Introduction

This document is written to serve as a guideline to help the user in developing the proper PCB design and surface mount process. Development effort and actual studies may still be needed to optimize the process in order to meet individual specific requirements.

Littelfuse's Q2L Quad Flatpak - No Lead Package (QFN) is a near Chip Scale Package (CSP) that uses conventional copper leadframe technology. Mechanical, thermal, and electrical connections are made through the exposed lands on the bottom of the package. This construction enables the use of a stable thermal path and electrical ground through a robust mechanical solder connection to the PCB. Its miniature dimension and low profile (1.0 mm height on PCB) requires less board area which increases board density compared to traditional leaded surface mount packages.

The QFN packaged product allows for a decreased package size without sacrificing performance. This next generation package platform is ideal for high density circuits and for handheld electronic products.

Package Design

The QFN packages are designed in MAP (Matrix Array Package) leadframe format and individually singulated by using a saw process (see Fig. 1). It can provide customized body size and customized land format design for specific design needs and applications.

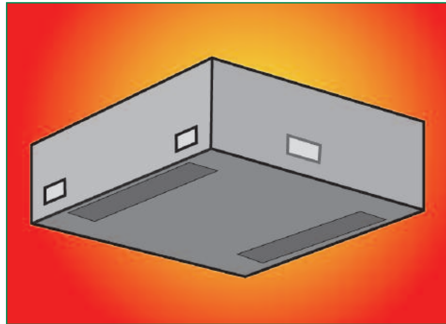

Figure 1. 3x3 QFN

PCB Design Guidelines

There are two different types of PCB pad configurations commonly used for surface mount leadless QFN packages:

1. *Non Solder Mask Defined Style (NSMD)*
2. *Solder Mask Defined Style (SMD)*

The NSMD contact pads have the solder mask pulled away from the solderable metallization, while the SMD pads have the solder mask over the edge of the metallization.

With the SMD pads, the solder mask restricts the flow of solder paste on the top of metallization that prevents the solder from flowing along the side of the metal pad (see Fig. 2A).

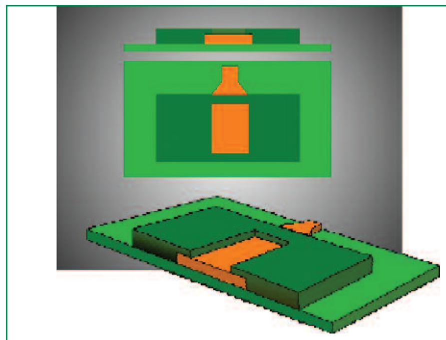

Figure 2A. SMD pad

This is different from the NSMD where the solder will flow around both the top and sides of the metallization (see Fig. 2B).

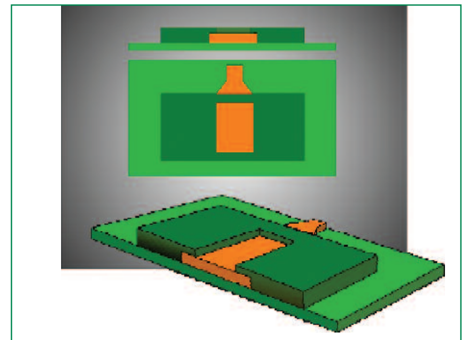

Figure 2B. NSMD pad

NSMD pads are recommended over SMD pads since the copper etching process is capable of a tighter tolerance than the solder masking process. Additionally, NSMD pads with solder mask opening larger than the metal pad size also improves the reliability of the solder joint as solder is allowed to wrap around the sides of the metal pads.

NSMD Pad Design Considerations

The solder mask should be located at least +/-3 mils (0.076mm) away from the edge of the solderable pad when dimensionally possible. This allows for solder mask registration tolerances and ensures the solder is not inhibited by the mask as it reflows along the side of the metal pads.

PCB Pad Pattern

The dimensions of the PCBs solderable pads should match those of the pads on the package (see Fig. 3A & 3B).

