Printing and Assembly Challenges for QFN Devices

Rachel Short
Photo Stencil
Colorado Springs

Benefits and Challenges

QFN (quad flatpack, no leads) and DFN (dual flatpack, no lead) are becoming more popular in new component releases. Their very small form factor allows smaller packages, better grounding, and better heat sink thermal properties compared to other SMT packages. Most QFNs have a metal pad on the underside of the part for grounding and heat conduction. DFNs have a similar center metal pad but have leads on only two sides. Typical thickness of the QFN devices is .85mm and the body range from 3mm up to 12mm, so the packages are very small and very light. The QFN leads and ground plane conductor are flat and in the same plane on the bottom of the package. Printing solder paste 1-1 with the ground plane can cause the QFN to float during reflow, thus miss-registering the leads on the QFN and the pads on the PCB. QFN float can be controlled by reducing the amount of solder paste printed on the ground plane. Typically a 50 to 60% reduction will solve the QFN float problem. However the aperture reduction must be done judicially. A Window Pane aperture is recommended for most cases. This allows the solder paste volatiles to easily escape during reflow without moving the QFN device. Figure 1 shows the benefits of window pane apertures for the ground plane.

The next challenge is the actual aperture size in the stencil. Figure 2 shows a 3mm QFN and a 4mm QFN device. Typical aperture widths as low as .175mm and aperture lengths as low as .4mm present a challenge to the printing process as far as percent paste transfer. The other challenge is the solder mask employed on the PCB. There are three types of solder mask design which are shown in Figure 3: 1 - SMD where the pad opening on the board is defined by the solder mask, 2 - 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), and 3 - NSMD – Window. In the last case there is no solder mask between pads so bridging between pads is more likely than with solder mask between pads.

Stencil and PCB Design Considerations

Table 1 shows stencil design guidelines for the three solder mask cases. This table shows the package size, the lead pitch, the number of I/O, the package lead dimensions, the recommended PCB pad dimensions, the recommended stencil aperture dimension, recommended stencil thickness, and resulting Area Ratio. For NSMD the stencil aperture is 1-1 with the PCB pad dimension. It should be noted that the recommended length of the pad on the PCB compared to the length of the lead on the QFN is .2mm larger. As seen the Area Ratio for a .125mm thick stencil is >.66 for all the examples listed. Aperture size for the SMD is .05mm smaller than the PCB pad. There are typically two reasons for this reduction. If the stencil is slightly misaligned to the PCB, paste could be printed on the solder mask. Also there might be high stress points if solder contacts the mask. The reduction in aperture size has reduced the Area Ratio making paste transfer more difficult. For Area Ratios below .66 Electroform stencils or Nano-Coated stencils are normally recommended. The final example in Table 1 is the NSMD –Widow. The pitch is .4mm leaving little room to put solder mask between pads on the PCB. Aperture size is also small giving a challenging Area Ratio for .125mm thick stencils; therefore .100mm thick stencils are normally recommended to provide a more robust stencil printing process window.

Another problem arises when using a NSMD window when the solder mask is higher than the pad on the PCB. In this case the solder paste is extruded through the stencil since the stencil is not in contact with the PCB pads during printing. This extruded paste will make contact with the bottom side of the stencil causing potential bridging during successive prints since there is no solder mask between neighboring pads. Stencil wiping after every print may help reduce this problem. An Example of a NSMD – Window PCB with solder mask above the height of the PCB pads is shown in Figure 4. One possible solution suggested by a customer[1] is a PCB side step stencil as shown in Figure 5. This is an Electroform stencil which is .08mm thick everywhere except in the QFN area inside the solder mask where it is .01mm thick.

In this case the mask opening was of the order of .125mm per side except on the ends of the pad rows where it was less. There are several limitations to this approach. Namely the spacing between the step and the solder mask is extremely small allowing for little miss-registration. Also the stencil is thinner for all other components except the QFNs which may yield insufficient paste. The first limitation could be addressed at the PCB design level by making the mask to pad clearance much larger; of the order of .25mm per side as well as leaving the ground plane without any solder mask surrounding.

Another possible solution is a single level stencil without step but with Nano-Coating on the aperture walls as well as on the bottom side (PCB side) of the stencil. Nano-Coatings have a property called fluxophobicity. Quite simply it is the stencils ability to resist the spread of flux on its surface. It is measured in the form of the 'Flux Contact Angle'. This is the angle that the flux will form when a drop is placed on the surface of the stencil. Nano coating not only increases the paste ability to release form 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. This property not only eliminates the need for frequent under board wiping but also reduces the occurrence for pad to pad bridging.

QFN 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 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 ranging in size from 3mm up to 12mm. Printing solder paste directly onto the QFN device is a more popular approach[2]. Figure 6 shows a repair stencil that fits into a holding tool which also holds the QFN in registration to this stencil. The top portion of a 7mm / 48 I/O .5mm pitch repair stencil and the top portion of a 10 mm / 72 I/O .5mm pitch repair stencil are shown. Note that the aperture area for the ground plane is reduced by approximately 50% by the window pane technique. Figure 7 shows paste printed on QFN pads using the tool that holds both the stencil and the QFN device. Figure 8 shows paste on the same QFN. After paste is applied the QFN is placed on the PCB and locally reflowed.

Conclusion

Although QFN devices present a challenge to the SMT assembly process with proper stencil design, proper stencil technology selection (Laser, Electroform, Nano-Coat), and proper PCB solder mask layout these challenges can be overcome. The most popular QFN repair seems to be to print solder paste directly onto the QFN leads and ground plane.

References:

1- Private communication with Greg Kloiber, Manufacturing Engineer Plexus Corp.
2- "BGA and QFN Repair Process" William E. Coleman, APEX 2008

General QFN References

1- "Rule of thumb guide for Practical DFN/QFN Printed Circuit Board and stencil Design" James R. Staley, Linear Technology application note
2- "PCB Land Pattern Design and Surface Mount guidelines for QFN Packages", Intersil Technical Brief TB389.6
3- "QFN Layout Guideline", Texas Instrument Application Report SLOA122

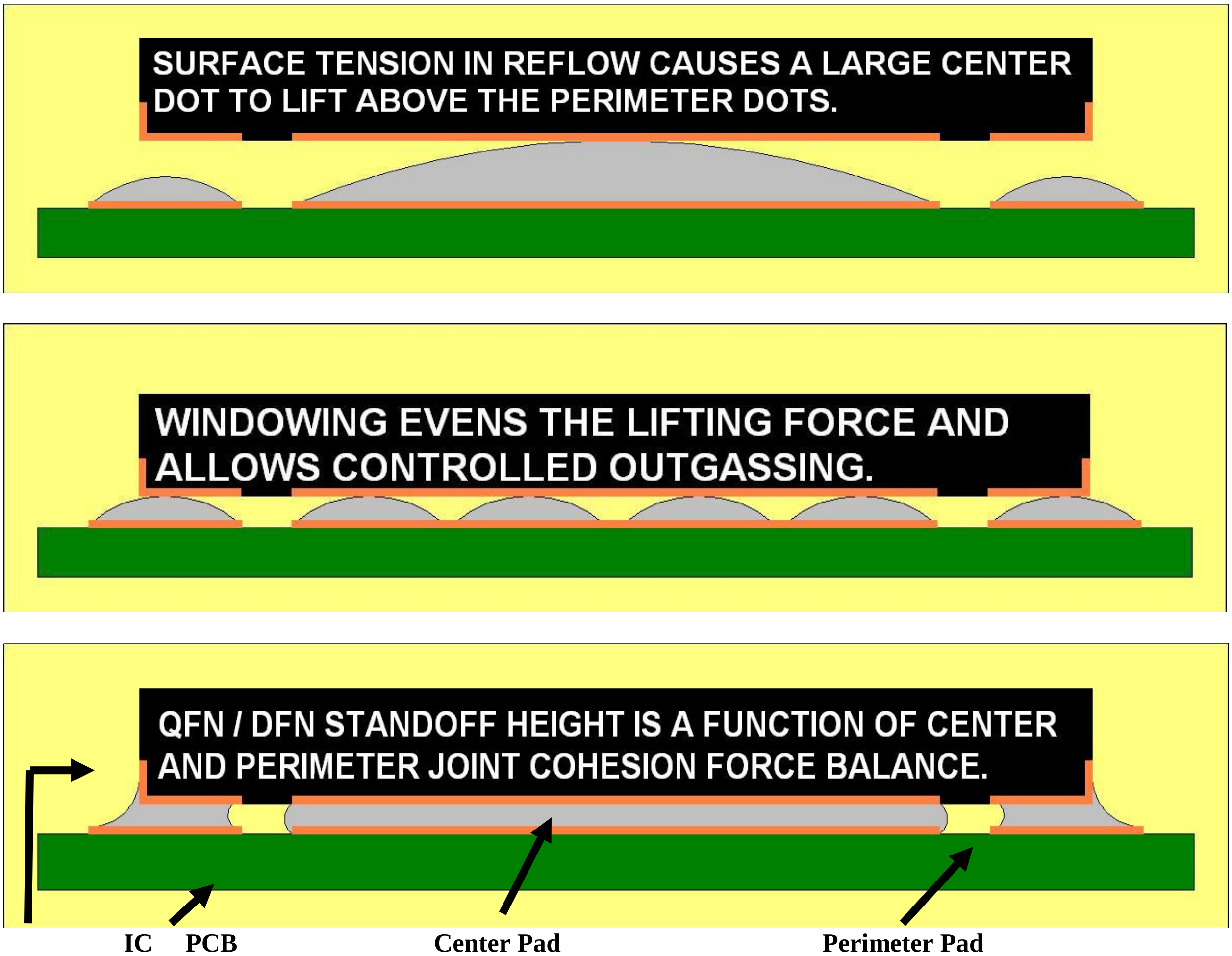


Figure 1 Benefit of Window Pane Ground Plane Apertures

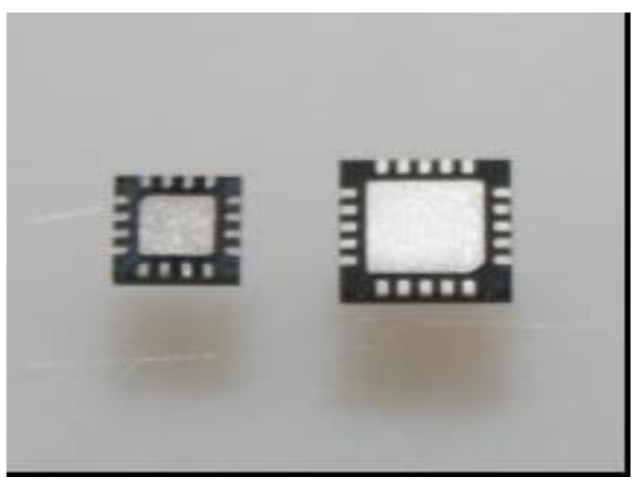


Figure 2 3 and 4 mm QFN Devices

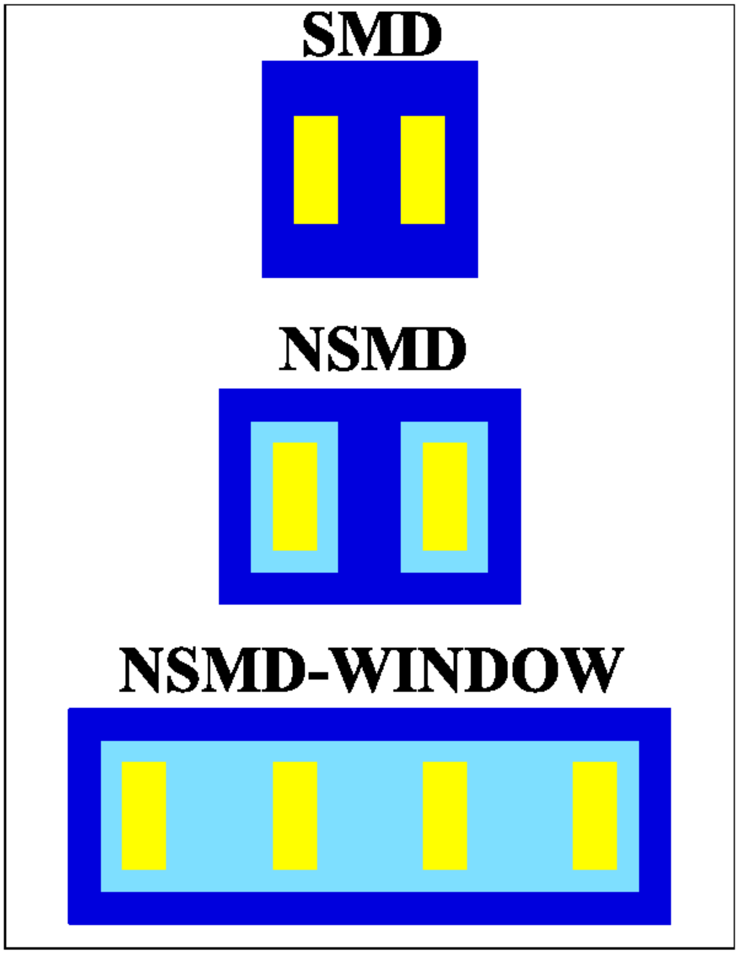


Figure 3 Solder Mask Designs

Table 1 QFN, PCB, Stencil Design Guidelines

Stencil Design for Typical QFN Apertures (NSMD)

