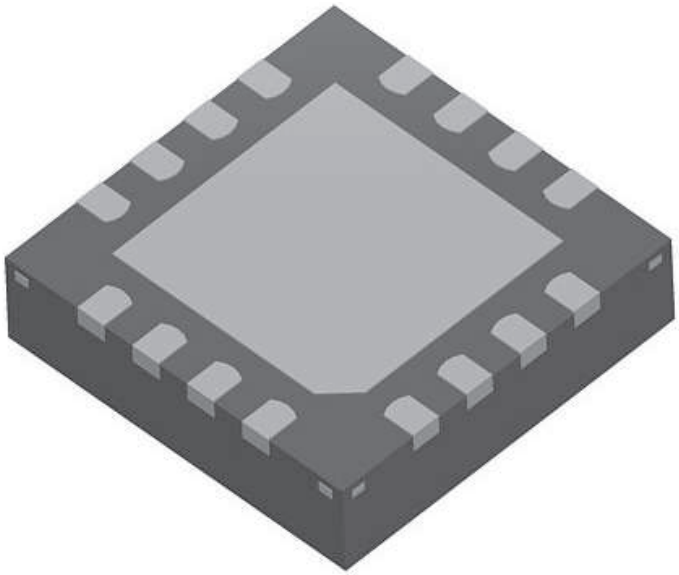
Application Report
SLUA271A–June 2002–Revised September 2007

QFN/SON PCB Attachment

Steve Kummerl, Bernhard Lange, Dominic Nguyen

ABSTRACT

Quad flatpack no leads (QFNs) and small-outline no leads (SONs) are leadless packages with electrical connections made via lands on the bottom side of the component to the surface of the connecting substrate (PCB, ceramic). This application report presents users with introductory information about attaching QFN/SON devices to printed circuit boards (PCBs).



Contents

List of Figures

List of Tables

1 Texas Instruments Quad Flatpack No Leads and Small-Outline No Leads

1.1 Introduction

Quad flat no leads (QFNs) and small-outline no leads (SONs) are thermally enhanced plastic packages that use conventional copper leadframe technology. This construction results in a cost-effective advanced packaging solution that helps to maximize board space with improved electrical and thermal performance over traditional leaded packages.

QFNs have solder lands on all four sides of the package. SONs typically have solder lands on two sides of the package.

QFN/SONs are available in a number of formats. QFN/SONs are molded and mechanically singulated from a matrix leadframe. Package size is determined by several key factors including die size, number of terminations, etc.

All QFN/SONs are leadless packages with electrical connections made via lands on the bottom side of the component to the surface of the connecting substrate [printed circuit boards (PCB), ceramic]. The standard QFN/SON package has an exposed pad that enhances the thermal and electrical characteristics, enabling high-power and high-frequency applications.

Note: This QFN/SON PCB application report is intended as a guide. Precise process development and experimentation are needed to optimize specific applications/performance.

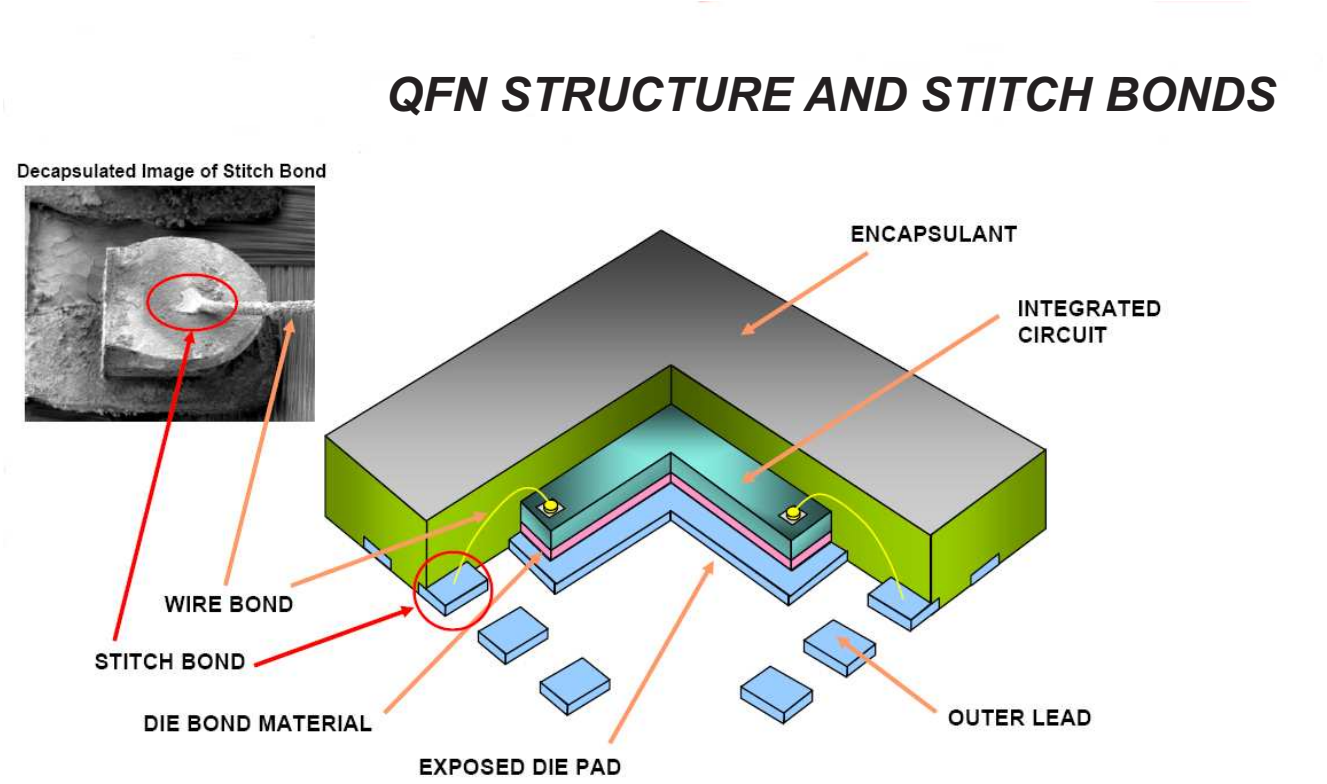


Figure 1. QFN Structure

2 Manufacturing Considerations

2.1 SMT Process

Many factors contribute to a high-yielding assembly process. A few of the key focus areas and their contributing factors are highlighted in Table 1.

Table 1. Essentials for Assembly Quality

Solder paste quality	Uniform viscosity and texture. Free from foreign material. Solder paste must be used before the expiration date. Shipment and storage temperatures must be maintained at the proper temperature. Paste must be protected from drying out on the solder stencil.
PCB quality	Clean, flat, plated, or coated solder land area. Attachment surface must be clean and free of solder-mask residue.
Placement accuracy	Tight tolerances are not usually required. QFN/SON packages self center as long as a major portion (more than 50 percent) of the lead finger is in contact with the solder-paste-covered land area on the board. Alignment marks (fiducials) on the PCB are helpful for verifying correct placement of parts.
Solder reflow profile	The solder reflow temperature is dependent on the PCB design, PCB thickness, and peak reflow temperature according to the moisture sensitivity level (MSL) of components, component density, and the recommended profile of the solder paste being used. A reflow profile must be developed for each PCB type using various QFN/SON packages. See the reflow profile in the solder reflow section (5.2). (see Figure 13).
Solder volume	Solder volume is important to ensure optimum contact of all intended solder connections.