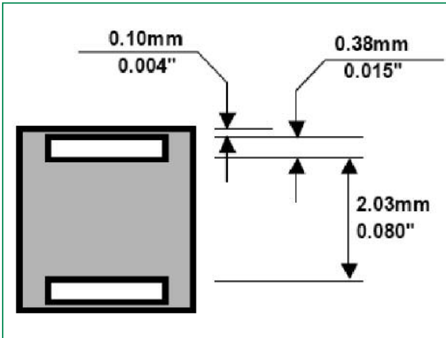

Figure 3A. OFN Footprint

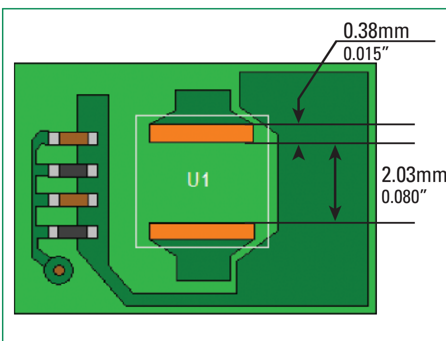

Figure 3B. PCB Layout

The 1:1 ratio between the package pad configuration and the PCB pad configuration is desired for optimal placement accuracy and reliability.

PCB Surface Finishes

The key factor in selecting an acceptable surface finish is to ensure that the land pads have a uniform coating. Irregular surface plating, uneven solder paste thickness, and crowning of the solder plating can reduce the overall surface mount yields.

There are two common surface finishes which are used for PCB surface mount devices. The first consists of an organic solderability preservative (OSP) coating over a copper plated pad. The organic coating assists in reducing oxidation in order to preserve the copper metallization for soldering.

The second recommended surface finish consists of plated electroless nickel over the copper pad followed by immersion gold.

Of all the coating and plating options available, Ni/Au is the most versatile, providing the gold thickness is controlled. Typically, 5um nickel, and between 0.05um and 0.1um gold are needed to prevent gold embrittlement which may affect the reliability of the solder joint.

Board Mounting Considerations

Solder Paste

The quality of the paste print is an important factor in producing high-yield assemblies. The paste is the vehicle that provides the flux and solder alloy necessary for a reliable and repeatable assembly process.

A low-residue, "no-clean" type 3 solder paste should be used in mounting QFNs. Typically, the choice of solder paste determines the profile and reflow parameters. Most paste manufacturers provide a suggested thermal profile for their products and must be referenced prior to manufacturing.

Solder Stencil

The stencil thickness, as well as etched pattern geometry, determines the precise volume of solder paste deposited onto the device land pattern. Stencil alignment accuracy and consistent solder volume transfer are critical for uniform reflow-solder processing.

Stencils are usually made of brass or stainless steel, with stainless steel being more durable. Apertures must be trapezoidal to ensure uniform release of the solder paste and to reduce smearing (see Fig. 4).

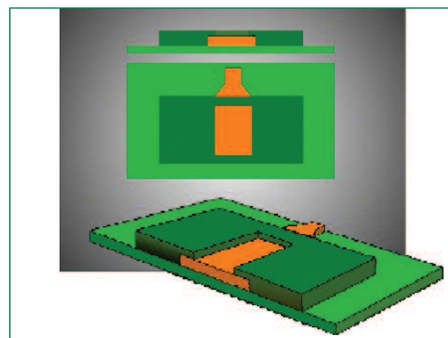

Figure 4. Solder Stencil Profile

The solder joint thickness of QFN lead fingers must be 0.050mm to 0.075mm. Thickness of the stencils is usually in the 0.100mm to 0.150mm.

The blade angle and speed must be fine-tuned to ensure even paste transfer. An inspection of stenciled board is recommended before placing the parts, because proper stencil application is the most important factor with regards to reflow yields further on in the process. As a guide, stencil thickness of 0.125mm for QFN components is recommended.

Lead Finger Stencil Design

The optimum and reliable solder joints on the perimeter pads should have 50 to 75 microns (2 to 3 mils) standoff height.

