COLUMN

A SHORT SCOOP

Stencils for QFNs

by Rachel Short
PHOTOSTENCIL LLC

QFNs (quad flatpacks, no leads) and DFNs (dual flatpacks, no leads) are becoming more popular component packages, in part because they are ideal for small, hand-held devices. As stencil manufacturers, our thoughts immediately turn to the question, how do we design and manufacture a stencil for these devices? What materials should be used? What types of coatings? How should the apertures be designed to get the best solder paste release? How can we accommodate the reduced footprint and keep-out areas? QFNs present many challenges to the assembly printing process, but the short scoop is that with proper stencil design, correct stencil technology selection (laser, electroform, and nano-coat), and consideration for the effects of PCB soldermask layout choices, these challenges can be overcome.

QFNs have a very small form factor—they are typically .85mm thick with a body size from 3mm up to 12mm. This allows for smaller and lighter packages. Most QFNs have a metal pad on the underside for grounding and heat conduction. The leads and ground plane conductor are flat and in the same plane on the bottom of the package. DFNs have a similar center metal pad, but have leads on only two sides. This ground plane provides excellent electrical conductivity and offers better heat sink thermal properties when compared to other SMT packages. All these advantages make QFNs a good choice for high density electronics. So let's discuss the problems that arise when using these components and what you can do to handle the printing challenges associated with fabrication.

QFNs, by their nature, create printing challenges. The package can float during reflow if there is too much solder. Aperture size is a problem because the apertures are short and narrow posing difficulties for paste transfer. Solder mask configurations chosen during the board design phase influence the stencil design. Board repair can also be difficult to accomplish.

The leadless QFN package sits almost flush against the board, but will float during reflow and can misregister if there is too much solder applied in critical areas of the component footprint. If the larger center thermal pad is printed 1:1, the reflowed solder can float the component due to the solder surface tension overcoming the weight of the package. The QFN/DFN stand-off height is a function of center and perimeter joint cohesion force balance. Surface tension during reflow causes the solder to dome up at the geometric center of the pad. The difference in the size of the larger center pad causes a large center dome to lift above the perimeter lead attachment pads. This height differential can cause the leads on the QFN and the pads on the PCB to misregister as the component floats on the dome formed by the center pad. Fortunately, QFN float can be controlled by reducing the amount of solder paste printed on the ground plane. Typically, a 50 to 60% reduction will

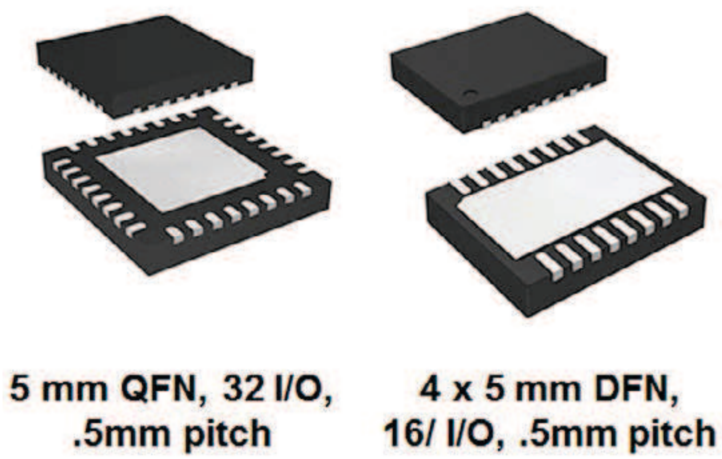

Figure 1: Examples of QFN/DFN.

solve the QFN float problem, but the aperture reduction must be done judiciously. We recommend that a window pane aperture be used for most cases. This allows the solder paste volatiles to easily escape during reflow without moving the QFN device.

Why are area ratio and paste transfer critical issues? The side walls of the aperture hold the solder paste inside the aperture while the area under the aperture (pad on PCB) pulls the solder paste out of the aperture and away from the aperture walls. Compared to the pad area beneath the aperture, the larger the wall area, the more difficult it is for the paste to release from the aperture walls. For QFN and DFN packages, the required apertures have widths as low as 0.175mm and aperture lengths as low as 0.4mm so the area ratio is low, which can result in poor paste transfer.

IPC 7525B Stencil Design Guidelines for Area Ratio Standards recommends the following types of stencils for specific area ratios (AR):

AR> .90:	Laser-cut, high precision Chem-etch, Chem-etch
AR .66 to .90:	Laser-cut, high precision Chem-etch
AR .5 to .66:	Electroform stencils

Roughness of the aperture wall only makes matters worse giving the paste something to bite onto and preventing good paste release. Aperture wall smoothness is a key issue for proper paste transfer, particularly when the area ratio is small. For area ratios below .66 electroform stencils or nano-coated stencils are normally recommended.

Nano-coating on the aperture walls as well as on the bottom side (PCB side) of the stencil is a good solution for single level stencils without a step. Nano-coatings have a property called fluxophobicity. It is the stencil's ability to resist the spread of flux on its surface and is measured in the form of the flux contact angle. This is the angle that the flux forms when a drop is placed on the surface of the stencil. Nano-coating not only increases the paste's ability to release from the apertures, but also to resist spreading on the bottom side of the stencil when the paste is extruded into a cavity created by the NSMD-Window. Nano-coating eliminates the need for frequent under board wiping and reduces the occurrence of pad to pad bridging.

