TN0019
Technical note

MEMS in QFPN package surface mounting guidelines

Introduction

This document is a general guideline about soldering MEMS products packaged in **Q**uad **F**lat **P**ackage **N**o lead surface mount.

Contents

1 General guidelines about soldering surface mount MEMS 3

2 PCB design guidelines 4

2.1 PCB design rules for QFN type “PB” 4

2.2 PCB design rules for QFN type “FL” 5

3 Stencil design and solder paste application 6

4 Process consideration 7

5 Solder heat resistance and environmental specification 8

Appendix A QFPN packages outlines 9

A.1 QFPN 7x7x1.8 mm, pitch 0.80, 28 land 9

A.2 QFPN 7x7x1.8 mm, pitch 0.50, 44 land 10

A.3 QFPN 7x7x1.8 mm, pitch 0.80, 28 land 11

A.4 QFPN 4x4x1.8 mm, pitch 0.50, 24 land 12

6 Revision history 13

1 General guidelines about soldering surface mount MEMS

As common PCB design and industrial practice, when considering MEMS devices soldering, there are always 3 elements to take into consideration:

1. PCB with its own conductive layers (i.e. Copper) and other organic materials used for board protection and dielectric isolation.
2. MEMS to be mounted on the board. MEMS senses a variety of physical stimuli as acceleration, velocity, etc., but it senses also the mechanical stress coming from the applicative board. This stress can be minimized with simple PCB design rules.
3. SOLDERING PASTE like SnAgCu. This soldering paste can be dispensed on the board with a screen printing method through a stencil. The pattern of the soldering paste on the PCB is given by the stencil mask itself.

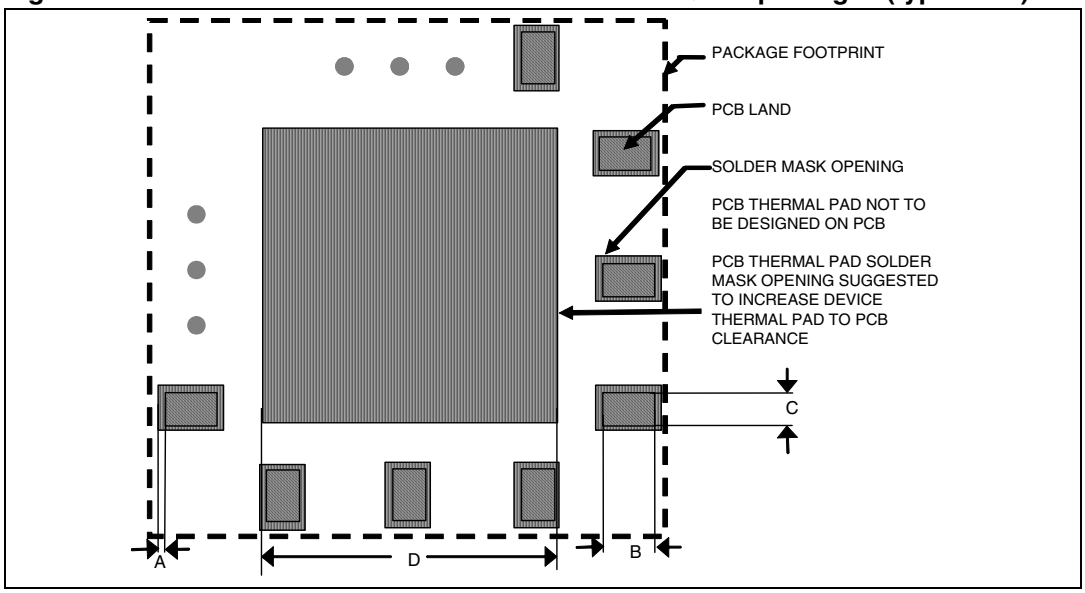
2 PCB design guidelines

PCB land and solder masking general recommendations are shown in *Figure 1* and *Figure 2*. Refer to device datasheet or appendix A for land count, size and pitch.

- It is recommended to open solder mask external to PCB land as per NSMD (non-solder mask defined) practice. Design of the solder resist mask in the pad area may be different for fine pitch devices, in this case a rectangle mask shape containing all pads may be recommended to avoid solder resist defects;
- It's mandatory for correct device functionality that some clearance is ensured to be present between accelerometer thermal pad and PCB. In order to obtain this clearance it is recommended to open the PCB thermal pad solder mask;
- The area below the sensor (on the same side of the board) must be defined as keep-out area. It is strongly recommended to not place any structure in top metal layer underneath the sensor (e.g.: PCB thermal pad not recommended);
- 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 a better control of solder paste reduction after reflow;
- For better performances overtemperature it is strongly recommended not to place large insertion components like buttons or shielding boxes at distance less 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.

