## PCB design techniques for reliable Land Grid Array assembly by Joel Dobler

## INTRODUCTION

This application note will discuss commonly encountered PCB assembly issues that are related to footprint design of Land Grid Array (LGA) packages and will offer guidance on avoiding these issues.

It can be difficult to design the LGA PCB footprints that produce high percentage assembly yields. Large LGA packages with high pin counts and multiple rows and columns, or entire rings of pins, make reliable PCB assembly even more difficult. Solder opens \& shorts between PCB pad and LGA pins are very common if the proper PCB footprint design is not used.
Gathering advice from several sources, experimental PCB footprints were generated and tried on two different LGA packages, including the ADAR1000 LGA. Test results from multiple lots of PCB fabricators and assemblers were promising with very high production yields. These results have led to a list of general guidelines when generating PCB footprints for LGA packages with pin dimensions of $0.25 \times .25 \mathrm{~mm}$ or similar, and 0.5 mm pitch.


Figure 1. Example of square LGA package

# TBD 

Figure 2.Example of rectangular LGApackage

## TABLE OF CONTENTS

Introduction ................................................................................. 1
Revision History .......................................................................... 2
Causes of Pcb assembly issues...................................................... 3
Solder Mask Variance and stencil invariance .......................... 3
Device Warping ....................................................................... 4
PCB footprint Guidelines ............................................................ 6

## REVISION HISTORY

3/2021-Revision 0 (Preliminary): Initial Version
General Guidelines for Footprint Design .....  .6
Design example: ADAR1000 pcb footprint .....  7
Conclusion. ..... 10

## CAUSES OF PCB ASSEMBLY ISSUES

There are several causes of PCB assembly issues. This application note will focus on the following:

1. Variance in Copper Pad and Solder Mask Aperture
a. Size and Location (Registration)
2. Solder Mask Variance and Non-Variance in Solder Stencil
3. Warping of the LGA package

## SOLDER MASK VARIANCE AND STENCIL INVARIANCE

Many PCBs use liquid photoimagable (LPI) solder masks that meet the IPC-SM-840 Class 3 standard. This was used in the examples shown in this application note. Depending on the fabrication lot, vendor, etc., the solder mask apertures can vary in size and be misaligned due to solder mask registration error. Varying solder mask apertures can be clearly seen in Figure 3, Figure 4, and Figure 5 for several of the footprint's pads. The most egregious examples are highlighted with red rectangles.
If the pads are solder mask defined, as is often the case for ground pads over a large ground plane, variation of the solder mask aperture can either increase or decrease the effective size of the copper pad, even though the copper itself it not varying.


Figure 3. PCB Footprint with clear problematic solder mask variations


Figure 4.. Examples (in red rectangles) of neighboring pads having solder mask variation


Figure 5. Addition examples (in red rectangles) of neighboring pads having solder mask variation

Solder paste variance is very small compared to a typical solder mask variance. Stencils are generally laser cut from stainless steel sheets by machines capable of minimum cut widths of 0.05 mm or better, and aperture tolerances of 0.013 mm or better. An example of a solder stencil is shown in Figure 6. Tolerances that are this tight usually ensure uniform volume of solder paste being deposited onto the exposed copper of the PCB.


Figure 6. Example of solder stencil
The combination of the solder mask varying and the stencil not varying can set up a situation in which PCB assembly is not reliable due to pin-to-pin shorts and/or pin opens.

Pin opens can be caused by a single pad having a larger aperture size while its neighboring pads have nominal or smaller than nominal apertures. This creates a situation of having a sufficient solder dam, but the deposited solder into the pad with the larger aperture spreads out and is shorter than it otherwise would be in a nominal sized aperture, due to the volume of solder being deposited by the solder stencil is the same for every pin. An example of this can be seen in Figure 7 showing a nominal aperture size, while Figure 8 shows an aperture that is larger than nominal.


