ADVANCED
ASSEMBLY

A GUIDE TO BALL GRID ARRAYS

TABLE OF CONTENTS

INTRO

The ball grid array (BGA) is a surface-mount chip package that is used to mount embedded devices (e.g. microprocessors) by melting balls of solder between the face of the device and the circuit board. Unlike the perimeter-only package type, which place soldering pins along the edge of the device, a BGA aligns its solder balls in a grid beneath the bottom surface of the device. As a result, this approach leaves a considerably smaller footprint on the PCB and induces better thermal and electrical properties than a perimeter-style mounting package. It's no surprise the format's popularity has grown in tandem with the continuous miniaturization of electronics.

And yet, BGAs are the boon and bane of engineers and printed circuit board designers the world over. Their unparalleled pin density and low lead inductance are essential in today's high pin count, high frequency integrated circuits. However, that same pin density and unique interface create a challenge unique unto themselves. While there are entire textbooks that cover the topic of BGAs, their use and fanout techniques, the quick overview provided here offers an engineer a good starting point for using BGAs in designs.

THE BASICS OF BGAs

While the basic BGA concept has remained the same, they have been changing in dimensions since their introduction, with smaller pitches and smaller outlines

While the basic BGA concept has remained the same, they have been changing in dimensions since their introduction, with smaller pitches and smaller outlines. Certain BGAs have no connections in the center, while others have pins all across the bottom of the package. For simpler BGAs, with greater pitch and space in the middle of the BGA, manual routing can be accomplished without creating a breakout pattern. As the pitch decreases and pin count goes up, the amount of connections required makes the semi-random placement of traces unfeasible. While increasing layers of the board can be a shortcut to make it simpler to route the traces, this creates increased cost and greater reliability concerns, which should be avoided. If it is possible to reduce the layer requirements by more thoughtful layouts, then do so, only increasing the layer count as a last option.

As the different ball patterns on the board give rise to different optimal fanouts, it's important to carefully look at the patterns and ask various questions:

- How far apart are the balls?
- Are they parallel and equidistant, or do they have greater spacing in one direction?
- Does the pattern change?
- What is your minimum trace width and spacing?

With answers to these questions in hand, you can decide on how to best approach the fanout yourself, or you can turn to the Internet to see if someone else has already developed a solution to this problem.

USING VIAS WITH BGAs

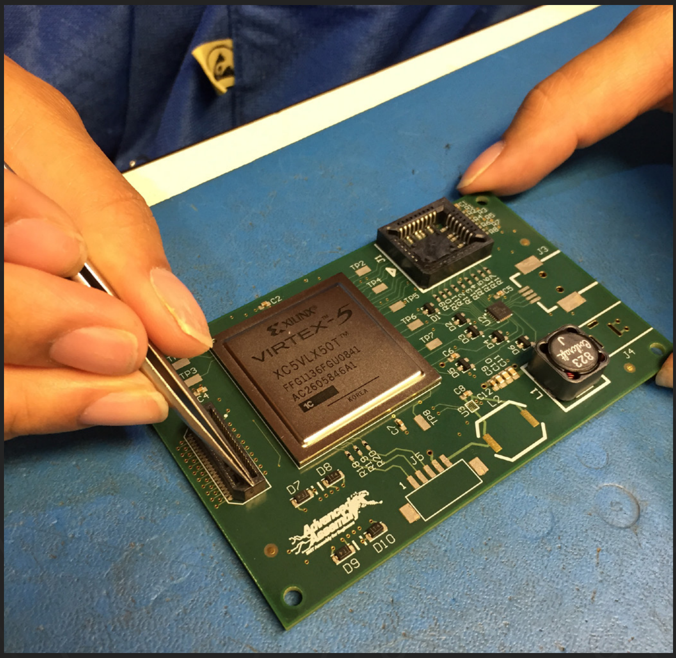
Due to their complex nature, BGAs are heavily dependent on vias, and it is essential to know how and where to use them. Where possible, to save money and increase reliability, use mechanical drilling for the vias. However, be aware that this will create a hole through all layers of the board when you may only want to connect to one or two layers. Blind microvias give you the option of only drilling down to the layer needed, or you can use laminated buried vias that can be used to change layers within the PCB. Unfortunately, this technology can be expensive and unreliable. Yet, at times, it’s indispensable. If it is not possible to properly breakout the BGA, contact your PCB manufacturer to ask about the company’s capabilities and cost for microvias.

Please Note: It is highly recommended that vias not be included in the track pads when using BGA in a PCB design. Vias and traces leading from the BGA that do not have a solder mask over bare copper are likely to cause solder thieving or poor solder joints, thus causing the BGA to fail. This is because solder balls arranged in a BGA will take any path they can when melting, often flowing away from its intended destination.