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

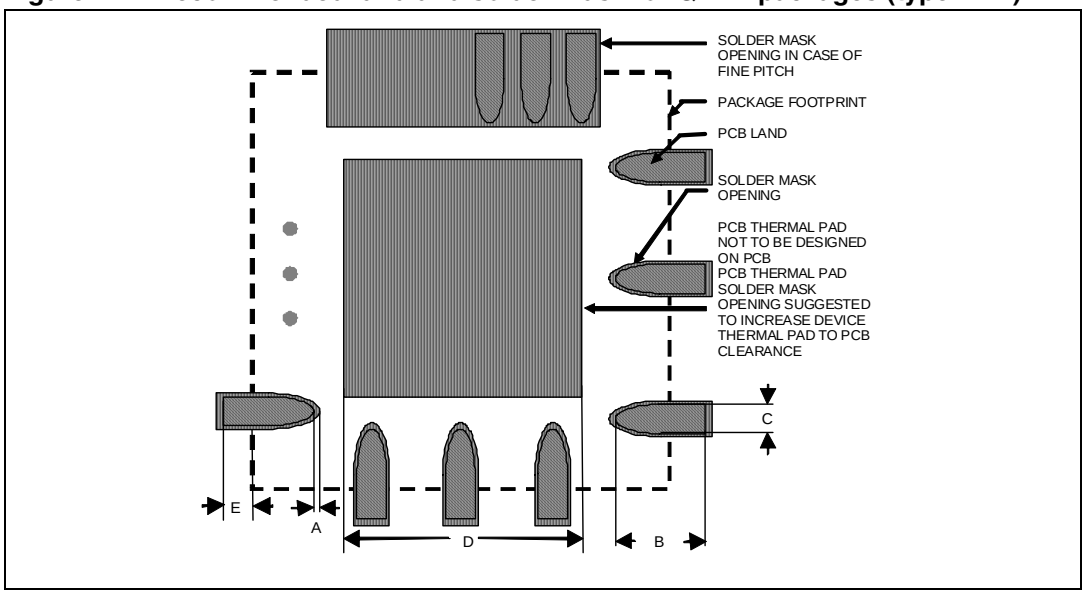
B = PCB land length = QFPN solder pad length + 0.1 mm

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 accelerometer mounting process

- Stainless steel stencils are recommended for solder paste application;
- A stencil thickness of 90 - 150 µm (5 - 6 mils) is recommended for screen printing;
- The final thickness of soldering paste should allow proper cleaning of flux residuals and clearance between sensor package and PCB;
- Stencil aperture should have rectangular shape with dimension up to 25 µm (1mil) smaller than PCB land;
- The openings of the stencil for the signal pads should be between 50% and 80% 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 (1mil) prior to application of the solder paste.

4 Process consideration

- In case of use of no self-cleaning solder paste it is mandatory proper washing of the board after soldering to eliminate any possible source of leakage between adjacent pads due to flux residues;
- The PCB soldering profile depends on the number, size and placement of components in the application board. It is not functional to define a specific soldering profile for the accelerometer only. Customer should use a time and temperature reflow profile that is derived from the PCB design and manufacturing experience.

5 Solder heat resistance and environmental specification

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.
The maximum ratings related to soldering conditions are also marked on the inner box label.

