

Keywords: QFN, Side Wettable, SMT, PCB design

APPLICATION NOTE 6382

SMT ASSEMBLY AND PCB DESIGN GUIDELINES FOR MAXIM'S STANDARD WIRE-BONDED QUAD FLATPACK, NO LEADS (QFN) PACKAGES

Abstract: This application note provides the PCB design and SMT assembly guidelines for Maxim's standard wire-bonded QFN packages, flip-chip QFN packages, and side-wettable QFN packages.

Introduction

The quad flatpack, no leads (QFN) package is a leadless, near chip-scale package (CSP), plastics-encapsulated package with copper lead frame as substrate. All QFNs are leadless packages where the connection is made through the leads (terminal pins) and exposed pads on the bottom surface of the package. These leads (terminal pins) and exposed pads can be directly soldered onto the PCB. This application note discusses the guidelines necessary for proper PCB design and SMT assembly when using Maxim's QFN leadless packages.

Package Type Cross Sections

Maxim's QFN package comes in two package singulation formats: punched QFN and sawn QFN. **Figure 1a** and **Figure 1b** show a package cross section of each format.

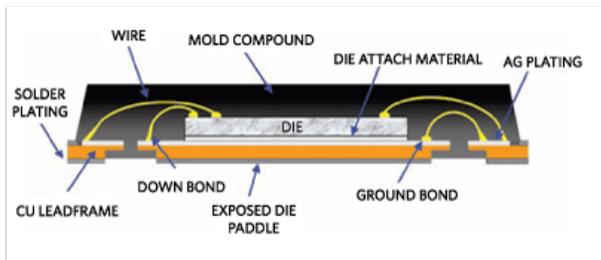


Figure 1a. Punched QFN package cross-section drawing.

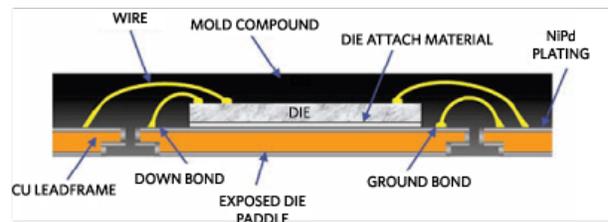


Figure 1b. Sawn QFN package cross-section drawing.

Maxim also offers flip-chip QFN (FC-QFN) packages, where the die is connected to the lead frame using solder ball or Cu pillar interconnects. **Figure 2** shows a sawn FC-QFN package cross section.

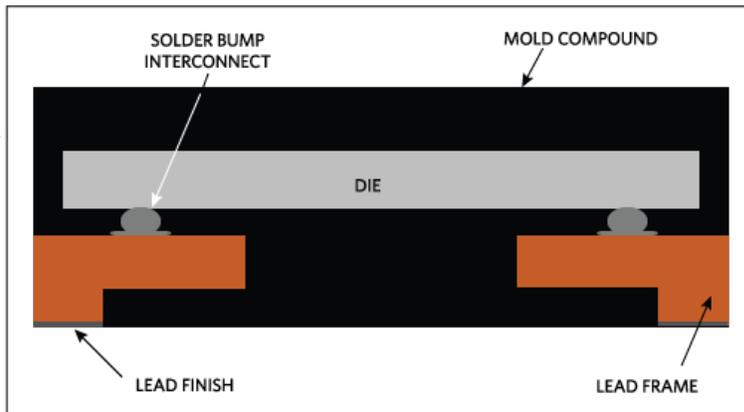


Figure 2. Sawn flip-chip QFN package cross section.

For select end applications, designers may want to visually inspect the solder joint formation on the QFN terminals using an automated optical inspection tool. For these select applications, designers should consider the side-wettable QFN package option.

Maxim's side-wettable QFN package options use geometrical solutions such as half-cut lead frame, half-etch lead frame, or dimpled lead frame to create an externally visible solder joint that can be inspected optically after reflow soldering. **Figure 3a** shows a dimpled side-wettable QFN drawing, and **Figure 3b** shows a half-cut side wettable QFN drawing.

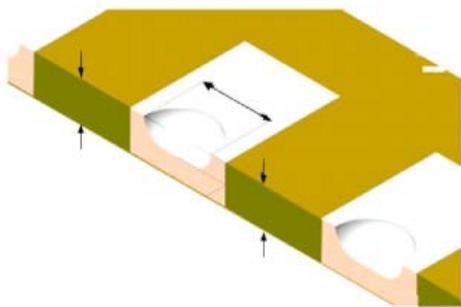


Figure 3a. Dimpled side-wettable QFN package drawing.