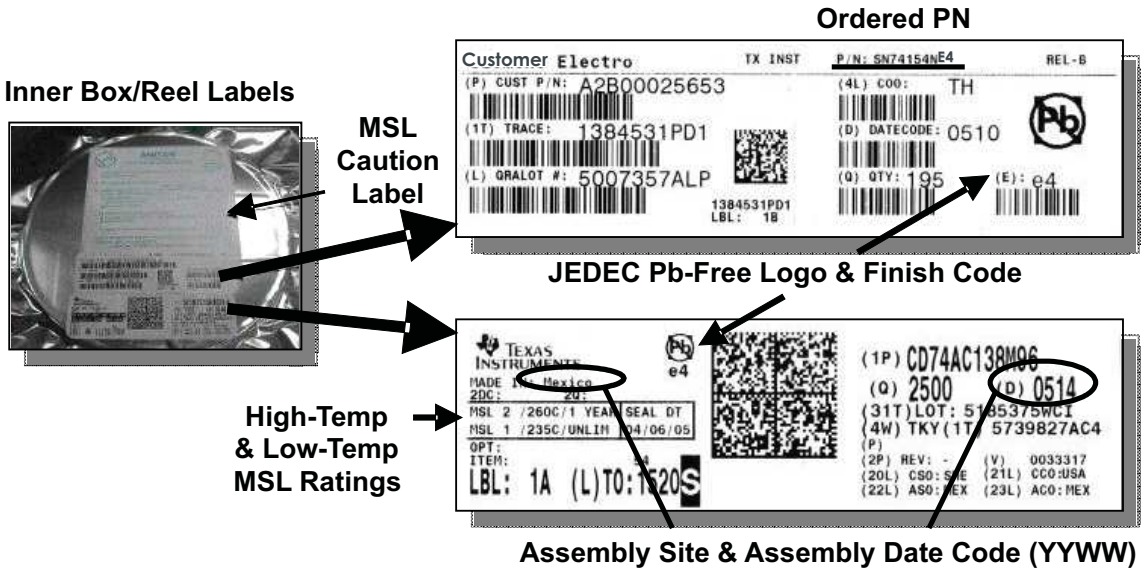


Figure 2. Packing Material Label Information With Moisture Sensitivity Level (MSL)

3 Printed Circuit Board (PCB) Design Guidelines

One of the key efforts in implementing the QFN/SON package on a substrate motherboard is the design of the land pad. The QFN/SON has lead fingers exposed on the bottom side of the package. Electrical and mechanical connections between the component and motherboard can be made by soldering the part using screen printed solder paste and reflowing after placement. To ensure consistent solder joint geometries, it is essential to design the land pattern considering the component exposed leadframe pattern.

3.1 Land Pad Styles

There are two basic designs for PCB land pads for the QFN/SON package—the copper defined or non-solder mask defined style (NSMD) and the solder mask defined style (SMD). The industry has debated the merits of both styles of land pads and although TI recommends the copper defined style land pad (NSMD), both styles are acceptable for use with the QFN/SON package.

NSMD pads are recommended over SMD pads due to the tighter tolerance on copper etching compared to solder masking. In addition, the NSMD pads help to contain the solder paste within the solder mask opening. NSDM, by definition, also provides a larger copper pad area and allows the solder to anchor to the edges of the copper pads, thus providing an increase in wettable surface area.

3.2 Land Pad Design

IPC-7351 is one of the industry standard guidelines for developing PCB pad patterns. Because the QFN/SON is a new package style, this application report is intended as a guide and should be used with the IPC-7351 in designing an optimum PCB land pattern.

Figure 3 identifies the various QFN/SON dimensions required to design a matching substrate pad pattern. Because most packages are square with dimension D = E and the leads are along the E direction for dual packages, the side-view dimensions (D, S, D2 and L) are used to determine the land length on the motherboard PCB/substrate. Figure 4 shows the motherboard PCB/substrate land pattern dimensions to be established. Table 2 offers a description of these dimensions.

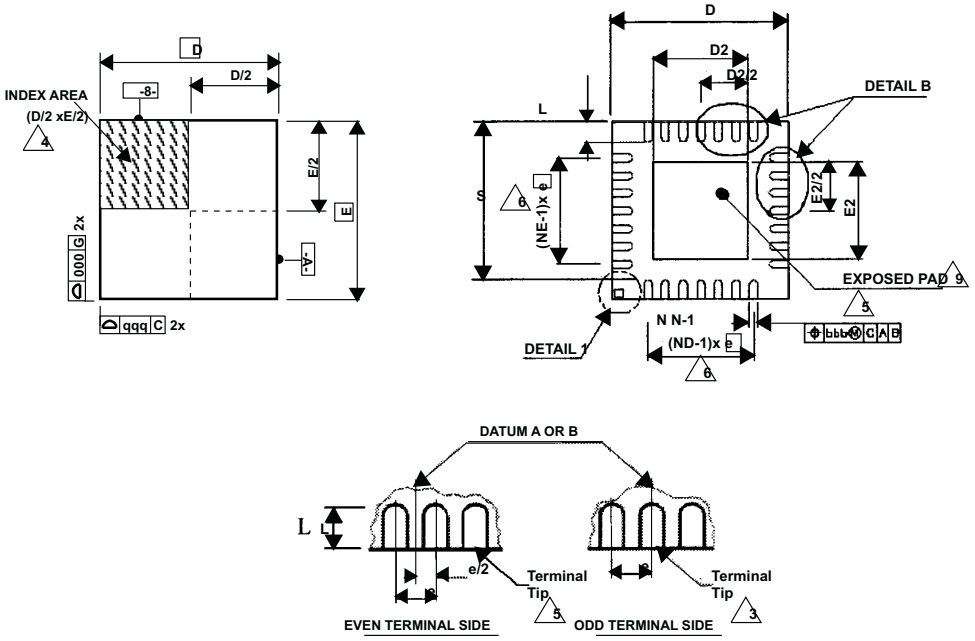


Figure 3. QFN/SON Outline Dimensions

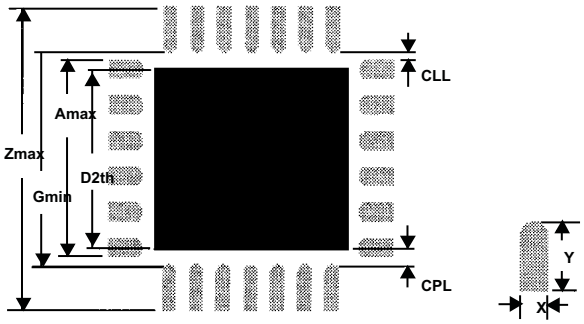


Figure 4. PCB Land Pattern

Table 2. PCB Dimension Definitions

SYMBOL	DESCRIPTION
A_{max}	Outer extents of pad widths per each side
Z_{max} (1)	Outside pad terminal dimension typically 0.8 mm larger than the package body as a starting point
G_{min} (1)	Inside pad terminal dimension
X	Lead pad width
Y	Lead pad height
C_{LL}	Corner pad edge to adjacent inside pad distance
C_{PL}	Central pad to inside edge of lead pad distance which should not be reduced below 0.2 mm

(1) The dimensions Z_{max} and G_{min} are the outside and inside pad terminal dimensions, respectively. X and Y identify the pad width and length, respectively. Clearance dimensions C_{LL} and C_{PL} are defined to prevent solder bridging.