The first step in achieving good standoff is the solder paste stencil design for perimeter pads. The stencil aperture opening should be designed so that the maximum paste release is achieved. This is typically achieved by considering the following two ratios:

1. *Area Ratio = Area of Aperture Opening / Area of Aperture Wall*
2. *Aspect Ratio = Aperture Width / Stencil Thickness*

For rectangular aperture openings, as required for this package, these ratios are:

Area Ratio $= LW/2T(L+W)$

Aspect ratio $= W/T$

where L & W are aperture length and width, T is stencil thickness.

For optimum paste release the area and aspect ratios should be greater than 0.66 and 1.5 respectively.

It is recommended that stencil aperture should be 1:1 to PCB pad sizes as both area and aspect ratio targets are easily achieved by this aperture.

The stencil should be laser cut and electropolished. The polishing helps in smoothing the stencil walls resulting in better paste release.

It is also recommended that the stencil aperture tolerances should be tightly controlled, especially for 0.4 and 0.5mm pitch devices, as these tolerances can effectively reduce the aperture size.

Package Placement and Alignment

The pick and place accuracy governs the package placement and rotational (theta) alignment. This is equipment/process dependent. Slightly misaligned pads (less than 50% off the pad center) automatically self-align during reflow due to surface tension of the solder (see Fig.5).

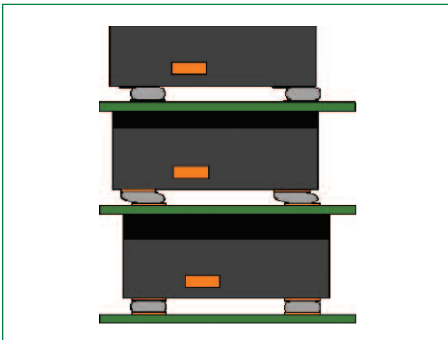

Figure 5. Package Self-Alignment at Reflow

Grossly misaligned packages (greater than 50% off the pad center) must be removed prior to reflow, as they may develop electrical shorts resulting from solder bridges, if they are subjected to reflow.

Solder Reflow

There are no special requirements when reflowing QFN components. As with all SMT components, it is important that profiles be checked on all new board designs.

In addition, if there are multiple packages on the board, the profile must be checked at different locations on the PCB. Component temperatures may vary because of surrounding components, location of the device on the board, and package densities.

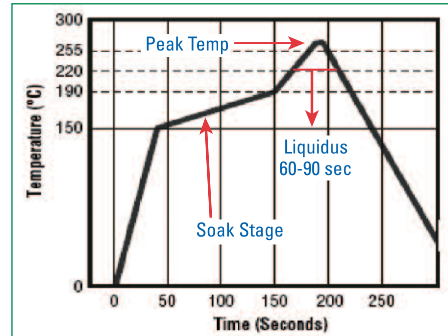

Figure 6. Typical Profile for Lead-Free Solder

Figure 6 is an example of a standard reflow profile for a lead-free solder paste. The paste manufacturer will determine the exact profile, since the chemistry and viscosity of the flux may vary.

In general, the temperature of the part should not be raised more than 2° C/sec during the initial stages of the reflow profile. The soak zone then occurs when the part is approximately 150° C up to 190° C and should last for 90 to 120 seconds. Extending the time in the soak zone will typically reduce the risk of voiding within the solder. The temperature is then raised and will be above the liquidus of the solder for 60 to 90 seconds depending on the mass of the board. The peak temperature of the profile should be 30° C to 40° C above the melting point of the solder. However, the temperature during reflow should not exceed the maximum temperature the package is qualified for according to Moisture Sensitivity Level Testing. Finally, Ramp Down Rate from peak temperature to room temperature should not exceed 4° C/sec.

5.6 PCB Cleaning

If a low-residue, "No Clean" solder paste is used, PCB cleaning is not required, and has little effect on QFNs. "No Clean" solder paste simply means that there are no harmful residues left on the board that could cause corrosion or damage to the components if left on the board.

