Application Note: 3D MagIC MLF Land Pattern and SMT Assembly Guidelines

Table of Contents

1 Introduction

This application note provides PCB designers with guidelines for successful mounting of PNI Sensor Corporation's 3D MagIC ASIC packaged in the MicroLeadFrame® package. The MicroLeadFrame package (MLF) is a near-CSP plastic encapsulated package with a copper leadframe substrate. This is a leadless package where electrical contact to the PCB is made by soldering the lands on the bottom surface of the package to the PCB, instead of the conventional formed perimeter leads. The ePad technology enhances the thermal and electrical properties of the package. The exposed die-attach paddle on the bottom efficiently conducts heat to the PCB and provides a stable ground through down bonds and electrical connections through conductive die-attach material.

Note: This document is a "guideline" to help the user develop a proper board design and surface mount process. Actual studies and other development effort may be needed to optimize the process to the user's surface mount practices and requirements.

Figure 1: Punch Singulated MLF Package Photo and Section Drawing

2 Considerations for Mounting MLF Package

For optimal 3D MagIC performance, special considerations should be followed in designing the user's PCB and mounting the package. For enhanced thermal, electrical, and board-level performance, the exposed pad on the package needs to be soldered to the board using a corresponding thermal pad on the board. Furthermore, for proper heat conduction through the board, thermal vias need to be incorporated in the PCB in the thermal pad region. The PCB footprint design needs to take into consideration dimensional tolerances imposed by the MLF package, PCB, and assembly.

A number of factors may have a significant effect on mounting the MLF package on the board. Some of these factors include: the solder paste coverage in the thermal pad region, the stencil design for the peripheral and thermal pad region, the type of vias, the board thickness, the lead finish on the package, the surface finish on the board, the type of solder paste, and the reflow profile. This application note provides guidelines covering these issues.

3 PCB Design Guidelines

As shown in Figure 2, the lands on the package bottom side are rectangular in shape with rounded edge on the inside. Since the package does not have any solder balls, the electrical connection between the package and the motherboard is made by printing the solder paste on the motherboard and reflowing it after component placement. In order to form reliable solder joints, special attention is needed in designing the motherboard pad pattern and solder paste printing.

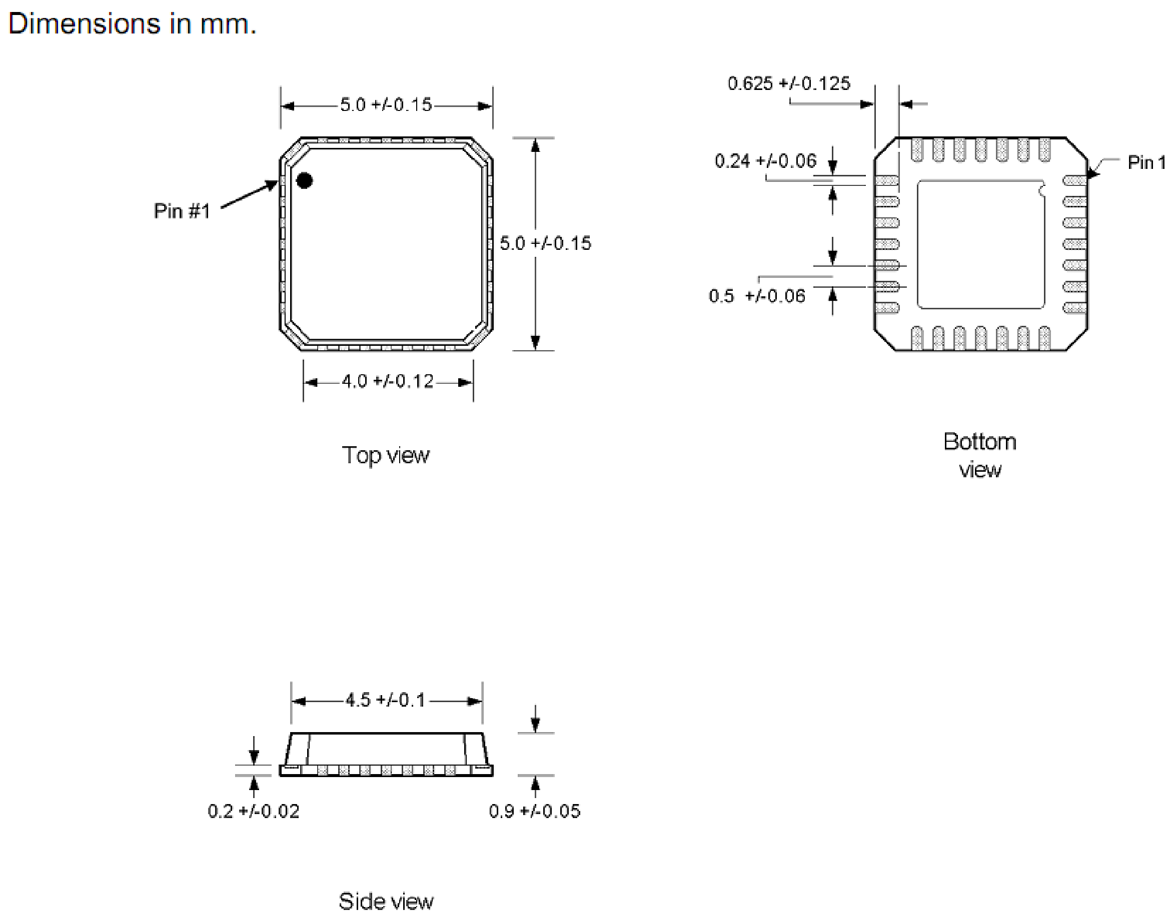


Figure 2: MLF28 package dimension

3.1 Perimeter Pads Design

Typically the PCB pad pattern for an existing package is designed based on guidelines developed within a company or by following industry standards such as IPC-SM-782. However, since MLF is a new package and the industry guidelines have not been developed yet for PCB pad pattern design, the development of proper design may require some experimental trials. For the purpose of this document, IPC's methodology is used here for designing PCB pad pattern. However, because of exposed die paddle and the package lands on the bottom side of the package, certain constraints are added to IPC's methodology. The pad pattern developed here includes considerations for lead and package tolerances. Figure 2 shows the package dimensions

3.2 PCB Pad Pattern

The PCB pad pattern dimensions are shown in Figure 3.