3.3 Lead Finger Pad PCB Design

It is recommended that the PCB lead finger pad be designed a minimum of 0,1 mm longer than the package land length (also known as the toe length) and be extended 0,05 mm towards the center line of the package (also known as the heel). TI uses a typical value of 0,4 mm toe length beyond the package body as a standard to optimize solder volume which should be considered as a model starting point for board designs. The PCB pad width must be a minimum 0,05 mm (0,025 mm per side) wider than the terminal width on the package (see Figure 5). However, to avoid solder bridging for components having lead pitches of 0,5 mm, the pad width should be reduced to 0,28mm or smaller. The 0,28 mm width was based on a 0,5mm pitch component having a max terminal width of 0,3mm per the mechanical drawing. In the event that a board supplier can't achieve a solder mask web between 0,28mm wide pads, the width should be reduced to accommodate the board supplier's solder-mask web tolerances. For board designs requiring the minimum land pattern, the stencil design parameters of aspect ratio and area ratio need to be considered in order to have a manufacturable land pattern design. Stencils typically are designed with a relationship to the land pattern, so both must be considered closely when minimizing. IPC-7525 outlines the necessary parameters when designing the stencil.

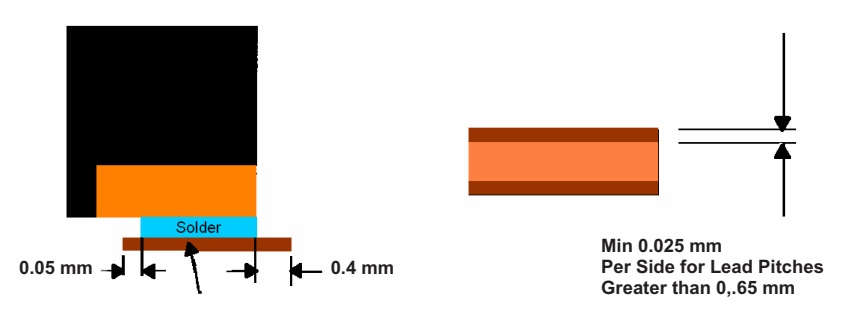


Figure 5. Substrate/PCB Lead Finger Geometry

3.4 Exposed Pad PCB Design

The construction of the exposed pad enables enhanced thermal and electrical characteristics. In order to take full advantage of this feature, the pad must be physically connected to the PCB substrate with solder. The published data for thermal performance, θ_{JA}, is based on a four-layer PCB incorporating vias that act as the thermal path to the layers, also known as a JEDEC High K board design per JESD51-7.

The dimensions of the thermal pad on the PCB should be equal to the exposed pad on the QFN/SON. Adequate clearance (CPL) is necessary to prevent solder bridging. Experiments have concluded that a minimum clearance of 0,2 mm is satisfactory for most designs.

3.4.1 Thermal Pad Via Design

The number of vias represented in TI example land patterns located at the back of every data sheet should be considered as an example starting point. Not all applications require vias. This is dependent on the amount of power the device dissipates. If the board routing becomes too challenging to employ the use of thermal vias, contact your local TI representative for further guidance. The center thermal pad, however, should always be soldered to the board for mechanical strength and reliability. For thermally challenging applications, it is recommended that the thermal vias be placed on a pitch of approximately 1,0 mm. Per standard PCB manufacturing capabilities, 0,3 mm diameter drill holes are recommended as a starting point, but a smaller via offers less risk of solder volume loss. On applications where solder volume loss thru the vias is of concern, plugging or tenting can be used to achieve a repeatable process.

Vias that are plugged eliminate the risk of solder volume loss. If plugging the via is not economically suitable, tenting also can offer benefit. Tenting from the back or topside of the board may cause processing issues with some printed circuit board manufacturers due to chemistry entrapment during the plating process. In the case of plugging or tenting from the backside of the board, the voiding potential may increase due to the air trapped within the tented via. Check with your board fabrication site for recommendations.

Another solder-mask tenting technique is to tent from the top of the board. The via solder-mask diameter must be 0,1 mm larger than the via hole diameter when tenting. Another variation is to create a cross-hatch pattern of solder mask to create a predictable area of coverage. Trials have shown that via tenting from the top is less likely to produce random voids between the exposed pad and the PCB pad. See Figure 6 for x-ray results of various via tenting structures.

For finishes that use an organic solder preserve (OSP), not tenting any vias from the top or bottom of board has shown repeatable performance in soldering. In addition, optimizing the profile to achieve maximum flux activity during the critical reflow stage reduces the amount of voiding seen. Trials have shown that an excessive soak time, which activates the majority of the flux prior to the melting temperature of the alloy, results in large voids. Check with your solder paste manufacturer if an alternative profile can offer more flux activity through the critical melting phase of the alloy used.

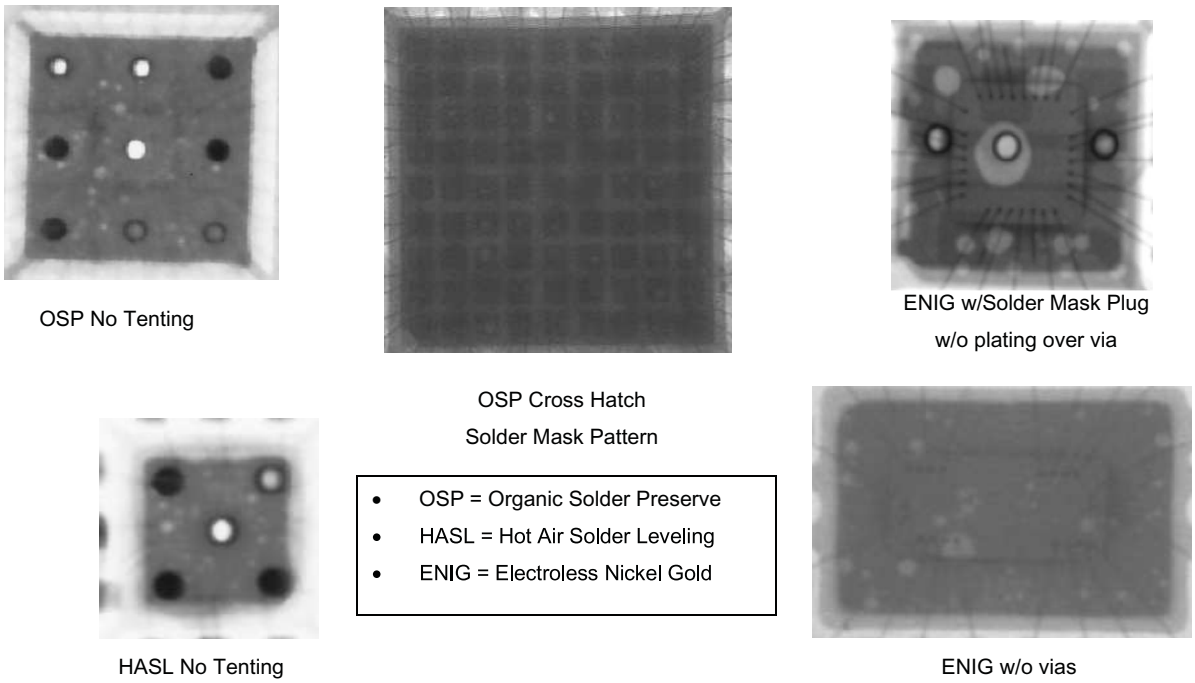


Figure 6. X-Ray Images for Reference

3.5 Solder Mask

As described at the beginning of this section, a non-solder mask defined pad (NSMD) is recommended over solder mask defined (SMD) to produce consistent solder joint geometries. The solder mask can be designed around each individual lead finger for lead pitches 0,65 mm and above. Check with your board fabrication site for solder-mask openings, but typically the NSMD openings are 0,1 mm to 0,14 mm larger than the lead finger pad size. For a lead pitch of 0,5 mm, it is recommended to design the solder mask around all pads on each side with a clearance of 0,05 mm or smaller, so consult your board fabrication site for tolerance requirements.