Figure 7. Side view of DUT pin-to-solder-to-pad interface; nominal solder mask aperture size creates proper connection to pin (not to scale; dimensions exaggerated for illustration purposes)


Figure 8. Side view of DUT pin-to-solder-to-pad interface; larger than nominal solder mask aperture size creates an open connection (not to scale; dimensions exaggerated for illustration purposes)
For pin-to-pin shorts, this usually happens with non-solder mask defined (NSMD) pads, and could be caused by the pin and adjacent soldermask apertures being larger than designed, thus their solder dam is smaller than expected and does not provide as much protection against shorts. Having a sufficient solder dam becomes especially important if the device is warping during assembly, even nominal apertures with NSMD pads could have pin-to-pin shorts when the device is warping (see next section).

## LGA DEVICE WARPING

Sometimes the solder mask variation on the PCB is very low as seen in Figure 9, yet assembly issues persist. If the die, the laminate and/or the package has a large coefficient of thermal expansion, this usually causes the DUT to warp significantly over temperature. A large amount of warpage (measured as deviation from coplanar or just coplanarity) could be enough to cause assembly issues if the board design is not optimized. Also, if the package is rectangular, the effects of the DUT warping can be asymmetrical and increased along the long axis of the rectangular package.
Shown in Figure 9 is a portion of a rectangular LGA package that was having issues being assembled to the PCB, which is shown in the bottom half of the picture. The solder mask variation was low, and the solder dam distance between pads seemed to be sufficient (the solder dam distance was 50 um by design, although effectively was larger as can be seen in Figure 9; the solder dam distance was changed to 165 um on a subsequent board revision). LGA pins are $0.3 \times 0.275 \mathrm{~mm}$ and PCB copper pad sizes are $0.305 \times 0.33 \mathrm{~mm}$ for scale.

Figure 10 shows an X-ray of the long side of the rectangular LGA after being assembled onto the PCB. As can be seen, the inner two (of the four) signal lines are shorted to the adjacent ground pads (circled in red); there is slightly darker gray can be seen between the ground and signal pads that are shorted together. Devices that have enough warpage can displace solder out of a PCB pad(s) and into the adjacent pad(s) if solder dam is not sufficiently large enough. This is illustrated in Figure 12.
In general, decreasing the solder mask aperture size, which increases the distance of the solder dam, helps with reliable assembly. A more methodical approach to designing LGA footprints for reliable assembly is presented in the next section.


Figure 9. Side-by-side comparison of the bottom of a rectangular LGA (left) vs. PCB footprint (right). Ground pads slightly bigger than signal pads due to solder mask recessing towards the ground plane. Solder dam designed to be 50um, but fabricated larger; for scale LGA pins are $0.3 \times 0.275 \mathrm{~mm}$ and $P C B$ copper pad sizes are $0.305 \times 0.33 \mathrm{~mm}$ for scale.

## Application Note



Figure 10. Post-Assembly. Signal-to-ground pad and ground-to-ground pad shorts; look for the slightly darker gray between the signal pads and adjacent ground pad, this is the solder short.


Figure 11. Side view of DUT pin-to-solder-to-pad interface prior to reflow; if the device did not warp during reflow, the chance of problems during assembly would be low (not to scale; dimensions exaggerated for illustration purposes)


Figure 12. Side view of DUT pin-to-solder-to-pad interface post reflow; the DUT warping and insufficient solder dams causing solder displacement, which in turn causing shorts to adjacent pads (not to scale; dimensions exaggerated for illustration purposes)

## PCB FOOTPRINT GUIDELINES

The goal is to design the PCB so that the effect of PCB variances and package distortions can be minimized or nulled out completely. The following guidelines attempt to address the issues mentioned thus far.