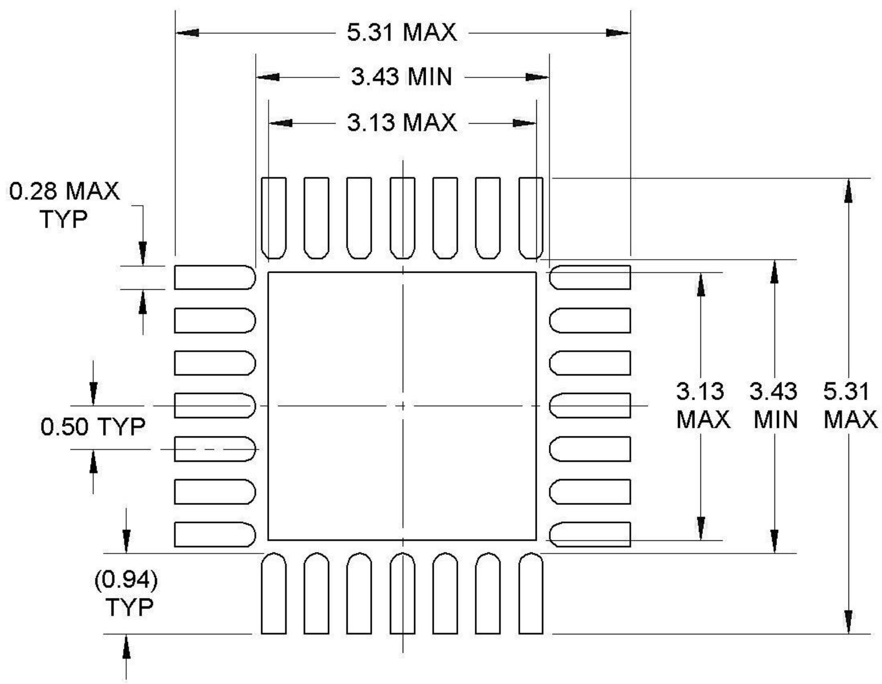


Figure 3: PCB Land pattern (millimeters)

3.3 Thermal Pad and Via Design

The MLF package is designed to provide superior thermal performance. This is partly achieved by incorporating an exposed die paddle on the bottom surface of the package. To take full advantage of this feature, the PCB must have features to effectively conduct heat away from the package. This is typically achieved by incorporating thermal pad and thermal vias on the PCB. However, because PNI's devices are low power and incorporate thermally stable measurement techniques, a die paddle land pattern feature on the PCB and associated thermal vias are optional.

SUNSTAR传感与控制 http://www.sensor-ic.com/ TEL:0755-83376549 FAX:0755-83376182 E-MAIL:szss20@163.com

Normally, the size of the thermal pad should at least match the exposed die paddle size. However, depending upon the die paddle size, this size needs to be modified in some cases to avoid solder bridging between thermal pad and the perimeter pads.

In order to effectively transfer heat from the top metal layer of the PCB to the inner or bottom layers, thermal vias need to be incorporated into the thermal pad design. The number of thermal vias will depend on the application and power dissipation and electrical requirements. Although more thermal vias improve the package thermal performance, there is a point of diminishing returns as additional thermal vias may not significantly improve the performance.

3.4 Solder Masking Considerations

The pads on the printed circuit board are either solder mask defined (SMD) or non solder mask defined (NSMD). Since copper etching process has tighter control than solder masking process, NSMD pads are preferred over SMD pads. Also, NSMD pads with solder mask opening larger than the metal pad size also improves the reliability of solder joints as solder is allowed to wrap around the sides of metal pads. Because of these reasons, NSMD pad is recommended for perimeter lands.

The solder mask opening should be 120 to 150 microns larger than the pad size resulting in 60 to 75 micron clearance between the copper pad and solder mask. This allows for solder mask registration tolerances, which are typically between 50 to 65 microns, depending upon the board fabricators' capabilities. Typically each pad on the PCB should have its own solder mask opening with a web of solder mask between two adjacent pads. Since the web has to be at least 75 microns in width for solder mask to stick to the PCB surface, each pad can have its own solder mask opening for lead pitch of 0.5mm or higher. However, for 0.4mm pitch parts with PCB pad width of 0.25mm, not enough space is available for solder mask web in between the pads. In such cases, it is recommended to use "Trench" type solder mask opening where a big opening is designed around all pads on each side of the package with

No solder mask in between the pads, as shown in Figure 4. It should also be noted that the inner edge of the solder mask should be rounded, especially for corner leads to allow for enough solder mask web in the corner area.