Package	Pitch	I/O	package lead width	package lead length	PCB	PCB	Aperture NSMD	Aperture NSMD	Stencil Thickness	Area Ratio
3mm	.5mm	12	.23mm	.55mm	.23mm	.75mm	.23mm	.75mm	.125mm	0.70
4mm	.5mm	20	.25mm	.40mm	.25mm	.60mm	.25mm	.60mm	.125mm	0.71
7mm	.5mm	44	.25mm	.55mm	.25mm	.60mm	.25mm	.60mm	.125mm	0.71
10mm	.5mm	72	.23mm	.40mm	.25mm	.60mm	.25mm	.60mm	.125mm	0.71
12mm	.5mm	80	.25mm	.55mm	.25mm	.75mm	.25mm	.75mm	.125mm	0.75

Stencil Design for Typical QFN Apertures (SMD)

Package	Pitch	I/O	package lead width	package lead length	PCB	PCB	Aperture SMD	Aperture SMD	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	.60mm	.20mm	.55mm	.125mm	0.59
10mm	.5mm	72	.23mm	.40mm	.25mm	.60mm	.20mm	.55mm	.125mm	0.59
12mm	.5mm	80	.25mm	.55mm	.25mm	.75mm	.20mm	.70mm	.125mm	0.62

Stencil Design for Typical QFN Apertures (NSMD window)

Package	Pitch	I/O	package lead width	package lead length	PCB	PCB	Aperture SMD	Aperture SMD	Stencil Thickness	Area Ratio
4mm	.4mm	28	.175mm	.45mm	.175mm	.710mm	.175mm	.710mm	.125mm	0.56
4mm	.4mm	28	.175mm	.45mm	.175mm	.710mm	.175mm	.710mm	.100mm	0.70

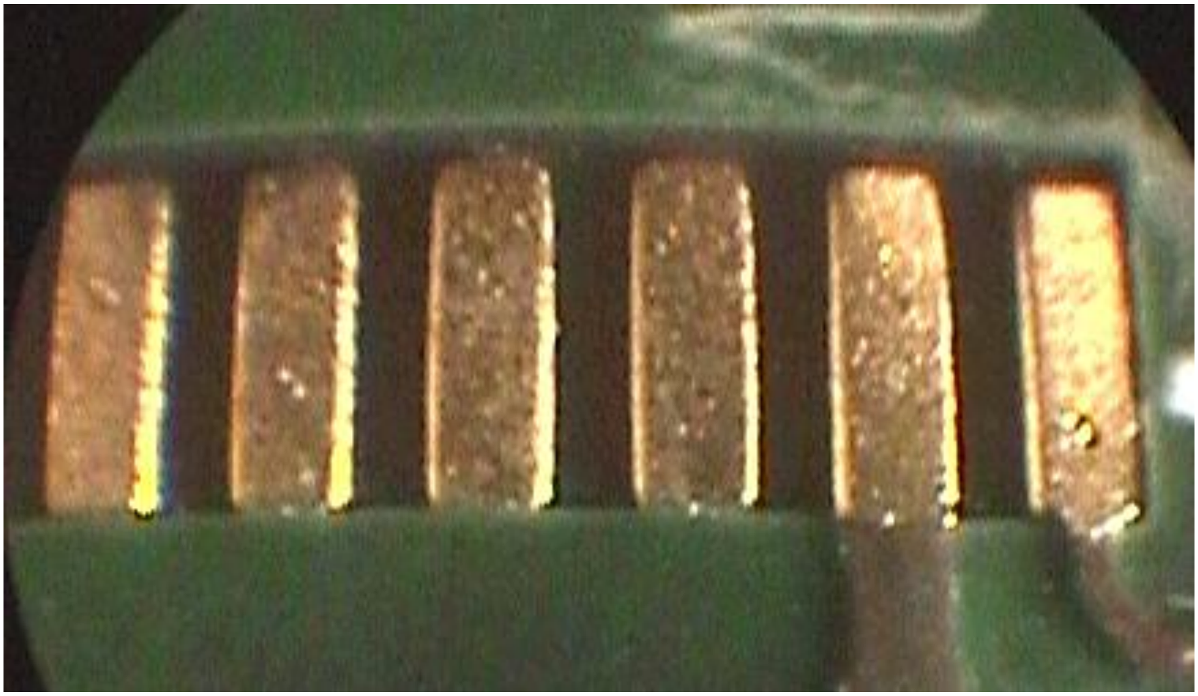


Figure 4 NSMD-Window with Mask to pad gap of .03mm

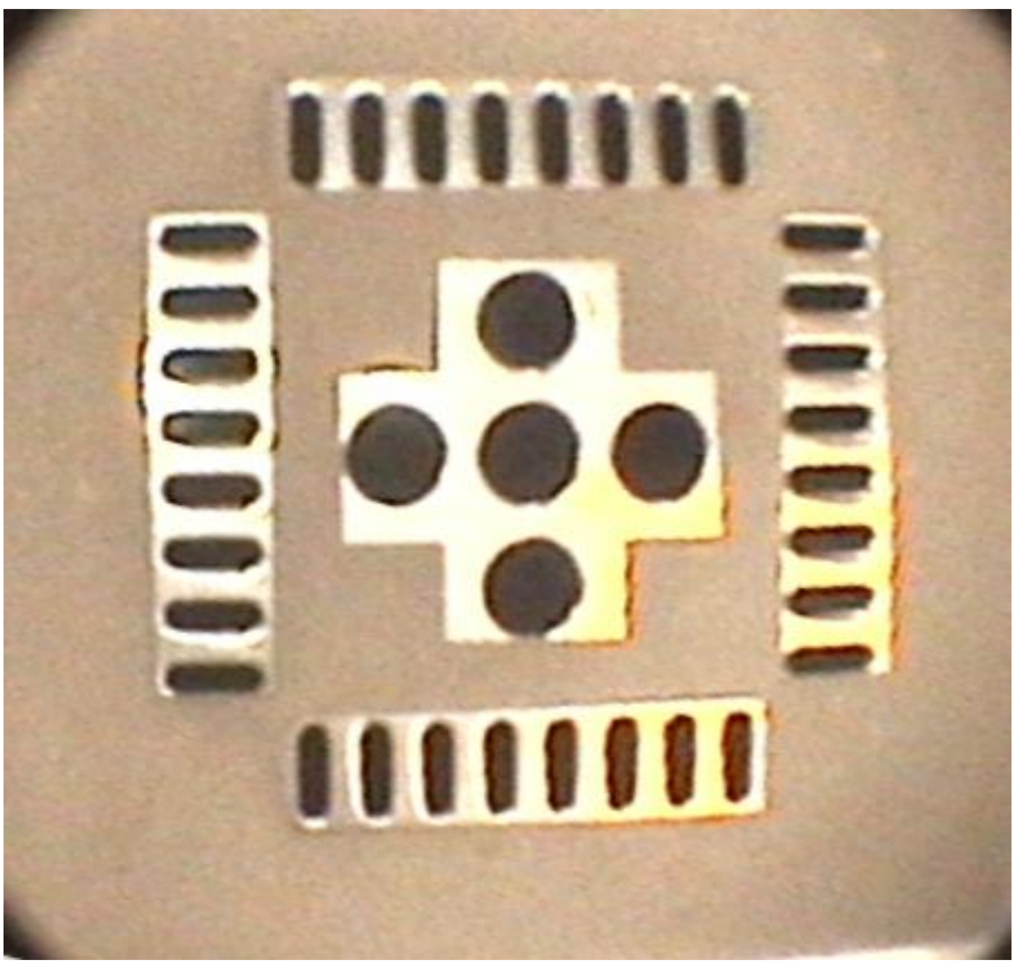
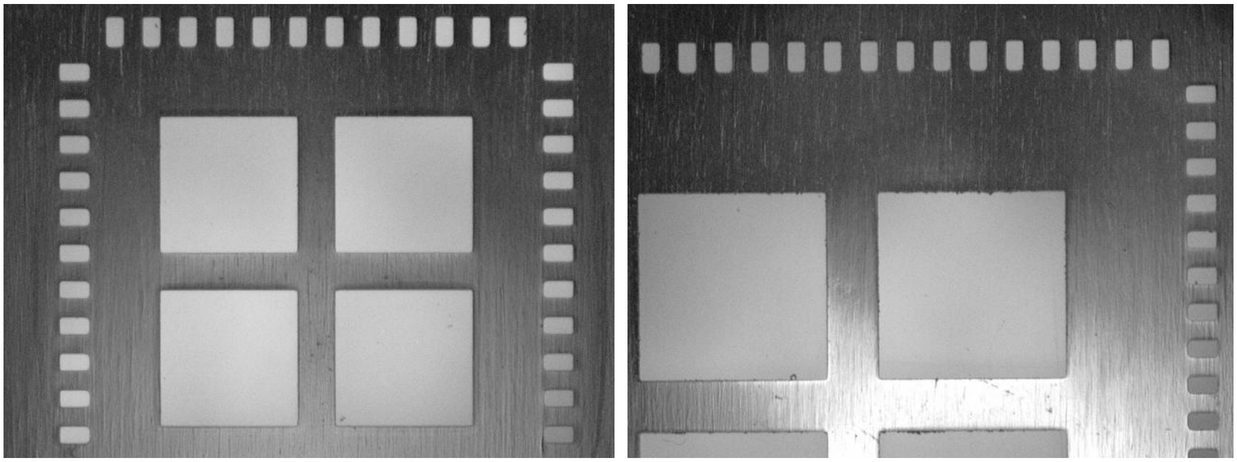


Figure 5 Step Electroform Stencil on PCB side - .1mm thick around QFN apertures and .08mm elsewhere for all other apertures.



7mm repair stencil **10mm repair stencil**

Figure 6 QFN Repair stencil with window pane ground plane apertures

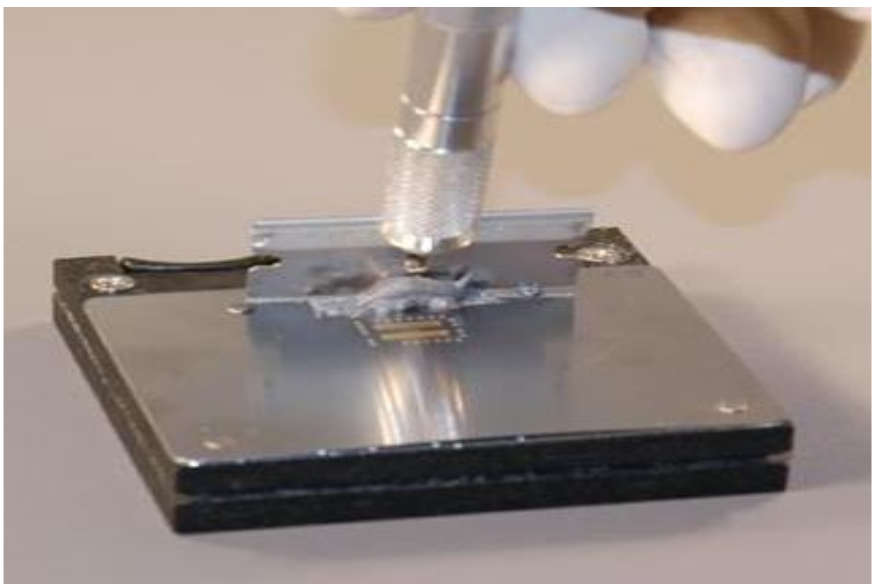


Figure 7 Printing Solder Paste on QFN device while in holding tool

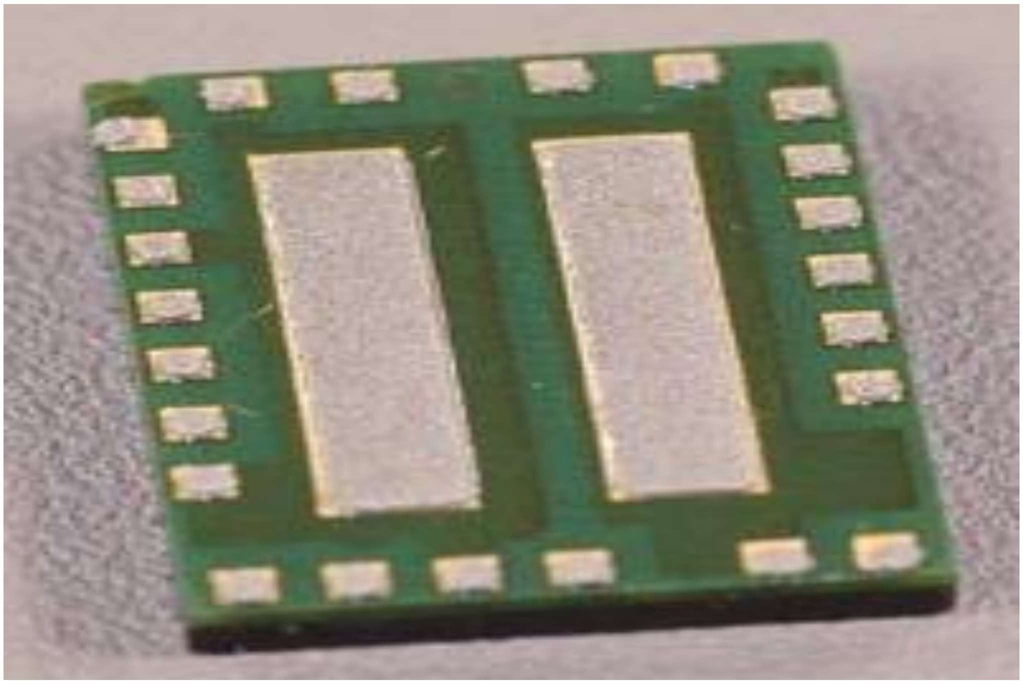


Figure 8 Solder paste printed on QFN before placement on PCB for rework

Printing and Assembly Challenges for QFN Devices

Rachel Miller Short

Photo Stencil

Colorado Springs, CO

Definition:

