

Establishing BGA Acceptance Criteria

By Don Miller

The use of ball grid array (BGA) and other area-array devices is quickly becoming a standard element in modern printed circuit board (PCB) design. This leaves many electronics assemblers faced with an inspection dilemma—ensuring correct assembly and achieving process yields when traditional verification methods no longer are sufficient.

Today, an increasing number of manufacturers are selecting X-ray systems to satisfy their inspection requirements. By using X-ray inspection, the characteristics of hidden solder joints of BGAs, microBGAs, and flip-chip devices can be checked in a simple, reliable, and cost-effective manner early in the production run. Until BGAs were incorporated into product designs, most PCB manufacturers did not use X-ray systems in their inspection process. Traditional methods, such as visual inspection, electrical tests including manufacturing defect analysis (MDA), and in-circuit and functional tests were used to test PCB components.

These methods do not provide adequate detection of hidden problems such as voids, cold solder joints, and poor solder adhesion. Only X-ray inspection can detect these problems effectively in addition to monitoring the quality process and providing the immediate feedback required for proactive process control.

BGA Acceptance Criteria

The issue of acceptable criteria for installed BGA devices on PCBs is of prime importance (Figure 1). At this time, there is no well-defined or documented industry standard available to use as criteria for determining acceptable conditions for reflowed BGA/PCB joints.

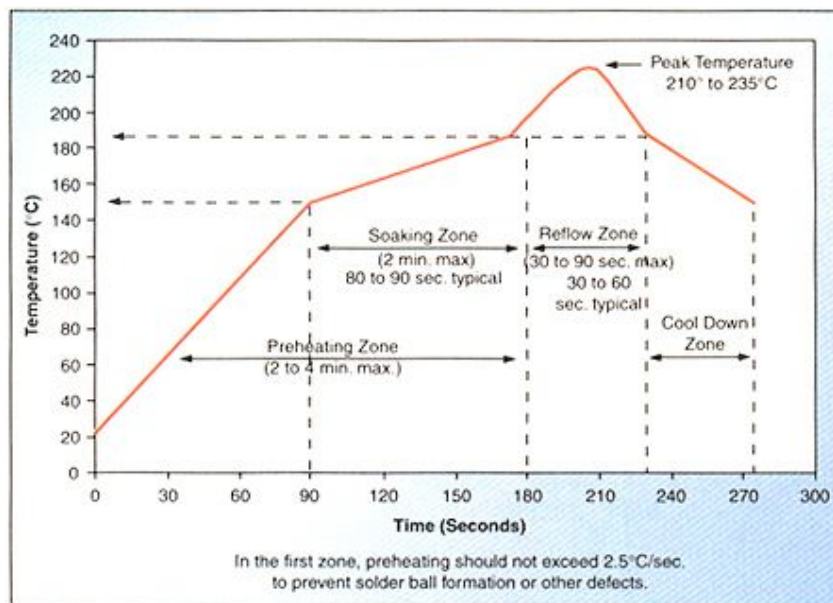


Figure 1: Typical reflow profile

Several methods have been proposed for determining the quality of these joints. Techniques such as microscopic evaluation, ultrasound, X-ray laminography, acoustic microimaging, electrical tests, and standard transmissive X-ray have been considered or are in use today.

Yet the only practical method to evaluate a reflowed BGA connection in a production environment is transmissive X-ray. Visual evaluation checks peripheral rows of BGA joints, but inner solder joints under the BGA package cannot be evaluated visually. Ultrasonic techniques require extensive interpretation, X-ray laminography is prohibitively expensive for most applications, and electrical tests sometimes provide inconclusive data regarding the quality of the soldered connection.

Other more esoteric techniques are new, largely untested, or not in general use. Many of these techniques supply indicators of the quality of the soldered joint but do not provide sufficient information to qualify the soldered condition of all the BGA/PCB joints.

For example, peripheral visual inspection gives a general understanding of the joint conditions on the exterior rows of the BGA only. It is not a valid determination of how well the internal rows of solder joints are reflowed and whether the internal rows actually are properly joined to the circuit pads.

Transmissive X-ray, which can be used to evaluate internal rows, depends on the X-ray system's capability to resolve data. As an example, consider a 0.050" dia ball resting on a 0.050" circuit pad. If the internal voids in the solder joint cannot exceed more than 20% of the ball diameter to meet the acceptance criteria, the X-ray system must resolve a 0.005" dia void as a minimum. Some systems resolve voids as small as 10 microns in diameter.

With a good solder reflow profile and correct PCB layout, the solder paste will reflow properly and form a meniscus encapsulating the solder ball (Figure 2).

Under conditions where the reflow profile is not quite correct, the flux in the solder paste will not properly outgas. The result will be voids in the solder connection or inadequate reflow of the solder, causing poor or cold solder joints.