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

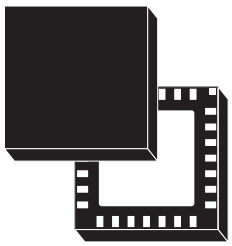
Appendix A QFPN packages outlines

A.1 QFPN 7x7x1.8 mm, pitch 0.80, 28 land

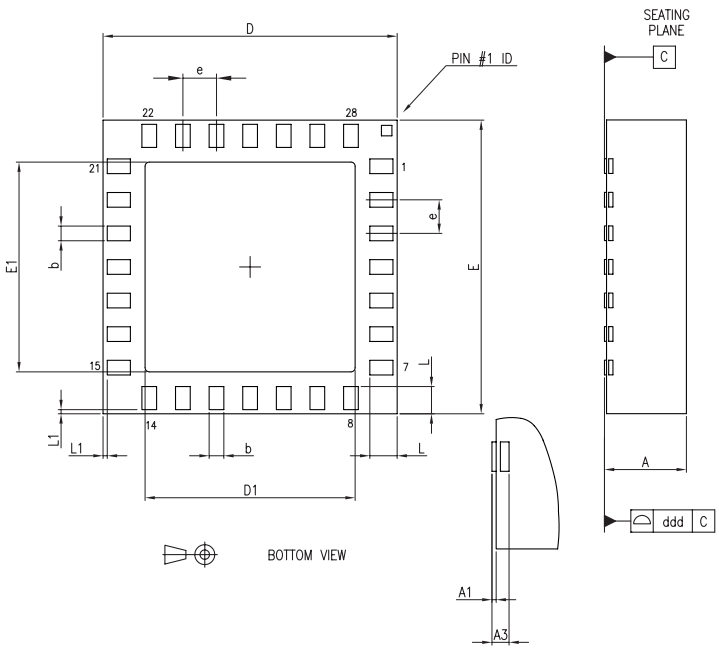
Figure 3. QFPN 7x7x1.8 mm, pitch 0.80, 28 land

DIM.	mm			inch		
	MIN.	TYP.	MAX.	MIN.	TYP.	MAX.
A	1.70	1.80	1.90	0.067	0.071	0.075
A1			0.05			0.002
A3		0.203			0.008	
b	0.30	0.35	0.40	0.012	0.014	0.016
D	6.85	7.00	7.15	0.270	0.275	0.281
D1	4.90	5.00	5.10	0.192	0.197	0.20
E	6.85	7.00	7.15	0.270	0.275	0.281
E1	4.90	5.00	5.10	0.192	0.197	0.20
e		0.80			0.0315	
L	0.45	0.55	0.65	0.018	0..022	0.025
L1			0.10			0.004
ddd			0.08			0.003

OUTLINE AND MECHANICAL DATA



QFPN-28 (7x7x1.8mm)
Quad **F**lat **P**ackage **N**o lead

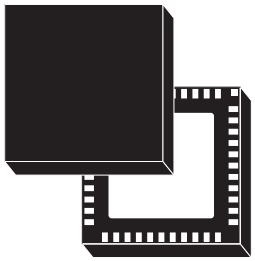


A.2 QFPN 7x7x1.8 mm, pitch 0.50, 44 land

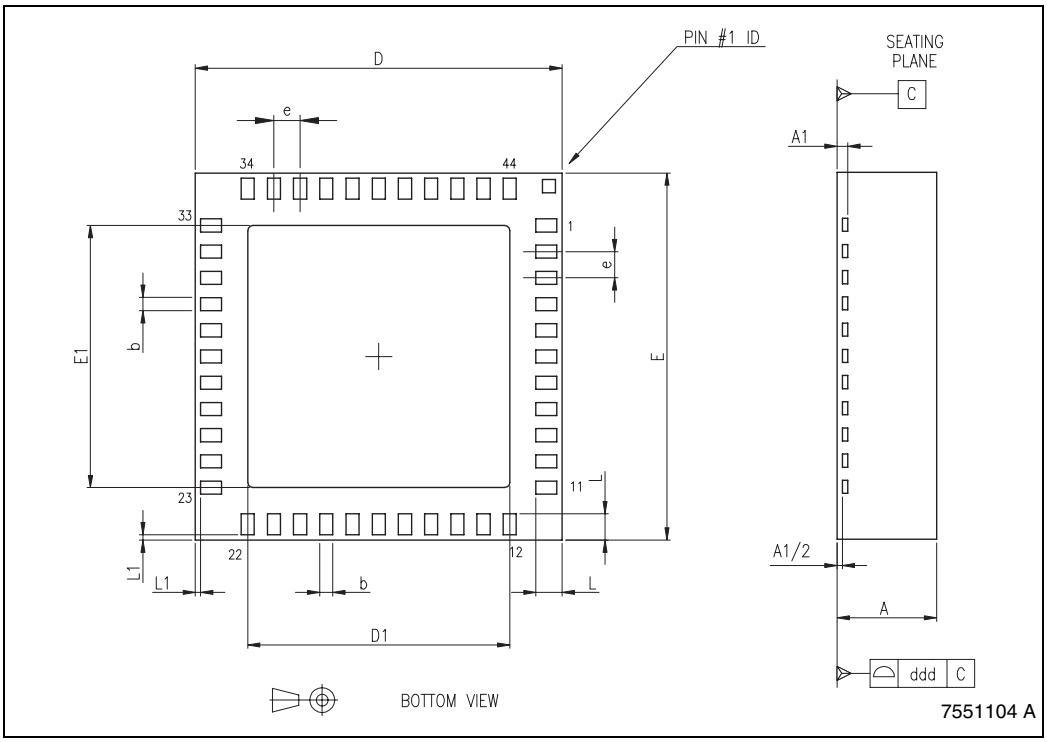
Figure 4. QFPN 7x7x1.8 mm, pitch 0.50, 44 land