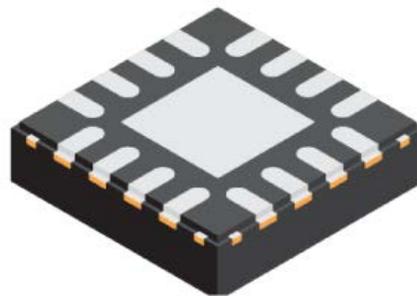


Figure 3b. Half-cut side-wettable QFN package drawing.

Package Surface Mount Considerations

Special considerations are needed to properly design the PCB to mount the package. For enhanced thermal, electrical, and board level performance, the exposed pad on the package needs to be soldered to the board using a corresponding thermal pad on the board.

For proper heat conduction through the board, thermal vias need to be incorporated into the PCB in the thermal pad region. The PCB footprint design must be considered from dimensional tolerances due to package, PCB, and assembly. Many factors can have significant effects on mounting the package on the board and the quality of solder joints. Designers must consider these factors, which include the following:

- Solder paste coverage in the thermal pad region
- Stencil design for peripheral and thermal pad region
- Type of vias, board thickness
- Lead finish on the package, surface finish on the board
- Type of solder paste and reflow profile

This application note provides designers with the guidelines for board design considering these factors. It should be emphasized that this document is strictly a guideline to help the user in developing the proper motherboard design and surface mount process. Actual studies as well as development effort may also be needed to optimize the process as per the user's surface-mount practices and requirements.

PCB Design Guidelines

PCB Pad Design

A well-designed and manufactured printed circuit board (PCB) is required for optimum manufacturing yields and product performance.

Two types of land patterns are used for surface-mount devices:

- a. Solder Mask Defined (SMD) pads have solder mask openings that are smaller than metal pads.
- b. Non-Solder Mask Defined (NSMD) pads have solder mask openings that are larger than the metal pads.

Maxim recommends the use of NSMD pads because they provide a larger metal area for the solder to anchor to the edges of metal pads on the board for improved solder joint reliability.

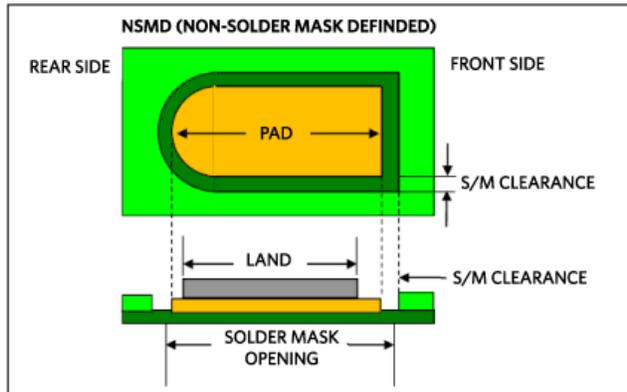


Figure 4. Cross section drawing of NSMD. Note: There should not be any mixture of SMD and NSMD pads within the same QFN footprint. Maxim recommends the inclusion of fiducial marks in proximity to the QFN package to facilitate component placement.

PCB Land Pattern Dimensions

Maxim's land pattern drawings for QFN packages can be downloaded from our website at: www.maximintegrated.com/packages. IPC land pattern calculator (IPC-7351) can be downloaded from the IPC website at: www.ipc.org.

Maxim does not recommend designing vias, untented with solder mask, close to the package lead terminal and underneath the corners of the package where tie bars terminate. Refer to IPC-7351 for "Land Pattern to Via Relationship" design recommendation.

PCB Pad Pattern for Lead Terminals

As a guideline, lead finger length dimension (Y) should be designed using the following considerations:

- 0.20mm to 0.5mm longer (extending outwards) than package terminal length for good solder filleting (for standard QFN packages)
- 0.25mm to 0.5mm longer (extending outwards) than package terminal length for good solder filleting (for side-wettable QFN packages)
- 0.05 mm extended towards center line of package

For lead finger width dimension (X) should be designed using the following considerations:

- Wider by 0.05mm than package terminal width (i.e., 0.025mm per side), for lead pitch greater than 0.65mm
- Match terminal width to minimize risk of solder bridging, for lead pitch 0.65mm and below

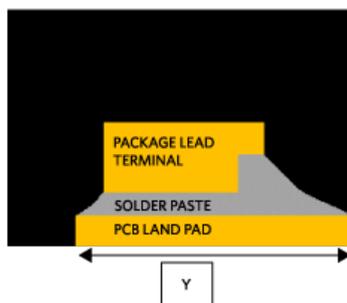


Figure 5a. Lead terminal length.