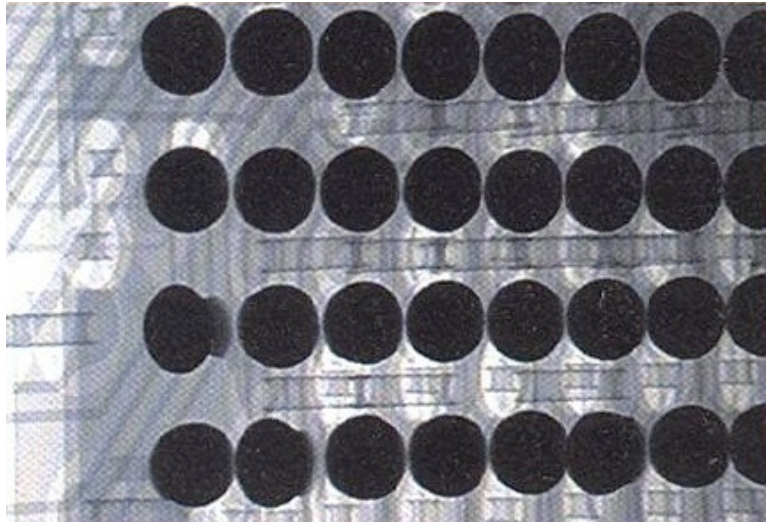


Figure 2: Oblique views showing open pad/BGA connections

Solder Reflow Profiling

Improper solder reflow or void conditions can be minimized in certain ways. First, a proper reflow profile must be developed and verified. The best approach uses a pass-through type profiler to determine the temperature range and profile at the BGA ball/pad junction as the unit travels through the reflow oven. This device, which consists of a datalogger with thermocouples attached to the PCB, records the temperature profile at the ball/pad location on the PCB.

The solder reflow profile is important to assure that the reflow profile for the solder paste and the thermal profiles of the PCB are correct. A scrap bare PCB and dummy or scrap BGA parts may be used to make this profile and evaluation. Confirming that the temperature profile is within acceptable limits to achieve proper solder reflow is of paramount importance.

The beginning reflow profile development program should consider the thermal mass of the PCB with components installed on the board. At least three passes through the solder reflow oven should be performed to establish the base reflow profile. Thermocouples should be placed at multiple locations on the product to ensure that the correct profile for solder reflow has been exhibited. If the initial PCB has dummy parts installed, the PCB should be X-rayed at several positions with top-down and oblique views in at least two directions as each unit is processed.

X-Ray Evaluation

After each X-ray and specifically on the very first populated PCBs, it is appropriate to evaluate the X-rays for lack of reflow, exemplified by rough, unsoldered areas near the edge of the pad, opens, shorts, bridges, and voids. These conditions indicate a low temperature range that does not allow the paste to reflow adequately. Shorts/bridges may be evidence of too high a temperature excursion, permitting the solder to be liquid too long and allowing it to flow off the pad and short to an adjacent pad.

Voids are an area where subjective evaluation is required. Even under conditions where a void is detected, the joint still may be adequately soldered to the pad. The ideal condition is no voids in the solder joint. However, voids may appear as the result of trapped flux pockets, contamination, and uneven dispersion of tin/lead or flux in the solder paste. Also, a warped PCB may contribute to inadequately soldered connections. Open soldered joints also may exist

In many cases, only an oblique view of the PCB can show these conditions. Due to the solder density, a top-down view may not disclose an open ball/pad junction.

The question now: How many voids and what size voids can be tolerated and still allow the PCB to function properly? Some literature indicates that a single void as large as 50% of the diameter of the ball is acceptable if the ball is encapsulated by reflowed solder (Figure 3).

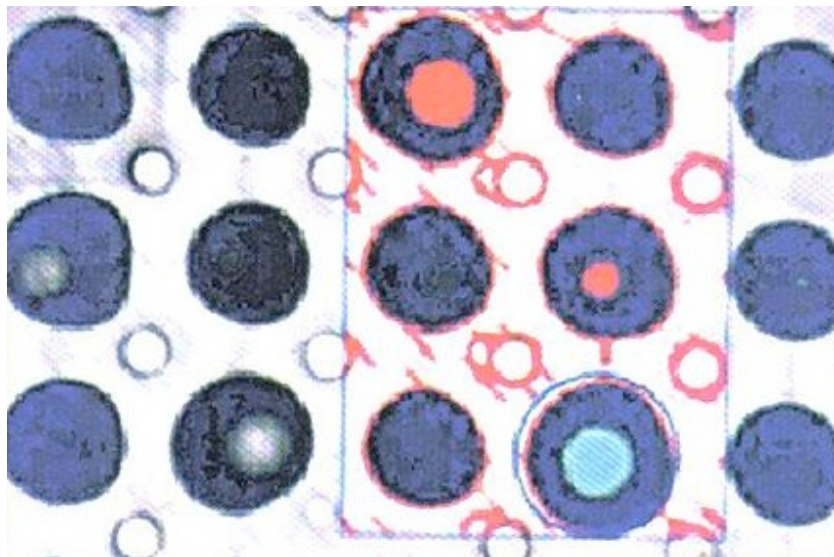


Figure 3: Balls with single voids exceeding 20 percent of ball diameter

Electrical contact is through the solder, and if the ball is adherent to the pad, a 50% void still may permit the BGA to work, although to a very marginal standard. Electrical performance may not be impaired, but thermal and mechanical stress issues must be considered.