QFN (Quad Flat Pack, No Leads)

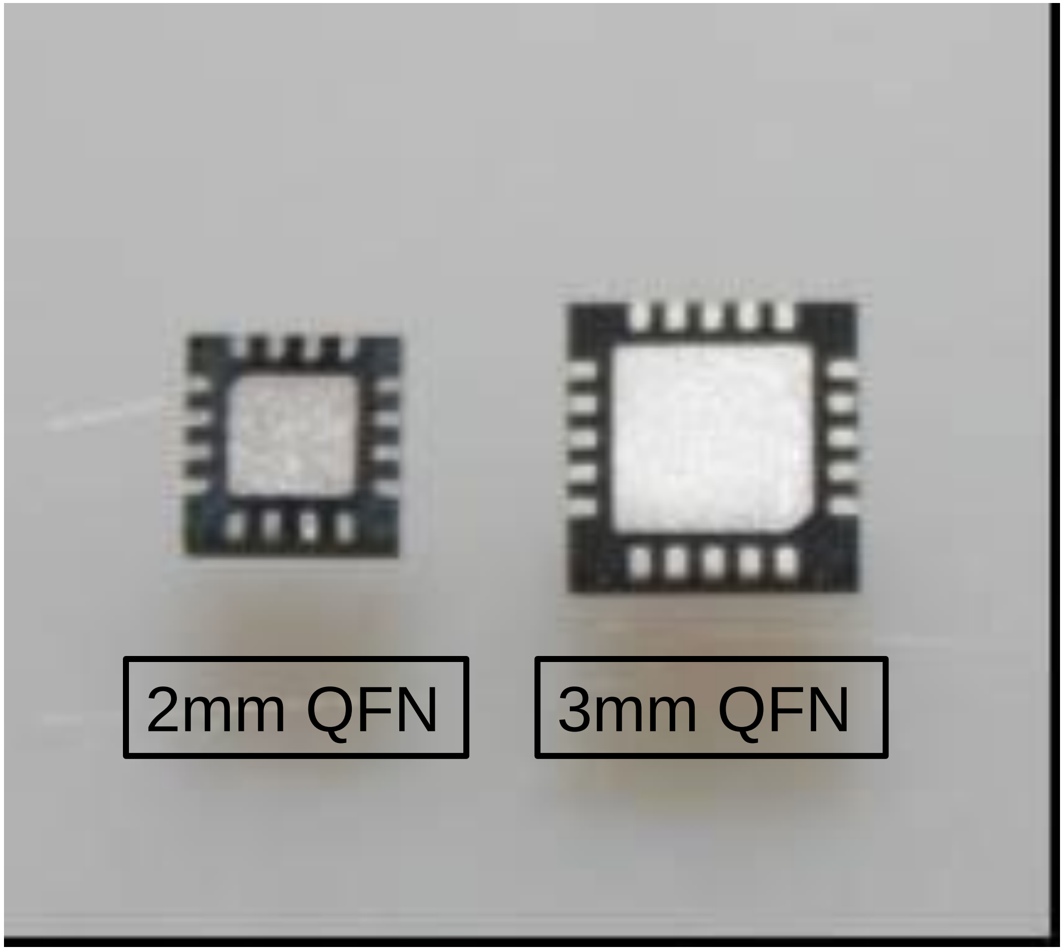
DFN (Dual Flat Pack, No Leads)

Leads and Ground Plane are flat on bottom of package

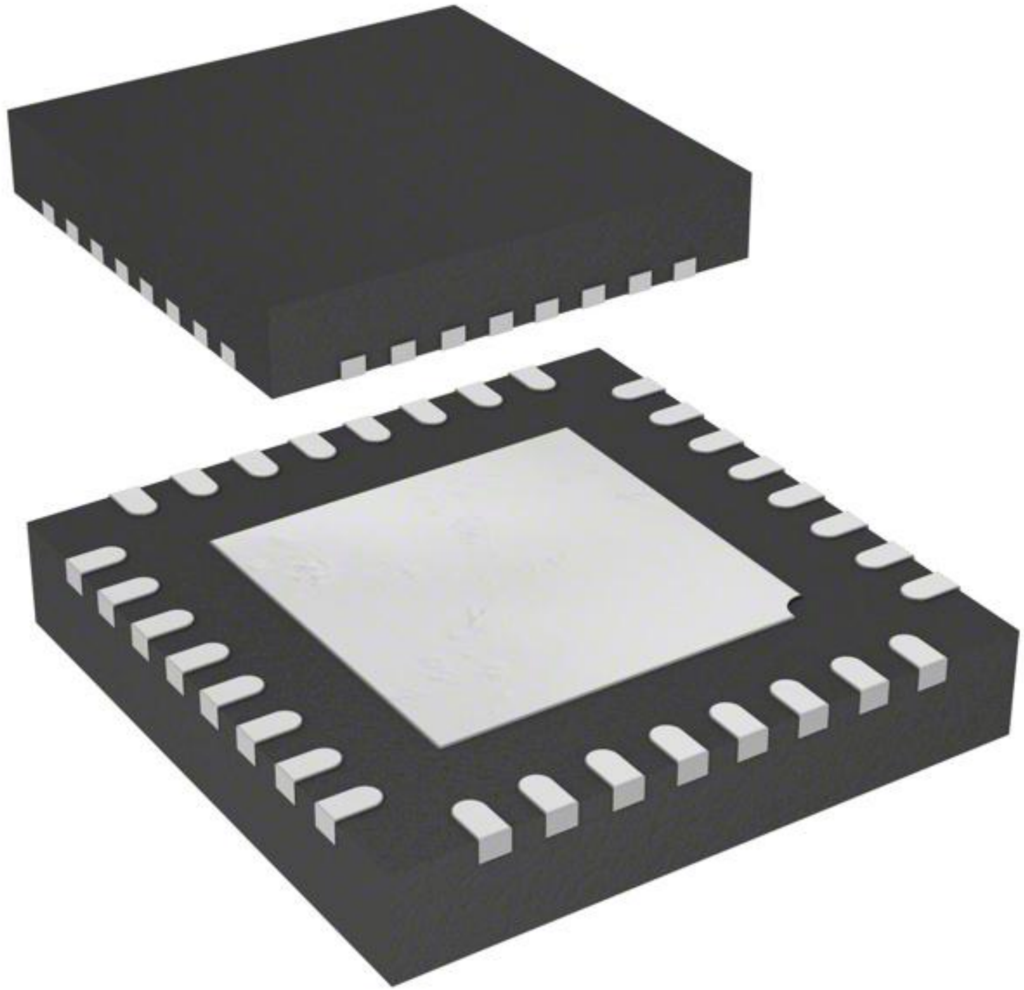
Typical Size range 2mm to 12mm

Typical Thickness .85mm

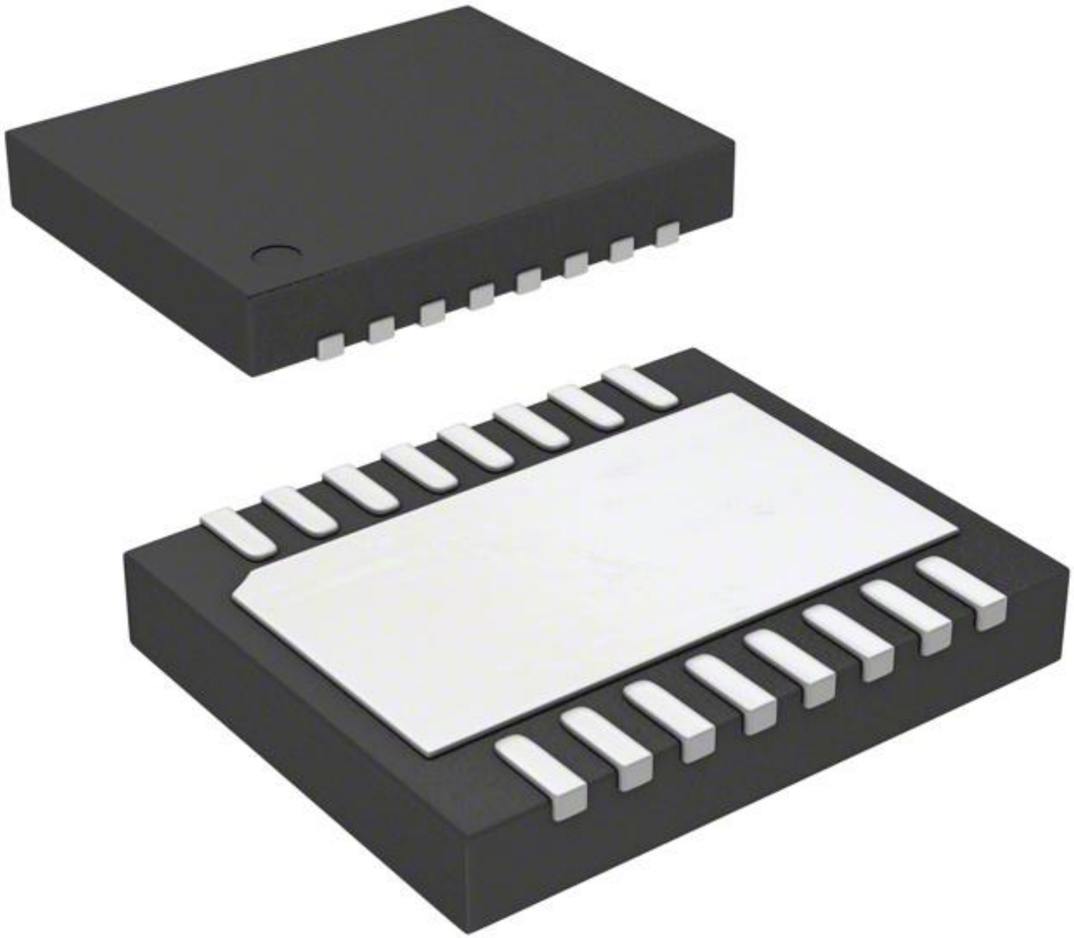
Typical Pitch .4mm / .5 mm



QFN Packages



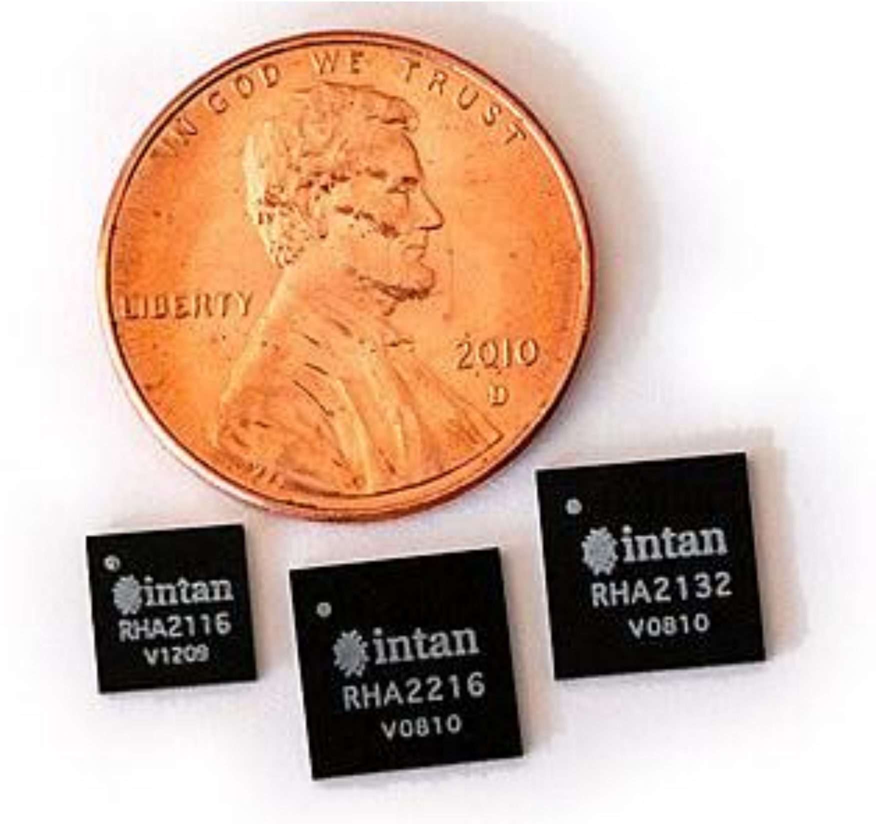
5 mm QFN, 32 I/O, .5mm pitch



4 x 5 mm DFN, 16/I/O, .5mm pitch

Advantages of QFN Devices

- Very Small Form Factor
- Ground Plane affords excellent thermal and electrical Conductivity
- Packages are small and light
- Ideal for small Hand Held Devices