## GENERAL GUIDELINES FOR FOOTPRINT DESIGN

These guidelines are applicable to LGA packages with pins dimensions between 0.25 mm square and $0 \times 3 \times 0 \times 33 \mathrm{~mm}$, with 0.5 mm pitch, and similar geometries. The guidelines are:

1. PCB pads should be $20 \%$ larger in each direction relative to the nominal DUT package pin dimensions, as shown in Figure 13 and Figure 14. This ends up being approximately 0.05 mm larger for packages with 0.25 mm square pins
a. Oversizing the pad helps with registration issues
2. PCB pad to solder mask opening should be a $1: 1$ ratio. This is neither a solder mask defined nor a non-solder mask defined solution, but rather a hybrid solution
a. Keeping the $1: 1$ ratio helps with having sufficient solder dame between the pads
3. On the paste mask/solder stencil, break up the ground paddle opening into several smaller openings, such that they are similar in size to the pins on the part
a. This evenly distributes solder over the large exposed copper sections of the PCB
4. Remove the silkscreen DUT outline
a. Removes any possibility of the outline affecting the planarity of the DUT
5. Remove any silkscreen outline of components that are near the DUT
6. Move any silkscreen text, pin 1 designators, etc. away from DUT
a. A 1.85 mm keep out region from nearest DUT pad is recommend
7. Move components as far away as possible from DUT
a. A 2.3 mm keep out region from nearest DUT pad is recommended

Note: \#5-\#7 affect the assembly less than the first four guidelines and fall into the best practices category. Nonetheless, we still recommend that they be implemented where possible and to a degree that PCB space allows.
Packages with smaller pin and pitch geometries were not investigated, however the guidelines should still be applicable, at least as a starting point for the footprint design. Advanced board design and fabrication techniques may need to be used to account for the solder mask variation when using appreciably smaller pad geometries. This topic is outside scope of this application note, but one strategy to combat this is to use fiducial markers near the LGA as discussed in the Additional Recommendations section.


Figure 13. Top View PCB Pad and solder mask aperture oversize guidelines


Figure 14. Side View PCB Pad and solder mask oversize guidelines relative to the DUT


Figure 15. Recommended keep out region from DUT for components, and silk screen designators; U1 and Pin 1 designators at least 1.85 mm from DUT, nearest components $\mathrm{C1}$ and C 4 are 2.3 mm from DUT and do not have silkscreen outline

## DESIGN EXAMPLE: ADAR1000 PCB FOOTPRINT

The following details the design of the ADAR1000 footprint and surrounding area on the ADAR1000-EVALZ evaluation board.

## ADAR1000 Package Outline Drawing

The ADAR1000's package is a 7 x 7 mm LGA package, that has an inner \& outer ring of pins and an exposed ground paddle in the center. Each pin is nominally 0.25 mm square, with variation of $\pm 0.05 \mathrm{~mm}$. The nominal exposed ground paddle dimensions are $4.25 \times 3.25 \mathrm{~mm}$. The remaining dimensions of the bottom side of the package are shown in Figure 16 below. Not shown is the pin recess depth which is 0.02 mm from bottom of package to pin.


Figure 16. ADAR1000 Package Outline Drawing (Bottom View); dimensions shown in millimeters

## ADAR1000 Recommended PCB Footprint Design

Applying the guidelines for reliable assembly, the following was done to the ADAR1000 PCB footprint:

1. The PCB pads were oversized by $20 \%$, which is 0.05 mm larger than the nominal size of the ADAR1000 pins. This makes the pad size is $0.3 \times 0 \times 3 \mathrm{~mm}$. This is shown in Figure 17 and Figure 18.


Figure 17. ADAR1000 PCB recommended footprint with $0 \times 3 \times 0 \times 3 \mathrm{~mm}$ pads selected in white, the rest of layer 1 metal shown in aqua.


Figure 18. Zoomed view of Figure 17.
2. The solder mask aperture dimension was set equal to that of the PCB pads, which is $0.3 \times 0 \times 3 \mathrm{~mm}$. This is shown in Figure 19.