SUNSTAR传感与控制 http://www.sensor-ic.com/ TEL:0755-83376549 FAX:0755-83376182 E-MAIL:szss20@163.com

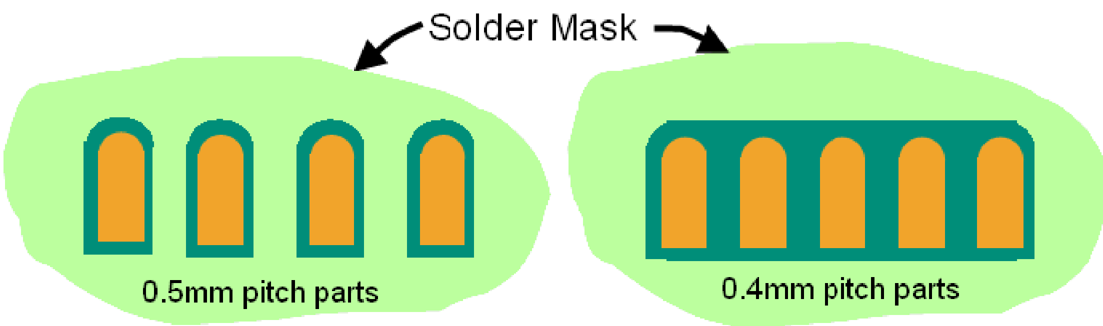


Figure 4: Illustration of Solder Mask Web

For the cases where thermal land dimension is close to the theoretical maximum discussed above, it is recommended that the thermal pad area should be solder mask defined in order to avoid any solder bridging between the thermal pad and the perimeter pads. The mask opening should be 100 microns smaller than the thermal land size on all four sides. This will guarantee a 25 micron solder mask overlap even for the worse case misregistration.

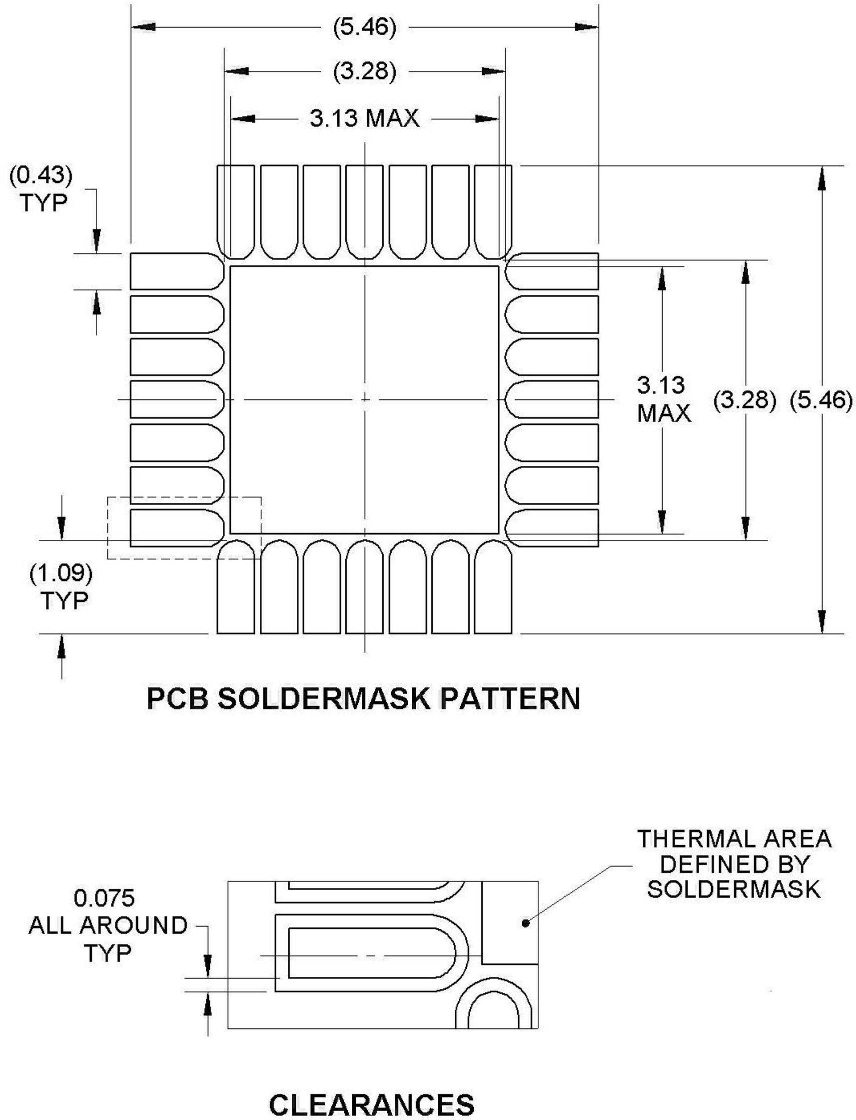


Figure 5: MLF28 Solder mask Dimensions (millimeters)

4 Board Mounting Guidelines

Because of the small lead surface area and the sole reliance on printed solder paste on the PCB surface, care must be taken to form reliable solder joints for MLF packages. This is further complicated by the large thermal pad underneath the package and its proximity to the inner edges of the leads. Although the pad pattern design suggested above might help in eliminating some of the surface mounting problems, special considerations are needed in stencil design and paste printing for both perimeter and thermal pads. Since surface mount process varies from company

SUNSTAR传感与控制 http://www.sensor-ic.com/ TEL:0755-83376549 FAX:0755-83376182 E-MAIL:szss20@163.com

to company, careful process development is recommended. The following provides some guidelines for stencil design based on Amkor's experience in surface mounting MLF packages.

4.1 Stencil Design for Perimeter Pads

The optimum and reliable solder joints on the perimeter pads should have about 50 to 75 microns (2 to 3 mils) standoff height and good side fillet on the outside. A joint with good standoff height but no or low fillet will have reduced life but may meet application requirement. The first step in achieving good standoff is the solder paste stencil design for perimeter pads. The stencil aperture opening should be so designed that maximum paste release is achieved. This is typically accomplished by considering the following two ratios:

- Area Ratio = Area of Aperture Opening/Aperture Wall Area, and
- Aspect Ratio = Aperture width/ Stencil Thickness

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

- Area Ratio = LW/2T(L+W), and
- Aspect Ratio = W/T

Where L and W are the aperture length and width, and 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 the stencil aperture should be 1:1 to PCB pad sizes as both area and aspect ratio targets are easily achieved by this aperture. The opening can be reduced for lead pullback option because of reduction of solderable area on the package. The stencil should be laser cut and electro polished. The polishing helps in smoothing the stencil walls which results 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.