Flexing of the PCB as a result of stress from normal environmental use could result in a solder joint with a void toward the outside of the ball/pad junction or a large void to crack. Voids greater than 35% of the BGA diameter or that show up on the outside of the joint during X-ray should be considered as an unacceptable condition.

If a void is completely encapsulated, there is much less likelihood of a solder-joint crack occurring since the stresses generally would be applied equally to the ball/pad joint. Nevertheless, a void larger than 35% of the diameter of the ball is indicative of a process-related problem and should not be accepted.

PCB Layout

Since success of the soldered joints on a BGA is a function of the PCB, there are three critical areas to consider in PCB layout.

Thermal Management

The layout of the PCB must consider its thermal mass. For example, grouping BGAs close together in one area of the board can cause a thermal imbalance in the way the PCB reacts while in the reflow oven. During reflow, it is important to bring the entire board, evenly, to the solder liquid phase.

By grouping a number of large BGAs in one section of the board, excessive heat may be required. This heat could result in burning the components in areas of the PCB that are less heavily populated. Conversely, if the less heavily populated areas of the PCB reach a faster solder liquid phase, the BGAs may not have adequately reached the point where the fluxes have outgassed from the joints. This causes voids or lack of fusion of the balls to the pad.

Vias

Vias or through-holes often are designed into a PCB. It is extremely important that any vias adjacent to a BGA pad be very well tented with solder mask. Failure to tent vias can lead to thermal problems, and any excess solder can run from the pad to the via, causing a short from the pad to any adjacent via. (Figure 4)

Solder Pad Geometries and Diameters

The trend toward miniaturization and higher array densities associated with chip-scale packages is having a direct impact on solder pad geometry and diameters. Likewise, BGAs come in different sizes, shapes, and complexities. The continuing reduction in package size, solder-pad geometries, and diameters inevitably will lead to the need for higher-resolution inspection techniques. It also will create continuing problems associated with BGA/PCB joint adherence.

A Proposed Acceptance Standard

An Acceptance standard will help the x-ray inspection system identify many typical solder problems that can be associated with the use of BGA devices. They include:

Voids

Voids in the soldered joint shall not exceed 20% of the diameter of the ball, with no single void showing on the exterior of the solder joint. Multiple voids can appear in the joint providing the sum total of the voids does not exceed 20% of the diameter of the ball.

Unsoldered Joints

Unsoldered joints are not permitted in the ball/pad/BGA joints.

Bridging and Shorts

There shall be no shorted or bridged solder joints unless they are specifically designed into the underlying circuitry or into the BGA.

X-Ray Evaluation

PCBs containing BGAs shall be evaluated using an X-ray system capable of resolving a hole whose diameter is less than 100 microns. The X-ray system must be capable of permitting the unit under test to be viewed top-down and obliquely in two directions.

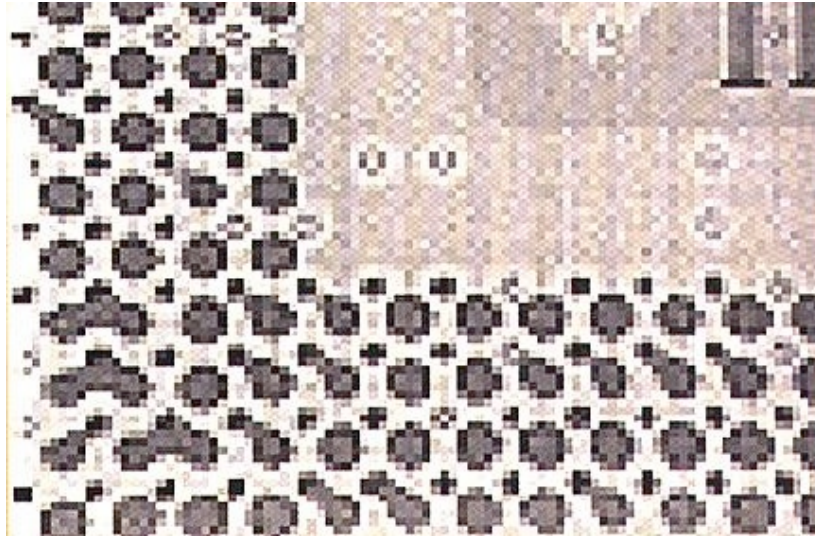


Figure 4: Untented vias causing solder shorts form solder ball via

Conclusion

X-ray inspection systems are a proven tool in detecting hidden solder joints, helping establish and control the manufacturing process, analyzing prototypes, and identifying process faults. They are effective, cost-efficient and, unlike MDAs, in-circuit testers, and vision inspection systems, can quickly identify shorts, opens, voids, and misalignment in solder balls, on boards with BGAs, and other area-array packages.