However, some types of "No Clean" solder paste may not be satisfyingly free from contamination on the final board, so it is recommended that an experiment should be conducted to examine whether eventually the flux residues are required to be removed.

Solder Joint Inspection

Inspection of QFNs on a PCB is commonly accomplished with the use of an X-ray inspection system.

In most cases, 100% inspection is not performed. Typically, X -ray inspection is used to establish process parameters and then to monitor the production equipment and process. The X-ray inspection system can detect bridging, shorts, opens, and solder voids.

In addition to searching for defects, the mounted device should be rotated on its side to inspect the sides of the solder joints. These joints should have enough solder volume with the proper stand-of height so than an "hour glass" shaped connection is not formed (see Fig. 7).

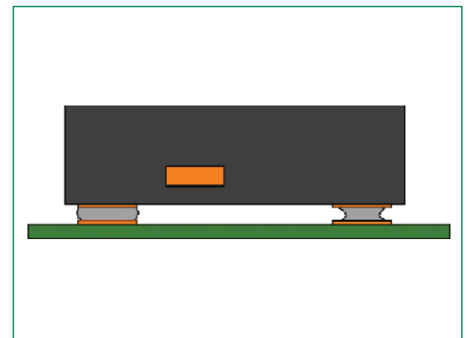

Figure 7. Desirable vs. Hour Glass Solder Joint

Rework Methodology

Due to the fact that the QFN is a leadless device, the entire package must be removed from the PCB if there is an issue with the solder joints. It is important to minimize the chance of overheating neighboring devices during the removal of the package since the devices are typically in close proximity with each other.

Standard SMT rework systems are recommended for this procedure since the airflow and temperature gradients can be carefully controlled. It is also recommended that the PCB be placed in an oven at 125° C for 4 to 8 hours prior to heating the parts to remove excess moisture from the packages.

Component Removal

The gas nozzle used during this process surrounds the device and seals against the board. The QFN is heated from the topside with hot gas while residual heat is exhausted up and away from adjacent components. The anti-crushing feature in the nozzle prevents excessive topside force from being applied to the QFN.

The entire assembly is also heated from the bottom side with an under-board heater to help prevent warpage.

Preheating the board to a fixed temperature before the component is heated also helps to ensure process repeatability.

Once the reflow process is complete, the nozzle vacuum cup is automatically activated and the component is slowly lifted off the pads. The vacuum cup in the nozzle is designed to disengage if the component has not fully reflowed for any reason. This prevents the potential for lifting pads.

Site Redress

Once the QFN has been removed, the residual solder that remains on the pads must be removed. The QFN PCB site is very fragile because of its small pad sizes. To avoid damaging the pads or solder mask, the site redress process must be performed very carefully. "No Clean" flux is applied to the site after component removal. Using a temperature-controlled soldering iron fitted with a small flat blade, gently apply solder braid that has been presoaked in flux over the PCB pads.

Residual flux is removed from the site with alcohol and a lint-free swab. This site is then inspected prior to the replacement process.

Component Replacement and Reflow

Due to the small pad configurations of the QFN, and since the pads are on the underside of the package, a manual pick and place procedure without the aid of magnification is not recommended. A dual image optical system where the underside of the package can be aligned to the PC board should be used instead.

Reflowing the component onto the board can be accomplished by either passing the board through the original reflow profile, or by selectively heating the QFN with the same process that was used to remove it. The benefit of subjecting the entire board to a second reflow is that the QFNs will be mounted consistently and by a profile that was already defined. The disadvantage is that all of the other devices mounted with the same solder type will be reflowed for a second time.

If subjecting all of the parts to a second reflow is either a concern or unacceptable for a specific application, then the localized reflow option would be the recommended procedure.

World Headquarters

Littelfuse, Inc.
800 E. Northwest Highway
Des Plaines, IL 60016, USA
www.littelfuse.com

FORM NO. EC642
© 2005 Littelfuse, Inc. Printed in U.S.A. August 2005

Specifications, descriptions and illustrative material in this literature are as accurate as known at time of publication, but are subject to change without notice.