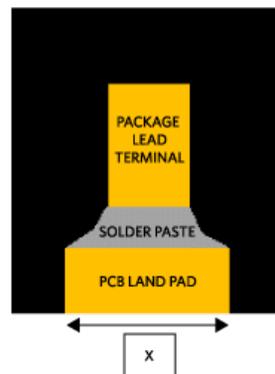


Figure 5b. Lead terminal width.

PCB Pad Surface Finish

The selection of an appropriate PCB pad surface finish is critical to ensuring optimum manufacturing of the final board assembly. **Table 1** compares several popular surface finishes.

Table 1. Surface Finish Options

Finish Name	Description	Recommendation	Comments
Copper OSP	Organic solderability preservative (OSP)	Acceptable	—
ENIG	Electroless nickel immersion gold	Acceptable	—
Silver Immersion	Thin layer silver electroless plated	Acceptable	Silver immersion thickness must be controlled (0.150mm to 0.625mm) to minimize micro voids at the solder joint
Gold Immersion	Thin layer gold electroless plated	Acceptable	—
HASL	Hot air solder leveling	Acceptable	Process must be controlled to provide solder coverage of small mask openings

Stencil Design Guidelines

The stencil thickness and pattern geometry determine the precise volume of solder paste deposited onto the device land pattern. Stencil alignment accuracy and consistent solder volume transfer is critical for uniform solder reflow. Stencils are usually made of stainless steel.

Design Recommendation:

- Stencil Thickness
 - 5 mils: Stencil aperture opening of 1 mil/side smaller than the pad size.
 - 4 mils: Stencil aperture opening should be 1:1 to PCB pad sizes.
- Stencil Fabrication
 - Laser-cut with electropolish for better release than the regular laser-cut stencil.
- Stencil Aperture
 - Tolerances must be tightly controlled, specifically for 0.5mm and 0.4mm pitch packages, as these tolerances can effectively reduce aperture size
 - Walls of the apertures should be as smooth, with rounded corners, and a trapezoidal cross section (**Figure 6**) enhances the release of solder paste from the aperture.
- QFN stencil aperture must meet the industry standard area ratio of > 0.66.



Figure 6. Stencil cross section.

Stencil Design for Thermal Pads (Exposed Pads)

Recommendation:

- Smaller multiple opening should be used instead of a single big opening to minimize voiding.
- 50% to 80% solder paste coverage
- Rounded corners to minimize solder paste clogging
- Positive taper with bottom opening larger than the top

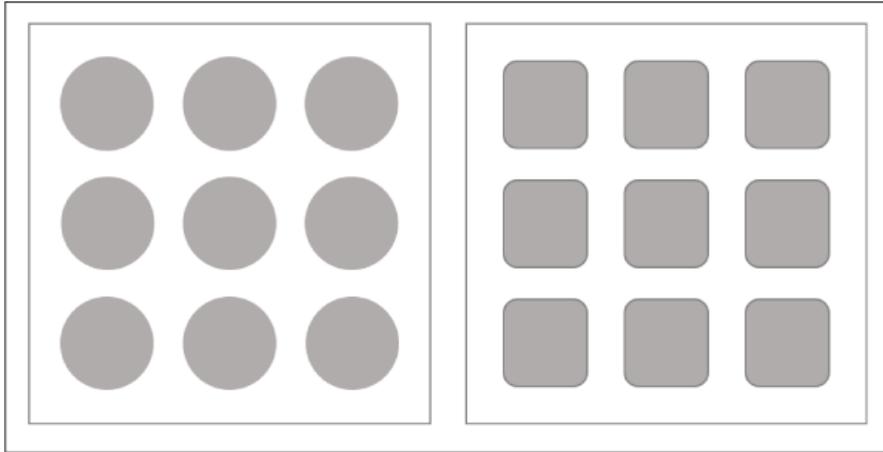


Figure 7. Stencil design for exposed pads.

Solder Paste

A low-residue, no-clean solder paste is commonly used in mounting QFNs. Water-soluble flux materials are not recommended as there is minimal standoff (distance of package to PCB) and it will, therefore, be very difficult to clean flux trapped underneath the package.

A Type 3 (or finer) solder paste is recommended for 0.5mm pitch printing. Nitrogen purge is recommended during solder reflow.