DIM.	mm			inch		
	MIN.	TYP.	MAX.	MIN.	TYP.	MAX.
A	1.700	1.800	1.900	0.0669	0.0709	0.0748
A1		0.203			0.008	
b	0.200	0.250	0.300	0.0079	0.0098	0.0118
D	6.900	7.000	7.100	0.2717	0.2756	0.2795
D1	4.900	5.000	5.100	0.1929	0.1969	0.2008
E	6.900	7.000	7.100	0.2717	0.2756	0.2795
E1	4.900	5.000	5.100	0.1929	0.1969	0.2008
e		0.500			0.0197	
L	0.450	0.550	0.650	0.0177	0.0217	0.0256
L1			0.100			0.0039
ddd			0.080			0.0031

OUTLINE AND MECHANICAL DATA



QFPN-44 (7x7x1.8mm)
Quad **F**lat **P**ackage **N**o lead

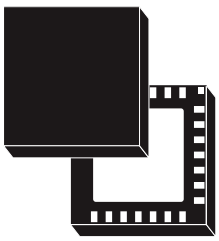


A.3 QFPN 7x7x1.8 mm, pitch 0.80, 28 land

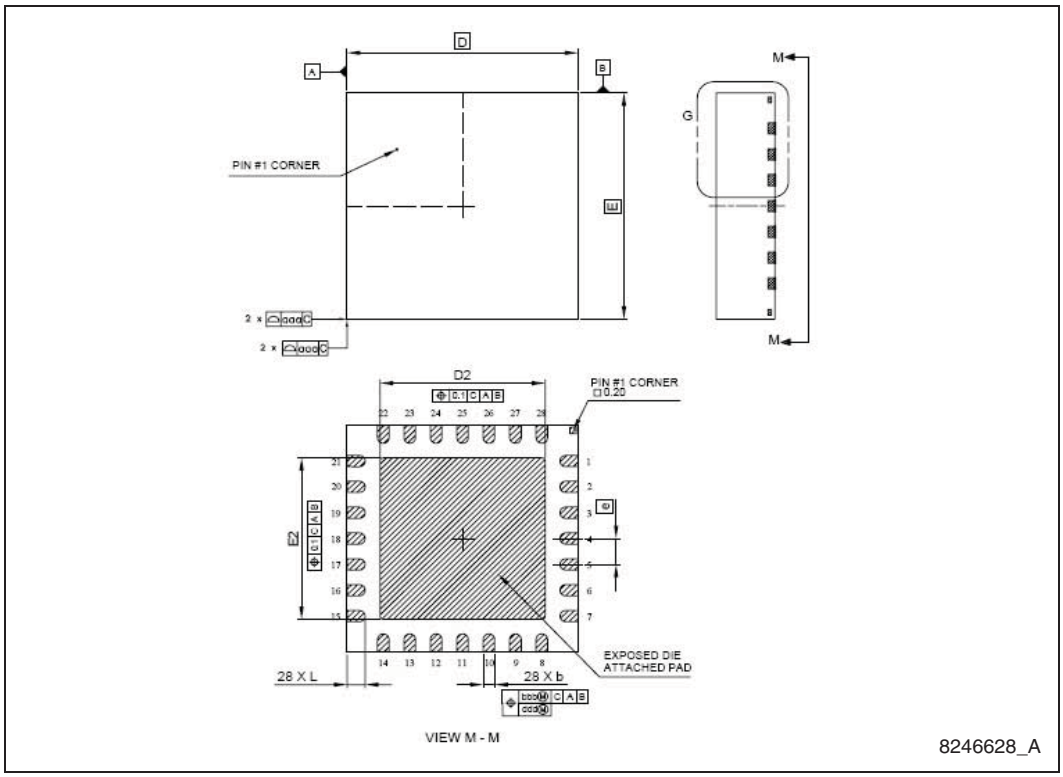
Figure 5. QFPN 7x7x1.8 mm, pitch 0.80, 28 land