4.2 Stencil Design for Thermal Pad

In order to effectively remove the heat from the package and to enhance electrical performance the die paddle needs to be soldered to the PCB thermal pad, preferably with minimum voids. However, eliminating voids may not be possible because of presence of thermal vias and the large size of the thermal pad for larger size packages. Also, out gassing occurs during reflow process which may cause defects (splatter, solder balling) if the solder paste coverage is too big. It is, therefore, recommended that smaller multiple openings in stencil should be used instead of one big opening for printing solder paste on the thermal pad region. This will typically result in 50 to 80% solder paste coverage. Shown in Figure 6 are some of the ways to achieve these levels of coverage.

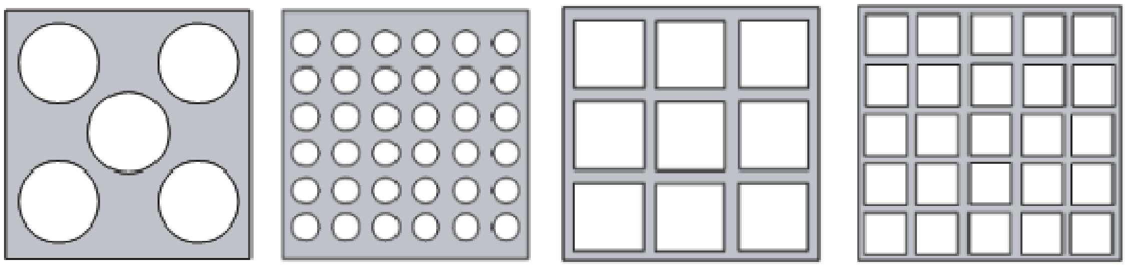


Figure 6: Typical pastemask patterns

4.3 Via types and solder voiding

Voids within solder joints under the exposed pad can have an adverse effect on high speed and RF applications as well as on thermal performance. As the MLF package incorporates a large center pad, controlling solder voiding within this region can be difficult. Voids within this ground plane can increase the current path of the circuit. The maximum size for a void should be less than the via pitch within the plane. This recommendation would assure that any one via would not be rendered ineffectual based on any one void increasing the current path beyond the distance to the next available via.

With regards to voids in the thermal pad region, it should be emphasized that the presence of these voids is not expected to result in degradation of thermal and electrical performance. No loss in thermal performance is predicted from thermal simulation for smaller multiple voids covering up to 50% of thermal pad area. It should also be noted that the voids in thermal pad region do not impact the reliability of perimeter solder joints.

Although the percentage of voids may not be a big concern, large voids in thermal pad area should be avoided. In order to control these voids, solder masking may be required for thermal vias to prevent solder wicking inside the via during reflow, thus displacing the solder away from the interface between the package die paddle and thermal pad on the PCB. There are different methods employed within the industry for this purpose, such as “via tenting” (from top or bottom side) using dry film solder mask, “via plugging” with liquid photo-imagible (LPI) solder mask from the bottom side, or “via encroaching”. These options are depicted in Figure 7. In case of via tenting, the solder mask diameter should be 100 microns larger than via diameter.

SUNSTAR传感与控制 http://www.sensor-ic.com/ TEL:0755-83376549 FAX:0755-83376182 E-MAIL:szss20@163.com

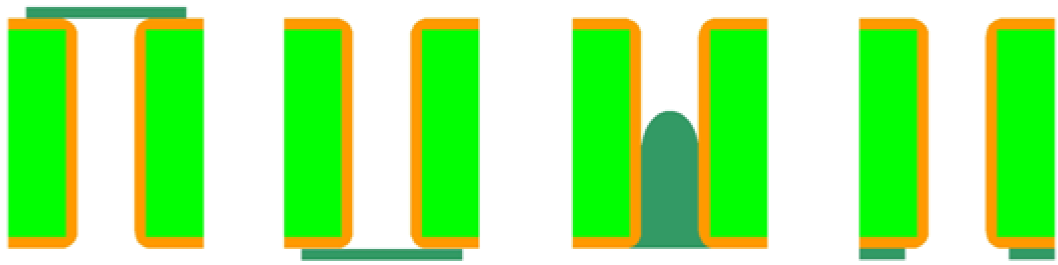


Figure 7: Via Hole Solder masking Treatment Examples

All of these options have pros and cons when mounting MLF package on the board. While via tenting from top side may result in smaller voids, the presence of solder mask on the top side of the board may hinder proper paste printing. On the other hand, both via tenting from bottom or via plugging from bottom may result in larger voids due to out-gassing, covering more than two vias. Finally, encroached vias allow the solder to wick inside the vias and reduce the size of the voids. However, it also results in lower standoff of the package, which is controlled by the solder underneath the exposed pad. Figure 8 shows representative x-ray pictures of MLF packages mounted on boards with different via treatments.

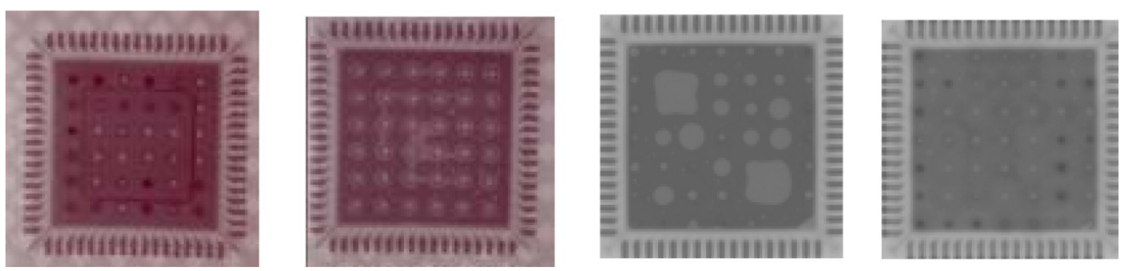


Figure 8: X-ray pictures of MLF packages mounted on boards with different via treatments

Encroached via, depending on the board thickness and amount of solder printed underneath the exposed pad, may also result in solder protruding from the other side of the board, as shown in Figure 9. Note that the vias are not completely filled with solder, suggesting that solder wets down the via walls until the ends are plugged. This protrusion is a function of PCB thickness, amount of paste coverage in the thermal pad region, and the surface finish of the PCB. Amkor's experience is that this protrusion can be avoided by using lower volume of solder paste and reflow peak temperature of less than 215°C. If solder protrusion cannot be avoided, the MLF components may have to be assembled on the top side (or final pass) assembly, as the protruded solder will impede acceptable solder paste printing on the other side of the PCB.

