

Chapter E: BGA

2. Conclusions and Guidelines

2.1 Technical Issues

2.1.1 Basic properties of BGA packages

The following table summarizes some key features comparing a conventional PGA package (pin grid array), to plastic QFP and BGA and TBGAs (Hwang p14).

Table 1. Typical key features of various packages (adopted from Hwang).

Feature	PGA	QFP	BGA	TBGA
Lead count	208	208	225	224
Pitch (mm)	2.5	0.5	1.27	1.27
Foot print (mm ²)	1140	785	670	530
Ratio of package to die	11	8	7	5
Inductance (nH)	5	6	4	3
Capacitance (pF)	7	0.5	1	1.5
Thermal resistance (°C/W)	2.5	0.5	10	1.5

2.1.2 Electrical performance

Low noise is achieved with TBGA and SBGA packages owing to ground planes incorporated into the structures. The SBGA is reported of being capable of performing at very high speeds (2 GHz) (Lau p48). With the basic PBGA, however, the most natural design for economical reasons is probably the worst package for electrical performance (Lau p351), and the standard OMPAC needs to be electrically enhanced if it is to be used in high-frequency applications, i.e. above some 50 MHz.. The conductors are essentially doubling over on themselves, separated only by the thickness of the substrate. This results in a very high mutual- and self inductance of the substrate, generally higher than of QFPs of comparable lead counts. Even so, the standard OMPAC design is most often adequate for general and moderate devices and applications. The main advantage using the PBGA, is of course the shorter and more widely spaced interconnects (solder spheres) giving an electrical connection of significantly lower inductance than traditional packages. Additionally, most high-performance PBGAs have greater potential to system performance increase than other package types mainly because of the smaller size and thus shorter conductor paths required.

Table 2 below is a comparison of typical PACKAGE features and related electrical parameters of a plastic BGA and a plastic QFP. The three electrical parameters capacitance, inductance, and resistance, are inherently found in every packaging concept. The resistance may cause signal line DC drops while contributing to charging delays in RC networks. On the other hand, the resistance may also reduce undesired noise at a system level. The capacitance is mainly responsible for signal delays and can be reduced by reducing the physical dimensions of the RC networks. The inductance also contributes to the switching noise and delays associated with the packages. A low dielectric constant is favourable both for signal delay and crosstalk, which is the coupled noise from busy signal paths to idle paths caused by mutual capacitive and inductive coupling. A typical value of the dielectric constant (ϵ_r) for plastic BGA substrates is in the range 3.5 to 5.

Table 2. Typical package features and electrical parameters comparing two plastic packages (After Lau p.371)

Attribute	PQFP208	PBGA225
Lead count	208	225
Pitch (mm)	0.5	1.5
Size (mm)	31x31	27x27
Shortest/longest signal path(mm)	10.2/13.7	6.8/22.5
Smallest Inductance (nH)	6.3-7.1	3.3-5.8

Largest Inductance (nH)	8.9-9.8	10-11.2
Inductive coupling (K_L)	0.68-0.70	0.12-0.25
Capacitance (pF)	1.61-2.38	1.28-1.31
Signal propagation delay (ps)	9-14.5	5-9.1
Switching (ΔI) noise (mV)	730	550
Crosstalk (mV)	510	90
Direct current resistance (m Ω)	70-80	20-24

2.1.3 Thermal performance (heat dissipation and cooling techniques)

A common perception about the BGA packaging concept is that it is superior to both QFP and PGA in terms of thermal performance. However, plastic BGA are not very different from plastic QFP or PGA packages, a slightly improved response may be seen owing to the overall shorter thermal paths associated with a BGA package. The thermal conductivity of a PBGA BT substrate is typically on the order of 0.2 W/m $^{\circ}$ C which is approximately ten times lower than that of the silver filled die attach adhesive used to bond the back of the chip to the substrate die pad. Plastic packages, especially the OMPAC type, suffers substantially in this respect from the overmould compound being very thick, up to 1 mm, further aggravated by a thermal resistance value of around 0.7 W/m $^{\circ}$ C. There are a number of heatsinks available to attach to the BGA package body, increasing the cooling capabilities. However, reliability studies involving PBGAs have shown that the heatsink itself may have a negative effect on the solder joint reliability because of the mechanically induced stress associated with the use of heatsinks (Mawer 1997).