IN GOD WE TRUST
LIBERTY
2010
D
intan
RHA2116
V1209
intan
RHA2216
V0810
intan
RHA2132
V0810

Printing Challenges:

A) Package can float during reflow

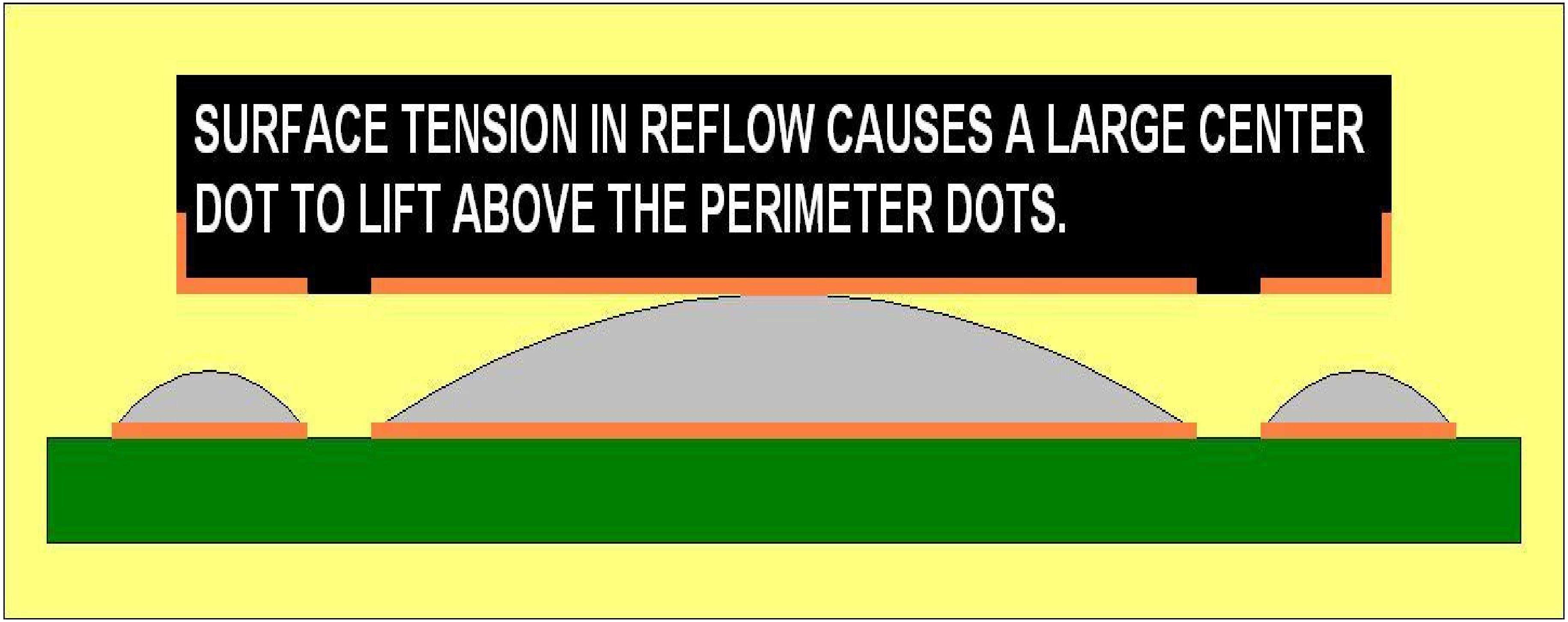
B) Solder Mask Configurations influence Stencil Design

C) Apertures typically short and narrow resulting in Low Area Ratio (can result in poor paste transfer)

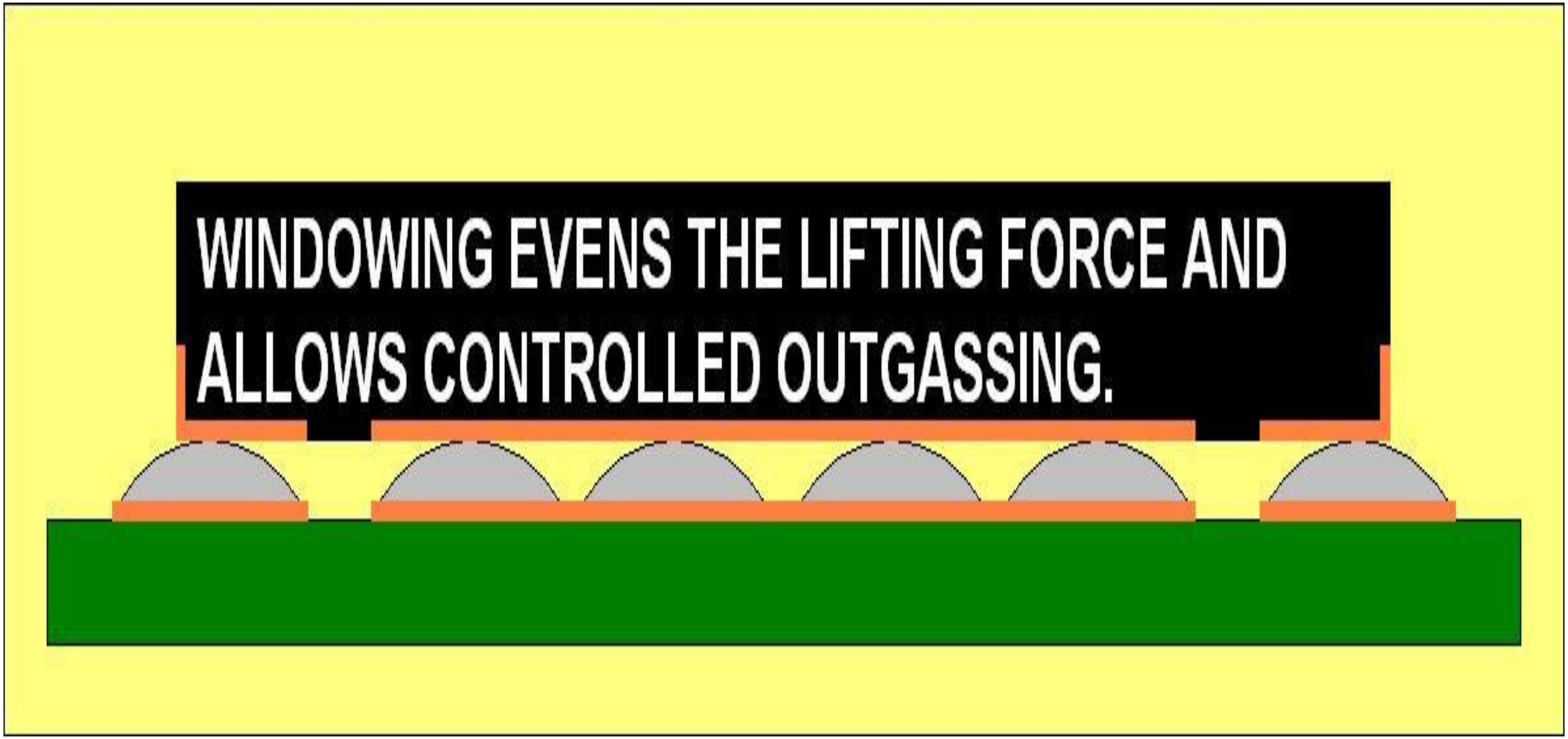
D) Solder Mask Height higher than PCB pads can cause stencil to PCB gasketing issues

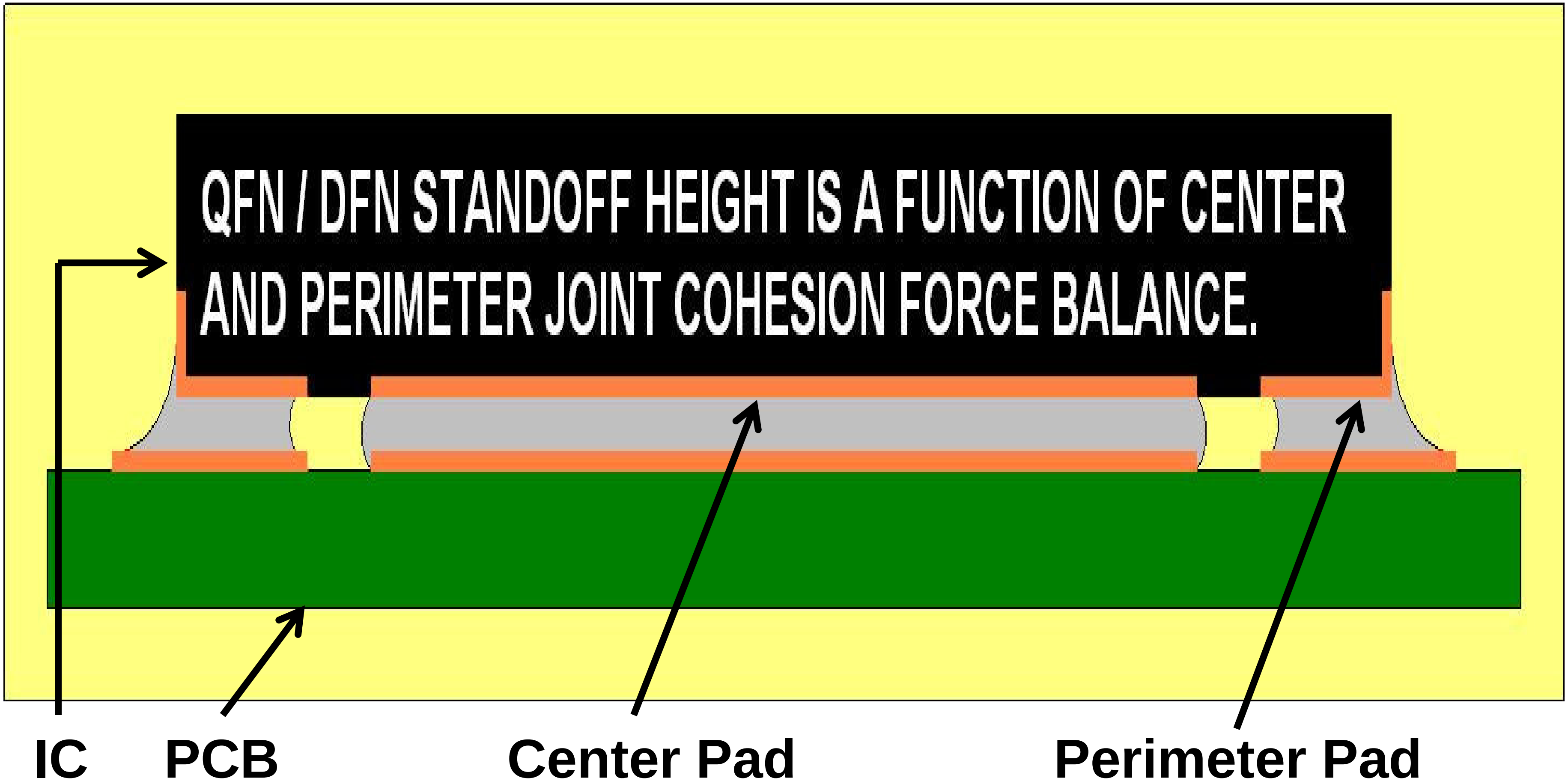
E) QFN Repair can be problematic

A) Package Float Problem occurs when solder paste printed 1-1 with ground plane pad on the PCB