SUNSTAR传感与控制 http://www.sensor-ic.com/ TEL:0755-83376549 FAX:0755-83376182 E-MAIL:szss20@163.com

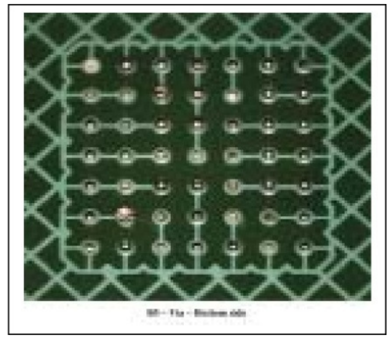


Figure 9: Solder Dispersion in Thermal Pad Vias

4.4 Stencil thickness and Solder Paste

The stencil thickness of 0.125mm is recommended for 0.4 and 0.5mm pitch parts and can be increased to 0.15 – 0.2mm for coarser pitch parts. A laser-cut, stainless steel stencil is recommended with electro-polished trapezoidal walls to improve the paste release.

Since not enough space is available underneath the part after reflow, it is recommended that "No Clean", Type 3 paste be used for mounting MLF packages. Nitrogen purge is also recommended during reflow.

4.5 Solder joint standoff height and fillet formation

The solder joint standoff is a direct function of amount of paste coverage on the thermal pad and the type of vias used for MLFs with exposed pad at the bottom. Board mounting studies sponsored by Amkor have clearly shown that the package standoff increases by increasing the paste coverage and by using plugged vias in the thermal pad region. This is shown in Figure 10 below.

The standoff height varies by the amount of solder that wets or flows into the PTH via. The encroached via provides an easy path for solder to flow into the PTH and decreases package standoff height while the plugged via impedes the flow of solder into the via due to the plugged via's closed barrel end. In addition, the number of vias and their finished hole size will also influence the standoff height for encroached via design. The standoff height is also affected by the type and reactivity of solder paste used during assembly, PCB thickness and surface finish, and reflow profile.

To achieve 50 micron thick solder joints, which help in improving the board level reliability, it is recommended that that the solder paste coverage be at least 50% for plugged vias and 75% for encroached via types.

SUNSTAR传感与控制 http://www.sensor-ic.com/ TEL:0755-83376549 FAX:0755-83376182 E-MAIL:szss20@163.com

The peripheral solder joint fillets formation is also driven by multiple factors. It should be realized that only bottom surface of the leads are plated with solder and not the ends and the bare Cu on the side of the leads may oxidize if the packages are stored in uncontrolled environment. It is, however, possible that a solder fillet will be formed depending on the solder paste (flux) used and the level of oxidation.

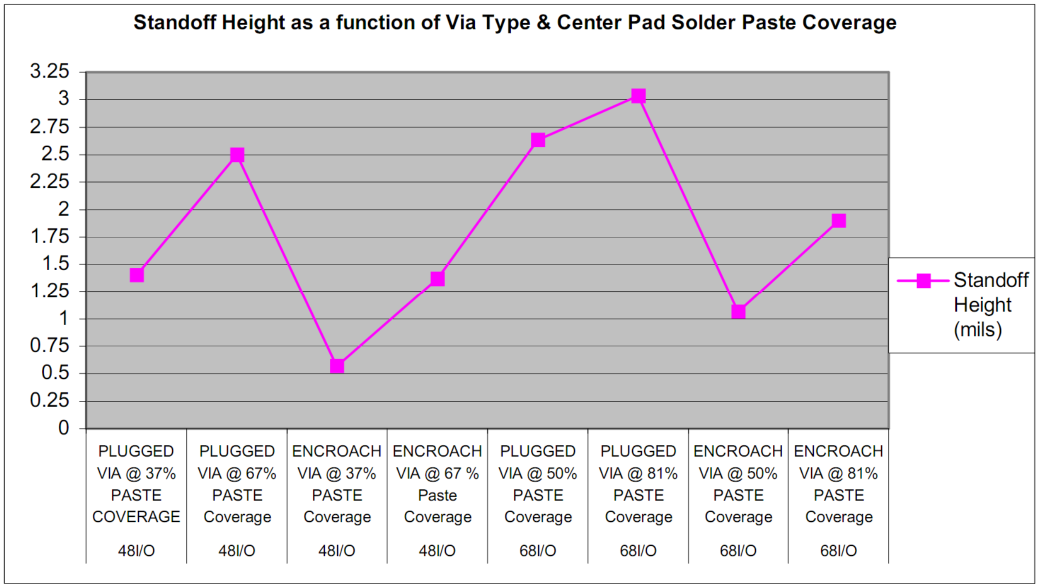
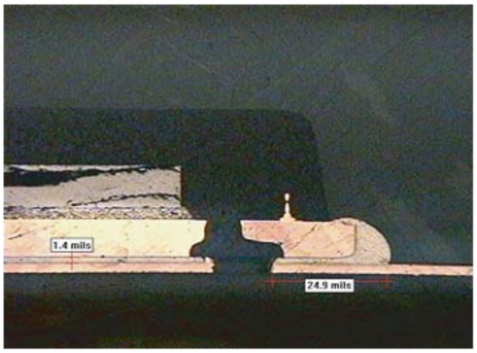


