

## Introduction

As Ball Grid Array (BGA) packages become increasingly popular, it is important to understand how they are affected by various board layout techniques. This document provides a brief overview of PCB layout considerations when working with BGA packages. It outlines some of the most common problems and provides tips for avoiding them at the design stage.

## Advantages and Disadvantages of BGA Packaging

One of the greatest advantages of BGA packaging over other new technologies is that it can be supported with existing placement and assembly equipment. BGAs also offer significantly more misalignment tolerance, less susceptibility to coplanarity issues and easier PCB signal routing under a BGA package (see Figure 14-1).

The primary drawback of BGA packaging is the inability to access the solder joints for testing and inspection (a later section in this document provides layout recommendations for testing). At best, only the outermost row of balls can be seen, and board size and other components often restrict even that view. The best option available for a complete inspection of the device is X-ray imaging. By this means, the user can visually assess shorted connections, missing balls, filled vias, and in some cases, opens (see Figure 14-2). Opens and partial opens (where the solder did not wet the entire pad) are more difficult to see and may require higher resolution equipment.

**Figure 14-1. Misalignment of BGA Balls vs. QFP Leads**

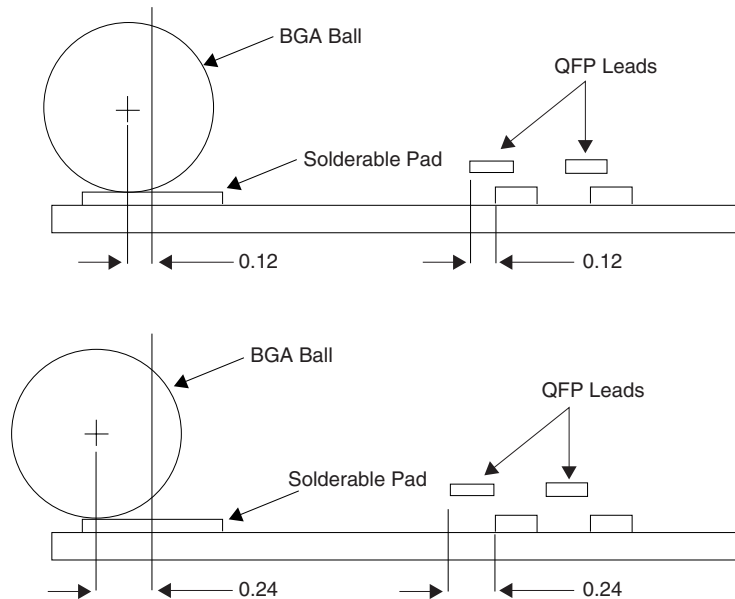
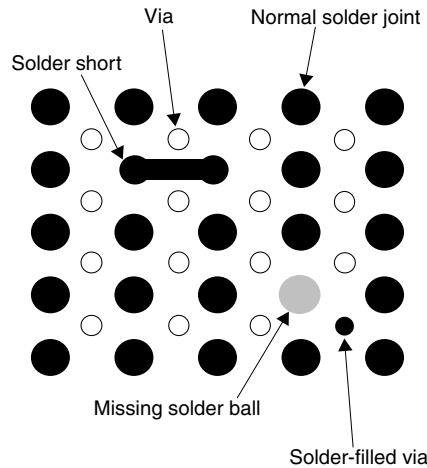


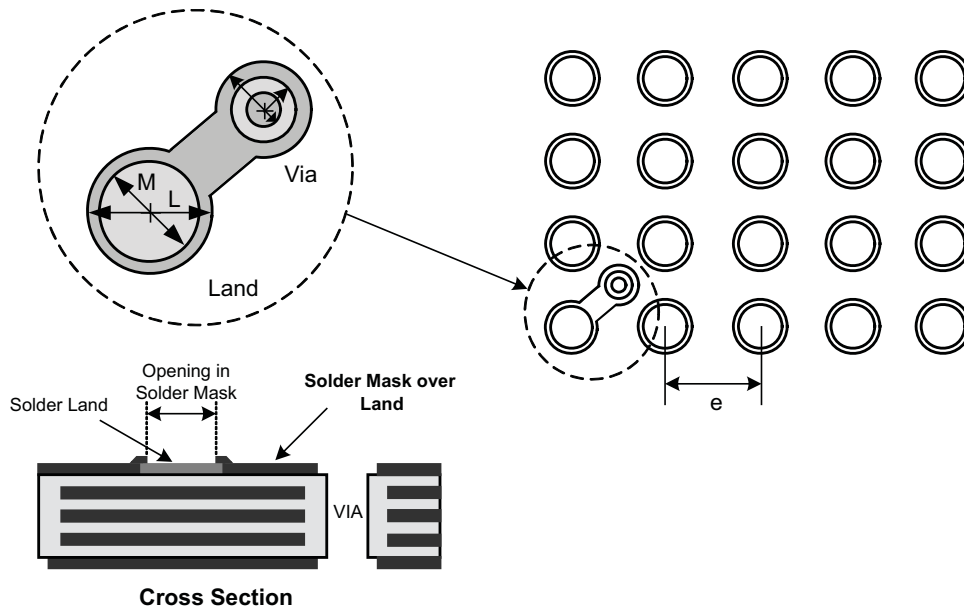
Figure 14-2. Example of How Defects May Appear in an X-Ray Image