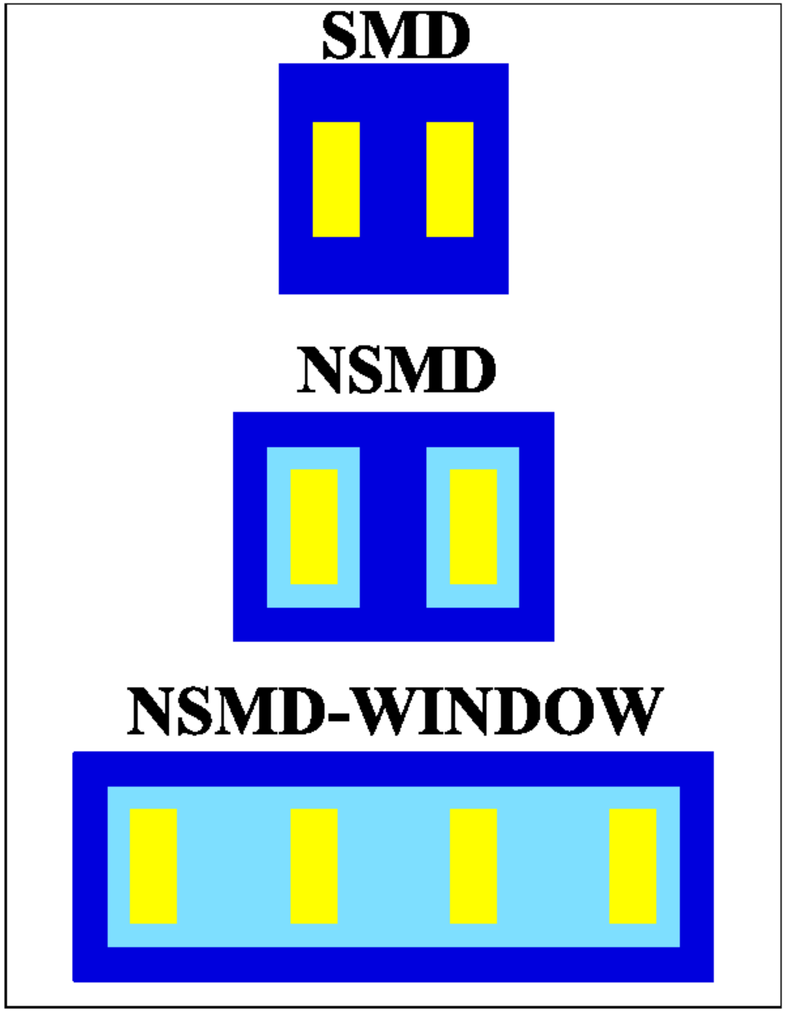
Window pane of the Ground Plane Aperture reduces aperture area by ~50% allows proper outgassing without moving the QFN part





Benefit of Window Pane Ground Plane Apertures

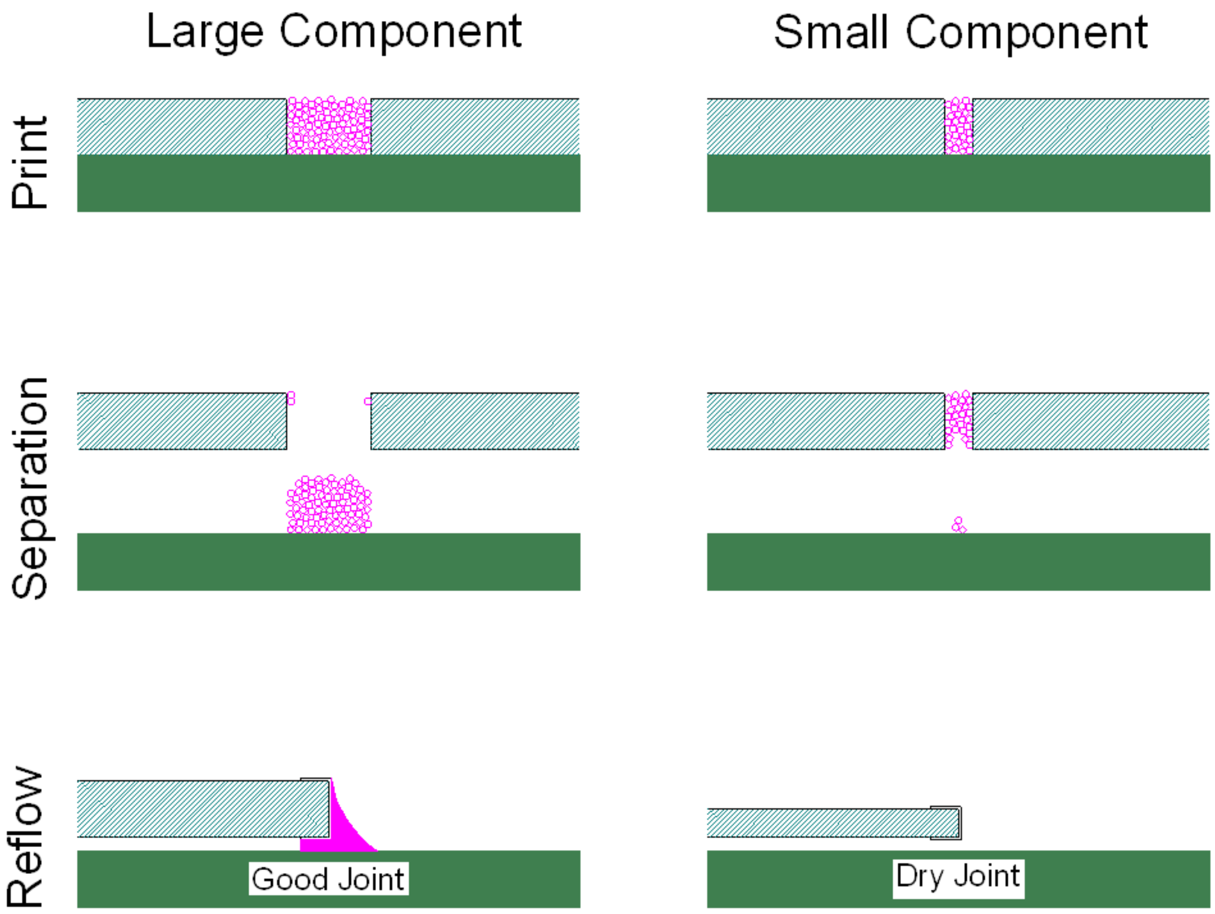
B) Solder Mask Configurations



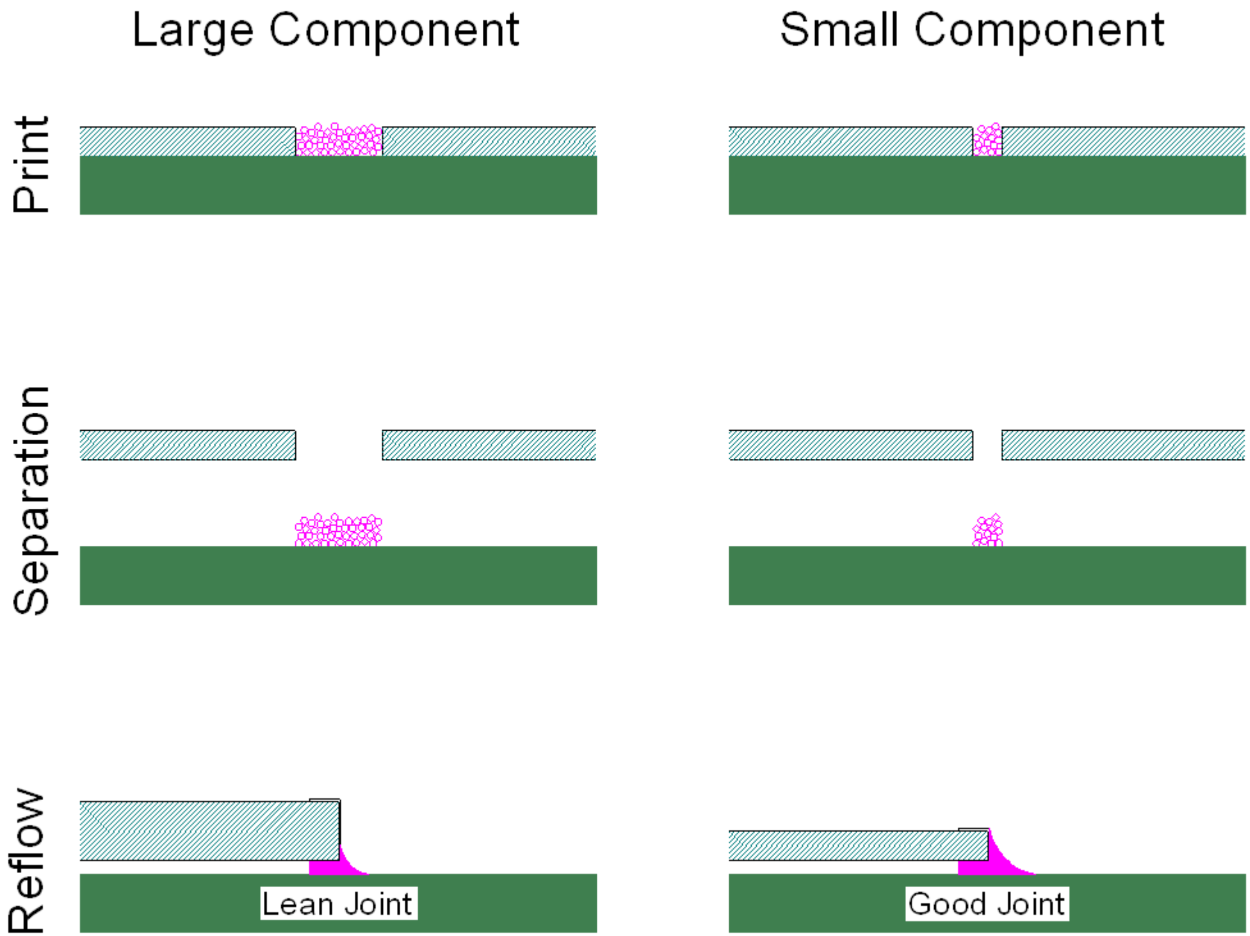
C) Why is area Ratio a paste transfer issue?

- Paste Transfer is a competitive process:
- The side walls of the aperture hold the solder paste inside the aperture.
- The area under the aperture (pad on PCB) pull the Solder paste out of the aperture away from the aperture walls.
- The larger the wall area compared to the pad area beneath the aperture the more difficult it is for the Paste to release from the aperture walls.
- Aperture Wall smoothness is a key issue for paste Transfer when the Area Ratio is small.

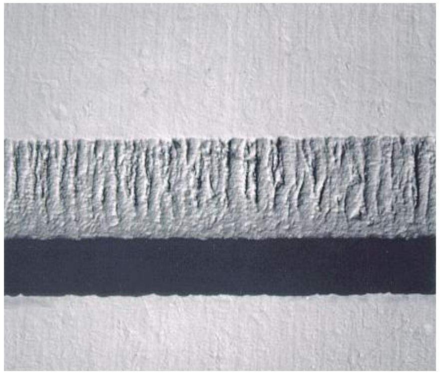
Thick Stencil



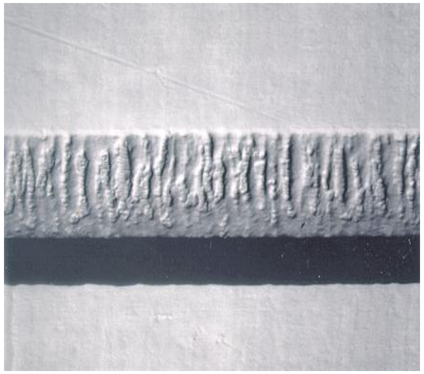
Thin Stencil



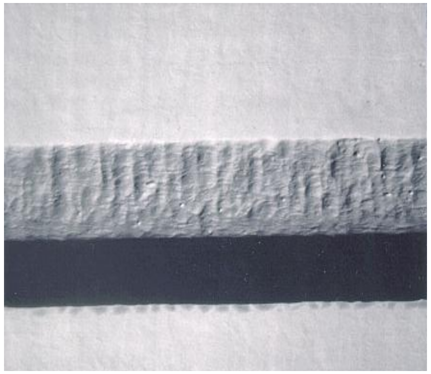
Laser



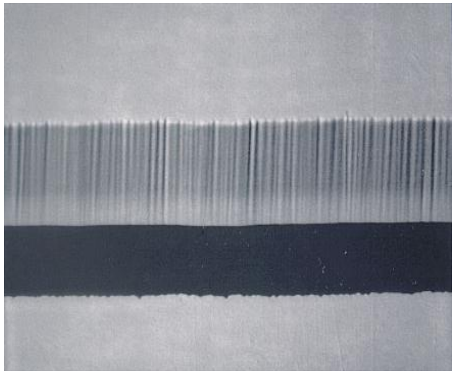
Laser EP



Laser EP / NP



E-FAB



IPC 7525B Stencil Design Guidelines for Area Ratio Standards

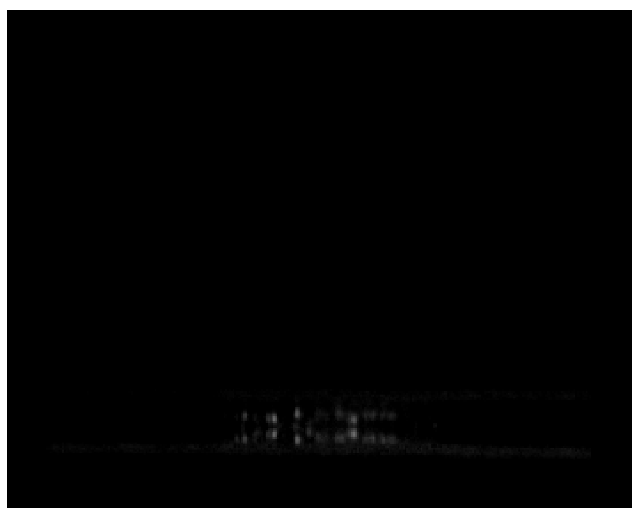
- Area Ratio (AR) = Area of Aperture Wall / Area of pad under aperture
- Stencil Technology recommended per AR range:
- AR> .90 Laser-cut, High Precision (HP) Chem-etch, Chem-etch
- AR .66 to .90 Laser-cut, HP Chem-etch
- AR .5 to .66 Electroform

*Note from IPC-7525B pg 5: Note that figures 3-1 through 3-4 are general design guides based on area ratios. It must be recognized that there are a wide variety of stencils available in the industry made with electroform, laser, chemical etch, and high-precision etching process. It is quite possible that some electroform stencils may not perform satisfactory with the area ratios between 0.5 and 0.66. On the other hand it is possible that some high-precision etched stencils (with post processing) or laser-cut stencils do perform satisfactory with the area ratios between 0.5 and 0.66.

