A guide to the BGA Package

1. What is a BGA?

The B(all) G(rid) A(rray) or BGA package invented by Motorola, is now a mainstream packaging technology. The most common example consists of a thin substrate of PCB material onto which the chip is mounted. Under the substrate is an array of solder balls forming the terminations. During reflow these balls fuse with corresponding pads on the Main PCB and form the joints.

2. Why have they caught on?

The BGA excels when it comes to high pin count devices, putting all terminations underneath the package instead of around the edges as they are on a QFP saves a lot of space allowing smaller products to be made. Using a 2-dimensional grid means that ball to ball spacing can be quite coarse compared to the lead pitch of a high pin count QFP - so less problems with solder shorts.

Consequently they are easier to solder, no legs to get damaged and they have a huge self centering effect due to the high solder surface tension effects caused by the array of solder balls.

High pin count QFP's by contrast either have to be bigger to accommodate the same number of edge mounted pinouts or the legs have to be extremely fine and damage prone.

So they are easy to handle and give very high assembly yields - consequently they have started to supplant other package styles in mass production.

3. Are they difficult to place by machine?

From a manufacturing perspective; a BGA is designed to be machine placed using vision systems to align the device to the grid of pads on the PCB. During reflow it has a very strong self centering effect due to the surface tension of all the solder balls – consequently it is quite tolerant of placement errors – as much as half a pitch of misalignment will usually not cause problems. Most machine systems place far more accurately than this.

4. But what about hand placing?

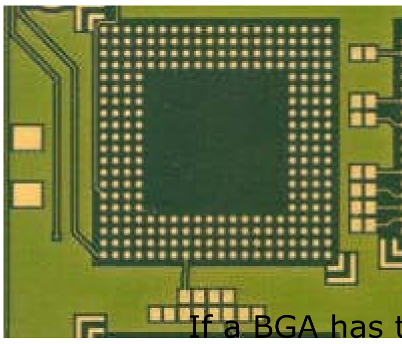
However, every silver lining etc: A BGA is not designed for hand assembly. Of course there will be a very small need to do so – for e.g. a prototype. Whilst this is tricky, it is not, as we shall see, impossible.

5. How do you know if a BGA has fully soldered - and don't you need an X-ray machine?

Once a BGA has been soldered it is impossible to visually inspect the joint – the only viable method is to use x ray or possibly fibre optic endoscopes - so how do you know if it has soldered properly? Well the question should be "why do you want to inspect it anyway?" People feel they need to inspect because they can't be sure of their soldering process. Most assemblers use convection ovens and despite all that the manufacturers claim, there is no way that hot air can penetrate fully under a BGA package that is sitting a millimetre or two from the board and heat every ball the same. The centre balls will inevitably be cooler than the outer ones. What actually has to happen is the package must heat through by conduction and often overheat the outer edge to ensure the centre sees the right temperature..

We use a different type of soldering (see 9) - and we can be certain of the soldering conditions - therefore we can say that since every ball has reached a known temperature - no matter where in the grid they are, the device will have soldered properly.

This is born out in practice - over 2000 BGA's soldered, no reported failures - and we don't have an x ray facility. One customer did x-ray a board and the results were perfect



6. Can a BGA be removed, reworked and replaced?

If a BGA has to be removed it cannot be done without destroying the balls beneath the device. Usually this means the device is scrapped although high value BGAs can be recovered by specialist companies who can re-ball the package so it can be used again. A typical cost of doing this may be £70 so clearly only worth doing on devices worth much more.

We are in the business of low volume manufacture so the BGA device initially presented us with some concerns. However, we have evolved methods of assembly that work and are viable. We have in fact, to date, (2013) placed in excess of 15000 devices – all by hand and without defect. We now have fully automatic placement capability through our MYDATA machine and semi auto placement on our Fritsch MicroPlacer.

7. What can be done at the PCB design stage to make life easier for the assembly company?

Alignment/inspection markings.

As the package itself obscures the grid of connections it is impossible to see if the package is in the right place. For this reason alignment indicators are extremely useful – see photo. Note how they have made two chevron marks on opposite corners – they have used two marks per corner to allow for two different package dimensions – most people only use a single mark per corner. Even a simple dot at each corner will do – two corners minimum but three or all is better still – see photograph.

Please Note that these marks MUST be in the copper itself, silkscreen printing is nowhere near accurate enough for this purpose.