Recommended lead(Pb)-free solder paste: SAC305 (Sn-Ag-Cu) alloys

Solder Paste Printing

An automatic or manual stencil/screen printer can be used to distribute the solder paste onto the PCB lands.

A design of experiments (DOE) should always be used to establish optimum printing parameters. Most assemblers find that the following parameter ranges serve as good starting points:

- Print head speed: 1–2 inches/second
- Squeegee pressure: 0.75–1.5 pound per inch of squeegee
- Under-stencil wiping: Every 3 boards
- Temperature: 23°C to 28°C
- Humidity: 30% to 60% RH

A stainless steel squeegee should be used. Multiple printing of solder paste should be avoided for fine pitch devices, as it may cause smearing of the solder paste.

Package Placement and Alignment

Conventional placement systems can be employed using either the QFN outline or the position of the leads as a placement guide. Placement guide using position of the leads tends to be more accurate but slower, and requires a complex vision processing system. Package outline placement method runs faster but is less accurate. The contract PCB assembler can determine the most acceptable method to be employed for this process. The QFN package is expected to a good self-centering ability similar to a BGA package.

Reflow Profile

Maxim QFNs are compatible with all industry-standard solder reflow processes. It is important that profiles be checked on all new board designs. Additionally, if there are tall components mixed on the board, the profile must be checked at different locations on the board. Component temperatures may vary due to surrounding components, locations of parts on the PCB, and package densities.

The reflow profile guidelines are based on the temperature at the actual lead to PCB land pad solder-joint location. The actual temperature of the solder joint is often different than the temperature settings in the reflow system. It is recommended that reflow-specific profiles be checked using thermocouples at the actual solder-joint locations and following the paste supplier's recommendations.

Nitrogen reflow is recommended to improve solderability and to reduce defects such as solder balls. **Figure 8** is one reflow example for Pb-free temperature profile per JEDEC J-STD-020.

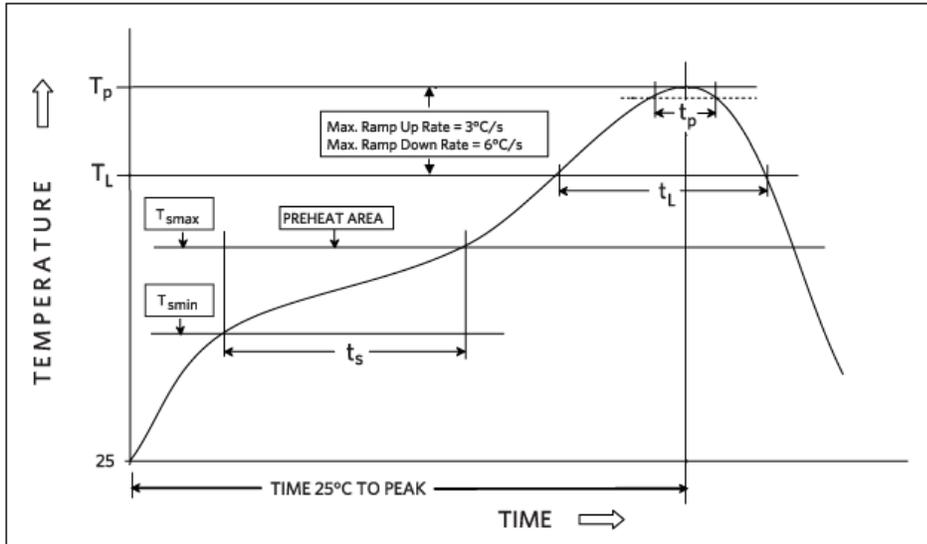


Figure 8. JEDEC lead-free reflow profile.

Table 2. Recommended Pb-free Reflow Profile

Lead (Pb) free reflow profile feature	Description	Pb-Free assembly recommendation
T_{Smin}	Min Soak Temperature	150°C
T_{Smax}	Max Soak Temperature	200°C
t_s	Time Between T_{SMIN} and T_{SMAX}	60–120 Seconds
T_L	Liquidous Temperature	217°C
t_L	Time Above T_L	60-150 seconds
T_P	Peak Package Body Temperature	260°C
t_p	Time Within 5°C of T_P	30 Seconds
T_L to T_P	Ramp-Up Rate	3°C/Seconds (max)
T_P to T_L	Ramp-Down Rate	6°C/Seconds (max)
—	Time 25°C to Peak Temperature	8 Minutes (max)

Solder Joint Inspection Guideline

Post-reflow inspection of QFNs on PCBs is typically accomplished by using transmission-type X-ray equipment. X-ray can be used for reflow process monitoring and as a failure analysis tool. A 2D X-ray system with oblique view at highest magnification (OVHM) is recommended as it can detect solder bridges, opens, and voids.

