PCB Land Design and Surface Mount for DFN2x5 Sawn Package

Introduction

DFN2x5 package is a plastic encapsulated package with a copper lead frame substrate. It offers good thermal and electrical performance, near chip scale footprint, thin profile and low weight. Electrical contact to the PCB is made by soldering perimeter leads and exposed die attach pad on the bottom surface of the package to the PCB. The exposed die attach pad on the bottom efficiently conducts heat to the PCB and provides a stable ground by electrical connection through a conductive die attach material. This technology enhances the thermal and electrical performance of the package. Figure 1 shows DFN2x5 package configuration.

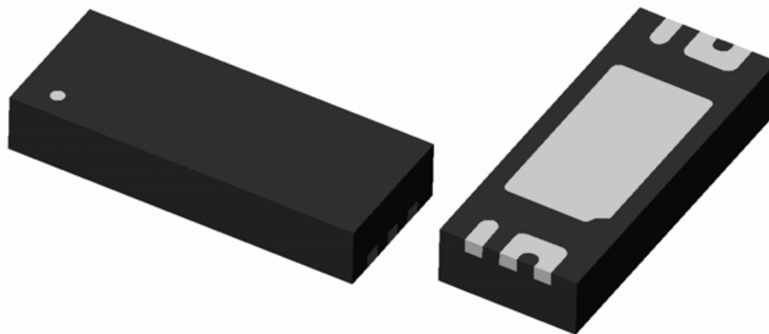


Figure 1, DFN 2x5 package configuration

PCB Land and Via Design

The PCB lands should be a little bit larger than the package's perimeter leads and pad on all sides. Inward corners may be rounded to match leads and pad shape. Dimension guidelines are shown in Figure 2 and Table 1

For I/O lead solder land design, we suggest to use a Non Solder Mask Defined (NSMD) approach, remain a small amount of solder mask between the I/O leads to avoid solder bridging. The outward extension (Tout) can be increased beyond 0.15 mm as this could potentially improve this external solder joint. Any increase in the inward extension (Tin) above 0.05mm must consider the effect on the isolation gap to the thermal land. The PCB land width should be 0.05 mm wider than lead width.

For PCB thermal land design, we suggest to use a Solder MASK Defined (SMD) approach. The recommended design gap between the PCB thermal land and I/O land is 0.15mm

DO NOT COPY WITHOUT ANY AUTHORIZATION OF ALPHA & OMEGA SEMICONDUCTOR, LTD.

minimum to avoid solder bridging and shorting. When space is available, gap of 0.25mm or more is preferred.

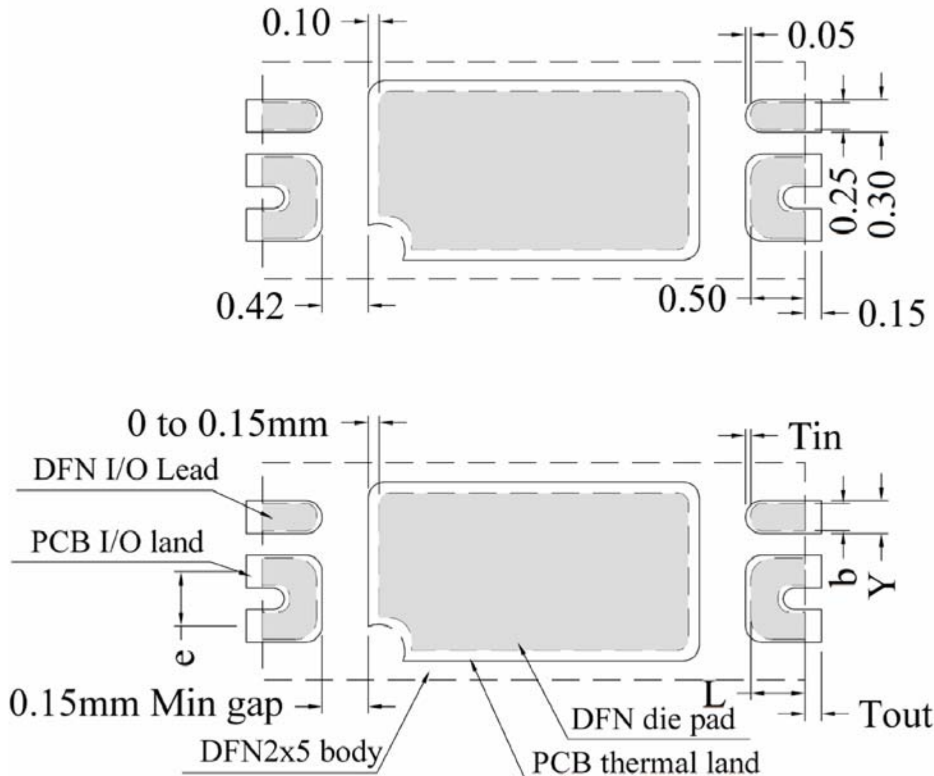


Figure 2, PCB Land Pattern Layout with DFN2x5 Package Overlay

Table1. Recommended Dimensions (unit: mm)

Package I/O Lead Nominal Dimension			PCB Land Nominal Dimension Guidelines		
Lead Pitch (e)	Lead Width (b)	Lead Length (L)	Land Width (Y)	Outward (Tout)	Inward (Tin)
0.5	0.25	0.5	0.3	0.15	0.05

Plated through via holes in the PCB thermal land should be 0.33mm in diameter and plugged. If via holes can not be plugged, suggest to cap vias on the backside of the board using solder mask material. This should allow vias to be filled with solder during reflow.

