Common BGA Rework Mistakes and Issues

Reworking a ball grid array (BGA) is one of the most challenging operations when it comes to printed circuit board (PCB) assembly or repair. The reason for this is because of the unique nature of BGAs: they are designed to permanently mate the PCB and its attendant integrated circuit and getting to the connecting pins (or heating the entire device to the heating point of the solder, possibly ruining it) is almost impossible.

One of the most common reasons for BGA repair and rework is the fact that the balls of tin solder (the "ball" in ball grid array) become unreliable due to the continuous heating and shrinking of the board or its components as caused by the normal heating cycle of the board. While the procedures for BGA rework are well-defined, there are still several different rework issues and mistakes that can happen over the course of a standard BGA repair or rework, leading to either a series of costly repairs or being forced to scrap the BGA entirely and start anew.

So what are some common BGA rework mistakes?

- **Inadequate operator training.** BGA repair technicians need to be well-trained, with a practiced, developed skill set. Part of this is being able to understand what materials they're working with, how they interact, and how their tool selection can impact this. A good chunk of this is a solid working knowledge of the process of BGA repair, as well as being able to evaluate a potential repair situation before work commences.

  So what are the steps of BGA repair?

  1. First, remove the components, such as the PCB. This is done by preheating the PCB until the solder becomes molten, and then removing the PCB with a vacuum.

     - One thing to be careful of here is damage to the BGA pads when the BGA itself is removed. If the BGA has not been heated properly or full saturation and flow of the solder has not been achieved, then attempting to remove the BGA pads can simply turn the entire component into an expensive mess.

  2. Next, the residual solder needs to be removed. The site should be fixtured to hold it down, then preheated again, with a vacuum used to remove the old and excess solder.

  3. Once all the excess solder has been removed, the BGA can be reballed. Reballing involves using a stencil to set new spheres of solder into the BGA.

  4. After the excess solder has been removed and the BGA has been reballed, the components and PCB can be re-soldered and reattached. This is much like the component removal from step 1, except that now solder can be printed onto either the PCB or BGA as necessary before soldering begins.

  While this process seems simple enough, there are several things that need to be considered when it comes to the finer points of BGA repair and it's these things that a repair technician needs to know when he or she undertakes a BGA repair. Technicians should be aware of things like the possibility of thermal mismatch between the PCB and BGA, the possibility of solder heat liquefying other components and correct thermal and flow management procedures, among others.

- **Improper preparation.** Whether it's a simple rework or a full-on BGA repair, the key to a job that lasts a long time is in how the technician prepares for it. As explained by the process above, there's a lot of preparation that goes into a proper BGA job and knowing that your technician is taking those steps can being some peace of mind.

  Long before he or she so much as touches the BGA, there is research that the technician needs to do and decisions that they need to make. They need to make an accurate assessment of the BGA's solder ball size so that they can make sure they're using the right size and then they need to check things like the coplanarity of the solder balls and the rest of the device. Once this research has been done, the technician can make a couple decisions, such as whether they want to use solder paste, what they think the right stencil is for the job, and choosing the right alloys and chemistry to ensure success.

After the research has been done and decisions have been made, there are several things a good technician can do before a repair that can make or break it, including:

- assessing damage to the solder mask
- checking for contaminated or missing pads at the PCB site
- preparatory steps, such as baking the moisture of the PCB and BGA and removing or otherwise shielding heat-sensitive components in order to prevent possible damage or reflow.

- **Incorrect equipment selection or improper material choices.** This goes hand in hand with the earlier point about improper preparation, but, as they say, you need the right tools to get the job done. The equipment a technician uses to fix BGA rework issues not only needs to be flexible and capable of performing a number of tasks, but also capable of standing up to repeated use in a controlled process.

  A technician needs to not only be able to choose the *right* equipment for a job, but also the *best* equipment for a job. Get equipment that gives you a combination of abilities and options, from closed-loop thermal sensing and control to the ability to deliver heat when and where the process requires. Make sure your equipment gives you the ability to handle products as necessary, both for part removal and part replacement.