SOLDERING BGAs

Bonding a device's BGA with the copper trace pads on a PCB requires the use of either a reflow oven or infrared heating. These tools produce an even distribution of heat that melts each solder ball individually, effectively keeping the package aligned with pre-defined alignment indicators (pads) while the solder solidifies and forms the actual connection between the embedded device and the board. The main disadvantage attributed with BGA usage is that soldering connections cannot be visually inspected once the device is mounted.

The pads for BGAs typically fall under two styles, solder mask defined pads and non-solder mask defined pads. The biggest difference is whether the solder mask encroaches on the pad or not. Generally, non-solder mask defined pads are Recommended because of the increased surface area for bonding, however the solder mask defined pads decrease the chance of bridging and may be needed with smaller pitches.



PLACING BGAs AND uBGAS

If the PCB design substitutes standard BGA packages with micro BGA (QBGA) packages, or ball grid arrays with a pitch of less than 0.5 mm, then paste flux may be used in place of solder paste. Keep in mind that uBGA's smaller frame does not inhibit the board from being machine assembled; automated Pick-and-Placed machines handle uBGAs in much the same manner as other surface-mount components.

There is an exception to this rule: Certain pick-and-place machines are incompatible with QBGAs that feature a mirror surface due to vision issues. In these circumstances, the QBGA must be handplaced. Contact the pcb assembly provider to determine if this may be an issue.

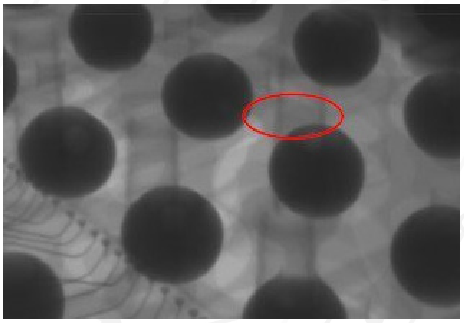
TESTING BGAs

Finding a solder fault beneath a device whose packaged has been soldered into place is practically impossible without special equipment. Nevertheless, it is advantageous to do so in order to ensure optimal PCB performance.

Inspection tools include a range of devices: X-ray machines, industrial CT scanning machines, special microscopes, endoscopes,

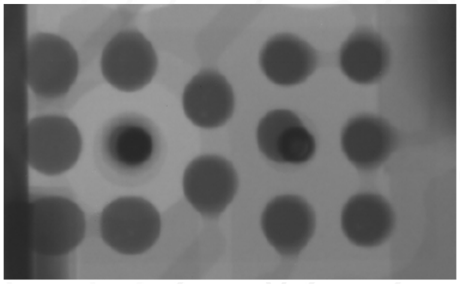
and JTAG. X-ray machines are arguably among the more efficient post BGA-bonding solutions due to their capacity for both 2D and 3D vision. The machines can see solder balls, excess solder, solder thieving, component misalignment, and bridging, as well as missing solder balls. This process helps verify that each solder ball is correctly placed, that each contact with the pads remain intact, and that no solder thieving has occurred. The following X-ray images are examples of BGAs and QBGAs with various defects.

Figure 1



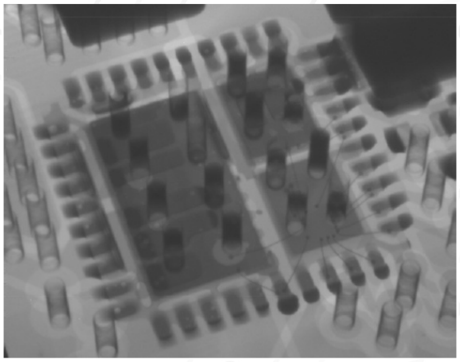
This 2D X-ray image depicts a QBGA suffering from solder thieving because its vias did not include a solder mask over bare copper. Notice how the solder balls have almost completely dissolved into the vias and created an open space.

Figure 2



3D X-ray images of BGAs permit an altogether different insight into the state of the BGA bonding. Here we see that the pad *is* visible at the top of each ball since the pad has a lower density compared to that of the ball.

Figure 3



In this particular 3D image, we can easily see that a short has occurred under a plastic package.

LIMITATIONS OF BGAs

Unfortunately, there are some limitations when taking X-ray images. For example, it's difficult to create a distinction between parts that are placed in the same location but on opposite sides of a board. Background metals such as POP parts may also create an interference with the X-ray that makes it image rendering difficult. As a result, exercising caution when aligning the BGA (or uBGA) with the PCB is ultimately the best practice; testing tools are simply meant to supplement accurate placement and provide a measure of quality assurance.

CONCLUSION

Despite the complexity and difficulty inherent with the use of BGAs, the increased performance and space savings frequently merit the challenges and potential headaches. As with all processes, time and practice will make the challenge of BGA design a second-nature skill that will become invaluable.

Have more questions? Reach out to us for a free pre-production assembly Q&A session with our engineers.

20100 E. 32nd Pkwy. #225 | Aurora, CO 80011 | www.aapcb.com | (800) 838-5650