Solder Paste Release Video

Chemical Etch

Laser Cut



Electoformed

Stencil Aperture Design for NSMD Mask

Typically 1-1 with the pad Layout on the PCB

Area Ratios are typically > .70 thus paste transfer is not an issue

Stencil Design for Typical QFN Apertures (NSMD)

Package	Pitch	I/O	package lead width	package lead length	PCB	PCB	Aperture NSMD	Aperture NSMD	Stencil Thickness	Area Ratio
3mm	.5mm	12	.23mm	.55mm	.23mm	.75mm	.23mm	.75mm	.125mm	0.70
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	.25mm	.40mm	.25mm	.60mm	.25mm	.60mm	.125mm	0.71
12mm	.5mm	80	.25mm	.55mm	.25mm	.75mm	.25mm	.75mm	.125mm	0.75

Stencil Aperture Design for SMD Mask

Typically a reduction to the pad Layout on the PCB
Area Ratios are typically <. 66 thus paste transfer is an issue.
Typically an Electroform Stencil is recommended

Stencil Design for Typical QFN Apertures (SMD)

Package	Pitch	I/O	package lead width	package lead length	PCB	PCB	Aperture SMD	Aperture SMD	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

Stencil Aperture Design for NSMD window Mask

Typically 1-1 with the pad width Layout on the PCB

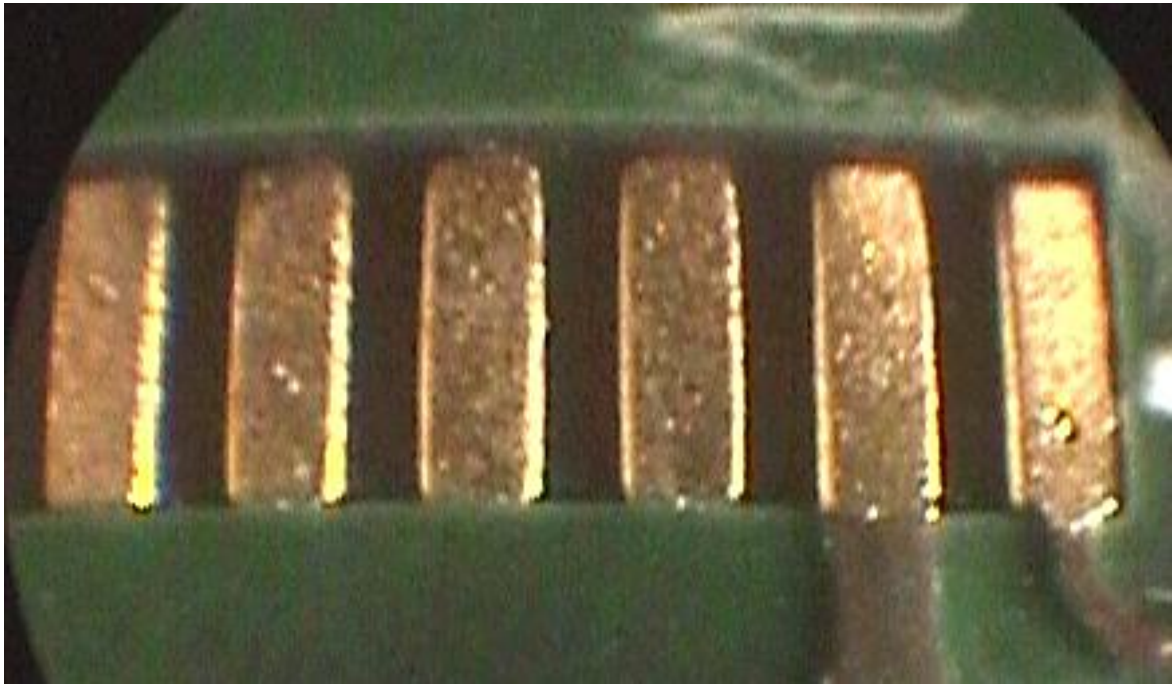
Most .4mm QFN's are NSMD window and even though the apertures are 1-1 with the pad width, Area Ratios are <.66.

Stencils with smooth aperture walls such as electroform are recommended.

Stencil Design for Typical QFN Apertures (NSMD window)

Package	Pitch	I/O	package lead width	package lead length	PCB	PCB	Aperture SMD	Aperture SMD	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

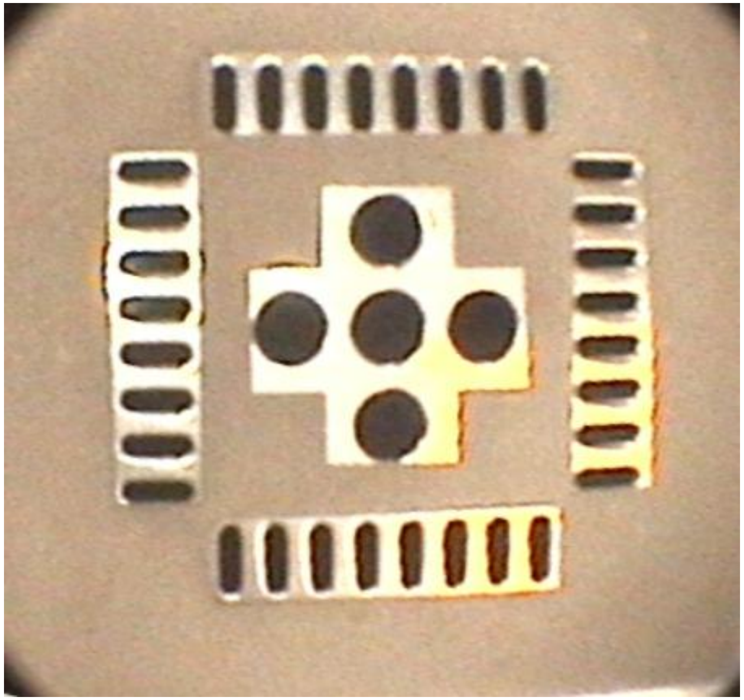
D) Solder mask Height is higher than PCB pads



NSMD Window with Mask to pad gap of .03mm

Problem: Stencil can’t make contact with pads causing paste smear on bottom side of stencil.

Step Electroform Stencil allows Stencil to go down into Solder Mask cavity to achieve good contact to the pads



Positioning of the Step Stencil to the PCB can be a problem

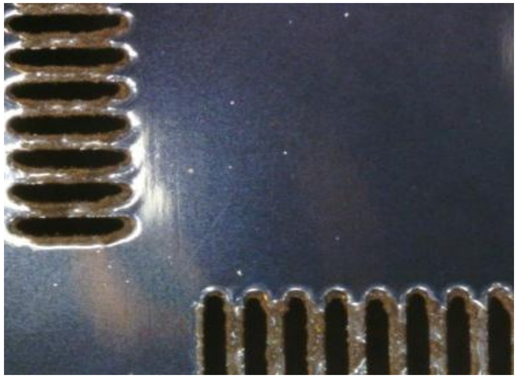
Step Electroform Stencil on PCB side - .1mm thick around QFN apertures and .08mm elsewhere for all other apertures.

Alternative Solution: Nanocoat .1mm thick Stencil w/o Step

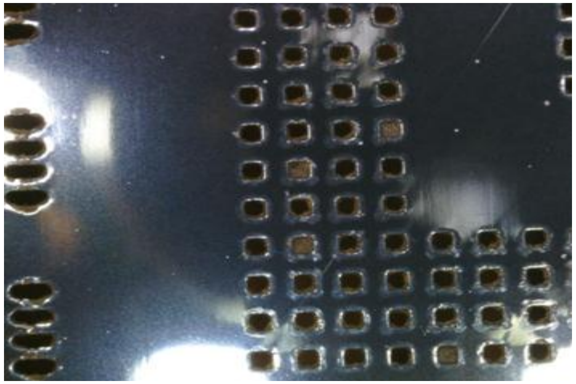
Nanocoat will reduce the spread of solder paste under the stencil even though the stencil is not making good contact to the PCB pad.

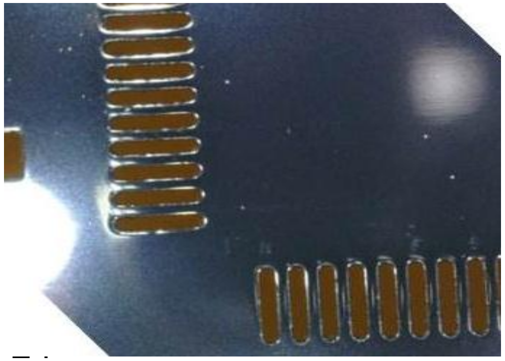
Cleaner Frequency Test Results

The images were taken on bottom of the stencil in 2 areas .5mm pitch uBGA and .5mm pitch QFP