Solder Mask Design

For surface mount package, two types of stencil designs are used
1, Solder Mask Defined (SMD): solder mask openings smaller than metal pads.
2, Non-Solder Mask Defined (NSMD): solder mask openings lager than metal pads

For the perimeter I/O lands, NSMD is recommended; it improves the reliability of solder joints as this allows the solder to wrap around the sides of the metal pads.

DO NOT COPY WITHOUT ANY AUTHORIZATION OF ALPHA & OMEGA SEMICONDUCTOR, LTD.

Because the spacing between the thermal pad and the perimeter land pads can be small, SMD is recommended for the thermal pad to prevent solder bridging.

0.125-0.150 mm thick stainless steel stencil is recommended for solder past application. The aperture walls should be trapezoidal and the corners are rounded for better paste release.

For the perimeter I/O lands, the stencil opening should be 0.05mm lager than the PCB land (0.025mm in each direction)

For thermal land area, it is recommended to screen the solder paste in an array of small openings rather than one larger opening. The total area of all the mask openings should be approximately equal to 50% of the total thermal land area. This will ensure good solder coverage with fewer voids.

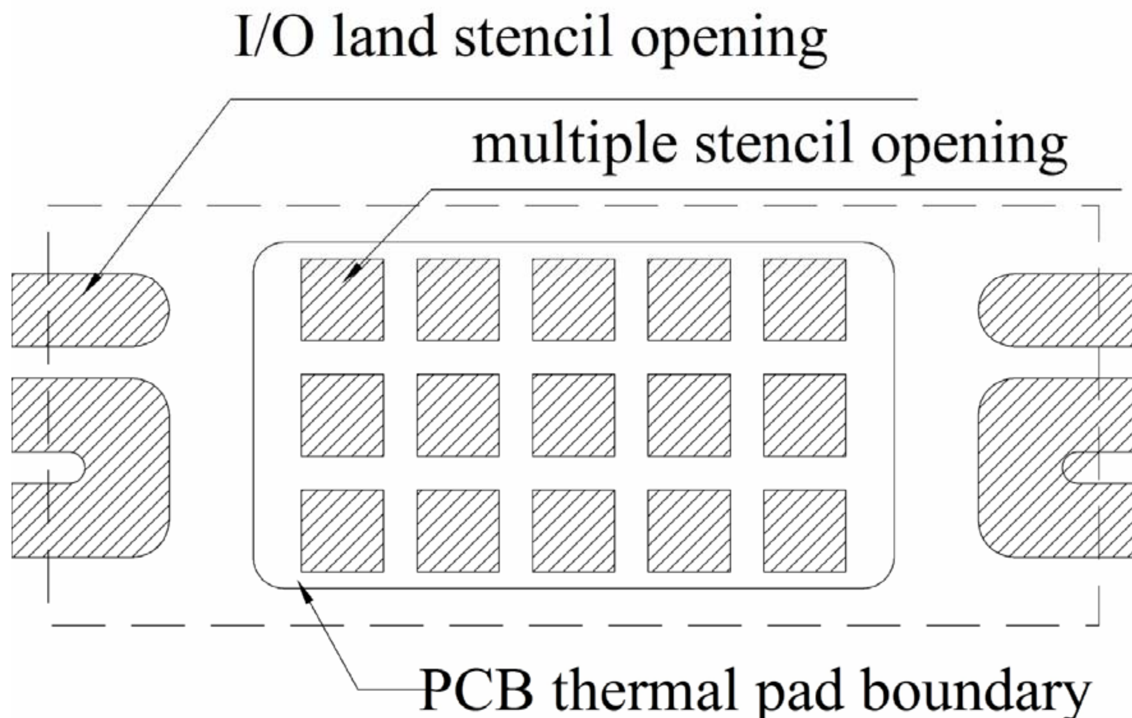


Figure 3, Suggested Stencil Design

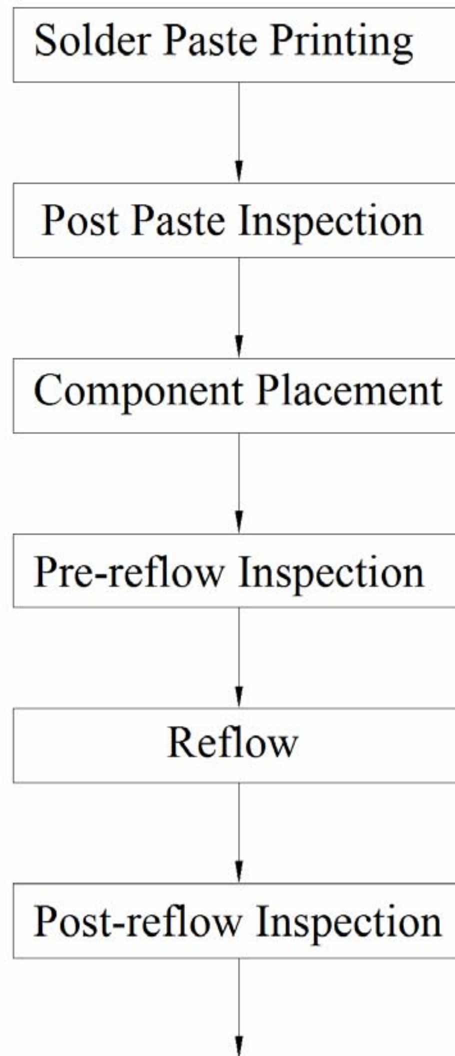
Solder Paste and Reflow Profile

Because DFN2x5 packages have a low profile and small terminal pitch, a No Clean, Type 3 solder paste and convection/IR reflow is recommended.

Refer to industrial standard lead free reflow profile, which can be used as starting point, the actual profile used will depend on the entire populated board and the solder used.

DO NOT COPY WITHOUT ANY AUTHORIZATION OF ALPHA & OMEGA SEMICONDUCTOR, LTD.

Surface Mount Assembly Flow



Rework Guidelines

After PCB assembly, the package should be inspected in transmission x-ray for the presence of voids, solder balling, or other defects underneath the package. Cross sectioning may also be required to determine the fillet shape and size, and the joint standoff height. Since solder joints are not fully exposed, any retouch is limited to the side fillet. For defects underneath the package, the whole package has to be removed. Rework of packages can be a challenge due to their small size, and in most applications, it will be mounted on smaller, thinner, and denser PCBs that introduce further challenges due to handling and heating issues. The following process provides a guideline and a starting point for the development of a successful rework process for this package.

Bake

DO NOT COPY WITHOUT ANY AUTHORIZATION OF ALPHA & OMEGA SEMICONDUCTOR, LTD.

Prior to any rework, it is strongly recommended that the PCB assembly be baked for at least 4 hours at 125C to remove any residual moisture from the assembly.

Component Removal