### PCB Layout

All Lattice BGA packages utilize Solder Mask Defined (SMD) pads. For optimized solder joint formation, the PCB pads should match the BGA solder pads (see Figure 14-3). For example, if the BGA has solder pads with 0.60 mm openings, so should the corresponding site on the PCB.

Figure 14-3. SMD Pad with Example Dimensions



### Plated Through Hole (Via) Placement

Probably the most critical aspect of BGA PCB layout is the consideration for Plated Through Hole (PTH) placement. If the pad is too close or on top of the hole, or if there is no solder mask covering the via, then it is possible for the ball solder and paste to melt and be wicked into the hole. If enough solder is lost into the hole, the result could be an open for that lead. While this type of defect can usually be detected in an X-ray image, it is best avoided at layout (see Figure 14-2).

## BGA Board Layout Recommendations

	Pitch 0.5mm csBGA	Pitch 0.8mm caBGA	Pitch 1.00mm (fpBGA, ftBGA, fpSBGA, fcBGA)				Pitch 1.27mm PBGA, SBGA
			All Other fpBGA, ftBGA, fpSBGA	100 fpBGA, 256 ftBGA	Organic fcBGA	Ceramic fcBGA	
Solder Land Diameter (L)	0.43	0.53	0.66	0.53	0.80	0.70	0.80
Opening in Solder Mask (M)	0.30	0.40	0.45	0.40	0.60	0.50	0.63
Solder Ball Land Pitch (e)	0.50	0.80	1.00	1.00	1.00	1.00	1.27

Note: The numbers in this table are intended to be used as an example only. The actual numbers are dependent on the PCB manufacturing tolerance.

## BGA Package Types

Package Type	Description
PBGA	<b>Plastic BGA</b> with 1.27 mm solder ball pitch. Die up configuration.
fpBGA	<b>Fine Pitch BGA</b> – Plastic BGA with 1.0 mm solder ball pitch. Die up configuration.
ftBGA	<b>Fine Pitch Thin BGA</b> – Thin plastic BGA with 1.0 mm solder ball pitch. Die up configuration.
caBGA	<b>Chip Array BGA</b> – Plastic BGA with 0.8 mm solder ball pitch. Die up configuration.
csBGA	<b>Chip Scale BGA</b> – Plastic BGA with 0.5 mm solder ball pitch. Die up configuration.
fcBGA	<b>Flip-Chip BGA</b> with 1.0 mm solder ball pitch. Die down configuration. May have a ceramic or plastic substrate.
SBGA	<b>Super BGA</b> – Similar to PBGA, but with an integrated heatsink plate. This package has 1.27 mm solder ball pitch and die down configuration. SBGA packages offer enhanced thermal dissipation capability.
fpSBGA	<b>Fine Pitch SBGA</b> – Super BGA with 1.0 mm solder ball pitch. Die down configuration.

## Further Information

For additional information, please visit the web sites listed below.

[www.amkor.com/Products/all\\_products](http://www.amkor.com/Products/all_products)

[www.latticesemi.com/lit/docs/package/amkor\\_bga\\_appnote.pdf](http://www.latticesemi.com/lit/docs/package/amkor_bga_appnote.pdf)

## Technical Support Assistance

Hotline: 1-800-LATTICE (North America)

+1-503-268-8001 (Outside North America)

e-mail: [techsupport@latticesemi.com](mailto:techsupport@latticesemi.com)

Internet: [www.latticesemi.com](http://www.latticesemi.com)

## Revision History

Date	Version	Change Summary
January 2005	01.0	Initial release.
November 2005	01.1	Figures updated.
June 2006	01.2	Removed NSMD content.
September 2006	01.3	Added note to BGA Board Layout Recommendations table. Reformatted BGA Package Types section in tabular format.