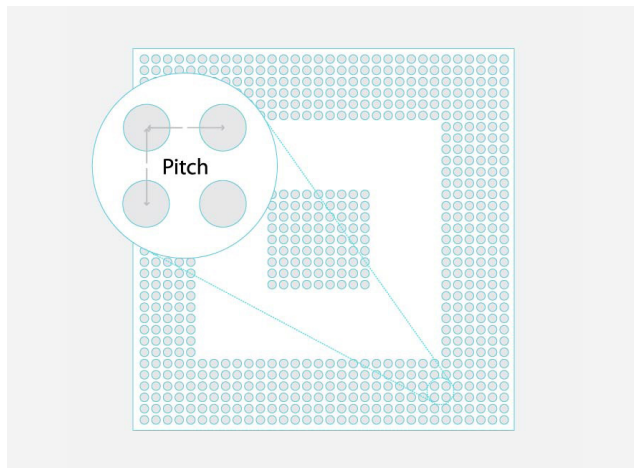


# Basics of Ball Grid Arrays (BGAs)

Ball grid arrays 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. These challenges need to be faced head on since the ball grid array (BGA) is prevalent in modern PCBs. 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 improving BGA designs.

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.



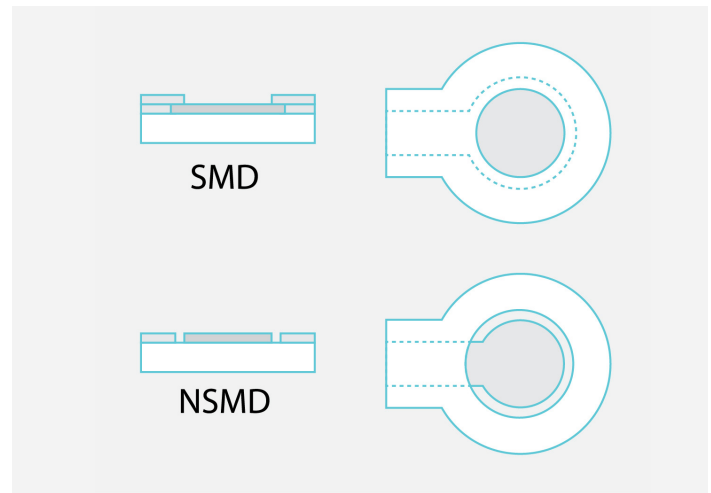
As the different ball patterns on the board give rise to different optimal fanouts or escape routing, it's important to carefully look at the patterns and ask various questions. How far apart are the balls. Are they parallel and equidistant, or do they have greater spacing in one direction. Does the pattern change. What is your minimum trace width and spacing.

*Advanced Assembly was founded to help engineers assemble their prototype and low-volume PCB orders. Based on years of experience within the printed circuit board industry, Advanced Assembly developed a proprietary system to deliver consistent, machine surface mount technology (SMT) assembly for 1-35 boards in 1-5 days. It's our only focus. We take the hassle out of PCB assembly and make it easy, so you can spend time on other aspects of your design.*

With answers to these questions in hand, you can either 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.

Due to their complex nature, BGA's 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, you should use mechanical drilling for the vias. However, this has the drawback of creating 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 is 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.

A BGA, having the leads beneath the package itself, is most easily soldered using a reflow process. Small balls of solder are attached to the part itself by the manufacturer which, when flowed, create the electrical and physical connection between the BGA and the PCB. Most assembly houses use automated X-ray inspection (AXI) to verify the solder joints since there is no other way to see the connections.



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.

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. ■