SkyHigh Memory

AN99178

Solder Mask and Trace Recommendations for FBGAs

Author: Rudy Sterner

AN99178 provides recommendations for solder masks used with FBGAs and escape considerations for traces.

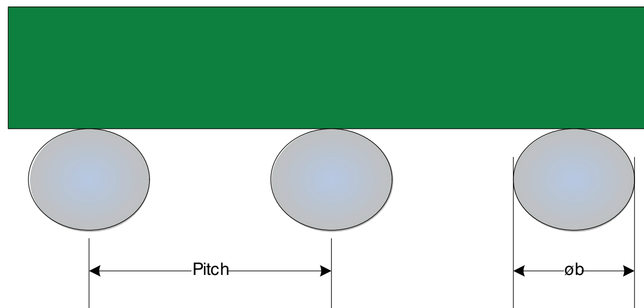
1 Hardware Architecture

1.1 FBGA Architecture

SkyHigh Inc. offers flash devices in a wide range of packages. The ball pitches of the FBGA packages range from 0.50 mm up to 1.00 mm. The ball diameters for each of SkyHigh's common ball pitches are given in Figure 1.

Figure 1. Solder-Ball Dimensions 0.50 mm to 1.00 mm Pitch

Pitch	0.50 mm			0.65 mm			0.80 mm			1.00 mm		
Diameter	Min	Nom	Max	Min	Nom	Max	Min	Nom	Max	Min	Nom	Max
Ball Diameter (øb)	0.25 mm	-	0.35 mm	0.45 mm	-	0.55 mm	0.35 mm	-	0.45 mm	0.50 mm	-	0.70 mm



1.2 Recommended PC Board Design

1.2.1 SMD & NSMD Dimensions

The solder-ball land pads on the printed circuit boards can be either solder mask defined (SMD) or non-solder-mask defined (NSMD). On boards with SMD configuration, the solder mask covers the outer edge of the solder pad. Solder is prevented from flowing over the edges of the pad by the solder mask. On boards with NSMD configuration, there is a small gap between the solder pad and the solder mask (Figure 2 on page 2). Solder flows into the gap between the pad and the solder mask.

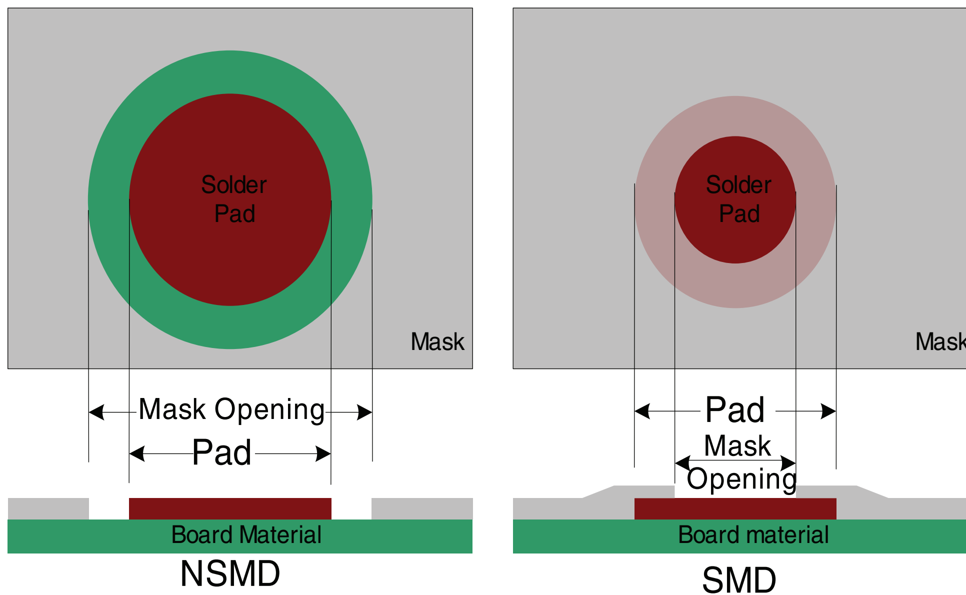
Because the entire solder pad is expose, the NSMD configuration results in a stronger bond between the solder pad and the ball. Additionally, the sides of the pad are wetted by the solder, which results in less stress concentration. Thus, NSMD is generally the recommended configuration.