The first step in removal of component is the reflow of solder joints. The reflow profile for part removal should be same as the one used for component 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 be avoided; air velocity of 15-20 liters per minutes is a good starting point. The PCB and components being removed should reach at least 55±5C^0 before applying the hot gas. The hot gas temperature can be approximately 425C^0. The hot gas can be applied with nozzle tip approximately 15-25 mm away from the component in a circular motion to bring the component and underlying solder to reflow temperature. As reflow temperature is approached it is sometimes useful to apply the heat edge on to the component in order to try and get the heat to go under the package to reflow the large thermal land. It is also helpful to wedge a tweezer tip under a corner of the component and supply a slight upward force so when the thermal land begins to reflow the component will lift up

Once the joints have reflowed, remove the component with a vacuum lift-off or tweezer. Because of their small size the vacuum pressure should be kept below 15 inch of Hg. This will avoid the land liftoff if all joints have not been reflowed and component is not lifted out.

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 the 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 recommendation should be followed.

Solder Paste Printing

Because of their small size and finer pitches, solder paste deposition requires extra care. However, a uniform and precise deposition can be achieved if miniature stencil to the component is used. The stencil aperture should be aligned with the pads under 50 to 100X magnification. The stencil should then be lowed onto the PCB and the paste should be deposited with a small metal squeegee blade. Alternative, the mini stencil can be used to print paste on the package side. No-clean flux should be used, as small standoff of package does not leave much room for cleaning.

Component Placement and Attachment

This package is expected to have a superior self-centering ability due to their small mass. As the leads are on the underneath of the package, split-beam optical system should be used to align the component on the board. This will form an image of leads overlaid on the mating

DO NOT COPY WITHOUT ANY AUTHORIZATION OF ALPHA & OMEGA SEMICONDUCTOR, LTD.

footprint and aid in proper alignment. Again, the alignment should be done at 50 to 100X magnification. This placement machine should have the capability of allowing fine adjustments in X, Y and rotational axes. The reflow profile developed during original attachment or removal should be used to attach the new component.

This guideline is general and only for customer reference. The customer must apply their experiences to optimize the PCB and solder mask design, and perform assembly process according to actual components, board and materials used.

DO NOT COPY WITHOUT ANY AUTHORIZATION OF ALPHA & OMEGA SEMICONDUCTOR, LTD.