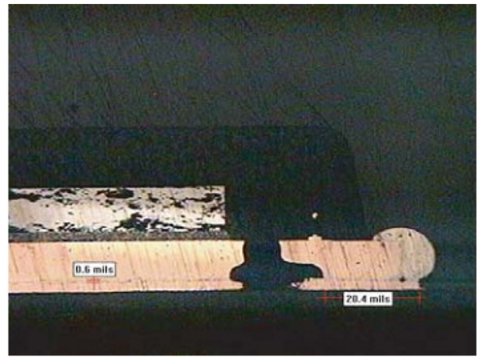
Figure 10: Standoff Height as a Function of Via Type and Paste Coverage

The fillet formation is also a function of PCB land size, printed solder volume, and the package standoff height. The land size listed in Table 1 and 2 along with 1:1 aperture will provide sufficient solder for fillet formation if the package standoff is not excessive. Since there is only limited solder available, higher standoff - controlled by paste coverage on the thermal pad – may not leave enough solder for fillet formation. Conversely, if the standoff is too low, large convex shape fillets may form. This is shown in Figure 11. Since center pad coverage and via type were shown to have the greatest impact on standoff height the volume of solder necessary to create optimum fillet varies. Package standoff height and PCB pads size will establish the required volume.

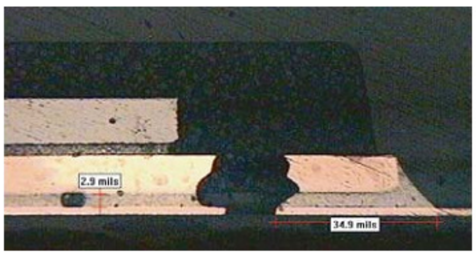
SUNSTAR传感与控制 http://www.sensor-ic.com/ TEL:0755-83376549 FAX:0755-83376182 E-MAIL:szss20@163.com



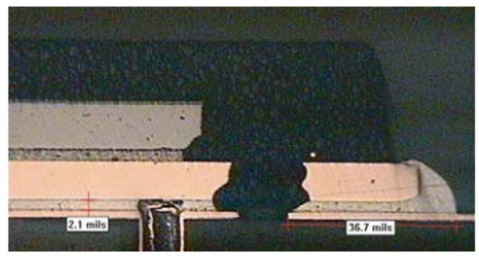
37% paste coverage, plugged via,
1.4 mil standoff



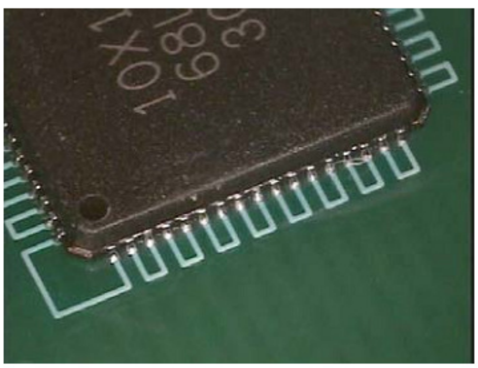
37% paste coverage, encroached via,
0.6 mil standoff



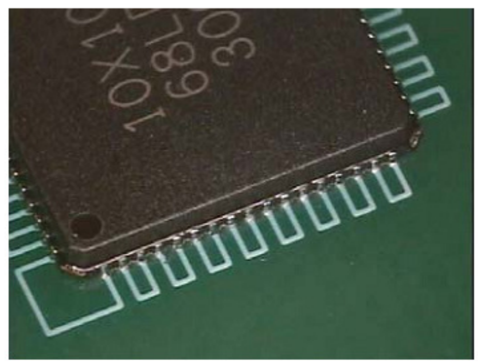
50% paste coverage, plugged via,
2.9 mil standoff



81% paste coverage, encroached via,
2.1 mil standoff



Large PCB pads, 81% paste coverage,
plugged vias



Small PCB pads, 81% paste coverage,
plugged vias

Figure 11: Solder fillet shape as a function of paste coverage in the thermal pad, via type, standoff and PCB land size

4.6 Reflow Profile

Reflow profile and peak temperature has a strong influence on void formation. Our packaging vendor Amkor has conducted experiments with different reflow profiles (ramp-to-peak vs. ramp-hold-ramp), peak reflow temperature, and time above liquids using Alpha Metal's UP78 solder paste (63%Tin /37%Lead). Some of the representative profiles are shown in Figure 12. Generally, it is found that the voids in the thermal pad region for

SUNSTAR传感与控制 http://www.sensor-ic.com/ TEL:0755-83376549 FAX:0755-83376182 E-MAIL:szss20@163.com

plugged vias reduce as the peak reflow temperature is increased from 210 °C to 215 – 220 °C. For encroached vias, it is found that the solder extrusion from the bottom side of the board reduces as the reflow temperature is reduced.