To maximize the solder mask in the corner regions, it is necessary to round the inner corner on each row (see Figure 7). This ensures sufficient solder mask in the corner of the PCB footprint design and also prevents the metal feature from encroaching beyond an air gap of 0,2 mm. In addition, it has been observed that some QFN configurations utilize an exposed metal pin 1 feature beneath the component, which has a potential of shorting to a via if present in this region. Therefore, TI recommends maintaining a routing and via keep-out area by pin 1 on all QFN designs to guard against this risk and offer more flexibility when selecting different suppliers (see Figure 7).

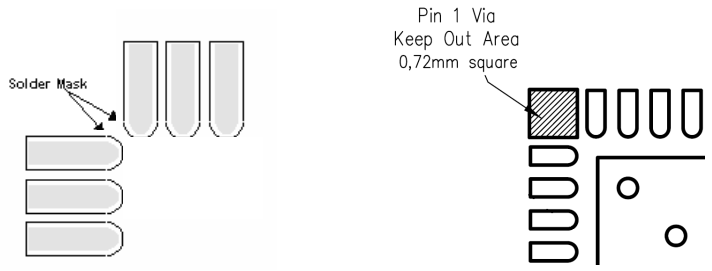


Figure 7. Substrate/PCB Solder Mask and Keep-Out Area Example

3.6 Surface Finishes

There are a variety of surface finishes commonly available. 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 overall surface mount yields. Bare copper with an organic solderability preservative (OSP) coating, electroless nickel/immersion gold, or electroplated nickel/gold finishes have been shown to provide an acceptable land pad surface. One type of surface finish that has shown irregular processing is referred to as a "dry film process." This is because the copper undercut effect caused during the dry film removal prevents optimal sidewall wetting during the reflow process.

The advantages of plating over OSPs are:

- Shelf life
- Permanent coverage of copper vias and other features not exposed to a solder process, and
- Contamination resistance

Even with these differences, OSPs have shown robust performance in industry. In summary, a controlled assembly process for QFN/SON soldering relies on a flat uniform attachment site. Achieving a flat uniform surface leads to a greater control of solder-paste print uniformity, resulting in an overall robust process.

3.7 Board Layout Considerations

There are a wide variety of QFN packages on the market that have varying thicknesses. Location of the thinner format QFN packages should be carefully considered when laying out the board design to avoid regions of extreme deflection thru manufacture (see Figure 8). Excessive bending of the substrate may lead to package damage and should be avoided in the assembly flow.

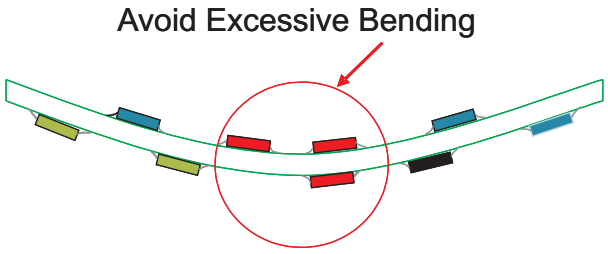


Figure 8. Avoid Excessive Bending

4 Solder Paste Screen Printing Process

4.1 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 solder (SN63/Pb37 or SAC alloy) paste is commonly used in mounting QFN/SON, however, water-soluble flux materials may be used as well if the residues can be adequately cleaned from the board. 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. Special SMD-specific solder pastes are being marketed by paste vendors that minimize voiding in the solder joint. If low-standoff parts such as CSPs and QFNs are to be used, highly aggressive solder fluxes are not recommended unless they can be cleaned from underneath the parts. TI recommends that the solder-paste manufacturer's recommended temperature profile be used to optimize flux activity within the MSL (moisture sensitivity level) guidelines for the most thermally sensitive component. Reference J-STD-033 for more details on MSL classification.

4.2 Solder Stencils

The formation of consistent solder joints is a necessity. The contrast between a large exposed pad and small lead fingers of the QFN/SON can present a challenge in producing an even standoff height. To this end, careful consideration must be applied to the stencil design.

The stencil thickness, as well as the stencil opening 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 nickel buildup or stainless steel, with a nickel buildup stencil offering a smoother side wall over a laser-cut stainless steel stencil. Apertures are typically trapezoidal, which helps to ensure uniform release of the solder paste and reduce smearing (see Figure 9).

The solder-joint thickness of QFN/SON lead fingers is typically between 0,050 mm to 0,075 mm, which has a direct relationship to the amount of solder printed on the center thermal pad area. Thickness of stencils used in manufacturing varies from 0,100 mm to 0,150 mm (0.004 in to 0.006 in) range, with a typical 0,125 mm stencil design for 0,5 mm pitch components. The actual thickness of a stencil is dependent on other surface-mount devices on the PCB, in addition to the area and aspect ratio of the minimum aperture used.

Use a squeegee with a durometer of 95 or harder (such as stainless steel). The blade angle and speed must be fine tuned to ensure an even paste transfer. An inspection of the printed solder paste is recommended before placing parts. A repeatable solder deposit is the most important factor for robust reflow yields further downstream in the process. As a guide, a stencil thickness of 0,125 mm (0.005 in.) for QFN/SON components is recommended. IPC-7525 outlines the necessary parameters to consider when designing the stencil.

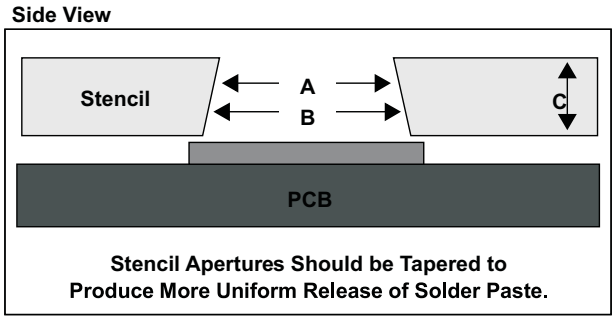


Figure 9. Solder Stencil Profile

4.3 Lead Finger Stencil Design

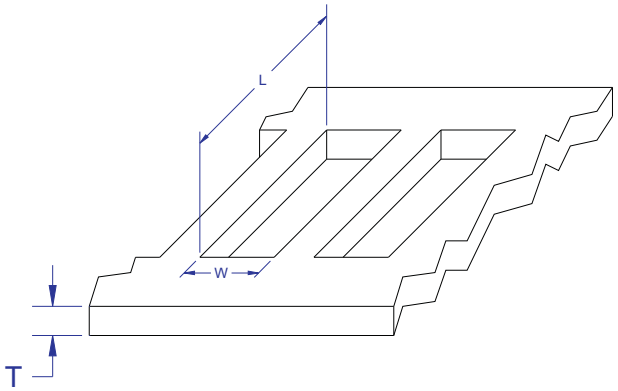
The stencil aperture is typically designed to match the PCB/substrate pad size, i.e., 1:1 (Note: 1:1 is recommended on periphery lead fingers and not the center thermal pad – see Section 4.4 for exposed pad recommendations). For fine-pitch QFN/SON components of 0,5 mm and below it may be necessary to reduce the stencil aperture by 20% to prevent shorting beneath the QFN, paying special attention that the area or aspect ratios are not exceeded. This reduction is recommended in cases where the SMT equipment placement force is not well regulated, leading to shorting from excessive solder-paste squeeze out. For 0,4 mm pitch components, a pad width of 0,2 mm is recommended to aid solder-paste printing and to achieve a gap of 0,2 mm between pads.

Lead finger stencil dimensions depend on the specific QFN/SON lead finger dimensions. For example, on a 0,5 mm pitch device with 0,85 mm x 0,28 mm wide pads, a stencil aperture of 0,23 mm x 0,8 mm and thickness of 0,125 mm should be used to achieve adequate print volume and area ratio requirements as outlined in IPC-7525 (see Figure 10). In cases where the board land pattern must be minimized, the stencil aspect and area ratios should be considered prior to board design.