2.1.4 Reliability issues

This section addresses reliability aspects of great importance to any user of BGA packages. Results from reliability tests, performed at IVF, under harsh automotive-like environmental conditions are presented, involving different BGA packages, including OMPACS, Super BGAs and Tape BGAs, as well as fine-pitch quad flatpacks serving as a reference. All packages were mounted to standard gold-plated FR-4 test boards, specifically designed for the tests. Daisy Chain packages were used to check the electrical performance of the solder joints. The environmental tests comprised temperature cycling from -40 to 100 $^{\circ}$ C, with and without simultaneous exposure to vibration (8 G, 10-500 Hz) during the high temperature phase.

The results show that non-soldermask defined (NSMD) pads on both board and package give superior performance for the PBGA361, compared to when solder mask defined (SMD) pads are used. The 1.27 mm pitch PBGA361 passed 5000 cycles and the PBGA256, as well as the QFP reference samples, passed 6000 cycles from -40 to 100 $^{\circ}$ C without failure. Cross-section analysis after 6000 thermal cycles, performed on 1.5 mm pitch PBGA225s, shows extensive cracking, which however could be avoided with the use of a suitable underfill.

Results from measurements of intermittent failures during temperature cycling of the 1.27 mm pitch PBGA361s, recorded by a high-speed event detector, are shown in Figure 7 below.

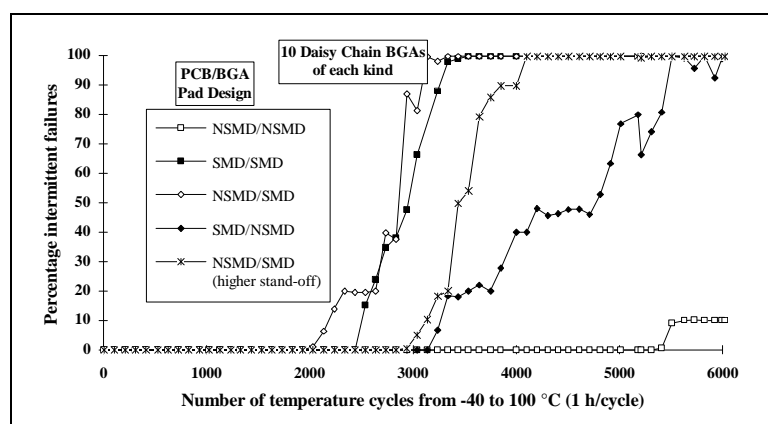


Figure 7. Pad design influence on reliability of 1.27 mm pitch PBGA361 during temperature cycling.

As evident from Figure 4, the most important factor for achieving high reliability is to have an NSMD pad on the BGA package substrate. The use of NSMD pads on both the board and the BGA package did not yield

any intermittent failures up to 5500 cycles (1 hour per cycle), while switching to SMD pads on the board caused failures to start at approximately 3000 cycles. However, if NSMD BGA packages are not available, it is not recommended to use NSMD pads only on the board, as this combination (NSMD/SMD) gives an even shorter life than SMD pads on both sides. The reason for the lower reliability with the NSMD/SMD combination is assumed to be a stress concentration to a weaker point at the package-ball interface. The stand-off is approximately the same in the two cases NSMD/SMD and SMD/NSMD and therefore cannot explain the different failure behaviour. However, it is clear from the diagram that an increased stand-off for the NSMD/SMD combination actually delays the onset of intermittent failures to a value slightly better than SMD/SMD.