Solder Joint Visual Inspection Recommendations

Standard QFN packages

Standard QFN packages do not have any externally visible features that allow one to check if the package has been successfully soldered to the board. As in any leaded package technology, the edges of the leads will have exposed bare copper. Bare copper is not sufficient to ensure side wetting.

Lack of solder wetting in this area should not be considered as a criterion for visual inspection rejection. Maxim does not guarantee formation on solder fillet (toe fillet) as the sides of these lead terminals will have an exposed copper post-package singulation.

Side-Wettable QFN Packages

The specifically introduced geometric features in a side-wettable package (half-cut/half-etch/dimple) allow for a visible solder fillet to form on the side of all peripheral pins (**Figure 9**). The fillet can be visually inspected if the pins are soldered successfully to the PCB pads. The full height of the geometrical structure needs to be covered by solder fillet.

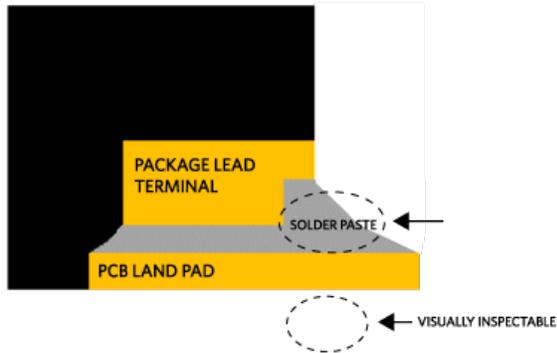


Figure 9. Half-cut side-wettable and dimpled side-wettable QFN.

PCB Cleaning

If a low residue, no-clean solder paste is used, PCB cleaning is not required. However, with different types of no-clean solder pastes available, Maxim recommends application-specific evaluations be performed to identify if any remaining residue needs to be removed from the boards.

Rework Guidelines

An appropriate rework station should be used for any rework on the parts.

Component Removal

Prior to any rework, it is strongly recommended that the PCB assembly be baked for at least 4 hours at 125°C to remove any residual moisture before component removal.

During the removal process, it is recommended that the board be heated from the bottom side using convective heaters, and hot gas or air should be used on the top side of the component. Special nozzles should be used to direct the heating to the component only and minimize heating of adjacent components.

Maximum part temperature should be above the liquidus temperature of 217°C, but should not exceed 260°C. Once the joints have reflowed, the vacuum lift-off should be automatically engaged during the transition from reflow to cool down. The vacuum pressure should be kept below 15in of Hg to ensure the component is not lifted out if all joints have not been reflowed.

Site Redress

Following component removal, the site must be cleaned properly without damaging the pads. Once the residual solder has been removed, the lands should be cleaned with a solvent. The solvent is usually specific to the type of paste used in the original assembly. Paste manufacturer's recommendations should be followed.

Solder Paste Printing

It is recommended that a miniature stencil be used to print solder paste on the PCB surface at the component site.

Follow stencil thickness, stencil design, solder recommendations, and screen-printing guidelines, as provided for original PCB assembly. Align stencil with the pads under 50X magnification to inspect the site before replacing with a new component (part).

Component Placement

QFN packages have superior self-centering ability similar to BGA packages. Because the leads are on the package's underside, a split-beam optical system should be used to align the component on the board. Doing so forms an image of the leads overlaid on the mating footprint and aids in proper alignment. A placement machine having a 0.05mm placement accuracy should be used.

Reflow

The same reflow profile developed during initial component attachment should be used to attach the new component followed by the post reflow inspection step.

More Information

[Technical Support ›](#)

[Samples ›](#)

More Information

For Technical Support: <https://www.maximintegrated.com/en/support>

For Samples: <https://www.maximintegrated.com/en/samples>

Other Questions and Comments: <https://www.maximintegrated.com/en/contact>

Application Note 6382: <https://www.maximintegrated.com/en/an6382>

APPLICATION NOTE 6382, AN6382, AN 6382, APP6382, Appnote6382, Appnote 6382

© 2014 Maxim Integrated Products, Inc.

The content on this webpage is protected by copyright laws of the United States and of foreign countries. For requests to copy this content, [contact us](#).

Additional Legal Notices: <https://www.maximintegrated.com/en/legal>