The area ratio of the stencil is critical for the printing to get good paste release. For very small apertures where the area ratio is less than 0.66, the stencil must be nickel formed. This type of stencil has superior release characteristics over stencils that have been produced by laser. Experiments have shown that nickel-formed stencils print with area ratios down to 0.57. Check with your stencil supplier for recommendations when designing an aperture with a challenging area ratio.

The aspect ratio relates to the manufacture of stencils. Stencil manufacturers require the aspect ratios (see Figure 10) to be greater than 1.5 (see IPC-7525).

The higher the area ratio, the better the solder paste will release, in addition to depositing more volume. Stencil thickness is inversely proportional to the area ratio. So the thinner the stencil, the higher the area ratio will be, ultimately resulting in a robust solder-paste release. (Ref IPC-7525)



$$\text{Aspect Ratio} = \frac{\text{Width of Aperture Opening}}{\text{Stencil Foil Thickness}} = \frac{W}{T}$$

$$\text{Area Ratio} = \frac{\text{Area of Aperture Opening}}{\text{Area of Aperture Walls}} = \frac{W \times L}{2 \times (L + W) \times T}$$

Figure 10. Aspect and Area Ratio Illustration

4.4 Exposed-Pad Stencil Design

The QFN/SON package is thermally and electrically efficient due to the exposed die attach pad on the under side of the package. The exposed die must be soldered down to the PCB or motherboard substrate.

It is good practice to minimize the presence of voids within the exposed pad interconnection. Total elimination is difficult, but the design of the exposed pad stencil is crucial. The proposed stencil design enables out-gassing of the solder paste during reflow and also regulates the finished solder thickness. Typically, the solder-paste coverage is approximately 50% to 70% of the pad area (see Figure 11). Designing an aperture that prints solder 1:1 with the exposed pad results in excessive metal volume that will “float” the part, causing opens and other manufacturing defects. In addition, the amount of voiding post reflow in the thermal pad solder joint should not exceed 50% in high power applications (to be verified using an x-ray). Based on a JEDEC High K board stackup, 25% has been determined to be a point of diminishing thermal performance returns, but TI prefers a limit of 50% to be set (reference JESD51-7).

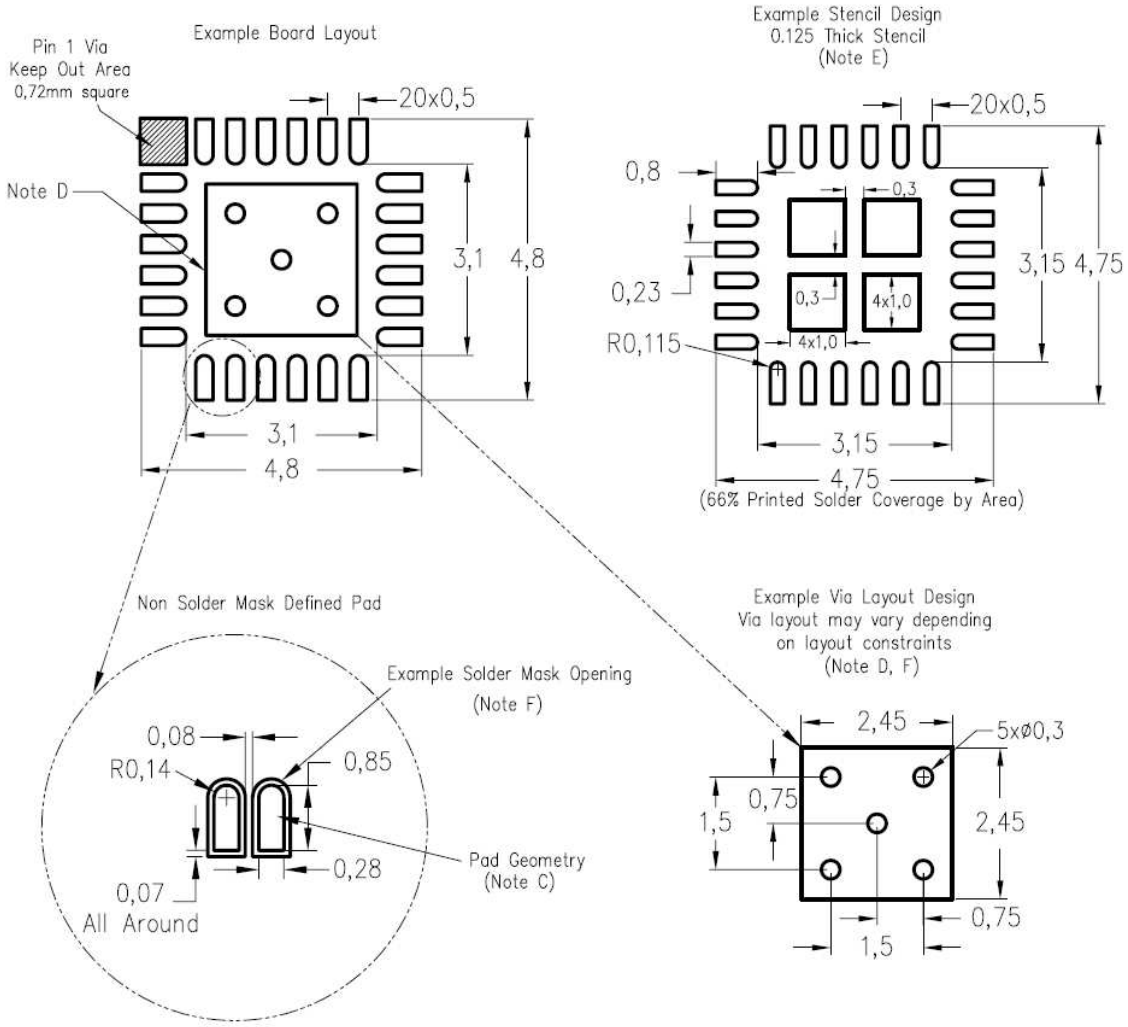


Figure 11. Example Land Pattern and Exposed-Pad Stencil Design

5 Package to Board Assembly Process

5.1 Placement and Alignment

The pick and place accuracy governs the package placement and rotational (theta) alignment. This is equipment/process dependent. Slightly misaligned parts (less than 50 percent off the pad center) automatically self align during reflow (see Figure 12).

Grossly misaligned packages (greater than 50 percent off pad center) must be removed prior to reflow, as they may develop electrical shorts resulting from solder bridges, if they are subjected to reflow. There are two popular methods for package alignment using machine vision:

- Package silhouette, also called “back lighting.” The vision system locates the package outline.
- Lead-frame recognition, also called “front lighting.” Some vision systems can directly locate the lead-frame pattern.

Both methods are acceptable for QFN/SON placement. The front lighting recognition type alignment tends to be more accurate, but is also slower because more complex vision processing is required of the pick and place machine.

The package silhouette method allows the pick and place system to run faster, but is generally less accurate. Both methods are acceptable and have been successfully demonstrated by major pick and place equipment vendors and contract assembly houses.

A starting placement force of 1.5N is recommended and should be minimized where possible. After a placement force is selected all four sides of the QFN package should be inspected to insure each side is seated in the solder paste. Excessive pressure may cause shorting due to solder squeeze out from underneath the part and in extreme cases may potentially crack the package. Location of the thinner QFN packages should be carefully considered. Excessive bending of the substrate may also lead to package damage and should be avoided in the assembly flow.

5.2 Solder Reflow

There are no special requirements necessary when reflowing QFN/SON 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 board. Component temperatures may vary because of surrounding components, location of the device on the board, and package densities.