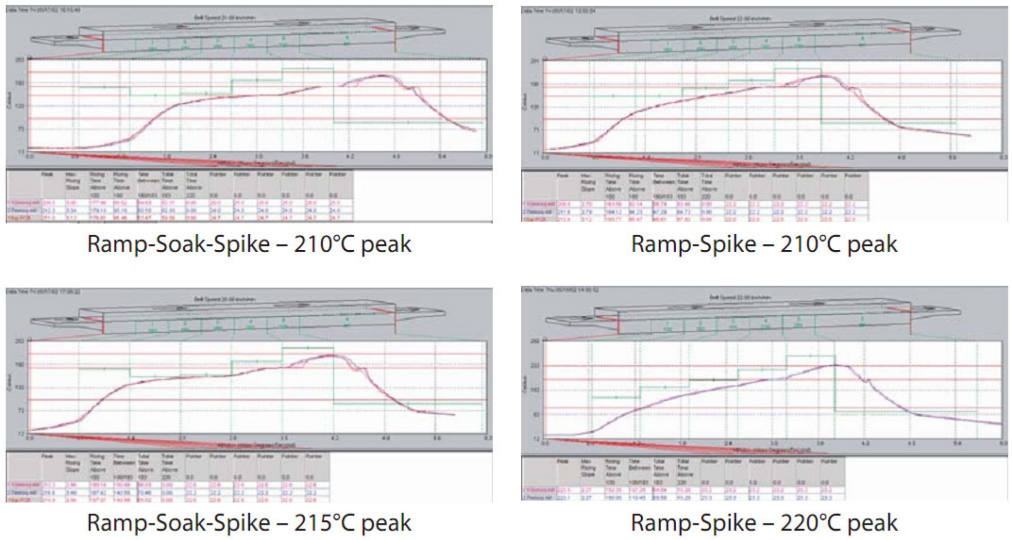
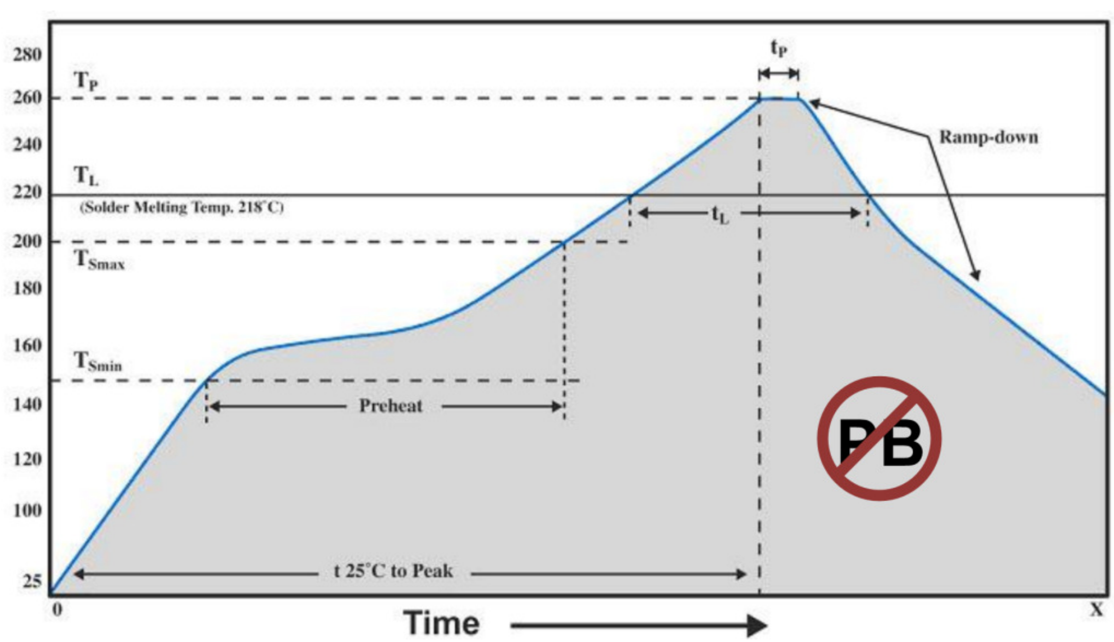


Figure 12: Various reflow profiles

4.7 Lead-free solder

For lead-free soldering of the MFL28 pacakage PNI recommends the reflow profile given in Figure 13 and Table 1.



SUNSTAR传感与控制 http://www.sensor-ic.com/ TEL:0755-83376549 FAX:0755-83376182 E-MAIL:szss20@163.com

Figure 13: Lead Free Reflow Profile

Table 1: Recommended Process Parameters

Parameter	Symbol	Value
Preheat Temperature, Minimum	T_{Smin}	150°C
Preheat Temperature, Maximum	T_{Smax}	200°C
Preheat Time (T_{Smin} to T_{Smax})		60-180 Seconds
Solder Melt Temperature	T_L	>218°C
Ramp-Up Rate (T_{Smax} to T_L)		3°C/second max
Peak Temperature	T_P	<260°C
Time from 25°C to Peak (T_P)		6 minutes max
Time Above T_L	t_L	60-120 seconds
Soak Time (within 5°C of T_P)	t_P	10-20 seconds
Rampdown Rate		4°C/second max

a. Meets lead-free profile recommendations (IPC/JEDEC J-STD-020)

5 Assembly Process Flow

Figure 18 shows the typical process flow for mounting surface mount packages to printed circuit boards. The same process can be used for mounting the MLFs without any modifications. It is important to include post print and post reflow inspection, especially during process development. The volume of paste printed should be measured either by 2D or 3D techniques. The paste volume should be around 80 to 90% of stencil aperture volume to indicate good paste release. After reflow, the mounted package should be inspected in transmission x-ray for the presence of voids, solder balling, or other defects. Cross-sectioning may also be required to determine the fillet shape and size and joint standoff height.

With regard to leaded solder; typical reflow profiles for No Clean solder leaded paste are shown in Figure12. Since the actual reflow profile depends on the solder paste being used and the board density, PNI does not recommend a specific profile for leaded solder. However, the temperature should not exceed the maximum temperature the package is qualified for according to moisture sensitivity level. The time above liquidus temperature should be around 60 seconds and the ramp rate during preheat should be 3^{o}C/second or lower.