The location of the open joints in the PBGA361 Daisy Chain packages, was found by manually probing the packages that had shown intermittent failures during temperature cycling, mainly packages with SMD pad designs. The result is depicted in Figure 8 below, which with remarkable consistency shows that electrical integrity is lost predominantly in joints near the die edge. Each black dot in Figure 8 denotes an open circuit in either or both of the two joints forming that particular pair of joints.

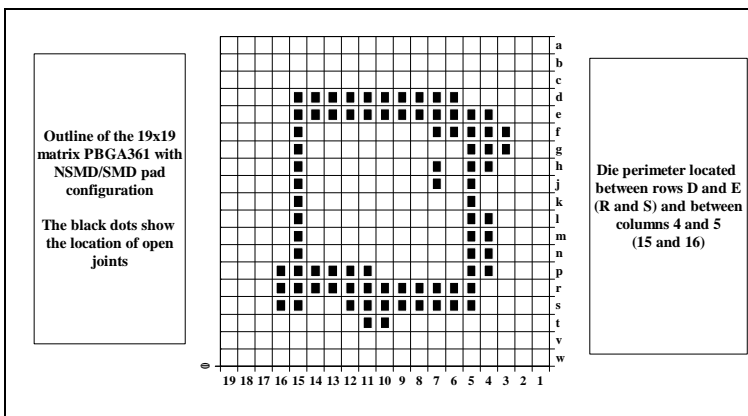


Figure 8. Location of open joints in the Daisy Chain circuit of a PBGA361 with NSMD/SMD pad design after 5000 temperature cycles from -40 to 100 °C (1 hour/cycle) [Rörgren 1998].

The occurrence of intermittent failures under temperature cycling conditions was also measured using another test board, to which different types of BGA packages had been mounted. 16 test boards were hooked up to the event detector described above. No failures were observed for neither the PBGA256 nor the QFP reference samples during the 6000 cycle long test. A few intermittent failures were detected for the SBGAs and the TBGA, as given in Table 3. Six out of the 16 test boards had also been exposed to vibration during the high-temperature phase of 250 thermal cycles (three boards from cycle 1032 to 1282, and three more from 2037 to 2287). This, however, did not seem to have any effect on the devices, i.e. the failures either occurred before vibration exposure (SBGA) or within the control group not subjected to vibration (TBGA).

Table 3. Intermittent failure registration during temperature cycling from -40 to 100 °C.

Device Type	No. of Pins	No. of Devices Tested	No. of Cycles to First Failure	No. of Failed Devices at 6000 cycles
PBGA	256	32	-	0
PQFP	160	16	-	0
PQFP	256	16	-	0
SBGA	256	32	527	5
SBGA	352	15	383	4
TBGA	240	31	3478	15

Recent studies involving ceramic BGAs generally show lower reliability than for plastic packages when mounted to organic substrates like FR-4, even though the CBGAs do not possess coplanarity problems as commonly found with PBGAs (Ghaffarian 1998).

2.2 Production Issues

2.2.1 Design rules and compatibility

As shown in 2.1.4. the reliability of the BGA joint is heavily dependant on the package pad design, and to a lesser extent, the motherboard pad design. The question "Solder mask defined or non-solder mask defined pad?" is of crucial importance, and the answer needs some thought. Figures 9 and 10 illustrates the differences between these SMD and NSMD pads, as they are most commonly referred to.