To maximize the self-alignment effect of QFN/SON, it is recommended that the maximum reflow temperature specified for the solder paste not be exceeded. A good guide is to subject the PCB to a temperature ramp not exceeding 4°C per second.

The reflow profile guidelines (see Figure 13) are based on the temperature at the actual solder-pad to PCB land-pad solder joint location. The actual temperature at the solder joint is often different than the temperature settings in the reflow/rework system, due to the location of the system thermocouple used to monitor the temperature.

Specific production reflow and rework systems vary depending on manufacturer and model. Therefore, system specific profiles must be established using thermocouples at the actual solder joint locations.

TI has tested and qualified QFN/SONs for three reflow operations per JEDEC JSTD-020. This allows one reflow operation per side of the PCB (assuming the use of a double-sided PCB) and one rework operation if necessary.

TI recommends that the solder-paste manufacturer’s temperature profile be used to optimize flux activity within the MSL guidelines for the most thermally sensitive component. Reference J-STD-033 for more details on MSL classification.

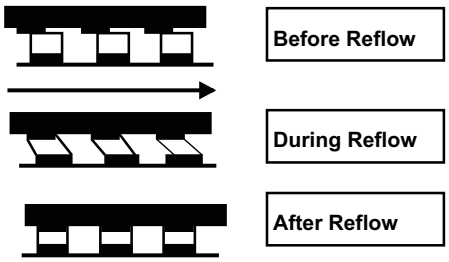


Figure 12. Package Self Alignment at Reflow

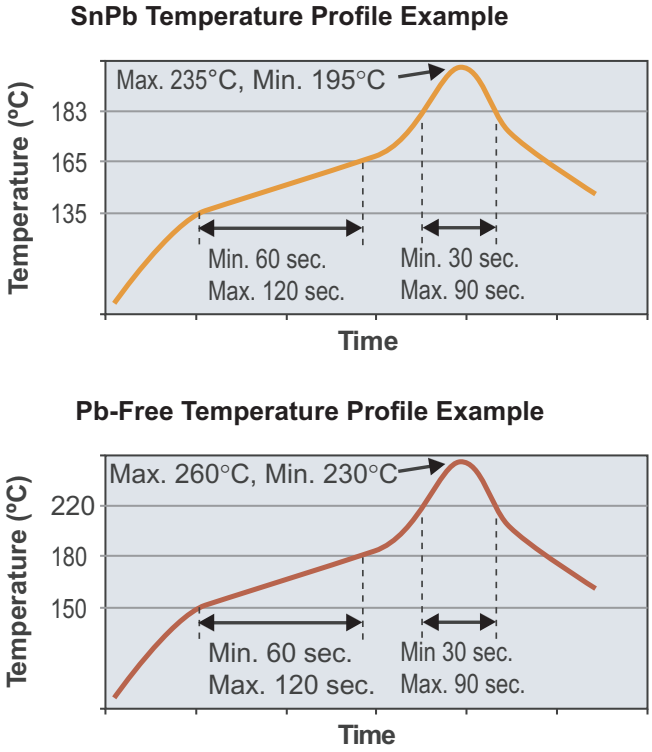


Figure 13. QFN/SON Pb and Pb-Free Example Reflow Profiles

5.3 PCB Cleaning

If a low-residue, no-clean solder paste is used, PCB cleaning typically is not required. With the elimination of materials containing chlorofluorocarbon (CFC), most companies have moved to either a no-clean or aqueous flux-based system. No-clean fluxes and solders have been formulated to minimize the harmful effects the residues left on the board may cause in the form of corrosion. Because there is a wide variety of no-clean solder pastes available, application-specific evaluations must be performed to identify if any remaining residues can cause harm to the assembly. Contact your solder-paste supplier for testing performed and recommended-use conditions. The cleaning process for water-soluble fluxes should be evaluated with special attention given to cleaning underneath the QFN component. Due to the wide variety of cleaning mediums on the market outside of TI's control, each customer should discuss material interactions with their supplier prior to cleaning. In addition to cleaning, the drying of circuit boards of any residual cleaning medium must be considered to prevent potential issues such as corrosion.

5.4 Inspection

Inspection of QFN/SONs on a PCB typically is accomplished by using transmission type x-ray equipment.

In most cases, 100-percent inspection is not performed. Typically, x-ray inspection is used to establish

process parameters and then to monitor the production equipment and process. Transmission x-ray can detect bridging, shorts, opens, and solder voids. There are many different types of x-ray inspection equipment available and functionality varies. X-ray inspection system features range from manual to automated optical inspection (AOI). Different systems also provide single-dimensional or multidimensional inspection capabilities.

As explained in the Solder Reflow section of this application report, QFN/SONs self align to the land pad using surface tension during the solder reflow process. As a result, it is very unlikely that a QFN/SON is marginally misaligned. If a misalignment does occur, it is likely to be by an entire pad. This effect makes it possible to do a gross visual alignment check after the reflow. Fiducial marks on the PCB can aid visual checks of the PCB and also are useful for manually placing units during any rework.

The exterior fillet formation may vary based on manufacturing factors, such as flux activity, solder volumes, and overall standoff height. Due to factors outside of TI's control, no assurance for consistent solder fillet heights on the package sides during assembly can be made. IPC-A610D does not require a side fillet, since the side terminations are not plated. While it may appear the wetting is reduced on the side of the component, the solder joint underneath is not affected (see Figure 14).

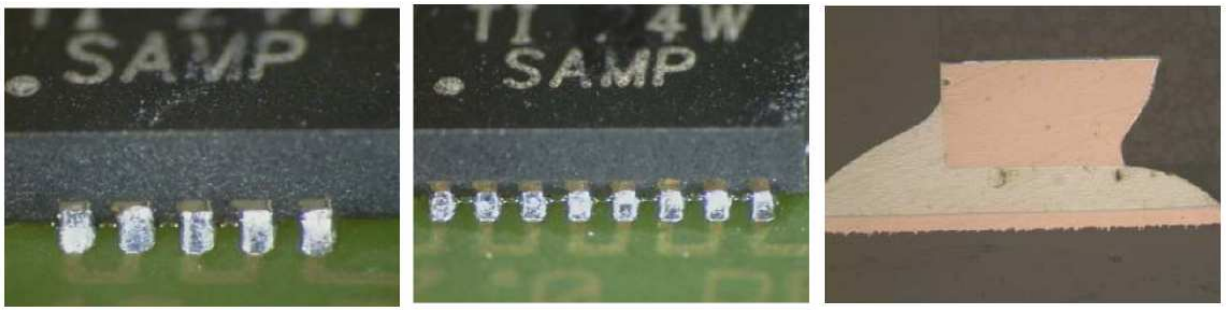


Figure 14. Illustration of Typical Fillet Formation

6 Rework Guidelines (Hot Gas Convection and Manual)

Rework equipment has continued to progress rapidly to address chip scale packages. Many manufacturers use a single rework station to incorporate multiple rework process steps, such as component removal, site redress, solder paste/flux application, alignment, component placement, and reflow. The advancement of beam-splitting imaging for alignment/placement and other areas, such as characterizing and storing individual component reflow profiles, has greatly simplified the rework process.

With the direction of QFN/SONs allowing more functionality/features on smaller products, one of the areas of concern is thermal separation of adjoining components during the rework process. Some manufacturers have addressed this concern by designing hot gas nozzles, which maintain the keep-out zone area around the rework component to thermally isolate adjacent components during the reflow process.