> ***Nano-coatings have a property called fluxophobicity. It is the stencil's ability to resist the spread of flux on its surface and is measured in the form of the flux contact angle.***

The solder mask windowing technique employed on the PCB can be a substantial determining factor for achieving the best result. There are three types of solder mask designs used for QFN and DFN packages:

a) SMD, where the pad opening on the board is defined by the solder mask

b) NSMD, where the pad itself defines the boundary of the pad and the solder mask is pulled back off the pad (typically .05 to .075mm per side)

c) NSMD-Window. In this last case there is no solder mask between pads so bridging between pads is more likely than with solder mask between pads.

Table 1 (a, b, and c) shows stencil design guidelines for these three solder mask cases. It shows package size, lead pitch, the number of I/Os, package lead dimensions, the recommended PCB pad dimensions, stencil aperture dimension, stencil thickness, and the resulting area ratio. For NSMD the stencil aperture for the pin connections should be 1:1 with the PCB pad dimension.

The last problem to be considered for the QFN and DFN package is unit repair. The first step to repair a defective QFN device is to remove the defective device from the PCB and clean the excess solder from the PCB pads. Solder paste is then printed either on the PCB or on

Stencil Aperture Design for SMD Mask										
Package	Pitch	I/O	Pkg Lead Width	Pkg Lead Length	PCB	PCB	Aperture	Aperture	Stencil Thickness	Area Ratio
3mm	.5mm	12	.23mm	.55mm	.23mm	.75mm	.18mm	.70mm	.125mm	0.57
4mm	.5mm	20	.25mm	.40mm	.25mm	.60mm	.20mm	.55mm	.125mm	0.59
7mm	.5mm	44	.25mm	.55mm	.25mm	.75mm	.20mm	.70mm	.125mm	0.62
10mm	.5mm	72	.23mm	.40mm	.23mm	.60mm	.18mm	.55mm	.125mm	0.54
12mm	.5mm	80	.25mm	.55mm	.25mm	.75mm	.20mm	.70mm	.125mm	0.62

Table 1a: Typically there is a reduction to the pad layout on the PCB. Area ratios are usually <.66, thus paste transfer is an issue. An electroformed stencil is normally recommended.

Stencil Aperture Design for NSMD Window Mask										
Package	Pitch	I/O	Pkg Lead Width	Pkg Lead Length	PCB	PCB	Aperture	Aperture	Stencil Thickness	Area Ratio
4mm	.4mm	32	.175mm	.45mm	.175mm	.610mm	.175mm	.560mm	.125mm	0.53
4mm	.4mm	32	.175mm	.45mm	.175mm	.610mm	.175mm	.560mm	.100mm	0.67

Table 1b: Typically 1-1 with the pad width layout on the PCB. Most .4 mm QFNs are NSMD window and even though the apertures are 1-1 with the pad width, area ratios are <.66. Stencils with smooth aperture walls such as electroformed stencils are recommended.

Stencil Aperture Design for NSMD Mask										
Package	Pitch	I/O	Pkg Lead Width	Pkg Lead Length	PCB	PCB	Aperture	Aperture	Stencil Thickness	Area Ratio
3mm	.5mm	12	.23mm	.55mm	.23mm	.75mm	.23mm	.75mm	.125mm	0.7
4mm	.5mm	20	.25mm	.40mm	.25mm	.60mm	.25mm	.60mm	.125mm	0.71
7mm	.5mm	44	.25mm	.55mm	.25mm	.75mm	.25mm	.75mm	.125mm	0.75
10mm	.5mm	72	.23mm	.40mm	.23mm	.60mm	.23mm	.60mm	.125mm	0.71
12mm	.5mm	80	.25mm	.55mm	.25mm	.75mm	.25mm	.75mm	.125mm	0.75

Table 1c: Typically 1-1 with the pad layout on the PCB. Area ratios are typically >.70, thus paste transfer is not an issue.

the bottom of the QFN prior to placing the QFN on the PCB and locally heating to reflow the solder paste and solder the device in place. Mini stencils are normally used to print paste on the PCB. This can be a difficult and tedious task for very small QFN devices. Printing solder paste directly onto the QFN device is a more popular approach to solving the rework problem.

While the basic challenges of continually denser package attachments are not new, the specifics of how to effectively deal with the new component types requires subtle tailoring. You can see that when it comes to optimizing your design for QFN and DFN packages there are a lot of things to consider, from the initial design of the solder mask opening to your selection of the proper stencil type and finish. As component packages continue to evolve and push the envelope, fabricating with new attachment patterns will pose challenges to the manufacturer for continued production of high quality, high yield assemblies. At Photo Stencil, we are always available to provide the assistance and experience needed to optimize a stencil for your unique design requirements. **SMT**

Rachel Short is vice president of sales and marketing at PhotoStencil LLC. She may be reached via e-mail, or by phone at 719-304-4224.