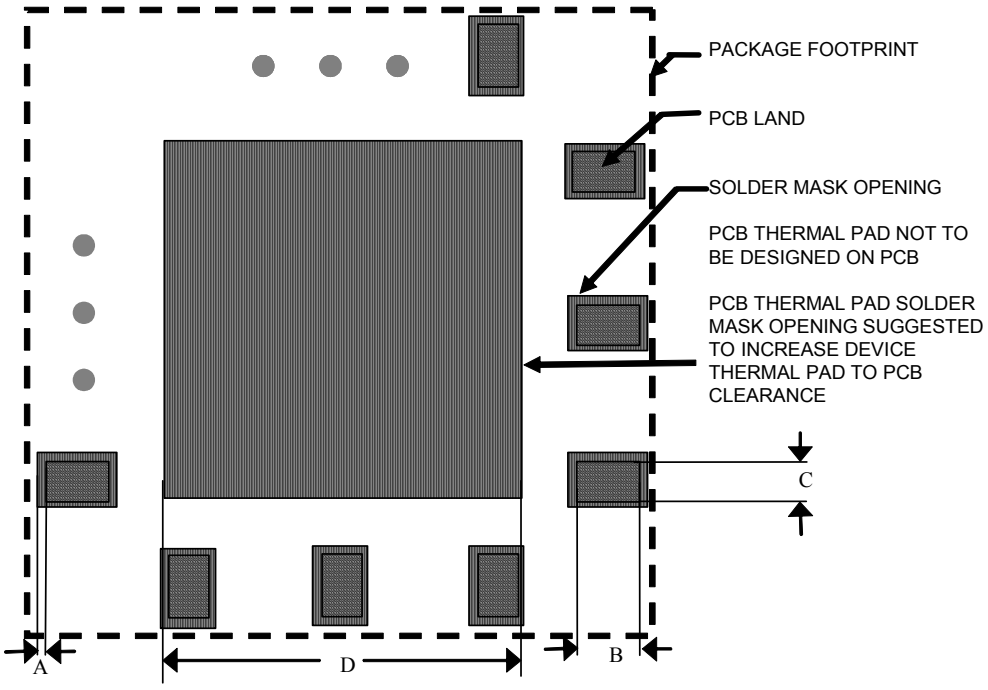
2 PCB design guidelines

PCB land and solder mask general recommendations are shown in Figure 1 and Figure 2. 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.
- Bottom side pin #1 indicator (if not defined by thermal pad shape), is electrically connected to the die pad. Leave pin 1 indicator unconnected during soldering.
- 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 for QFN type “PB”

Figure 1. Recommended land and solder mask for QFPN packages (type “PB”)



A = Clearance from PCB land edge to solder mask opening ≤ 0.1 mm to ensure that some solder mask remains between PCB pads

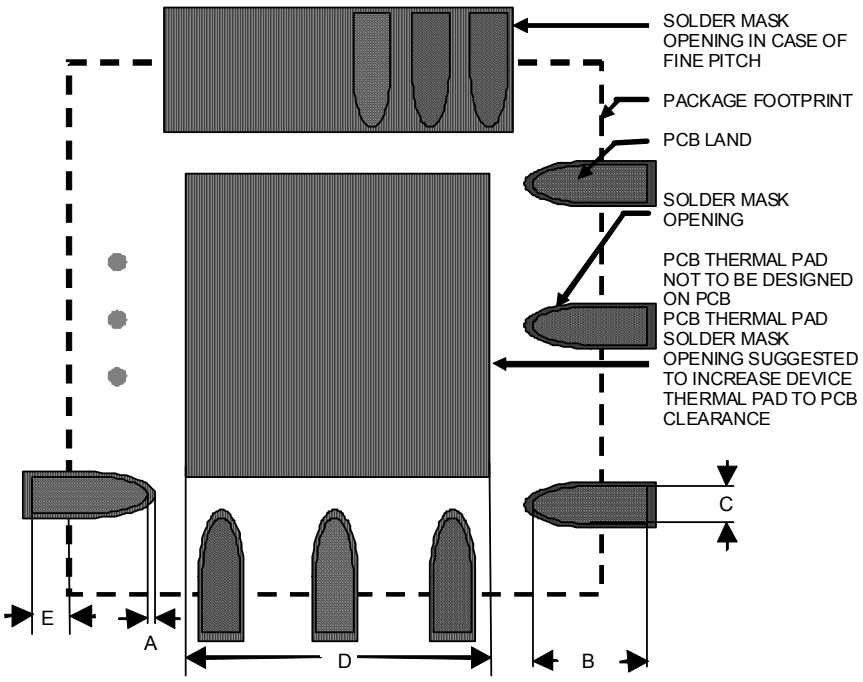
B = PCB land length = QFPN solder pad length + 0.1 mm

C = PCB land width = QFPN solder pad width + 0.1 mm

D = PCB thermal pad solder mask opening = QFPN thermal pad side + 0.2 mm

2.2 PCB design rules for QFN type “FL”

Figure 2. **Recommended land and solder mask for QFPN packages (type “FL”)**



A = Clearance from PCB land edge to solder mask opening <= 0.1 mm to ensure that some solder mask remains between PCB pads

B = PCB land length = QFN solder pad length + 0.3 mm

C = PCB land width = QFN solder pad width + 0.1 mm

D = PCB thermal pad solder mask opening = QFN thermal pad side + 0.2 mm

E = PCB land outer extent = QFN package edge + 0.25 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.
- 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 different grades of ECOPACK packages, depending on their level of environmental compliance. ECOPACK specifications, grade definitions and product status are available at: www.st.com. ECOPACK is an ST trademark.

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.

QFN packages for accelerometers are qualified for soldering heat resistance according to JEDEC J-STD-020C, in MSL3 condition.

Revision history

Table 1. **Document revision history**

Date	Version	Changes
08-Nov-2006	1	Initial release
01-Apr-2010	2	Updated Section 1: General guidelines for soldering surface mount MEMS and Chapter 2: PCB design guidelines. Added Chapter 2.2: PCB design rules for QFN type “FL”, Section A.3: QFPN 7x7x1.8 mm, pitch 0.80, 28 land and Section A.4: QFPN 4x4x1.8 mm, pitch 0.50, 24 land.
10-Dec-2020	3	General revision of technical note

Contents