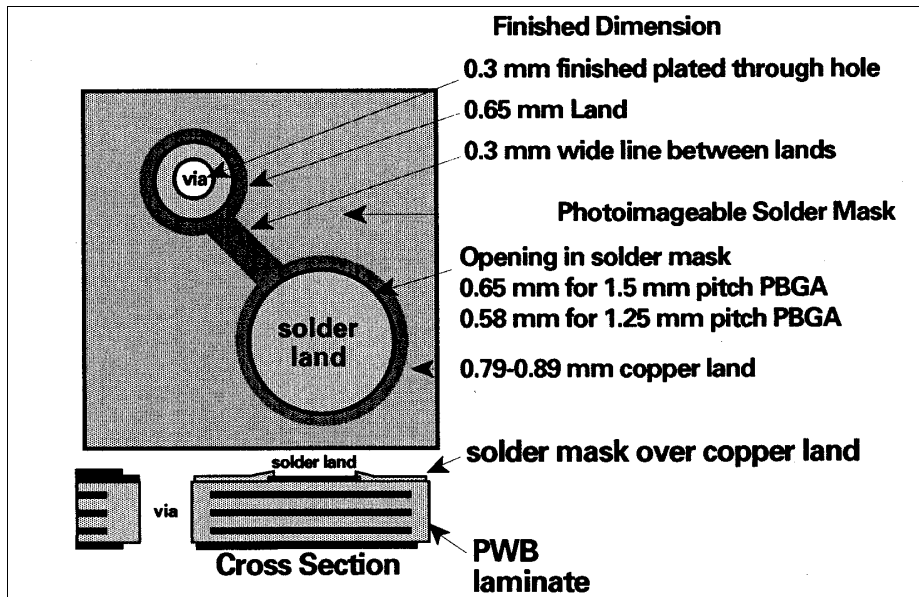


Figure 9. Solder Mask Defined (SMD) pad design for PBGA.

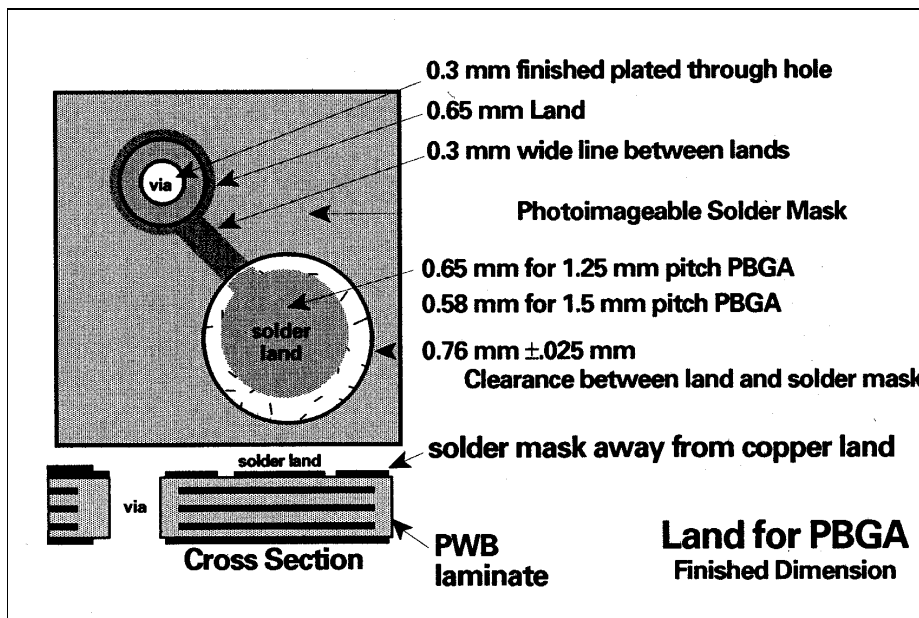


Figure 10. Non-Solder Mask Defined (NSMD) pad design for PBGA.

Clearly, the use of NSMD at BOTH package and motherboard gives optimum reliability and hence longer life owing to a lower stress concentration in the joints (Rörgren 1998). The use of SMD pads results in a higher joint stand-of, which with the same pad configuration would improve performance, but early failures are encountered as cracks are initiating at the edge of the solder mask. However, when NSMD BGA packages are not available, it is not very wise to chose NSMD pads on the motherboard. It is therefore recommended to use the same type of pad designs and of course NSMD if available. Determine the type and diameter of the BGA pads and use the same type on the motherboard, maybe somewhat smaller since

stress concentration is higher on the BGA side. Designing with NSMD pads can be more difficult since tolerances get more critical and routing may get tighter. An increased risk of motherboard pad lift (delamination) has also been discussed for NSMD pads.