Figure 19. ADAR1000 Solder Mask with $0.3 \times 0.3$ mm pad openings \& $3.378 x$ 4.369 mm ground paddle opening
3. The solder stencil was designed such that the ground paddle opening is broken up into smaller apertures that are similar in size to the pins of the LGA. The solder stencil design is shown in Figure 20
a. Figure 21 shows a picture of wet solder paste deposited from a stencil with the ground paddle opening broken up into several smaller apertures


Figure 20. ADAR1000 PCB Paste Mask (Solder Stencil) with $0.28 \times 0.28 \mathrm{~mm}$ pad openings \& $0.406 \times 0.28 \mathrm{~mm}$ ground paddle openings


Figure 21. Example of wet solder paste (pre reflow) on a copper ground pad that has been deposited by a solder stencil with multiple smaller apertures for the ground pad. Signal pins shown at the top of the picture for reference.
4. The silkscreen outline around the ADAR1000 was removed
5. The silkscreen outlines around the components that are within 2.3 mm from the ADAR1000 were removed.
6. Any silkscreen text or features near the DUT like a Pin 1 designator were moved away from the DUT the recommend distance of 1.85 mm .
7. The closest component was set to 2.3 mm from the ADAR1000

Guidelines 4 through 7 are illustrated in Figure 22 and Figure 23. As can be seen in Figure 22 the nearest component (C21) is 2.3 mm away and the nearest silkscreen feature (C21 outline) is 1.85 mm away. The next nearest silkscreen feature is the 'C35' text. Both the 'DUT' silkscreen text and the Pin 1 designator were moved well outside the 1.85 mm keep out region.

The ADAR1000 evaluation board allowed for keeping the nearest components and silkscreen features well away from the ADAR1000 This keep out region is not always possible on application boards, but the board designer should keep this guideline in mind when designing their board, and observe these guidelines as best as possible..


Figure 22. ADAR1000 PCB keep out regions; silk screen features are at a minimum of 1.85 mm from DUT pads; nearest populated component is 2.3 mm from DUT pads


Figure 23. ADAR1000 PCB keep out regions; silk screen features are at a minimum of 1.85 mm from DUT pads;

## Additional Recommendation

To help with alignment of the LGA and give an indication if there are any pad and/or solder mask registration issues is to add fiducial markers near the LGA. Placing these fiducial markers near the LGA will in theory show if there are any local issues; it is not guaranteed that the same issues affecting the LGA will affect other parts on the PCB that are distant to the LGA. Adding fidiucial markers is done only if there is enough empty space on the PCB.
Although not implemented on the ADAR1000 evaluation board, 1 mm diameter copper fiducial markers with 2 mm diameter solder mask apertures were implemented on the board that assembled the rectangular 12.5 x 7 mm LGA. On this board, two fiducial markers were placed on diagonally opposite corners of the LGA, approximately 4 mm away from each corner to allow enough space for routing. Figure 24 shows the upper left hand fiducial, placed just beside a line that is partially routed on the top layer.


Figure 24. Fiducial marker shown on upper left-hand corner of $12.5 \times 7 \mathrm{~mm}$ LGA; solder mask is red, copper is green; solder mask over copper appears as orange. An identical fiducial marker was also put on the lower right hand corner of the LGA.

## CONCLUSION

## Results

Following the LGA PCB footprint design guidelines presented in this application note, two experimental footprints were implemented with 2 separate LGA packages. The first package was the ADAR1000. The second package was $12.5 \times 7 \mathrm{~mm}$, with $0.3 \times 0.275 \mathrm{~mm}$ pins, with 0.5 mm pitch.
The board designs with the experimental footprints were sent to multiple PCB fabricators and subsequently sent to multiple PCB assemblers. Boards from all lot combinations had very high production yields compared to previous attempts.

Although these LGA footprint guidelines do not address every assembly problem, these experiments have proven they do reduce or even null the effects of solder mask variation and DUT warping during reflow.