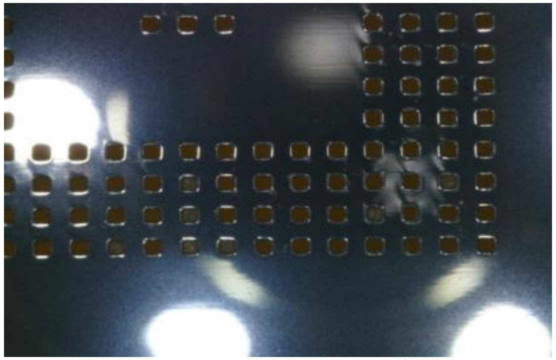


← Electroform without Nano-coat after 14 prints without wipe →



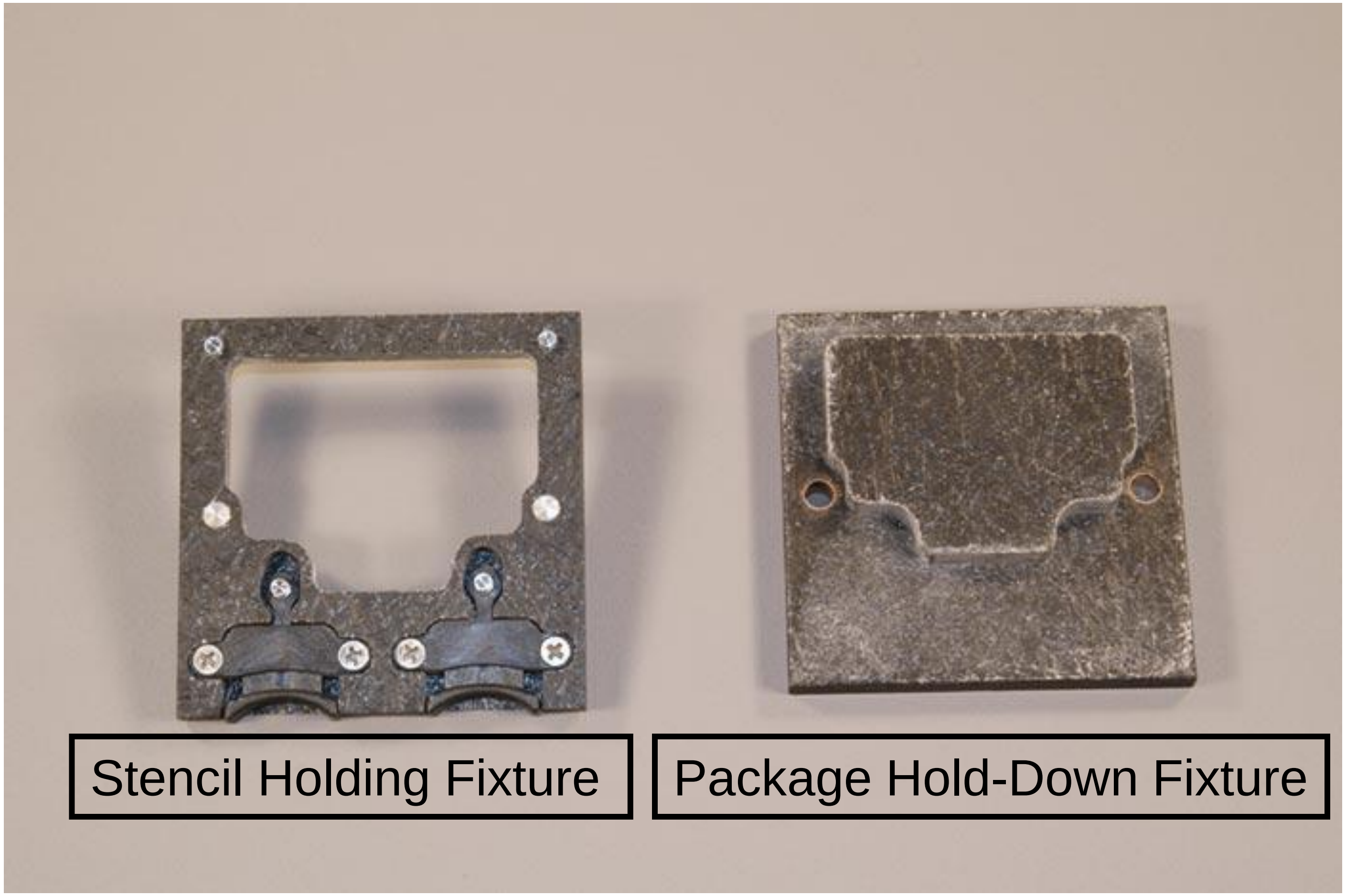


← Electroform with Nano-coat after 40 prints without wipe →



E) QFN Repair Options

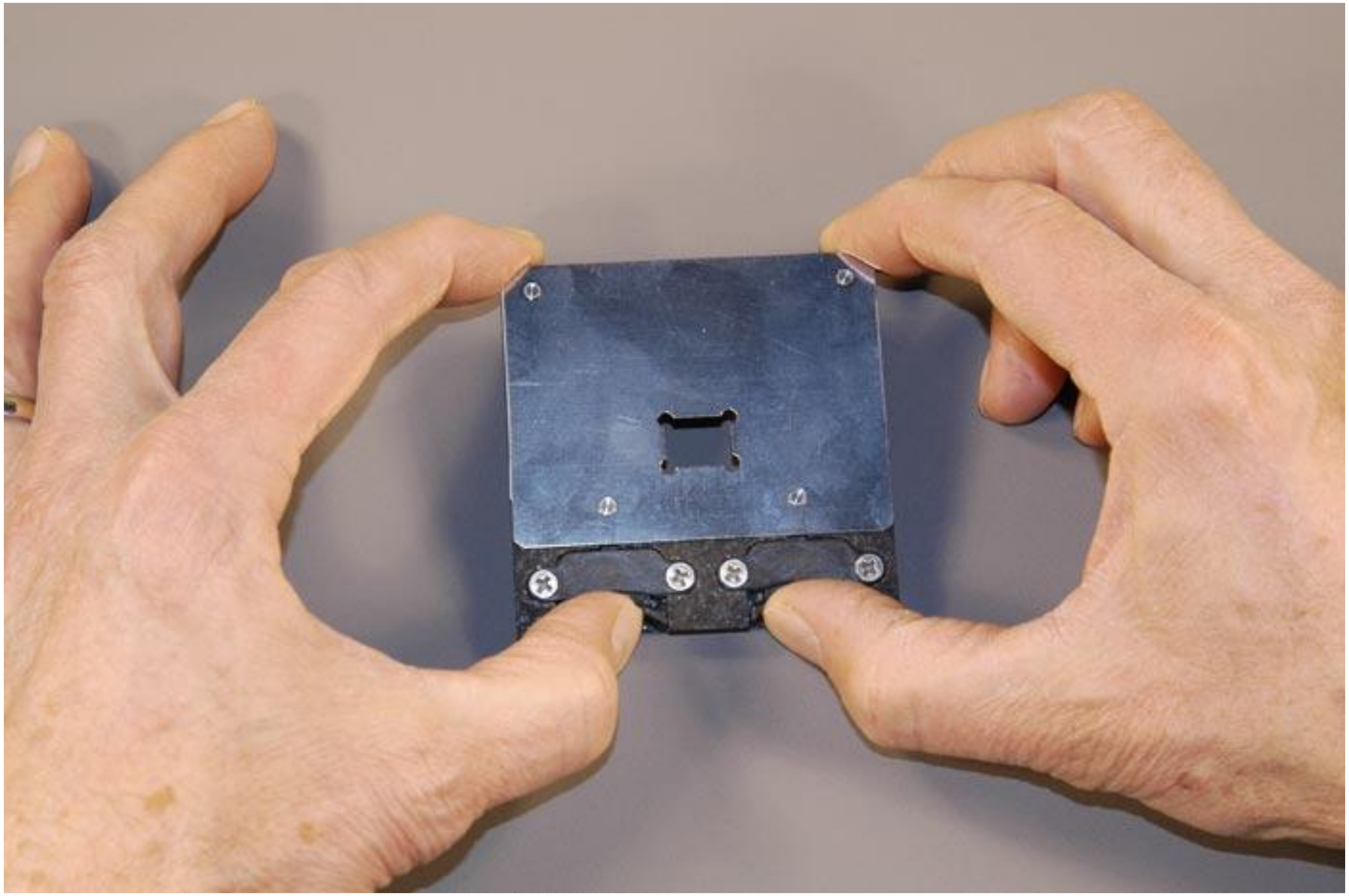
QFN Repair Tool



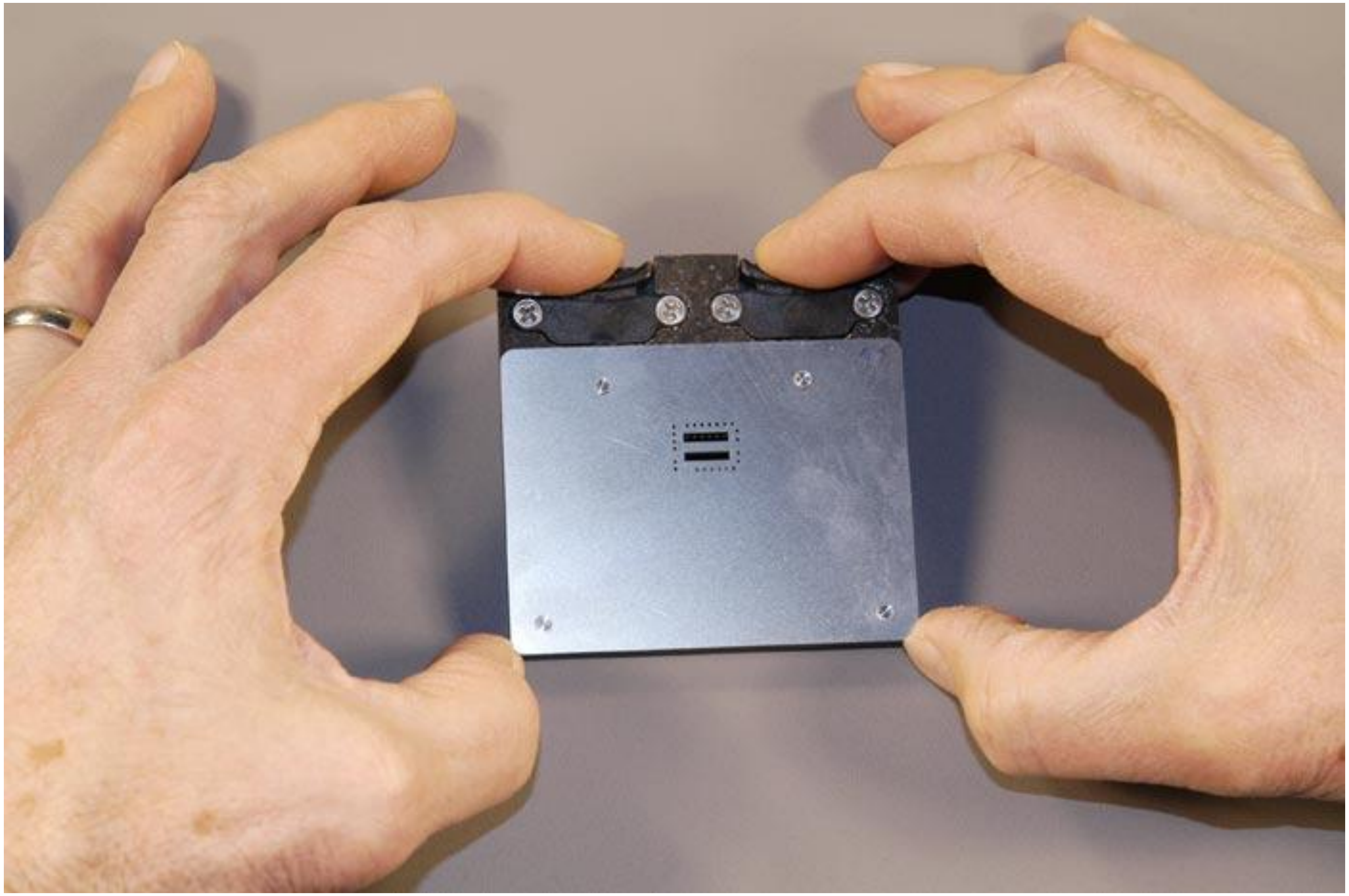
Stencil Holding Fixture

Package Hold-Down Fixture

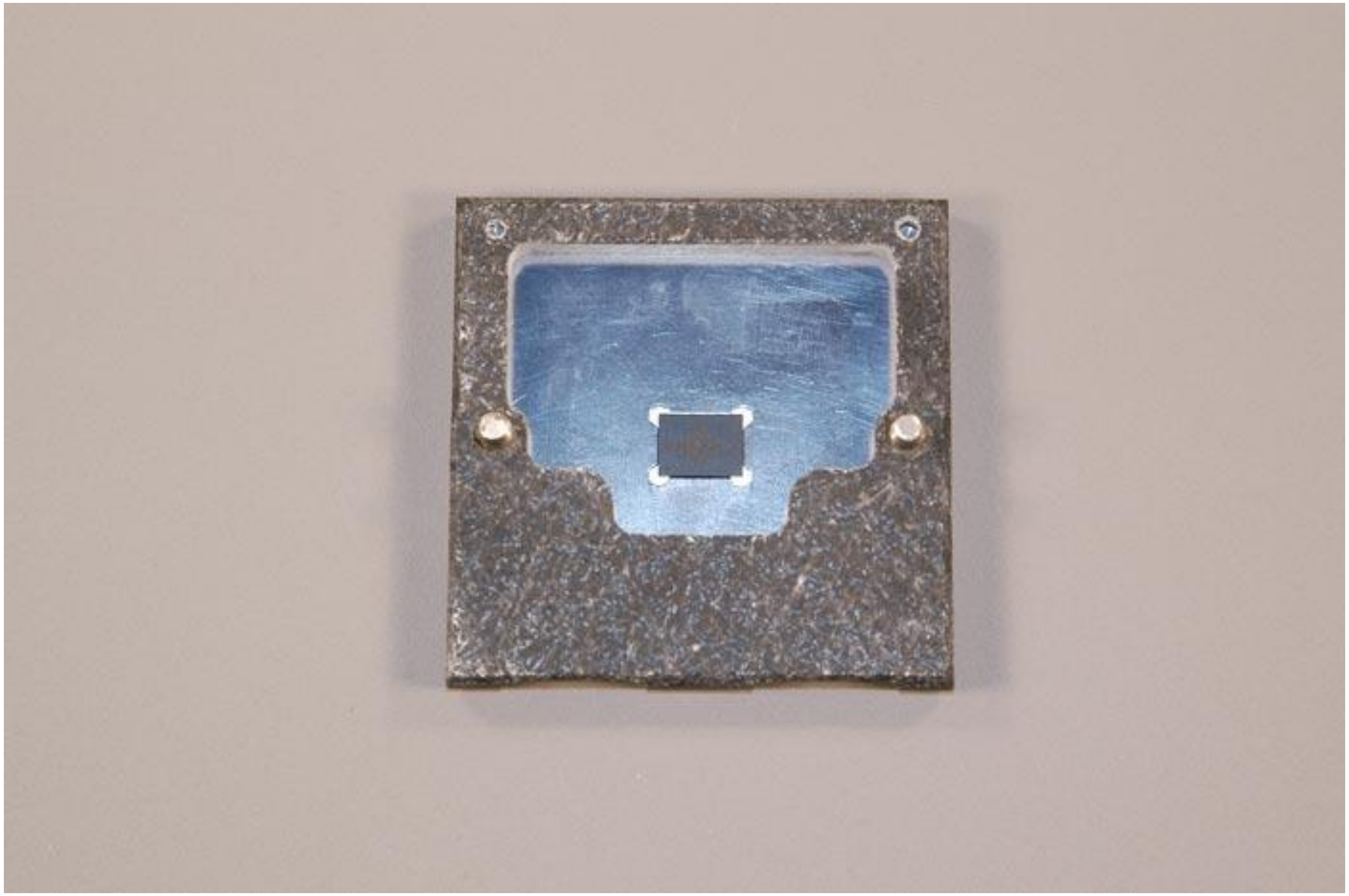
Package Holding Fixture



Stencil placed on alignment pins



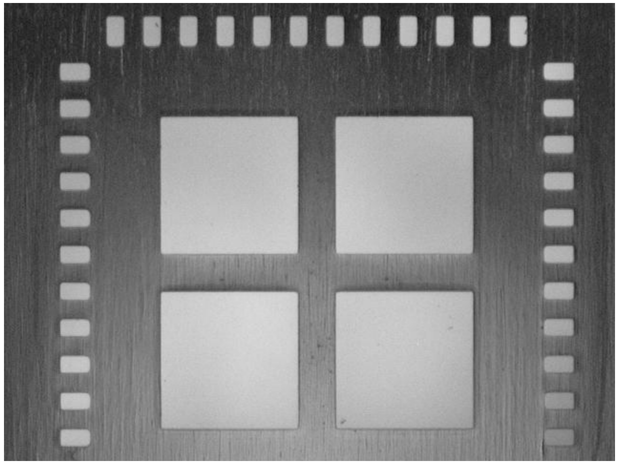
QFN placed in Package Holder



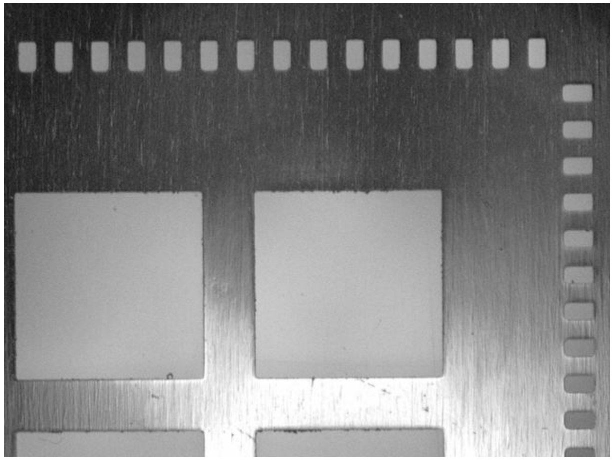
Package Hold-Down fixture is snapped into place

QFN held in place just prior to printing solder paste

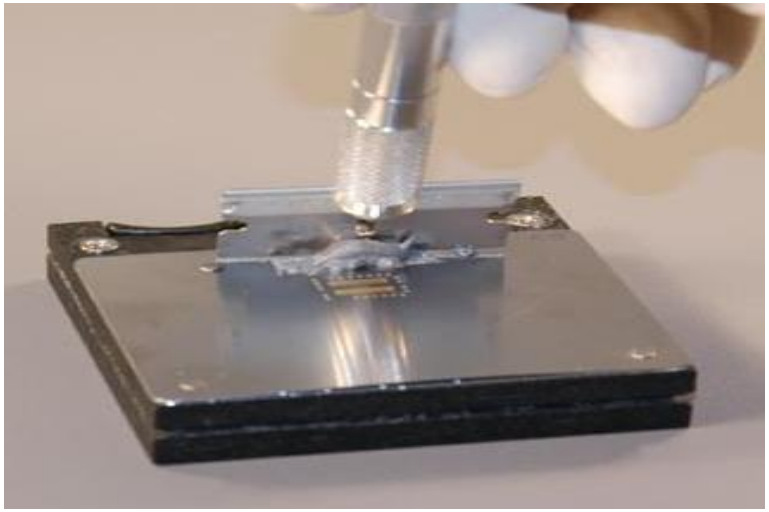
7mm repair stencil



10mm repair stencil

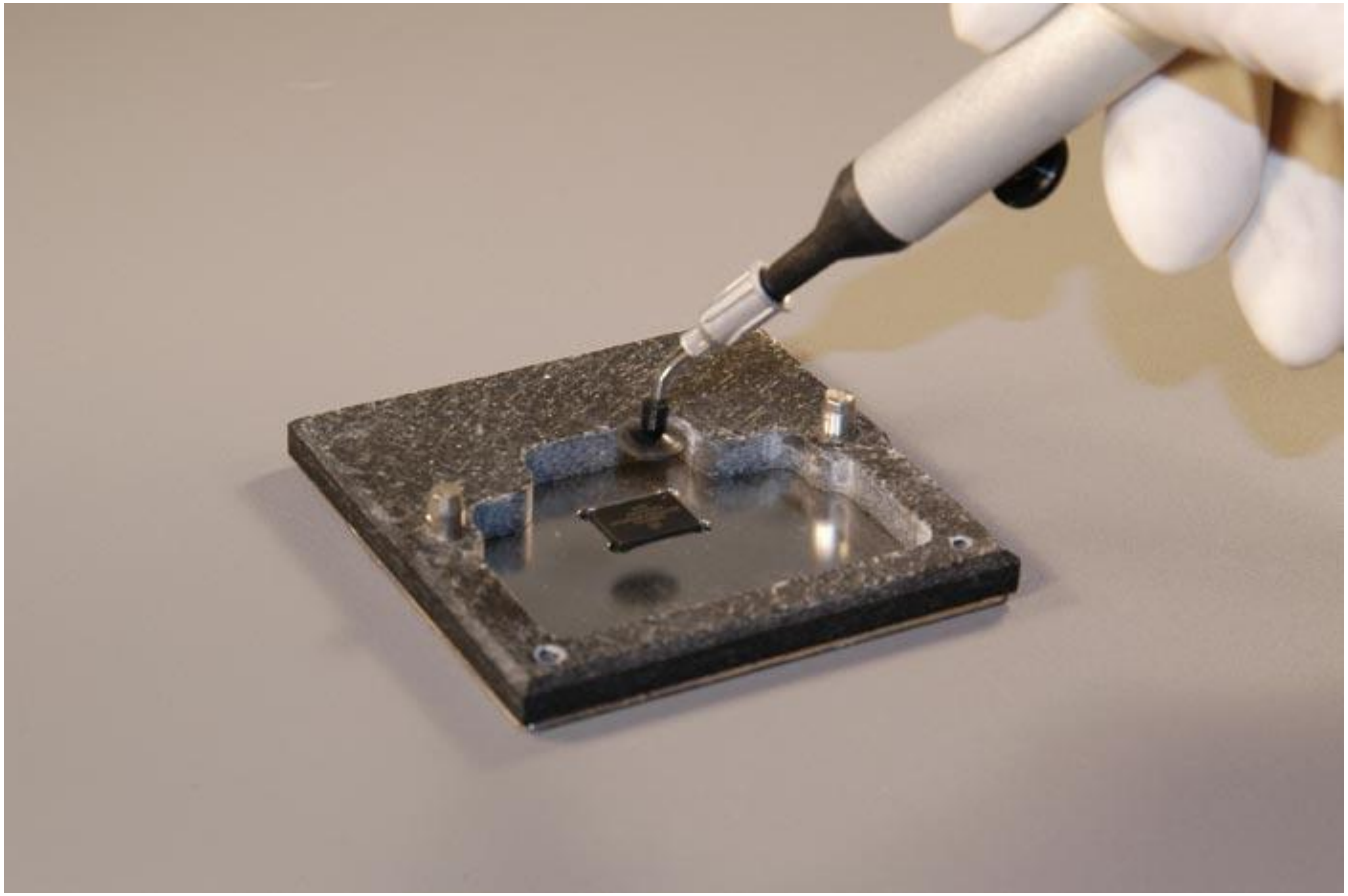


Printing Solder Paste on QFN device while in holding tool

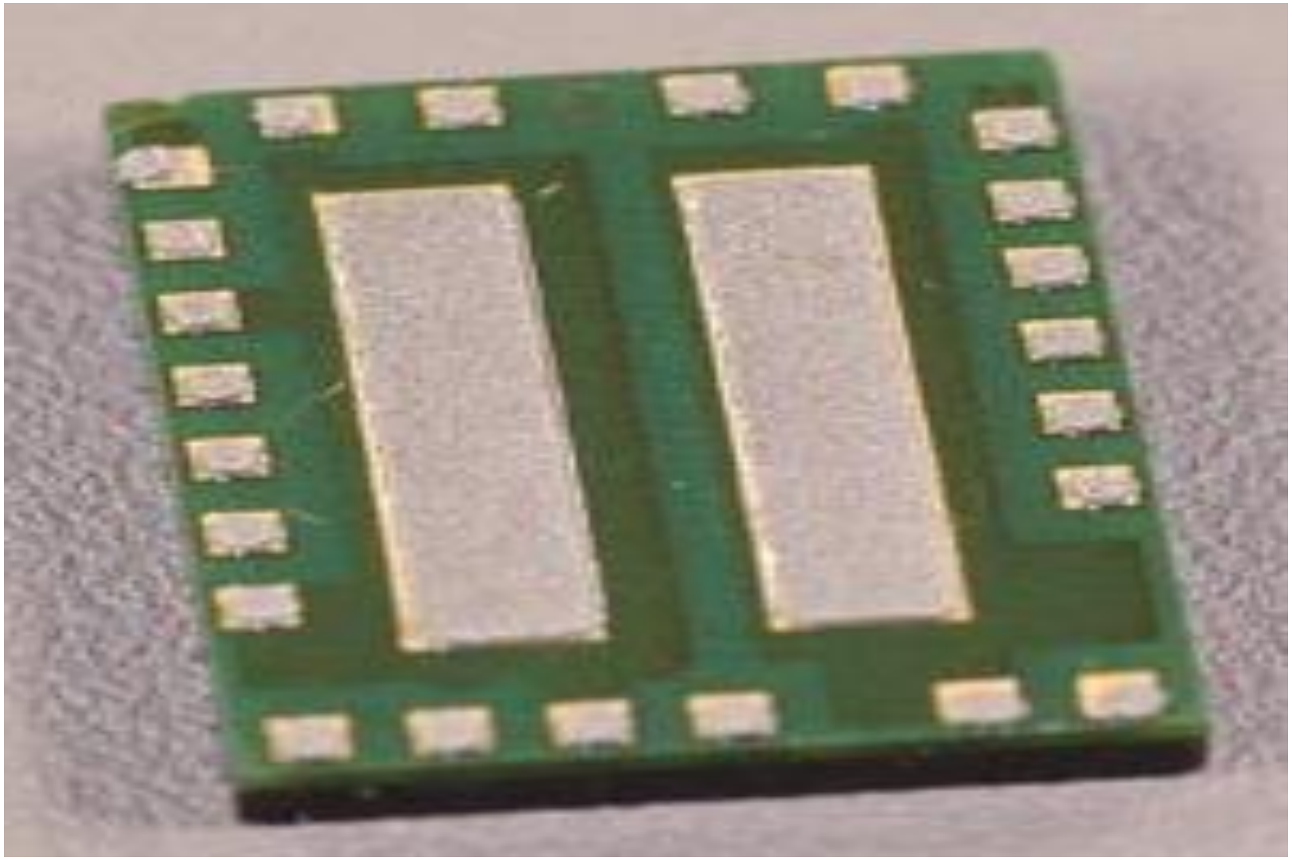


After Solder Paste Printing

Package Hold-Down fixture is removed and Vacuum Pick places QFN on PCB



Solder paste printed on QFN before placement on PCB for rework



Conclusion:

- Although QFN devices present a challenge to the SMT assembly process with proper stencil design, proper stencil technology selection (Laser, Electroform, Nano-Coat), and proper PCB solder mask layout these challenges can be overcome.

- The most popular QFN repair seems to be to print solder paste directly onto the QFN leads and ground plane.

References:

•Private communication with Greg Kloiber, Manufacturing Engineer Plexus Corp.

•"BGA and QFN Repair Process" William E. Coleman, APEX 2008

General QFN References
•"Rule of thumb guide for Practical DFN/QFN Printed Circuit Board and stencil Design" James R. Staley, Linear Technology application note

•"PCB Land Pattern Design and Surface Mount guidelines for QFN Packages", Intersil Technical Brief TB389.6

"QFN Layout Guideline", Texas Instrument Application Report SLOA122

Thank You

Questions??

Contact Information

Email: rshort@photostencil.com