As a general rule on SMD configurations, it is good practice to make the solder mask opening the same size as the diameter as the solder ball. On NSMD configurations, the solder pad can be between 80% and 100% the solder ball diameter. The solder-mask opening should be 0.15 mm larger than the solder pad. This gives ample space for excess solder. Table 1 shows commonly used dimensions for SMD and NSMD configurations.

Table 1. Common Solder Pad Dimensions

		Pitch				Units
		0.50 mm		0.65 mm		
Configuration		Min	Opt.	Min	Opt	
SMD	Solder Pad		0.45		0.65	mm
	Solder Mask		0.35		0.55	mm
NSMD	Solder Pad	0.28	0.45	0.44	0.55	mm
	Solder Mask	0.43	0.50	0.59	0.70	mm
		Pitch				Units
		0.80 mm		1.00 mm		
Configuration		Min	Opt	Min	Opt	
SMD	Solder Pad		0.55	0.34	0.47	mm
	Solder Mask		0.45	0.27	0.34	mm
NSMD	Solder Pad	0.36	0.45	0.56	0.70	mm
	Solder Mask	0.51	0.60	0.71	0.85	mm

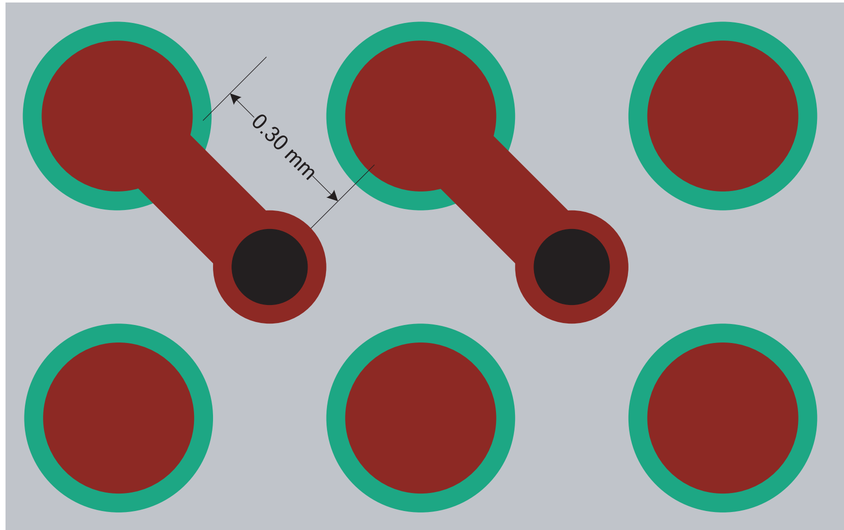
Figure 2. SMD and NSMD Configurations



1.2.2 Vias

Avoid placing vias at the solder-ball pad. This causes solder wicking inside the hole and can result in misshapen solder joints with electrical opens. The vias should be placed away from the pad a minimum of 0.30 mm from the pad as shown in Figure 3.