Dim.	mm		
	Min.	Typ.	Max.
A	1.75	1.80	7.85
A1	0.00		0.05
A3		0.203 ref	
b	0.30	0.35	0.40
D		7.00 bsc	
D2	4.90	5.00	5.10
E		7.00 bsc	
E2	4.90	5.00	5.10
e		0.80 bsc	
L	0.50	0.55	0.60
aaa		0.10	
eee		0.08	

Outline and mechanical data



QFPN-28 (7x7x1.8 mm)
Quad Flat Package No lead

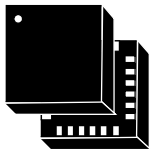


A.4 QFPN 4x4x1.8 mm, pitch 0.50, 24 land

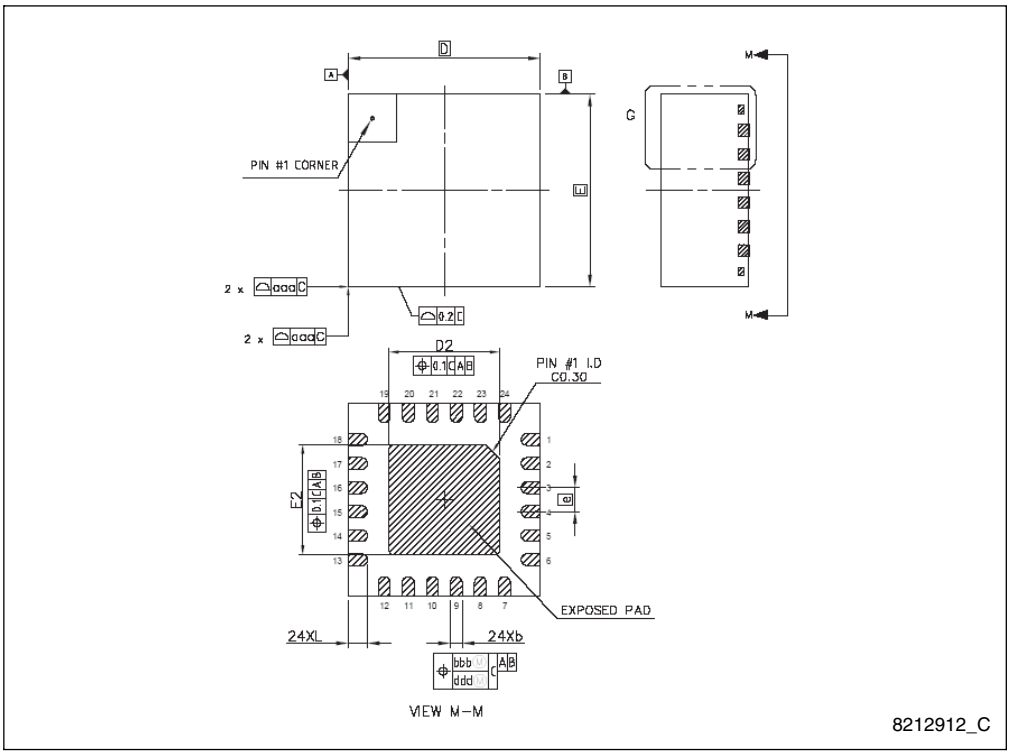
Figure 6. QFPN 4x4x1.8 mm, pitch 0.50, 24 land

Dim.	mm		
	Min.	Typ.	Max.
A	1.75	1.80	7.85
A1	0.00		0.05
A3		0.203 ref	
b	0.20	0.25	0.30
D		4.00 bsc	
D2	2.20	2.30	2.40
E		4.00 bsc	
E2	2.20	2.30	2.40
e		0.50 bsc	
L	0.35	0.40	0.45
aaa		0.10	
eee		0.08	

Outline and mechanical data



QFPN-24 (7x7x1.8 mm)
Quad Flat Package No lead



6 Revision history

Table 1. Document revision history

Date	Revision	Changes
8-Nov-2006	1	Initial release.
01-Apr-2010	2	Updated *Section 1: General guidelines about 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*.