Original equipment manufacturers have differing requirements when it comes to solder-paste and flux-only applications during the rework process. For those who require solder paste, microstencils and squeegees have been developed. These microstencils are aligned using the same beam-splitting imaging as the component placement. Microsqueegees allow for simple, uniform solder-paste coverage across the microstencil. Section 6.3 outlines the technique of screen-printing solder to the component prior to placement.

6.1 Compound Removal

It is highly recommended to bake out the board prior to rework to reduce the risk of delaminating either the board or part. Removing the component is by far the easiest part of the rework process. Once the thermal profile is optimized, the process parameters are used to remove the device.

The gas nozzle used during this process surrounds the device and seals against the board. The QFN/SON is heated from the top side with hot gas, while residual heat is exhausted up and away from adjacent components. The anti-crushing feature in the nozzle prevents excessive top-side force from being applied to the QFN/SON. The entire assembly is also heated from the bottom side with an underboard 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. The application of flux is recommended for QFN/SON removal.

Table 3 lists generic guidelines to remove QFN packages assembled on a 0.056-in FR4 board. It is recommended to modify heating profiles for different board thicknesses and equipment used. Parts must not exceed the peak temperature as listed on the MSL label. Parts and boards must be baked out prior to rework to reduce the risk of delamination.

Table 3. Pb-Free Solder Component Removal

1.	Apply flux paste to component.
2.	Align nozzle over part to be removed.
3.	Maintain nozzle 0.050-in over device. Care must be taken to prevent over travel of the vacuum tip, which may damage the part or vacuum tip when measuring this distance.
4.	Preheat board to 90°C, nozzle warming up 20% air flow, 125°C.
5.	Soak stage–20% air flow, 225°C, 90 s
6.	Ramp stage–20% air flow, 335°C, 30 s
7.	Reflow stage–25% air flow, 370°C, 65 s
8.	Enable vacuum at the end of the reflow cycle, lower vacuum nozzle, and remove part.
9.	Cool down stage–40% air flow, 25°C, 50 s
10.	Turn off the vacuum and remove part from nozzle.
11.	Using any metal tweezers or rough handling can damage the part, and render it unanalyzable
12.	Do not reuse the part after it is removed.

6.2 Site Redress

Once the QFN/SON has been removed, the residual solder that remains on the board pads must be removed. The QFN/SON PCB site is very fragile due to its extremely small pad sizes. To avoid damaging the pads or solder mask, the site redress process must be performed very carefully. 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. The site is then inspected prior to the replacement process. Note that the removed part should not be reused. The added volume of solder on the removed part causes coplanarity issues and inconsistent solder volume. Performing site redress on the components periphery leads may also cause unforeseen damage.

6.3 Component Replacement and Reflow

The first step prior to replacement is to apply solder paste either to the board or directly to the component. Depending on the application, one of these two methods can be more advantageous, due to factors such as board density and ease of access to the rework site to be evaluated on a case-by-case basis. Both methods require the use of a mini stencil, with the following illustrations outlining the steps required to screen print solder directly onto the component. In Figure 15, the solder paste is shown applied to the part using a fixture courtesy of Photo Stencil™.

A component insertion tool is used to ensure proper registration of lightweight QFN/SONs onto the printed circuit board. In addition, the insertion tool eliminates manual handling of the component, which can deposit unwanted skin oil on the component pads.

Optical systems used for alignment consist of a beam-splitting prism combined with an inspection microscope or video camera fitted with a zoom lens. This optical system allows the operator to see a magnified image of the bottom side of the QFN/SON superimposed over the corresponding PCB land pattern.

Alignment and placement of the QFN/SON must be accurate to within 0,1 mm. When the QFN/SON is correctly aligned, the X-Y table is locked to prevent further movement. The nozzle (Figure 16) is lowered until it lightly contacts the board. The nozzle vacuum is automatically deactivated and the thermal reflow cycle begins. PC-based software provides the process control necessary to ensure repeatable results. Once the cool-down stage is complete, the nozzle is raised and the assembly is removed for inspection.

Figure 15 shows generic guidelines to remove QFN packages assembled on a 0.056-in FR4 board. It is recommended to modify heating profiles for different board thicknesses and equipment used. Parts must not exceed the peak temperature as listed on the MSL label. Parts and boards must be baked out prior to rework to reduce the risk of delamination.

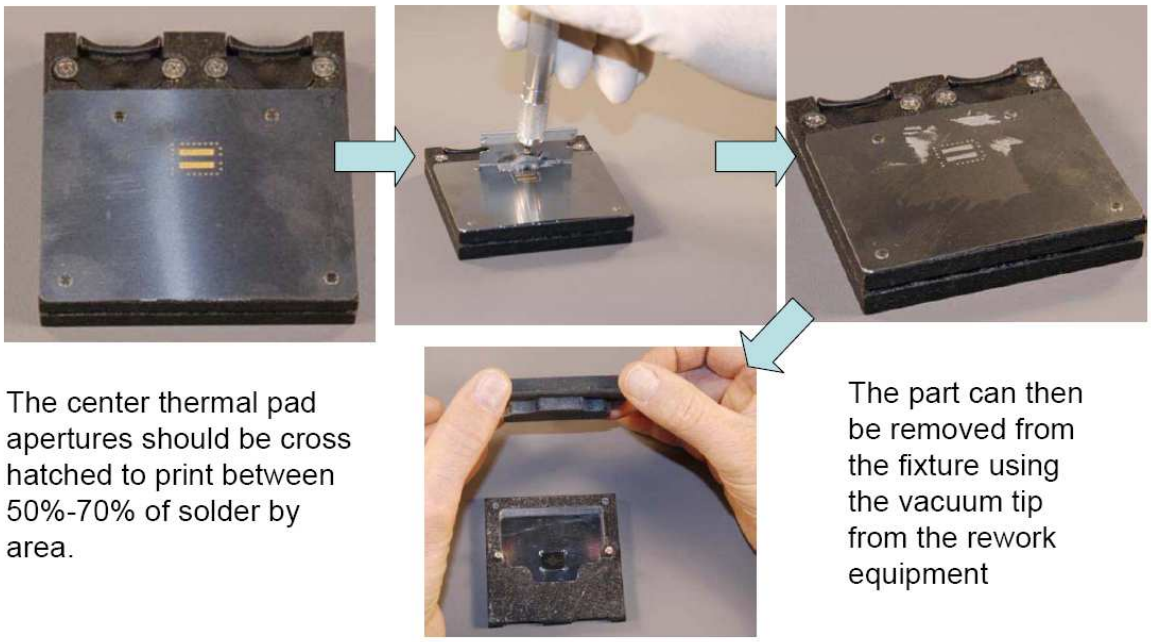


Figure 15. Example Mini Stencil

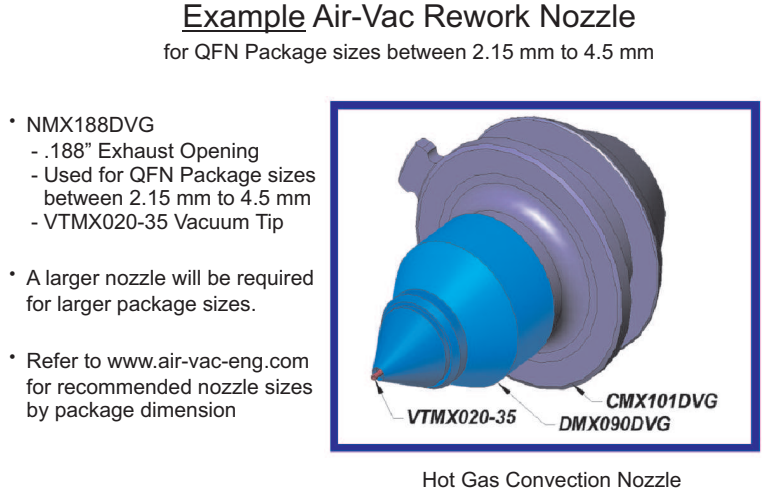


Figure 16. Example Hot Gas Convection Nozzle

Table 4. Pb-Free Solder Component Placement

1.	Apply solder paste to component or board using mini-screen printing tool shown in Figure 18.
2.	Align device over pads.
3.	Place device on board. Care must be taken to prevent over travel during placement, which may damage the part or vacuum tip.
4.	Raise nozzle 0.050-in.
5.	Preheat board to 90°C, nozzle warming up 20% air flow, 125°C.
6.	Soak stage–20% air flow, 225°C, 90 s
7.	Ramp stage–20% air flow, 335°C, 30 s
8.	Reflow stage–25% air flow, 370°C, 65 s
9.	Cool down stage–40% air flow, 25°C, 50 s

