DVD-96C
Ball Grid Array Rework

Below is a copy of the narration for DVD-96C. The contents for this script were developed by a review group of industry experts and were based on the best available knowledge at the time of development. The narration may be helpful for translation and technical reference.

Introduction

This program will detail the procedures required for the rework of ball grid array components, also known as BGAs. BGAs differ from other types of electronic components, such as PGAs, PLCCs and QFPs, in that they don't attach to the circuit board with leads. Instead, they use row upon row of tiny metal balls or columns which are soldered to a matching set of lands on the board. These rows make up a "grid" of connection points.

BGAs come in a variety of configurations, usually square and rectangular. The materials are plastic, called PBGAs; ceramic, called CBGAs; and metal tape, called TBGAs. BGAs offer a number of advantages over leaded packages such as Quad Flat Packs, or QFPs. Because of their high density and high I/O count, they take up less board space than QFPs. They are also easier to assemble and have better electrical and thermal performance.

BGAs are more robust than leaded components such as QFPs. This means that lead coplanarity concerns are minimal. Also, most BGAs are self centering or self correcting during reflow and have higher first pass yields than QFPs.

Disadvantages include the inability to visually inspect BGA solder joints and the necessity of utilizing alternate inspection and testing procedures. In addition, PBGAs and TBGAs are generally moisture sensitive like other plastic components, and ball replacement is difficult on an otherwise good device.

The most important concern explained in this video is that conventional rework techniques may no longer be applicable. Before we go into the actual BGA rework procedure, let's examine some of the rework issues so you'll have a better understanding of what's involved.

PBGAs, like other plastic components, are susceptible to absorbing moisture from the environment. This can cause a catastrophic mechanical failure called the "popcorning effect."

Popcorning describes the abrupt vaporization, or eruption of moisture trapped in a package or component during a reflow soldering process. Solutions to the moisture problem include baking, or storing components in packages or containers with desiccant or nitrogen.

Another rework issue is solder paste printing. Most BGA solder defects are traceable to printing. Correct solder paste volumes are essential. Excessive solder volume increases the height of the solder fillet. This can produce geometries with stress concentration or bridging. Too much solder increases the center of gravity, thereby increasing chances for cracking. Insufficient solder volumes can produce weaker, low reliability solder connections.

Rework temperature profiles are critical and must be developed and carefully followed for BGA removal and replacement. To develop temperature profiles, the circuit board should be monitored at critical locations. This includes sides, internal die and adjacent components.

In terms of preheating, local underside spot preheating can cause multilayer PWB warpage. Large wide area preheaters are the most effective. Typically, BGA boards should be preheated to 200-250 degrees Fahrenheit. The specific time for preheating will depend on variables such as the type of BGA and circuit board. Also, the PWB may need to be supported to avoid sag and warp.

There is also a concern for adjacent component heating during the rework operation. Venting air away from adjacent components will reduce this potential problem. Another option is shielding the component.

Finally, some PBGA and CBGA solder balls may be removed with the package and the balance remains on site. Reballing techniques for PBGAs must be developed to save good devices.

Removal

Now let's take a look at the two removal processes for BGAs -- hot gas and conduction. For the purpose of this video we'll use the PBGA which is the most common.

We'll examine the hot gas, or convective process first. If desired, apply flux to the perimeter, or inject it beneath the BGA. This can assist in heat transfer. The flux should be compatible with the flux used during the assembly process. Next, mount the circuit board on the rework station. Align the nozzle with the BGA to be removed. Then preheat the board to the desired temperature.

Using a predefined temperature profile, apply the heat to the top side of the component. Allow the temperature to safely ramp up to the set point where the BGA will reflow. Once reflow is achieved, the device can be lifted off with a vacuum pick-up.

Now that we've seen the proper removal of a BGA, let's look at what could go wrong during the removal operation. If there is not enough heat, the solder won't reflow and the BGA can't be removed. This can occur if you have an improper temperature profile. Excessive mechanical stress, thermal exposure or convection pressure can cause damage to the BGA or the circuit board. For example, excessive mechanical stress can warp the board. If the nozzle is too far away from the BGA, it can adversely affect reflow times or the heating of adjacent components.

Let's look at the proper hot gas BGA removal procedure one more time using a different machine.

The second method of BGA removal uses conductive heat. First, set the temperature of the dual thermal vacuum pick handpiece and install the appropriate tip and vacuum cup. Tip size is based on the size of the BGA.

Lower the handpiece until the tip is aligned on the BGA. After the predetermined profile time which results in complete solder melt, activate the vacuum and remove the BGA.

Site Preparation

In this section we'll discuss the site preparation prior to component replacement. Before we start, let's briefly take a look at a typical procedure that can be used when the BGA requires reballing.

We begin by removing the old solder from the BGA with a solder extractor. Next, set the temperature, timer and internal blower to the settings specified in your system manual. Apply flux to the BGA lands. Place the solder ball carrier on the BGA and secure with the frame. Place the BGA on a heat resistant flat surface making sure the solder balls are facing up.