  Making sure that you have the best equipment available and that you don't cut corners can cut down on some of the BGA rework issues that you might face in the future. Some of the BGA rework mistakes that can be avoided due to proper material choices and proper equipment selection include solder joints voided by excessive solder. Solder joints can be voided due to choosing the improper type of solder paste for the material in question, which can result in the BGA needing residual solder removed and the process beginning again if the voiding is over 25 percent. Other possibilities include flat-out rejection of the BGA in question or simply additional rework, especially if the solder attachment points are compromised.

- **Poor development of the BGA rework profile.** Developing a proper thermal profile for the individual components in your BGA rework or repair is one of the most important things you can do, and knowing what constitutes a proper profile and what doesn't can not only save you time and effort, but cut down on the incidence of rework and repair issues.

  Since the process for BGA rework and repair is a repeated process, if you don't have proper thermal profiles for your components, you'll struggle with putting together a successful, repeatable BGA repair or rework process.

  Not only that, but having a poorly-developed thermal profile means that there's an increased chance of damage to the BGA, the PCB, or the other components, which means you'll spend more time reworking the same site. The best way to develop a good thermal profile is to ensure that your thermocouples are properly placed (especially when it comes to the PCB) and then analysis the data that they provide you. It's only with good, solid data that you can put together a workable, repeatable thermal profile.

- **Collateral heat damage.** Collateral heat damage is one of the things that can happen if the thermal properties of your chosen materials are improperly profiled. As the name implies, not only can collateral heat damage cause damage to nearby components, but it can also lead to a host of other BGA rework issues, depending on which component was damaged and how.

  Having solder from adjacent components reflow after it has been set can lead to problems as well, including but not limited to pad and lead damage, solder oxidation and starved joints, any of which can open you up to other problems as things progress.

A good technician needs to not only be aware of where he or she is directing heat, but also *how much* heat they're directing. A temperature that's good for one component may well be catastrophic for another, depending on the materials involved. Having a well-developed thermal profile for all of your components, as well as tight process control, can help minimize the migration of heat away from the component being reworked, preventing future headaches in the process.

- **Insufficient or improper post-placement inspection.** BGA components can be complex pieces of tech and the world beneath them can often seem like some mysterious, confusing place. No longer. With the rise of x-ray inspection machines, it's become that much easier for a technician to find and diagnose common BCA rework issues. Issues that would once require costly disassembly to diagnose, such as poor alignment, poor placement or excessive voiding are now able to be diagnosed by simply looking at an x-ray.

  However, a technician still needs to be trained in the use of an x-ray machine. He or she needs to be able to properly understand the image that the machine is showing and then interpret the data provided by that image.

  One of the most common problems that can be spotted through the use of good post-placement inspection techniques is an improperly-oriented BGA. Fixing the orientation means more rework cycles and the increased possibility of not only heat damage to the BGA itself, but also to surrounding components.

One other issue when it comes to BGA rework and repair is just overall BGA design. Manufacturers nowadays are tending more and more toward ultra-fine (0.3 mm) pitch BGAs in their devices, with pitch being defined as the space between the center of one ball in the BGA to the center in the next. As the space between the two balls decreases with the drive toward smaller and smaller components, manufacturers have striven to try and increase functionality within the same amount of real estate.

However, this can be complicated by not using the correct design specifications. Since most BGAs in current use are designed to be 0.5 mm or 0.4 mm, trying to design a 0.3 mm BGA by using these same specifications is going to result in design missteps, especially when it comes to the layout of vias and lands. The "land" of a PCB is what the BGA device balls sit up against and eventually get soldered to. It is connected via a circuit called a "fan out" to the "via," which distributes on/off and power signals from the device to its peripherals.

These design specifications can be themselves complicated by the fact that, in producing 0.5 and 0.4 mm BGAs, the industry is already pushing its manufacturing capabilities to their limits. In recent times, tolerances have usually been in the +/- 3 mil range, with the new push to produce 0.3 mm BGAs pushing things closer to the +/- 1 mil range. The tightened tolerances of these new specifications require not only additional equipment for validation, but for rework, as well. 3D x-ray is almost certainly going to be required in the future, along with advance placement and rework stations.

Board fabrication and assembly are already challenging enough. Not only does the PCB designer need to show diligence when it comes to working with fine-pitch BGAs, but so too do rework and repair technicians, especially with regard to possible complications such as the type of the solder paste or underfill used, the type and thickness of stencil used for placing the BGA balls and other general equipment tolerances.