2.2.2 Tools and investments

In general, no special tooling or equipment is needed in order to use BGA packages in a conventional SMT assembly line. Hence, no additional investments are generally needed, except for circumstances under which inspection has to be made by means of x-ray (see 2.2.5). Thus, ordinary equipment such as stencil printers, pick & placement machines, and reflow ovens are usually more than adequate from a production point of view. Rework and repair stations intended for fine pitch QFPs are equally well suited for BGA rework tasks.

Equipment for real-time x-ray analysis, in-line or for batch use, are available from a number of equipment manufacturers. Some of the most well known are listed below. The price of even a simpler piece of x-ray equipment can be rather steep. There are possibilities to rent x-ray equipment for process set-up or there are several research facilities or institutes throughout the world providing x-ray analyses on an hourly charge.

- Feinfocus Röntgentechnik GmbH, Germany (<http://www.feinfocus.com/>)
- HP Four Pi (x-ray Laminography), USA (http://www.tmo.hp.com/tmo/pia/wirelessiia/PIAProd/English/wireless_other_xray.html)
- Nicolet Imaging Systems, USA (<http://www.nicimg.com/>)

2.2.3 Assembly issues

Assembly using BGA packages is, in principle, rather straight forward. The SMT processes used are essentially the same as for fine-pitch QFPs. Companies already using BGAs have reported significantly improved yield figures as well as increased productivity levels. As an example, switching from 0.5 mm QFP to BGA permitted a major multinational manufacturer to lower the assembly defect ratio from 200-300 ppm down to only 3 ppm. However, there are some areas of concern if care is not taken to observe all the particulars (and they are often changing) about new BGA packaging concepts. To minimize the occurrence of assembly defects, examples of which are given in 2.2.4, some rules-of-thumb may be worth adhering to.

- Check both PCB and BGA planarity, as warpage often results in open or otherwise defective joints
- Select appropriate solder paste (flux, particle size etc) compatible with solder mask and chemicals used
- Select suitable reflow profile with respect to paste and components. Measure the BGA joint temperature
- Make sure the PBGA packages are moisture-free - or have them baked
- Assure good wetting is achieved, or you may encounter voids and solder balls
- Always use fresh paste. Do not exceed shelf life or open time.
- Remember: Solder paste is hygroscopic. Check ambient conditions regularly.

Solder paste deposition

The most common way to apply solder paste for BGA assembly is by means of screen or stencil printing. The process is in principle the same as for QFPs. Since BGAs typically use a pitch far coarser than for (fine-pitch) QFP, the stencil used can also be thicker. However, when dealing with eutectic solder spheres, it is actually not the BGA that determines the thickness of the stencil but other, mainly, fine-pitch components. This is because roughly 90 percent of the solder volume of the final BGA joint comes from the reflowed solder sphere itself. Some manufacturers even use only a tacky flux for attaching BGAs, and it works very well, but this is not generally recommended since the solder paste deposit helps overcome coplanarity problems which are frequently encountered PBGAs. When using non-eutectic solder spheres, as is the case for TBGAs, the paste deposit and hence stencil thickness becomes a much more important parameter. S < a rule of thumb, select a stencil thickness as a compromise with major emphasis on the finest pitch used on the board.

Just as for ordinary SMT, there are several different types of stencils to choose from. Chemically etched stencils are the most economic and established alternative, while additively processed stencils, often referred to as electroformed, are the state of the art for very fine pitch applications - and thus rather costly.

Following the argument above, the BGA technology itself does not require the use of any exotic (costly) stencil type owing to the rather large pitch and with it the possibility to use relatively coarse-grain solder pastes. Also bear in mind that a laser cut stencil, being a very wise choice for ultra fine pitch QFPs, becomes quite expensive for large I/O BGA purposes since laser cutting is a sequential process and each hole in the mask adds to the total cost. Thus, choose a conventional, 200 µm thick, chemically etched stencil, unless you in addition to BGAs also are putting very fine pitch packages on the assembly. If so, let the very fine pitch determine the choice.