A pin 1 mark that is not obscured by the package – this can be done in silkscreen. It's amazing how many pin one marks vanish once the package is down…

DOs and DON'Ts and things that are OK

Resist defined pads are OK

Don't put vias in BGA pads – unless they are microvias. The solder ball will wick down the hole by capillary action and you WILL get an open. These are non-repairable and not covered by our warranty.

DO make sure vias on short stubs have a resist barrier between via and BGA pad – or the same thing will happen.

Wetting indicator pads (dog bone or tear-drop shaped) are OK if you want to use them to us – but not so popular now. They were intended to allow an x ray photo to reveal that the ball has wetted the pad by distorting its shape.

8. What works best - printed paste or flux only?

Two main methods for fitting BGAs are in use:

Printed Solder Paste

The main method is to print paste to all BGA pads along with all the usual SMT parts, the device is placed onto the paste and reflowed with the rest of the parts.

Having solder paste is said to take up minor co planarity errors if the device or PCB is warped although this sis debatable. This method is fine for machine or vision assisted placing as any smears of the printed solder paste can lead to short circuits just where they are least wanted.

The real risk of using printed paste is that if it is too much – or gets smeared – a solder ball can become large enough to touch its neighbour and form a short that is impossible to remove – or see unless it is on the outer edge or you have x ray facilities.

Flux only method.

This is approved by Motorola (the inventor of the package) and is the method usually used if reworking a package onto an otherwise populated board. IF a board is already loaded with parts it is not usually possible to re-paste the BGA pads – although micro stencils are made for this purpose. Instead flux is applied to the pads or the BGA balls themselves and the joint is made during reflow by the solder from the ball flowing onto the pad. Some people think that the lack of solder paste may increase the likelihood of an open is the device or board is warped but in practice we have never had one in over 2000 parts.

A characteristic of this method is that the package will sit a little lower on the PCB, as the solder ball has not been increased in volume by the printed paste.

9. Thoughts on BGA soldering.

After much analysis – I decided that what worried people most about BGAs was not alignment – that turned out to be easier than everyone first thought. No, it was "has the flipping thing soldered?" You can't have a quick squint under the microscope so how do I really know that the balls right underneath have gone?

SOLDERING METHODS

So what methods are in use – and why are people uncertain?

Most people in the SMT world had switched to convection (hot air) ovens long before BGAs arrived. Unfortunately if they had stuck with the first generation infrared systems they'd have been better off.

Problem is a BGA has all its connections underneath – the BGA body to PCB gap is a millimetre or so. Now you have to get even heating right under the BGA – just one cold spot means a defective joint. Hot air – even if turbulent just is not going to penetrate that well into such a narrow gap. So what do solderers do? They either increase the temperature to ensure that every part of the device is hot enough or increase the heating time so as to allow the package time to heat through by conduction. Hence why IR is better – it heats the package rather than tries to blow heated air under a narrow opening.

However, there is a third heating method – the one we use. It is guaranteed to heat every part of every device evenly, it is impossible to overheat from its specified temperature, it completely surrounds the job in an oxygen free, totally inert environment which helps the flux do its job better still. What is this heating method: it is now called "condensation reflow" although many old hands at SMT know it as Vapour Phase soldering.

A quick description is that the process uses a special chemical (basically a fluorocarbon) that boils at a known temperature – we use a 215-degree BP. The boards to be soldered are placed in a chamber in the bottom of which is a sump of this fluid, which is heated. AS it heats up it produces steam – which just like it does in your kitchen condenses on any surface cooler than itself. As it condenses it give sup its heat to the cooler item. Steadily the cooler PCB gets hotter – until eventually (having passed the solder melting point) it reached the same temperature of the steam. At that point no more steam can condense – a special heat probe detects this point and shuts off the heat source. The liquid stops boiling, the PCB can be removed from the steam chamber to cool. The vapour blanket is totally inert and heavier than air so all oxygen is displaced from the joints – so the flux only has to clean the joint not also cope with the oxidation occurring during normal reflow in air. Less active fluxes can be used.

Also the vapour penetrates everywhere – around tall objects, down between things and crucially for us – under BGAs. The whole board and parts are evenly heated, all around and from both sides. In a conventional linear oven the hot front moves along the board so that at any time there is a wide variation in temperatures across the board – this can lead to distortion.

Condensation reflow is not without its critics (who often seem to make rival technology ovens surprisingly). The first generation VP systems plunged a board at ambient temp into a steam blanket – imagine walking into a Turkish baths – instant thermal shock. The current generation machines like ours start with the fluid at ambient too – it gently heats up and the temperature rises slowly – our system shows a rise of below 2 degrees a second – considered ideal by the component manufacturers.

END.