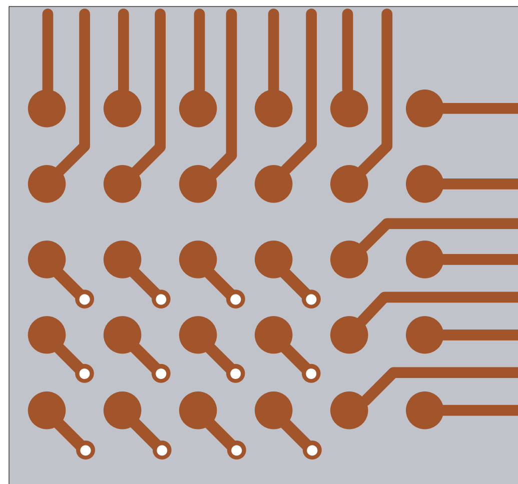
Figure 3. Recommended Solder Mask



1.2.3 Escape Paths

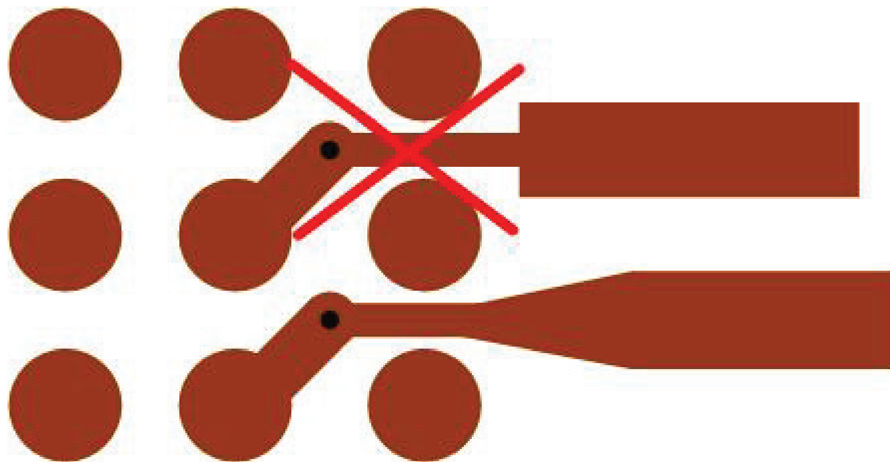
Maintaining good signal integrity is always a top priority in the consideration of escape paths. It is a good idea to limit one signal trace between any two adjacent solder balls (Figure 4).

Figure 4. Escape Routes



Sometimes it is necessary to adjust the width of the trace at the escape path to allow it to fit between adjacent pads. Avoid abrupt changes in width, such as shown in the top of Figure 5. This results in an abrupt change in the trace impedance that can increase reflections and radiated noise. The preferred approach is to gradually taper the trace as shown in the bottom of Figure 5. The impedance change is also gradual and related reflection issues are avoided.

Figure 5. Trace Width Adjustment at the escape Point



2 Conclusion

Mounting flash devices offered by SkyHigh can be accomplished following commonly used techniques.

- Use either the SMD or NSMD configuration.
  - When given a choice NSMD is preferred.
- Do not place vias directly under the solder ball. Offset the via by about 0.3 mm.
- On any single layer, allow only one, signal line to escape between any 2 adjacent solder pads.
- Taper the end of wide signal lines to minimize reflections.
- This note is meant as a guideline. Always, consult with the board manufacturer for the latest board mounting techniques.

3 References

1. PCB Design and SMT Assembly Guidelines for PBGA and FPBGA Packages: http://www.skyworksinc.com/products_display_item.asp?did=4734
2. AND8075/D Board Mounting Considerations for FCBGA Packages: www.onsemi.com/pub/Collateral/AND8075-D.PDF
3. Application Note 1126 BGA (Ball Grid Array): http://www.national.com/an/AN/AN-1126.pdf#page=1

Document History Page

Document Title: AN99178 - Solder Mask and Trace Recommendations for FBGAs Document Number: 001-99178				
Rev.	**ECN No.**	**Orig. of Change**	**Submission Date**	**Description of Change**
**	–	–	11/14/2007	Initial version
*A	4955647	MSWI	10/09/2015	Updated in Cypress template
*B	5873125	AESATMP8	09/06/2017	Updated logo and Copyright.
*C		MNAD	06/03/2019	Updated to SkyHigh format