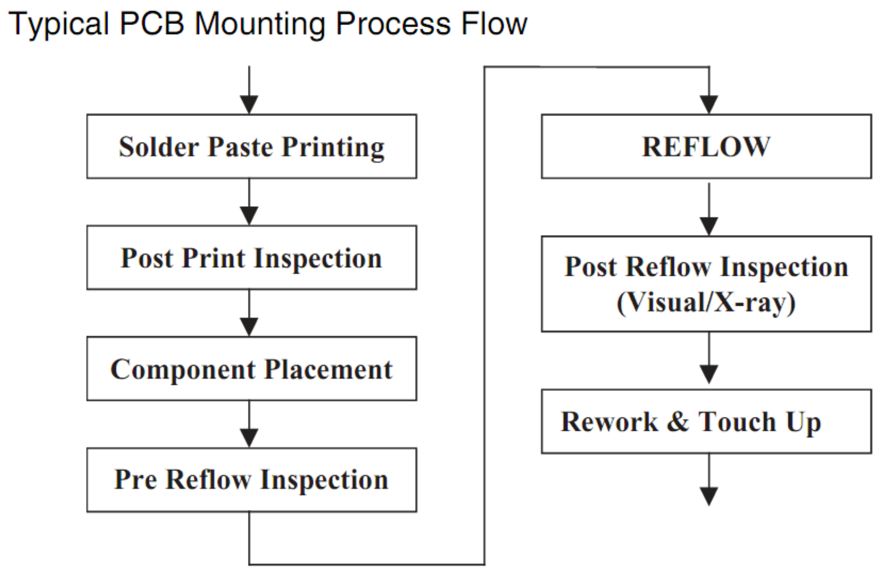


Figure 14: Typical PCB mounting process flow

6 Rework Guidelines

Since solder joints are not fully exposed in the case of MLFs, any retouch is limited to the side fillet. For defects underneath the package, the whole package has to be removed. Rework of MLF packages can be a challenge due to their small size. In most applications, MLFs will be mounted on smaller, thinner, and denser PCBs that introduce further challenges due to handling and heating issues. Since reflow of adjacent parts is not desirable during rework, the proximity of other components may further complicate this process. Because of the product dependent complexities, the following only provides a guideline and a starting point for the development of a successful rework process for these packages.

The rework process involves the following steps:

- Component Removal
- Site Redress
- Solder Paste Application,
- Component Placement
- Component Attachment

These steps are discussed in the following in more detail. Prior to any rework, it is strongly recommended that the PCB assembly be baked for at least 4 hours at 125°C to remove any residual moisture from the assembly.

SUNSTAR传感与控制 http://www.sensor-ic.com/ TEL:0755-83376549 FAX:0755-83376182 E-MAIL:szss20@163.com

6.1 Component Removal

The first step in removal of component is the reflow of solder joints attaching component to the board. Ideally the reflow profile for part removal should be the same as the one used for part attachment. However, the time above liquidus can be reduced as long as the reflow is complete.

In the removal process, it is recommended that the board should be heated from the bottom side using convective heaters and hot gas or air should be used on the top side of the component. Special nozzles should be used to direct the heating in the component area and heating of adjacent components should be minimized. Excessive airflow should also be avoided since this may cause CSP to skew. Air velocity of 15 – 20 liters per minute is a good starting point.

Once the joints have reflowed, the Vacuum lift-off should be automatically engaged during the transition from reflow to cool down. Because of their small size the vacuum pressure should be kept below 15 inch of Hg. This will allow the component not to be lifted out if all joints have not been reflowed and avoid the pad liftoff.

6.2 Site Redress

After the component has been removed, the site needs to be cleaned properly. It is best to use a combination of a blade-style conductive tool and desoldering braid. The width of he blade should be matched to the maximum width of the footprint and the blade temperature should be low enough not to cause any damage to the circuit board. Once the residual solder has been removed, the lands should be cleaned with a solvent. The solvent is usually specific to the type of paste used in the original assembly and paste manufacturer’s recommendations should be followed.

6.3 Solder Paste Printing

Because of their small size and finer pitches, solder paste deposition for MLFs requires extra care. However, a uniform and precise deposition can be achieved if miniature stencil specific to the component is used. The stencil aperture should be aligned with the pads under 50 to 100X magnification. The stencil should then be lowered onto the PCB and the paste should be deposited with a small metal squeegee blade. Alternatively, the mini stencil can be used to print paste on the package side also. A 125 microns thick stencil with aperture size and shape same as the package land should be used. Also, no-clean flux should be used, as small standoff of MLFs does not leave much room for cleaning.

6.4 Component Placement

MLF packages are expected to have superior self-centering ability due to their small mass and the placement of this package should be similar to that of BGAs. As the leads are on the underside of the package, split-beam optical system should be used to align the component on the motherboard. This will form an image of leads overlaid on the mating footprint and aid in proper alignment. Again, the alignment should be done at 50 to 100X magnification. The placement machine should have the capability of allowing fine adjustments in X, Y, and rotational axes.

6.5 Component Attachment

The reflow profile developed during original attachment or removal should be used to attach the new component. Since all reflow profile parameters have already been optimized, using the same profile will eliminate the need for thermocouple feedback and will reduce operator dependencies.

7 Disclaimer

This document contains general guidelines that PNI Sensor Corporation received from its package vendor. PNI does not make direct recommendation for board design nor does it take legal liability and responsibility for the information in this document. These guidelines are based on IPC standard: IPC-SM-782, Surface Mount Design and Land Pattern Standard. This standard has since been superseded by standard IPC-7351B Generic Requirements for Surface Mount Design and Land Pattern Standard. Please refer to the IPC website for more information. See references at the end of this document.

Sketch drawing in this application note is reference only. It may not represent all MLF application.

The terms MicroLeadFrame, MLF and the Amkor Technology logo are all registered trademarks of Amkor Technology, Inc.

8 References

Application Notes for Surface Mount Assembly of Amkor's MicroLeadFrame (MLF) Packages http://www.amkor.com/go/packaging/document-library
IPC-SM-782 Surface Mount Design and Land Pattern Standard. IPC-7351B Generic Requirements for Surface Mount Design and Land Pattern Standard http://www.ipc.org