6.4 Manual Rework Considerations

If manual touchup is required, it is highly recommended to bake out the board and replacement components prior to rework, which reduces the risk of delaminating either the board or part. Using a pre-heater (see Figure 17) is also strongly advised to reduce the risk of temperature overshoot. By using a pre-heater, the soldering iron tip size and potential for temperature overshoot can be reduced, resulting in a robust manual soldering iron process.

IPC7711 recommends that the lowest possible tip temperature should be used initially. It is also recommended that the smallest tip size be used, which mitigates temperature overshoot and reduces the risk of delaminating either the board or component.

Damage can result when a soldering iron comes in contact with the periphery lead, and pressure is applied to the side of the QFN part. Avoid contacting the side of the QFN part when performing manual rework with a soldering iron or damage will occur (see Figure 18). If manual soldering is used, the soldering iron tip should only contact the board pad without contacting the side of the component. In general a temperature-controlled hot gas convection repair method is recommended over manual soldering because the risk of applying excessive force is eliminated. The maximum rework temperature as measured on the package should not exceed the MSL rating as specified on the shipping label. See Figure 2 for an example of the MSL label.

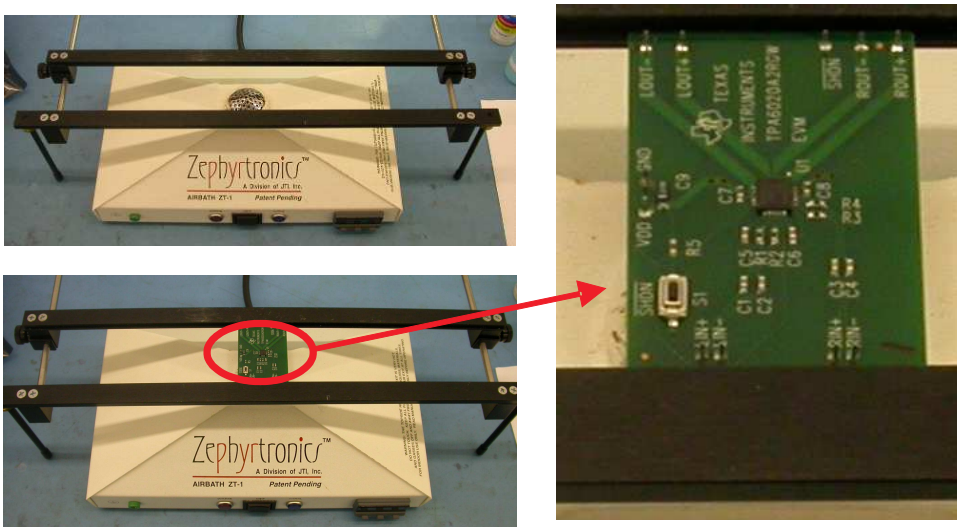


Figure 17. Pre-Heater

Figure 18. Manual Rework Damage

IMPORTANT NOTICE

Texas Instruments Incorporated and its subsidiaries (TI) reserve the right to make corrections, modifications, enhancements, improvements, and other changes to its products and services at any time and to discontinue any product or service without notice. Customers should obtain the latest relevant information before placing orders and should verify that such information is current and complete. All products are sold subject to TI's terms and conditions of sale supplied at the time of order acknowledgment.

TI warrants performance of its hardware products to the specifications applicable at the time of sale in accordance with TI's standard warranty. Testing and other quality control techniques are used to the extent TI deems necessary to support this warranty. Except where mandated by government requirements, testing of all parameters of each product is not necessarily performed.

TI assumes no liability for applications assistance or customer product design. Customers are responsible for their products and applications using TI components. To minimize the risks associated with customer products and applications, customers should provide adequate design and operating safeguards.

TI does not warrant or represent that any license, either express or implied, is granted under any TI patent right, copyright, mask work right, or other TI intellectual property right relating to any combination, machine, or process in which TI products or services are used. Information published by TI regarding third-party products or services does not constitute a license from TI to use such products or services or a warranty or endorsement thereof. Use of such information may require a license from a third party under the patents or other intellectual property of the third party, or a license from TI under the patents or other intellectual property of TI.

Reproduction of TI information in TI data books or data sheets is permissible only if reproduction is without alteration and is accompanied by all associated warranties, conditions, limitations, and notices. Reproduction of this information with alteration is an unfair and deceptive business practice. TI is not responsible or liable for such altered documentation. Information of third parties may be subject to additional restrictions.

Resale of TI products or services with statements different from or beyond the parameters stated by TI for that product or service voids all express and any implied warranties for the associated TI product or service and is an unfair and deceptive business practice. TI is not responsible or liable for any such statements.

TI products are not authorized for use in safety-critical applications (such as life support) where a failure of the TI product would reasonably be expected to cause severe personal injury or death, unless officers of the parties have executed an agreement specifically governing such use. Buyers represent that they have all necessary expertise in the safety and regulatory ramifications of their applications, and acknowledge and agree that they are solely responsible for all legal, regulatory and safety-related requirements concerning their products and any use of TI products in such safety-critical applications, notwithstanding any applications-related information or support that may be provided by TI. Further, Buyers must fully indemnify TI and its representatives against any damages arising out of the use of TI products in such safety-critical applications.

TI products are neither designed nor intended for use in military/aerospace applications or environments unless the TI products are specifically designated by TI as military-grade or "enhanced plastic." Only products designated by TI as military-grade meet military specifications. Buyers acknowledge and agree that any such use of TI products which TI has not designated as military-grade is solely at the Buyer's risk, and that they are solely responsible for compliance with all legal and regulatory requirements in connection with such use.

TI products are neither designed nor intended for use in automotive applications or environments unless the specific TI products are designated by TI as compliant with ISO/TS 16949 requirements. Buyers acknowledge and agree that, if they use any non-designated products in automotive applications, TI will not be responsible for any failure to meet such requirements.

Following are URLs where you can obtain information on other Texas Instruments products and application solutions:

Products		**Applications**	
Amplifiers	amplifier.ti.com	Audio	www.ti.com/audio
Data Converters	dataconverter.ti.com	Automotive	www.ti.com/automotive
DSP	dsp.ti.com	Broadband	www.ti.com/broadband
Interface	interface.ti.com	Digital Control	www.ti.com/digitalcontrol
Logic	logic.ti.com	Military	www.ti.com/military
Power Mgmt	power.ti.com	Optical Networking	www.ti.com/opticalnetwork
Microcontrollers	microcontroller.ti.com	Security	www.ti.com/security
RFID	www.ti-rfid.com	Telephony	www.ti.com/telephony
Low Power Wireless	www.ti.com/lpw	Video & Imaging	www.ti.com/video
		Wireless	www.ti.com/wireless

Mailing Address: Texas Instruments, Post Office Box 655303, Dallas, Texas 75265