Align the BGA to the nozzle. Adjust the nozzle until it is one millimeter above the BGA. Start the reflow cycle. When the reflow cycle is complete, lift the nozzle and allow the solder to solidify. Then let the BGA cool.

Next, remove the frame. Apply water to the top of the solder ball carrier and soak for about thirty seconds. The water will loosen and dissolve the water soluble paper. Then remove the paper, and clean and bake the BGA component if necessary. The baking is done to drive out moisture.

Now let's prepare the site on the circuit board. The first step is to remove residual solder from the lands. This can be accomplished using a continuous vacuum extractor or a soldering iron and solder braid.

It is important to be very careful and gentle to avoid damage to the lands or the solder mask. Also, make sure the solder doesn't go into the circuit board holes so there is no bridging of the solder joints.

One technique is to come straight up and down with the solder braid rather than moving it across the lands. The moving across action can cause damage to the lands. Also, it is recommended to remove the iron and the braid at the same time to prevent lifted lands.

Next, if required, clean the site with an approved solvent and wipe dry with a clean cloth. When the site preparation is complete, the lands should be free of all residual solder and have a thin, level and uniform coating of solder on all the lands.

Next, we'll look at solder paste requirements. It may not be necessary to add solder paste if the new BGA has been manufactured with eutectic solder balls. This is because the solder balls will meet to form the fillets.

If solder paste is required, the application can be accomplished with a mini-stencil or a pre-programmed paste dispenser.

When using a mini-stencil, first align and position the mini-stencil on the BGA site. Next, apply the solder paste. Use a mini squeegee to print the paste on the site. Then lift the stencil off the site and inspect for paste uniformity. Make sure you check the requirements for specific stencil and paste characteristics based upon whether you are reworking a PBGA, CBGA or TBGA.

The second option is to dispense the solder paste with a preprogrammed dispenser. This is normally done with a syringe using a needle with an open diameter.

Replacement

Now we'll take a look at replacing the BGA. We begin by making sure that solder paste has been printed properly onto the site, or that the BGA has eutectic solder balls.

The first step is component alignment which can be accomplished manually or with a vision system. The manual method aligns the nozzle with the work site of the PWB. This is done with a template. The template is placed around the BGA site and the nozzle is aligned to it. The template is then removed and the BGA is inserted into the nozzle. If eutectic solder balls are used, flux is applied to the land areas since no solder paste has been added.

When a vision system is used, the BGA is first inserted into the nozzle. The vision system utilizes a beam splitter to look at the BGA and the circuit board and superimpose the two images. The circuit board is moved until the images line up. PBGAs will self center during reflow and do not require precise alignment. However, if the PGBA is too far out of alignment, it will increase chances for defects such as bridges and opens. After alignment is achieved, lower the BGA nozzle to the circuit board and begin the reflow cycle. Again, a pre-defined replacement preheat and reflow profile is utilized. After the reflow cycle is complete, allow the BGA to cool before removing the circuit board.

Now let's take a look at what can go wrong during this process. Excessive heat can cause BGA components to totally collapse and cause shorts. It can also cause board delamination. Insufficient heat can cause opens, shorts, cold solder joints, and insufficient wetting. Nozzle alignment can affect reflow time. Too much gas or mechanical pressure can cause damage to the BGA or the circuit board.

Inspection

This last section examines inspection requirements and takes a look at possible solder defects. Unlike other surface mount components such as QFPs, 100 percent visual inspection of BGAs is not possible.

A limited type of visual inspection can be done which requires inspection optics. This inspection is subjective in nature and is limited to the outer perimeter balls. In this manner, external wetting can be verified as well as proper component alignment. But it is not possible to determine the acceptability of the hidden solder joints.

These hidden solder joints require X-ray equipment. There are two methods of X-ray -- direct transmission and laminography.

The direct transmission method can detect bridges, excessive solder, missing balls, insufficient solder and misaligned components. However, only limited wetting data can be obtained.

Another X-ray method is laminography. In laminography the BGA can be moved through the beam so that different areas can be viewed. The result is a more graphical representation than direct transmission.

Now, let's take a look at the defects that can be detected with x-rays. Misalignment occurs when the balls of the BGA are not properly lined up with the lands on the circuit board.

Another gross defect occurs when there is only partial or no reflow causing the solder joints to be incomplete. X-rays can also detect missing balls, voids, opens and shorts.

The key is correlating the inspection results with defects and processes. The critical processes include solder paste application and reflow. Monitoring the solder volume and the reflow temperature will go a long way in minimizing BGA defects.

This program has described the general procedures required for BGA rework. You've been introduced to the characteristics and rework issues of BGAs; seen the sequences for performing rework including component removal, site preparation, and component replacement; and examined the inspection requirements.

The quality of every soldering operation can affect the functioning of an entire electronic system. Careful attention to detail will help ensure that your company's products work properly and reliably.