Device placement

The PBGA exhibits a strong self centering effect, so precise alignment is not critical. Up to 0.3 mm of misalignment has been reported not to cause any excessive bridging after reflow. In most cases, conventional placement systems can be employed, using either the package outline or the position of the solder spheres as a placement guide. Vision systems may also be used, increasing the productivity, though a few problems have been encountered. Firstly, it has been reported that the colour of the back side, i.e. the substrate laminate) of PBGs may vary from manufacturer to manufacturer or even worse, from batch to batch. This means that when automatic vision alignment is employed, the threshold or acceptance level must be adjustable, and adjusted if varying colours are encountered. Furthermore, some early vision systems were not capable of using all the solder spheres for optical alignment, nor were they able to check whether all spheres were present. These older systems, typically using only the outer row of solder spheres, are now becoming obsolete and are getting replaced and in the near future this problem should not be heard of any more. A modern vision system should also be able to optically align a BGA package with corner balls removed and/or with additional thermal balls without any interference to the alignment procedure.

Reflow soldering

The method of choice for soldering BGAs is of course mass reflow soldering. This can be performed in several different ways using nitrogen, air, IR, full or partly convection, in almost the same ways as when soldering QFP assemblies. One difference is the thermal mass of a BGA package, in general being somewhat higher. This means that there is a slight risk of having too low a temperature in the middle of a BGA while small chip components still get a little too hot. This is particularly true if IR and/or ceramic packages are used. With a properly run convection oven, a temperature difference of 5 - 8 degrees measured as described above should be attainable. To achieve this, a slightly slower belt speed may help, together with a little lower reflow peak temperature. However, measurements of the actual joint temperature are always necessary in order to optimize the process.

One other major issue of concern when using plastic BGA packages, is the fact that they indeed are sensitive to humidity. PBGAs are not hermetically sealed and, due to package construction with non-covered substrate edges, they will absorb moisture while in storage. If the package is heated too quickly, such as during reflow, the package will crack (the "popcorn" effect) from steam pressure build-up. It is therefore of utmost importance to assure that the plastic packages have been kept in their dry packs and have not been exposed to ambient conditions for more than the specified or accepted period of time. If you are not sure of the packages history; make sure you have them baked again, e.g. at 125 °C for 24 hours.

2.2.4 Inspection and Workmanship Standards

Assembly defects related to defective solder joints are generally thought to be detected in the final production stage, being either visual inspection or functional testing, or both. Failures not detected or any defects that will eventually lead to a failure with time or environmental exposure, are in this context treated as a reliability concern, as discussed in 2.1.4.

The most likely defects to observe upon inspection include all or most of the following:

- Placement accuracy
- Solder bridging (short-circuits)
- No contact (opens)
- Excessive or insufficient solder
- Solder balling (beads)

- Voids

Because of the "secret" nature of the BGA package, with the joints hidden under the device, x-ray inspection is really the only possibility to actually inspect all the joints. However, once the process has been set up and qualified using x-ray, a sound statistical process control should be sufficient to maintain the quality goal. Most of the defects usually encountered, except the hard failures "shorts" and "opens" need to be judged according to some sort of Workmanship Standard in order to qualify the solder joint as accepted or not. In addition, flux residues or other types of chemical contamination, as well as signs of cracking (pop-corning) must be taken into consideration. Of course, all of this. Including the acceptance criteria, is depending on the expected service environment of the product and may therefore differ from case to case.

2.2.5 Rework and Repair

Conventional rework stations are often capable of BGA rework as well. There are also equipment especially designed for BGA, incorporating both hot gas and IR heating, to take faulty BGA components off the board of putting new packages down. Redressing of the board site is most often necessary, before new paste (or flux) is applied by mini-stencilling or dispensing. A package removed for some other reason than being defect itself cannot be put back since the eutectic solder spheres will have melted while the package was removed. An obstacle to rework is the case with double sided assembly, generally making rework impossible if the board is densely populated and designed without repair in mind.

